

GRAPHICAL MACHINING PROCESS SIMULATOR

%1245

N1G0G54T03S250M1

N2G1X245.54Y45.87F030

N3X-34.768Y234Z-50

N4X-68.678Y234Z-55.534

N5X-77.879Y54.985Z-45.1

N6X-88.547Y66.786Z-7.865

N7X-94.856Y88.876Z-0.241

N8X-112.472Y89.534Z4.874



CONTENTS

INTRODUCTION	1-5
About the program	1-5
Hardware requirements	1-5
Package contents	1-5
Program installation and running	1-6
Files used by NCTuner	1-6
1 QUICK TOUR	1-7
2 PROGRAM DESCRIPTION	1-9
2.1 MAIN WINDOW	1-9
2.1.1 Main menu	1-9
2.1.2 Tool panels (Main panel)	1-11
2.1.3 System setup window	1-15
2.1.4 NC program loading	1-15
2.1.5 Editing tuning file to CNC system	1-16
2.1.6 Workpiece parameters	1-20
2.1.7 Current operation tool	1-21
2.1.8 NC program text window	1-22
2.1.9 NC program manager window	1-25
2.1.10 Showing the current values of registers	1-27
2.1.11 Selection of registers to be tracked	1-28
2.2 STREAM EDITOR	1-31
2.2.1 Create/finish file macro	1-31
2.2.2 Execute file macro	1-31
2.2.3 Stream editor functions	1-31

INTRODUCTION

About the program

NCTuner is a 32-bit application for Windows 95/98/NT.

Program's purpose – control and final accomplishment of NC programs for 2 - 3 axis milling machines with CNC. **NCTuner** offers a full range of tools for realistic simulation of machining process, tuning and editing.

Program **NCTuner** allows:

- simulate machining process,
- visually control machining quality,
- obtain a photo realistic image of the machined part,
- eliminate errors in the NC program,
- make all necessary changes to the NC program's text.

The initial data for machining process simulation is a NC program in the format of any CNC system, tuning file to the corresponding CNC system, parameters of the tool and shape of the workpiece. A being created model of the machined detail possesses of the high degree of authenticity and allows you to visually grade the quality of machining and detect possible errors.

The environment of NC program debugging allows you to easily find fragments of trajectory in the text of the NC program and the block of the program, which corresponds to the section of the trajectory. There are means for "Block by Block" debugging of a NC program.

For NC program's text editing, all functions of the modern multi-window text editor are available. When modifying a NC program, all related corrections are automatically added to the displayed trajectory of tool movements. The editor contains a set of specialized functions for NC program editing.

Hardware requirements

- Recommended amount of RAM – not less than 32 Mb. When RAM increases, computer's performance grows.
- Display resolution – 800x600 or higher; not less than 256 colors.
- Processor – Pentium 166 or higher.
- Windows 95/98/NT operating system.

Package contents

Standard **NCTuner** package contains the following items:

1. CD-ROM with the program.
2. Readers manual.
3. Electronic key preventing unauthorized copying.
4. License agreement.
5. Package box.

Program installation and running

To install the program to the computer, one should fulfill the following actions:

1. Insert the CD-ROM into the CD drive.
2. Being in Windows 95/98/NT operating system, run the **SETUP.EXE** program from the CD-ROM. This can be done for example using the "**Start**" menu, "**Run**" command. In the appeared dialogue box type the name of the required program (D:\SETUP.EXE in this case) in the field "**Open**", and click the "**Ok**" button or press "**Enter**".
3. During installation, different windows will be showing up on the screen. The following data should be typed in:
 - Name and title of the user's organization;
 - Folder where to install the program (By default – C:\NCTuner);
 - Choose **Typical**, **Compact** or **Custom** installation (differ by the availability of system project examples and IGES-files);
 - Folder where program icons for executable files will be located to (By default – NCTuner).

To go on to the next step of installation it is necessary to click the "**Next**" button in the dialogues window.

4. After installation process is completed, a notification about the setup accomplishment will show up. The window should be closed by clicking the "**Done**" button.

Before running the program it is necessary to insert the electronic protection key (computer at that moment must be switched off) into the parallel port of the computer (a printer is usually connected to the parallel port). If a printer or any other device is connected to the parallel port, then its interface cable must be reconnected to the port through the empty slot of the electronic key. Presence of the electronic key does not influence the functional capabilities of the computer and neither its interaction with a printer or other device, connected to the parallel port.

To run the program choose <**Start**> <**Programs**> <**NCTuner**> menu options. Further, for one's convenience, it is recommended to put **NCTuner's** icon to the Desktop.

Note:

If when running the program it displays the message:

Electronic key not found!

it is recommended to check the correctness of connection of the electronic protection key to the parallel port.

Files used by NCTuner

- NCTuner.exe – executable module of the program.
- *.nct – NCTuner's project files.
- *.ppp – postprocessor tuning files to different CNC systems.
- *.stc – SprutCAM's project files.
- *.mcd –related to SprutCAM files of technological commands.

1 QUICK TOUR

To get to know the program fast, it is enough to perform the following typical sequence of actions:

1. Load NC program. To do this, click the  button on the main panel. In the appeared window choose a NC program file and select a tuning file for it (in the lower part of the window). If necessary, change the folder to that where tuning files to CNC systems are kept, by pressing the  button in the lower part of the window.

2. If NC program is interpreted incorrectly, then this is because that either the tuning file to CNC system is improper for that NC program (to change the tuning file choose menu options **<File> <Change postprocessor>**), or the tuning file contains incorrect data (to correct it, choose options **<File> <Tune postprocessor>** of the main menu).
3. Choose convenient parameters of objects visualization in the graphical window, using the buttons on the visualization control panel



4. Open the NC program manager window by clicking the  button on the main panel and select the tool for every operation of the of the NC program. To do so, it is necessary to select by mouse or keyboard the required operation from the list, and press the  button. In the appeared window parameters of the tool for the selected operation are set up.

5. If necessary, make changes to the NC program and edit its text in the corresponding window. At that, trajectories of tool movements will be recalculated automatically.

6. Define parameters of the workpiece in the window, which opens by clicking the  button.

7. Select necessary simulation parameters. For instance; switch on visualization of the workpiece and the tool by pressing the  button.

and  buttons.

8. Run simulation by pressing the buttons on the simulation player panel . Simulation will be interrupted at break points, which can be located by pressing the

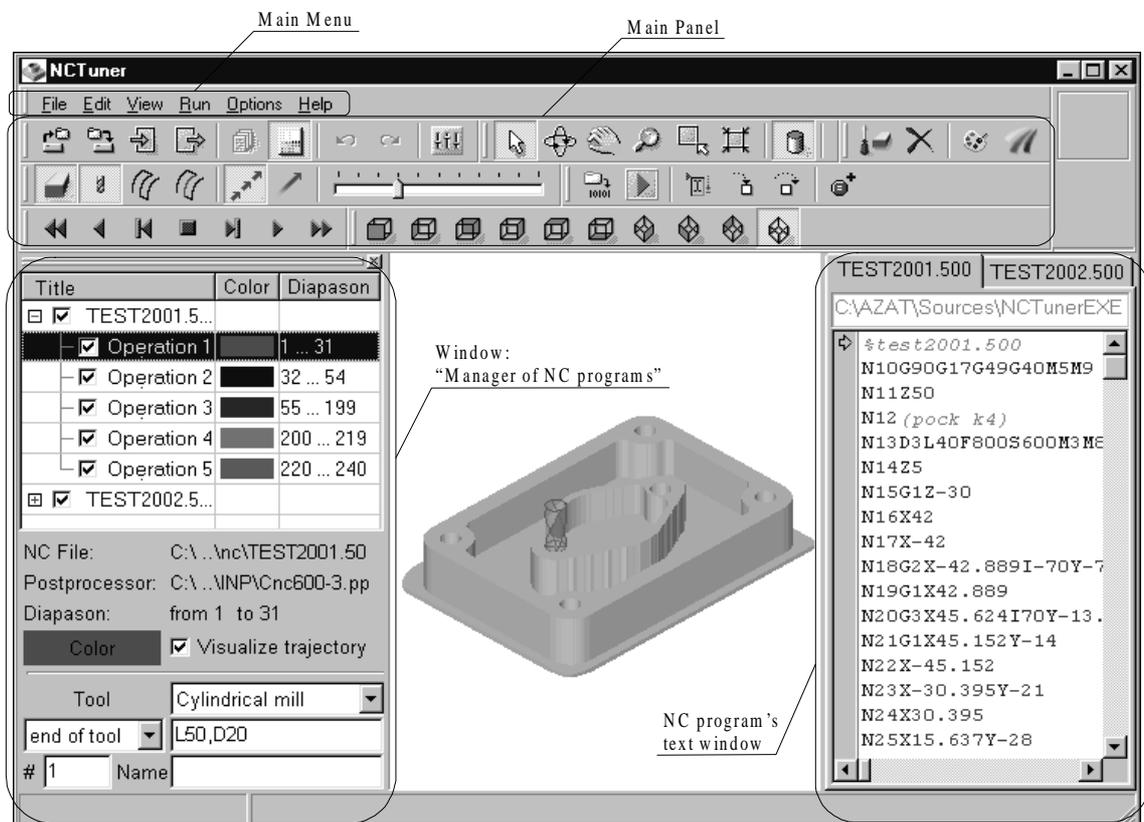
 button, or clicking the left mouse button in the space left to the NC program text. If the simulation results do not satisfy one's requirements, then all additional modifications can be added to the NC program again (refer to the step 5).

9. Save modified NC program by selecting the **<File> <Save NC>** options of the main menu.

2 PROGRAM DESCRIPTION

2.1 MAIN WINDOW

Main window of the program has the following view:



In the upper part of the window located main menu and tool panels (Main panel). In the center – graphical window of the system. Windows of the NC program manager and NC program text can be located in separate windows, or parked on the left, right or below from the main graphical window.

Visibility of the tool panels on the main panel and visibility of the separate windows can be controlled by the sub-options of the <View> menu option of the main menu. Alteration of the instrumental panels location, size and location of different windows is also possible.

2.1.1 Main menu

- <File>:
 - <New> – creates a new project, at that the current closes automatically.
 - <Open> – loads a project. The option is duplicated on the “Main panel”.
 - <Reopen> – loads a project from the list of the earlier loaded.
 - <Save> – saves the project under the current name. If the project has not been saved, a new file name will be requested. The function can be called from the “Main panel”.

- **<Save As>** – saves the project under a new name.
- **<Open NC>** – opens a NC program. The option is duplicated on the “Main panel”.
- **<Save NC>** – saves the NC program. The function can be called from the “Main panel”. If the NC program has not been saved, a new file name will be requested.
- **<Save NC As>** – saves the NC program under a new name.
- **<Close NC>** – closes the NC program.
- **<Tune Postprocessor>** – calls the “Postprocessor tuning window”. After modification of postprocessor tunings the trajectory of all loaded programs, referring to that postprocessor tuning file will be automatically recalculated.
- **<Change Postprocessor>** – allows you to change postprocessor tuning file for interpretation of the current NC program. The trajectory of NC program recalculates automatically.
- **<Exit>** – quits the **NCTuner** program. If the opened file has not been saved, then you will be offered to make a selection; either to save the project or quit without saving.
- **<Edit>**:
 - **<Undo>** – abolishes the previous action in the current NC program text.
 - **<Redo>** – repeats the earlier abolished modification in the current NC program text.
 - **<Cut>** – deletes the selected objects and puts them into the clipboard.
 - **<Copy>** – copies the selected objects into the clipboard.
 - **<Paste>** – inserts contents of the clipboard after the cursor.
 - **<Delete>** – deletes the selected block of text of the current NC program.
 - **<Select All>** – selects all text of the current NC program.
 - **<Find>** – activates the text search window.
 - **<Replace>** – activates the text replacement window.
 - **<Find Next>** – repeats search or replacement with the current options.
- **<View>**:
 - **<NC Program Manager>** – shows/hides any of the earlier loaded NC programs. When closing/opening the window of NC program text, corresponding trajectories of tool movements will be automatically shown/hidden in the graphical window.
 - **<Stream Editor Window>** – opens stream editor window. Unavailable if no NC programs is opened.
 - **<Registers' Values>** – opens window which displays the current values of registers. Not available without NC program.
 - **<Tool panels>** – activates tool panels (accordingly):
 - **File**
 - **Visualization Control**
 - **View Vectors**
 - **Workpiece Parameters**
 - **Simulation Mode**
 - **Simulation Player**
 - **NC Debugging**

- **<Run>**:
 - **<Compile>** – compiles the current NC program (recomputes tool movement trajectories, feedrates, and etc.). The menu option is duplicated on the main panel.
 - **<Run NC>** – simulation of machining for the current NC program. The menu option is duplicated on the main panel. The menu option is duplicated on the main panel.
 - **<Run to Cursor>** – runs simulation of NC program from the current position to the position of the cursor.
 - **<Run One Block>** – runs the current block with subprograms. The menu option is duplicated on the main panel.
 - **<Run Block>** – runs the current block without subprograms. The menu option is duplicated on the main panel.
 - **<Toggle/Delete Break Point>** – toggles/deletes break point in the current position of the cursor. Simulation of machining will be stopped after execution of the block, marked by a break point. The menu option is duplicated on the main panel.
- **<Options>**:
 - **<Edit System Setup>** – allows you to alter system setting. The function can be also called from the main panel.
 - **<Save System Setup>** – saves the current system settings in the configuration file (NCTuner.cfg).
 - **<Load System Setup>** – allows restoration of the system setting from the configuration file (NCTuner.cfg).
- **<Help>**:
 - **<Contents>** – displays help contents.
 - **<Help>** – activates help window.
 - **<SPRUT on the WEB>** – loads WEB-page of JSC “SPRUT-Technology”. URL address is: <http://www.sprut.ru>.
 - **<E-Mail to SPRUT>** – preparation of an e-mail to JSC “SPRUT-Technology”. E-Mail: st@sprut.bmstu.ru.
 - **<About>** – information about the system.

2.1.2 Tool panels (Main panel)

Tool panels are intended for the quick access to the main functions of file management, parameters and visual properties of a model. Tool panels are divided into several functional groups, and can be located on the main panel in an arbitral order.

The following tool panels are available in the **NCTuner** program:

■ Tool panel “File”

Tool panel File is purposed for the quick access to the basic functions of file management. Buttons duplicate some options of the main menu.



create new project.



open an existing project.



save the current project.

	create a new NC program.
	load a NC program from the existing NC programs list.
	save the current NC program.
	close the current NC program.
	open "NC program manager" window.
	undo previous action.
	redo previous cancelled action.
	open "System setup" window.

■ Tool panel "Visualization control"

The biggest part of the window occupies the "Graphical window". Geometrical model, trajectory of tool movements and objects of planar geometry are displayed in there. Management of the graphical output is performed by buttons located on the tool panel.

- After pressing the  button, selection of separate blocks of NC program will be possible right from the screen. When moving mouse pointer within the graphical window, closest to the pointer section of the trajectory of tool movements will be highlighted. For selection it is necessary to point at the required element, and press the left mouse button. At that, the cursor will be transferred to the corresponding block of NC program in the NC program text window.
- After pressing the  button, the window sets to the view vector modification mode. To rotate an object, it is necessary to press the left mouse button within the space of the graphical window and move the mouse holding that button pressed. If moving the mouse horizontally, the objects will be rotated round the horizontal axis. If moving the mouse vertically, the objects will be rotated round the vertical axis.

Selection of the view vector is also possible from the "View vectors" tool panel.

- After pressing the  button, the graphical window sets to the dynamical image displacement mode. To move objects, one should press the left mouse button within the space of the graphical window, and move the pointer holding the mouse button pressed.
- After pressing the  button, the graphical window sets to the dynamical scale modification. To change the scale, one should press the left mouse button within the space of the graphical window and move the pointer holding the mouse button pressed. When moving the pointer upwards, the image will be zoomed up; downwards – zoomed down. Moving the mouse horizontally shall not affect the current scale of the image.

- After pressing the  button, the graphical window sets to the “Selection of an area to zoom” mode. To select a square, area one should press the left mouse button within the space of the graphical window and move the pointer holding the mouse button pressed. After the button is released, the image within that area will be zoomed up to the whole graphical window, and the area selection mode will be finished.
- After pressing the  button, the system will automatically select such a scale, at which all geometrical objects are visible in the graphical window
- Button  allows switching on/off the shade mode

■ Tool panel “View Vectors”

Standard view vectors of the graphical window can be selected by pressing the buttons on “View vectors” tool panel



. Unlike the “Visualization control” buttons, all the buttons of the “View Vectors” tool panel have the predefined orientation of view vectors (accordingly XY, ZY, -XY, -ZY, X-Z, XZ, -YXZ, -XZY, YZX, XYZ).

■ Tool panel “Workpiece Parameters”

The following buttons are located on the “Workpiece Parameters” tool panel.



open the workpiece parameters definition window.



cancel simulation results.



determination of the workpiece color.



If the button is pressed, then tool tracks on a workpiece will be marked by different colors; if released, then a monochromatic machined detail is displayed.

■ Tool panel “Simulation mode”

Simulation Mode tool panel is intended for managing of the graphical output of different elements during the simulation mode.



show/hide the being machined detail.



show/hide the tool in the current position.



show/hide the trajectory of tool's movements.



switch on/off highlighting of the trajectory section, which corresponds to the block of a NC program, the cursor is currently on. (Current block highlighting mode)



If the button is pressed, then the graphical output renews after accomplishment of every block. (Block by Block imaging).



If the button is pressed, then the graphical output renews after the tool have passed a section of trajectory of the defined length (smooth movement), the length of trajectory sections and consequently the visual speed of the tool are defined by the displacement of the roller in the



space. (Smooth imaging).

If the button of “Block by Block imaging” and the button of “Smooth imaging” are released, then the image renews after accomplishment of the defined action of simulation, or after breaking the simulation at the break point

Imaging mode can be changed over during the simulation process.

■ Tool panel “Simulation player”



return to the beginning of the NC program.



return to the previous tool change point.



return back on one block of a NC program.



stop simulation.



go ahead on one block of a NC program.



go to the next tool change point.



go to the last block of a NC program.

Pressing any of the buttons of the simulation player cancels the being executed at that moment simulation command.

Execution of simulation cancels at the defined by a user break points.

■ Tool panel “NC debugging”

The following buttons are located on the “NC debugging” tool panel:



recompiles the current NC program (accordingly; recomputes toll movement trajectories, feedrates and etc.).



simulates the current operation of a NC program.



runs simulation of a NC program from the current position to the position of the cursor.



runs the current block with subprograms.



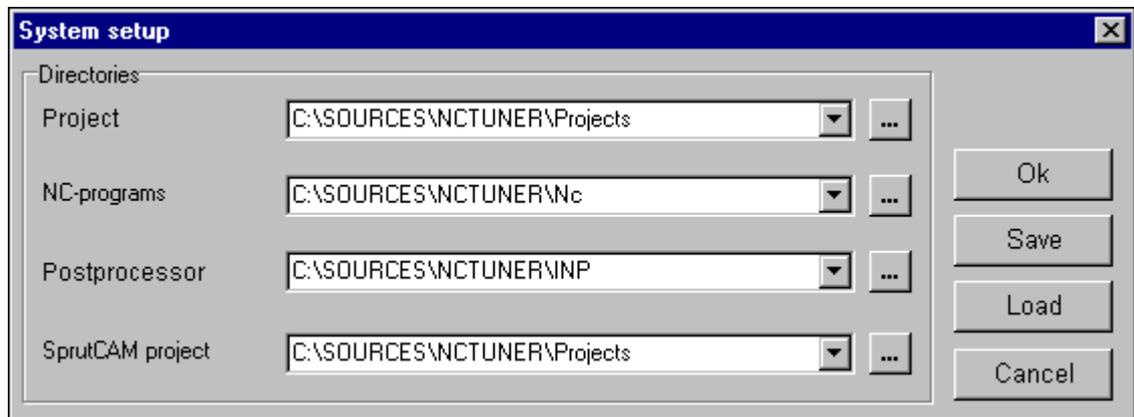
runs the current block without subprograms.



toggles/deletes break point, where program simulation will be stopped.

2.1.3 System setup window

System setup window activates by pressing the  button on the main panel or by selecting the <Options> <Edit system setup> options of the main window. Paths by default for folders which store system files



can be selected in this window.

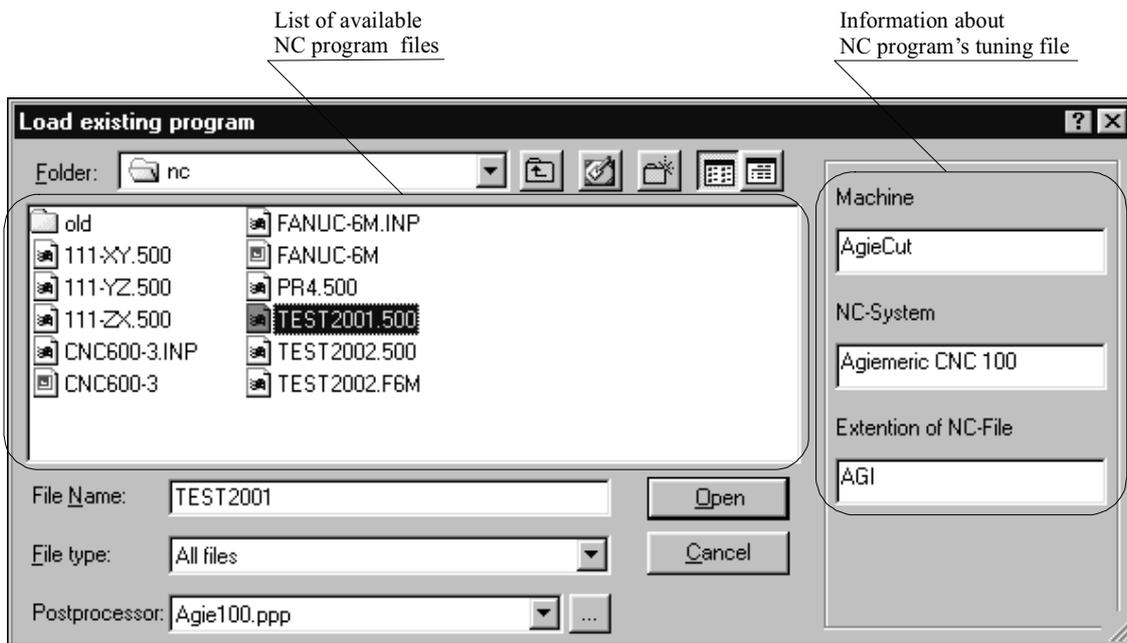
- In field “**Projects**” determine folders to be opened by default then loading saved projects of the **NCTuner** program (*.nct files).
- In field “**NC programs**” determine folders to be opened by default then loading NC program files.
- In field “**Postprocessor**” determine folders to be opened by default then loading postprocessor tuning files to different CNC systems (*.PPP files).
- In field “**SprutCAM project**” determine folders to be opened by default then loading SprutCAM project files (*.STC). Combined with the opened project files of technological commands (*.MCD) are loaded in accordance with the paths, defined in this project.

Paths can be corrected both manually and with the help of the path selection dialogues window, which activates by pressing the  button.

- Button <Ok> – closes the window and applies all settings to the current work sñance.
- Button <Save> – closes the window and saves all changes. New settings will be saved to the disc and will be valid not only for the current work sñance, but for the all following.
- Button <Load> – load settings earlier saved to the disc.
- Button <Cancel> – closes the window and abolishes all modifications made.

2.1.4 NC program loading

NC program load window can be opened by either pressing the  button on the main panel or by selecting the <File> <Open NC> options of the main menu. The view of the window is shown below:



In the left part of the window displayed the information about the name of the milling machine, CNC system, and the extension for the selected NC program file. (Data cannot be edited)

The required NC program can be selected from the list of available NC programs. For the correct interpretation of the NC program it is necessary to select the corresponding postprocessor tuning file from list of available (section Postprocessor). All tuning files, located in the defined folder are available in the postprocessor tuning files list (the folder by default sets in the System setup window). The folder with the postprocessor tuning files can be changed in the dialogues window, which opens by pressing the



button.

After pressing the <Open> button, the selected NC program will be loaded to the **NCTuner** program and compiled by the defined postprocessor tuning file. Pressing the <Cancel> button will lead to the closing of the window without loading of the selected files.

Notes:

Incorrect selection of a postprocessor tuning file may lead to the incorrect displaying of the tool movement trajectory

In order to the system be able to open folders containing postprocessor tuning files and NC program files automatically, one should define this folders in the corresponding fields of the system setup window.

2.1.5 Editing tuning file to CNC system

The window of editing data about a milling machine and a CNC system can be activated either from <File> <**Tune postprocessor**> options of the main menu, or from the pop-up menu of the NC program window.

Switching over pages of the window can be performed by selection of the corresponding tab sheet.

Upon closing the window by the <Save> button, all changes made will be saved to the current postprocessor tuning file, whereas upon closing the window by the <Save as> button, all changes will be saved to a new postprocessor tuning file. If the window is closed by the <Cancel> button, all changes are ignored.

■ Information about the milling machine

In the upper part of the window enters the name of a milling machine, the name of a CNC system and extension of the NC program file. The milling machine and CNC system names carry only the informative function. All NC programs, generated with use of an editable tuning file, will be recorded to the files with the defined extension.

Note:

In order to avoid conflicts with some operation systems, it is recommended to enter file extensions of NC programs consisting of three characters.

Maximal transitions along main coordinates are necessary for controlling of the correctness of a NC program. Should maximal allowed transition along any of coordinate axes be exceeded, a corresponding notice will be displayed.

■ Information about the CNC system

The identifiers, which forego the transition values are defined in the "Linear transitions" spaces ("X axis identifier", "Y axis identifier", "Z axis identifier"). In most systems, these identifiers are "X", "Y", "Z" correspondingly.

Positioning functions and switchings of the linear interpolation function are defined in the spaces "Positioning" and "Linear interpolation" accordingly. In most systems, these functions are "G0" and "G1".

In space “Feedrate function (fast transitions)”, identifier of switching of a feedrate and value of the rapid feedrate (F300 for example) are entered. In space “Subprogram identifier” identifier of calling of a subprogram are defined additionally.

Tick the “Space before identifier” field if want to insert blank space before a register identifier in a NC-program block.

Ticks in the “X, Y, Z coordinates”: “Absolute”, “Incremental” fields set the allowed type of transitions in a CNC-system: absolute only, incremental only or both - absolute and incremental. If the CNC-system allows both transitions, it is necessary to specify functions of switching of these modes in the “Function” space. Most frequently used functions are: “G90” – to turn on the absolute transitions and “G91” – for incremental).

The “Circular interpolation”: “X center”, “Y center” and “Z center”, “Circle radius”, “Arc angle” fields define the identifiers, which precede corresponding values (frequently used combination is: “I”, “J”, “K”, “R”, “A” accordingly).

The “CLW”, “CCLW” spaces define the functions of switching of the circular interpolation clockwise and counterclockwise correspondingly.

The “Circle division” space allows a user to define the maximal angular size of an arc, which is allowed in a NC-program (quarters, halves or full circles).

Ticks in the “Circle center coordinates”: “Absolute”, “Incremental” fields specify the method of the circle center definition. “Absolute only”, “Incremental only” and both “Absolute and Incremental” combinations are allowed. If the system allows only “Incremental”, then the center of a circle can be defined only in the incremental coordinates. If the system allows only “Absolute”, then the center of a circle can be defined only in the absolute coordinates. If the system allows “Absolute” and “Incremental”, then the center of a circle can be defined either in the absolute or incremental coordinates, or in the absolute and incremental. If the center of a circle can be defined by the absolute and incremental coordinates, then functions of switching of modes of the circle center definition are considered the same as the functions of switching of transition modes.

Note: *If definition of the data about a CNC-system is performed after the registers list formation, the system will prompt the registers identifiers in the transitions, coordinates, radius and arc angle fields.*

■ Register properties definition

#	Name	Id
1	N	N
2	INTERP	G
3	ABS_INC	G
4	G_FUNC	G
5	PLANE	G
6	KORECV	G
7	GPos	G
8	D	D
9	DX	D
10	X	X
11	PAUSE	I
12	XC	I
13	DY	D

Register name: INTERP
 Register identifier: G
 Scale: 1
 Register width: 2
 Decimals: 0
 Decimal point: Is absent
 Sign: No
 Leading zeroes: No
 Non-significant zeroes: No
 Designation: Interpolation

Value type	Value	Explanation
<input type="radio"/> Number	0	Positioning
<input checked="" type="radio"/> Enumerable	1	Linear
	2	Circular CW

Buttons: Save, Save As, Cancel

In the register properties definition window the following can be defined:

- Register name – name, with which the command processing programs are operating;
- Register identifier – symbols, which are written to a NC program block before register's value;
- Scale – value, on which register's value is multiplied at its writing to a NC program block;
- Register width – maximal number of symbols at writing register's value to a block;
- Decimals – number of signs in the decimal part of register's value;
- Decimal point – may have several options: is absent, is present, is present anyway, comma is present; if "is present" selected, then a decimal point will be only in those cases, when register's value has a decimal part; if "comma is present" selected, then a comma will be written to block instead of a point;
- Sign – whether to write register's value to a block. Possible options: No, "-" only, "+" only, "-" and "-" always;
- Leading zeros and Non-significant zeros – whether to write zeros to a block before and after register's value;

- Register's value can be either of the **numerical** or **enumeratable** type, what can be defined by the value type. If register's value has the enumeratable type, then in the chart, you can specify the possible values for the register and the comment for every value. Before running a NC program, the registers of the numerical type have the initial value equal to zero, and the enumeratable – the first from its list of values. During NC execution, at register's values registration, depending on the type of the register, either its numerical value or explanation corresponding to the current value be displayed in the corresponding window.

The registers list is located in the left part of the register properties window.

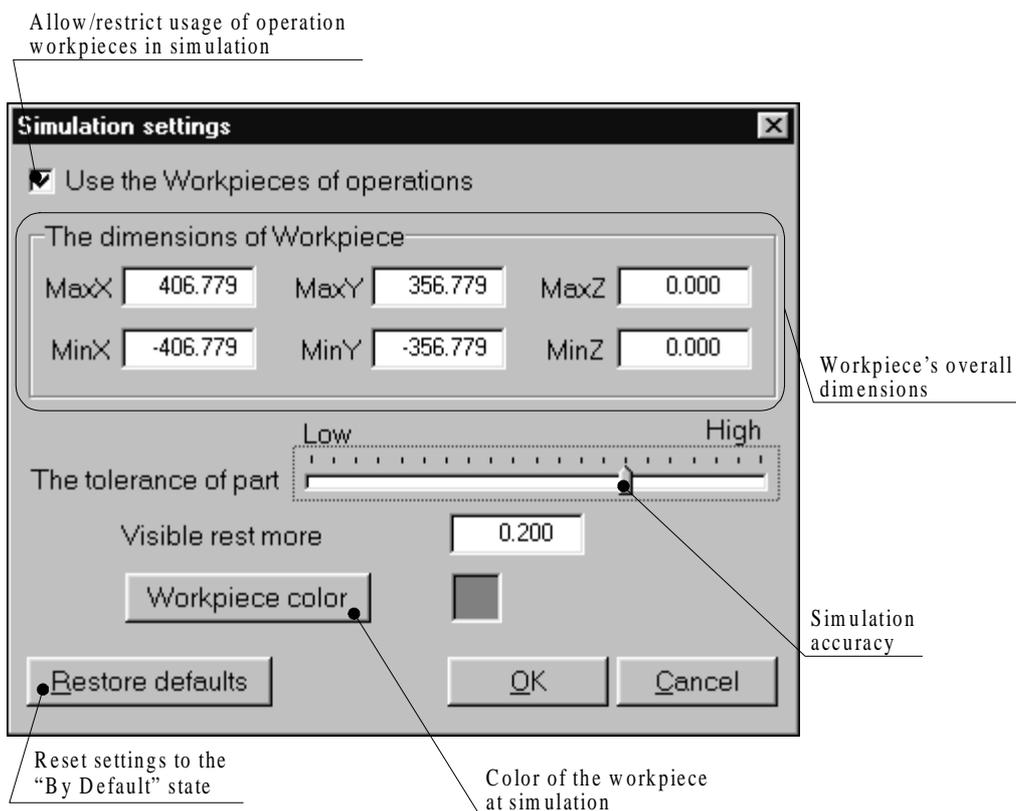
Upon clicking the right mouse button within the registers list area, a pop-up menu containing the following options will appear:

- **<Add new>** - adds a new register after the currently highlighted.
- **<Delete>** - deletes the selected register.
- **<Cut>** - transfers the selected register to the clipboard.
- **<Copy>** - copies the selected register to the clipboard.
- **<Insert>** - inserts a register from the clipboard after the currently highlighted.

To change the order of registers location, click the left mouse button upon the number of any register, then holding the button pressed, move the selected register to a new position, and then release the button.

2.1.6 Workpiece parameters

The workpiece parameters window can be opened by clicking the  button on the main panel.



The current simulation parameters are shown in this window. When simulation is started first time, the simulation parameters calculate automatically. In this case, the dimensions of a workpiece are equal to the sizes of the entire loaded geometrical model. If <Use the workpieces of operations> checkmark is ticked, then the model of workpiece is created using all profiles of workpieces in the technological process.

The <The tolerance of part> defines relative tolerance of the obtained model. A computer with high performance is required for simulation with high tolerance.

Note:

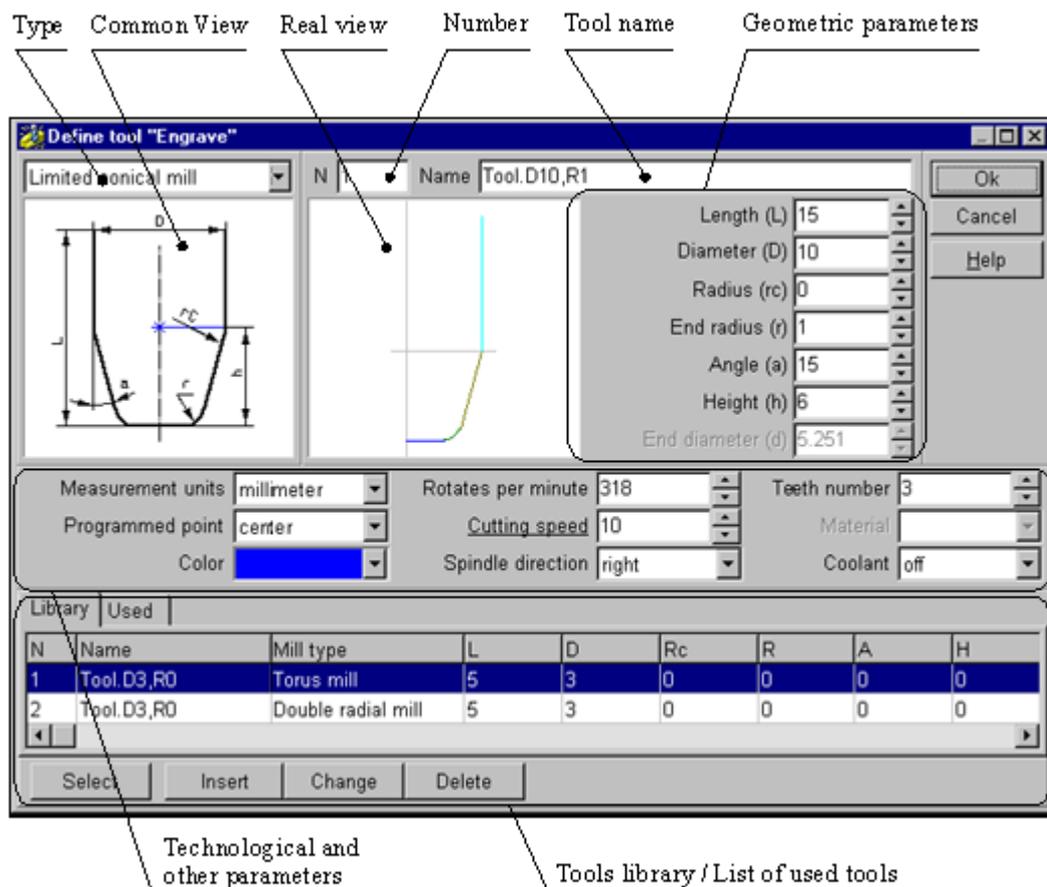
It is not recommended to set high tolerance if your computer resources are low.

Sizes of a workpiece can be defined manually; in this case, alteration of a geometrical model (its size) will not be traced. Click the <**Restore defaults**> button to restore dimensions of a corresponding geometrical model.

Color of a workpiece can be changed as well.

2.1.7 Current operation tool

This window opens by clicking the  button in the NC program manager window.



The system uses 8 types of tool that differ by its geometrical parameters:

1. Cylindrical mill: length, diameter;
2. Spherical mill: length, diameter;

3. Torus mill: length, diameter, radius of round;
4. Double radial mill: length, diameter, radius of cylinder part round, radius of end round;
5. Limited double radial mill: length, diameter, radius of cylinder part round, radius of end round, height;
6. Conical mill: length, diameter, radius of cylinder part round, radius of end round, angle;
7. Limited conical mill: length, diameter, radius of cylinder part round, radius of end round, angle, height;
8. Engraver: length, diameter, height, diameter of end;

If the value of any parameter brings a tool to the degeneration, this parameter will be highlighted red.

Common view and real view. They display the current tool. Common view shows the tool of chosen type with the principal sizes. Real view shows the tool in proportions that correspond to the current geometrical parameters and the type of the tool; only one side of tool is displayed.

Space <N> determines number of the tool in the NC program. When you need to use a tool that is already defined in the project you can choose it from the list of used tools.

Technological and other parameters:

- Measurement units of tool's parameters, millimeters and inches.
- Programmed point indicates for which point of a mill (central or end) NC program will be calculated.
- Color indicates which color will be used for painting the tool movement tracks for the current operation at its execution and color of the tool at machining simulation.
- Rotates per minute determine a frequency of rotating spindle. This parameter is recalculated when velocity of cutting or diameter is changed if before this a velocity of cutting was changed.
- Cutting speed determines the velocity of tool's transition upon a workpiece in meters per minute or feet per minute, depending on the selected measurement units. This parameter is recalculated when the rotation velocity or diameter are changed, if the rotation velocity was changed before.
- Spindle direction determine the direction of its rotating (left or right).
- Teeth number is a number of tool's teeth.
- Material is the material of the tool.
- Coolant assigns the necessity to supply the coolant when the current operation is running.

Tools library is used for keeping and quick loading beforehand determined tools. Tool number in the library does not correspond to the number in the NC program!

List of used tools similar to library of tools. With its help it is possible to choose earlier an defined tool from other operations.

2.1.8 NC program text window

NC program text window creates automatically when opening a new NC program. Several NC program text windows can be opened at one time. NC program text windows can be parked in the main system window on the left, right, or below from the graphical window. Every window contain the text of one NC program. Control over the visibility of text windows can

be performed from the main menu options **<View> <NC Program Window> <Name of NC File>**. Upon closing/opening a NC program text window, corresponding tool movement trajectories will be shown/hidden automatically.

NC program text can be edited by means of the built in text editor. When modifying the text of a NC program, tool movement trajectories will be recomputed automatically

If the “Current block highlighting mode” is on, then the section of the tool movement trajectory, being formed by the block of the NC program in the current cursor position, will be highlighted in the graphical window.

In the simulation mode, the block of a NC program, being executed at the current moment, (corresponds to the current tool position) is highlighted in the NC program text window.

Break points, at simulation of machining can be toggled/deleted either by using the **<Debug> <Toggle/Delete Break Point>** options of the pop-up menu or by clicking the left mouse button within the space, left to the NC program block, or either by pressing the **<F5>** key.

■ Pop-up menu:

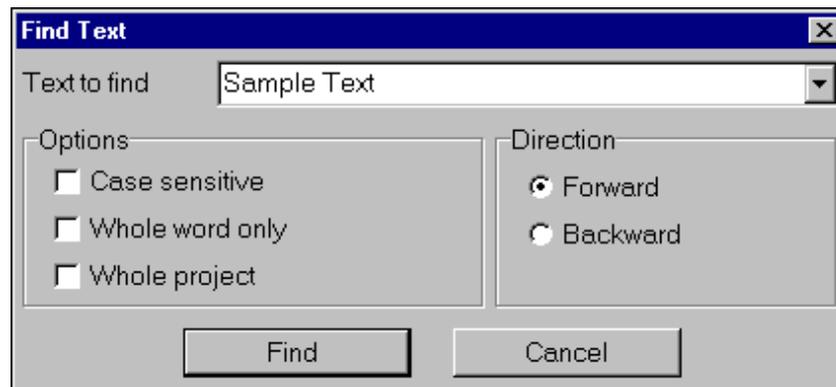
Upon clicking the right mouse button within the space of NC program text window, a pop-up menu appears. The window is intended for the quick access to the main functions of file management. The menu duplicates the main options of the main menu. The following functions are available:

- **<Undo>** – abolishes the previous action in the current NC program text.
- **<Redo>** – repeats the earlier abolished modification in the current NC program text.
- **<Cut>** – deletes the selected objects and puts them into the clipboard.
- **<Copy>** – copies the selected objects into the clipboard.
- **<Paste>** – inserts contents of the clipboard after the cursor.
- **<Toggle Bookmarks>** – toggles bookmarks in the text of NC program. To do so, it is necessary to put the cursor to the string of NC program text, select menu option **<Toggle Bookmarks>** and choose a number of the bookmark; or simultaneously press **<Ctrl+Shift+N_o of Bookmark>**. The bookmark will be put to the current cursor position. If the bookmark with such number is already exists, then bookmark’s parameters will be redefined. In order to delete bookmark, one should put the cursor to the string of that bookmark, and press **<Ctrl+Shift+N_o of Bookmark>** again.
- **<GoTo Bookmark>** – transfers the cursor from any place of NC program text to the desired bookmark. To do so, one should select **<GoTo Bookmark>** options of the pop-up menu and choose the number of the sought bookmark; or simultaneously press **<Ctrl+N_o of Bookmark>**. After that, the cursor will be transferred to the position of the chosen bookmark.
- **<Debug>** – option debugging duplicates buttons of the “Simulation player” tool panel, and also buttons “Run to Cursor” and “Toggle/Delete Break Point” of the “NC Debugging” tool panel.
- **<Save NC>** – saves the NC program. The function can be called from the “Main panel”. If the NC program has not been saved, a new file name will be requested.
- **<Save NC As>** – saves the NC program under a new name.
- **<Close NC>** – closes the NC program.

- **<Tune Postprocessor>** – calls the “Postprocessor tuning window”. After modification of postprocessor tunings the trajectory of all loaded programs, referring to that postprocessor tuning file will be automatically recalculated.
- **<Change Postprocessor>** – allows you to change postprocessor tuning file for interpretation of the current NC program. The trajectory of NC program recalculates automatically.
- **<Close Window>** – closes the current NC program text window.

■ Find text

Find text window can be activated either from **<Edit> <Find>** options of the main menu, or by simultaneous pressing “Ctrl+F” keys.



1. Enter the text to be found in space “Example”.
2. Choose search options:
 - “Case sensitive” – differentiates uppercase from lowercase when performing a search.
 - “Whole words only” – searches only the exact string, specified as the example.
 - “Whole project” – searches the example string in the entire project.
3. Choose search direction:
 - “Backward” – search from the current position to the beginning of the file.
 - “Forward” – search from the current position to the end of the file.

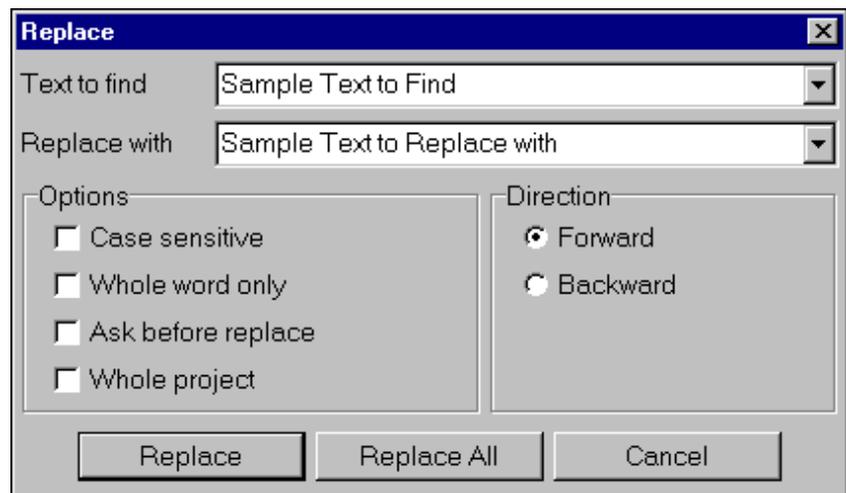
Upon clicking the **<Find>** button, search of the defined string will be started, and the window will be closed. In order to continue search with the previous options either simultaneously press “Ctrl+L” keys, or choose **<Edit> <Find Again>** items of the main menu. If the window is closed by the **<Cancel>** button, search will not be performed.

■ Replace text

Replace text window can be activated either from **<Edit> <Replace>** options of the main menu, or by simultaneous pressing “Ctrl+R” keys.

1. Enter the text to be found in space “Example”.
2. Enter the text to be replaced in space “Replace with”.
3. Choose search options:

- “Case sensitive” – differentiates uppercase from lowercase when performing a search.
 - “Whole words only” – searches and replaces only the exact strings, specified as the examples.
 - “Ask before replace” – Prompts you before replacing each occurrence of the search string. If it is disabled, the editor automatically replaces the search string.
 - “Whole project” – searches and replaces the example string in the entire project.
4. Choose search direction:
- “Backward” – search from the current position to the beginning of the file.
 - “Forward” – search from the current position to the end of the file.

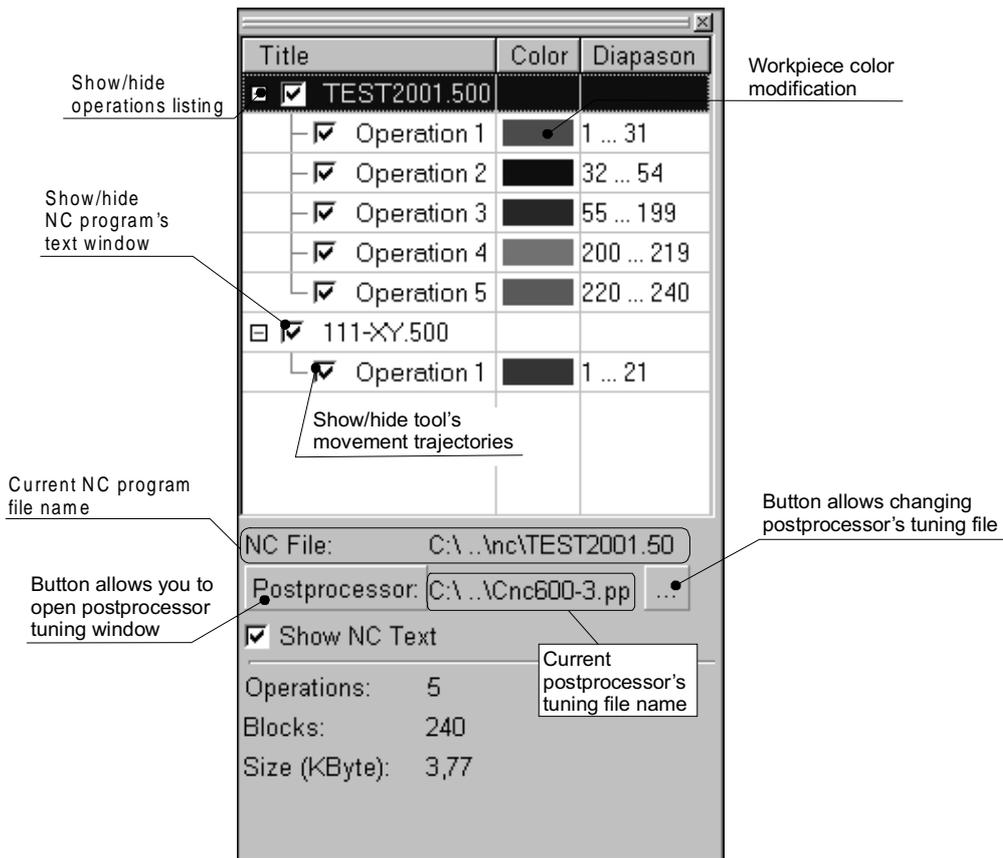


Upon clicking the <Replace> button a single text example will be replaced. Upon clicking the <Replace All> button, all found fragments matching the “replace text example” will be replaced. If the window is closed by the <Cancel> button, neither replacing nor searching will be performed.

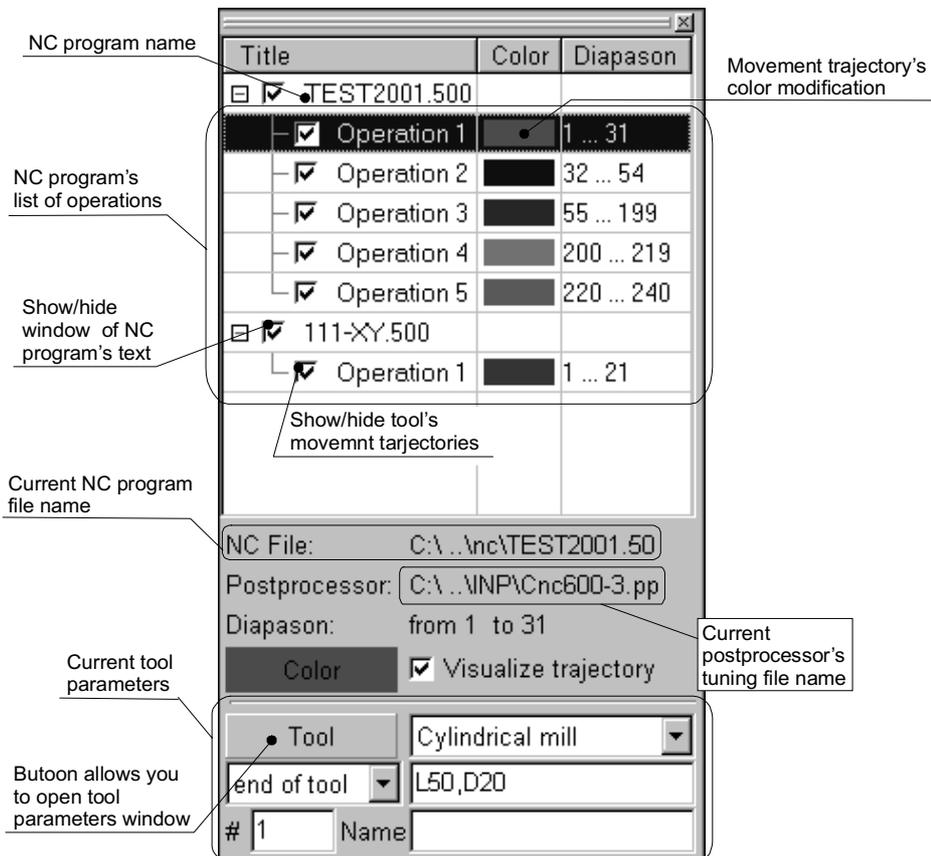
2.1.9 NC program manager window

NC program manager window opens either by pressing the button, or by selection of <View> <NC Program Manager> options of the main menu. In the window one can choose the color, trajectory visibility, tool parameters, and also correct postprocessor tuning or change postprocessor tuning file. NC programs are displayed in the hierarchical list. In the root nodes of the list, titles of NC program files are located. Every NC program is divided into separate operations (for example by tool changes), which are also displayed in the hierarchical list.

From the window you can interactively control visibility of trajectory sections joined into operations. To do so, tick/untick the field next to the corresponding operation or select <Show Trajectory> menu options from the pop-up menu. The menu calls by clicking the right mouse button within the area of the corresponding operation of the NC program manager window.



Modification of the color of trajectory sections is performed after clicking the left mouse button in the corresponding "Color" section. In the appeared window, choose a new color of the trajectory. Besides, modification of the trajectory color can be done through the <Trajectory Color> option of the pop-up menu. The menu calls by clicking the right mouse button within the area of the corresponding operation.



Main parameters of the current tool are shown in the lower part of the NC program manager window. Their modification is accessible directly from the window. To do so, one should choose the desired parameter, having either put the cursor to the corresponding field, or pressed the “” button (down arrow), select the type of the tool and the programmed point (central or end) from the pop-up list. One should note, that for editing from the NC program manager window, only the following parameters are accessible: tool name, tool number and geometrical parameters of the tool. In order to define other parameters, it is necessary to press the button. After that, the current operation tool window will be called.

Note: *If modification of tool parameters is made during simulation, then all simulation results will be reseted.*

■ Pop-up menu:

Upon clicking the right mouse button within the space of the NC program manager window, the pop-up menu appears. It is intended for the quick access to the main functions of the file management. The view of the pop-up menu depends on the type of the selected in the list element (NC program or operation). The menu partially duplicates main menu options. The following options are available:

- **<Show Trajectory>** – shows/hides the tool movement trajectory of the selected operation (the option available only for operations).
- **<Trajectory Color>** – allows you to change the color of the operation’s trajectory (the option available only for operations).
- **<Tool>** – activates the “Current operation tool” window (the option available only for operations).
- **<Show NC Text>** – shows/hides the text of the NC program for the current NC program file. Trajectories of all operations of the NC program will be simultaneously shown/hidden.
- **<Save NC>** – saves the NC program. The function can be called from the “Main panel”. If the NC program has not been saved, a new file name will be requested.
- **<Save NC As>** – saves the NC program under a new name.
- **<Close NC>** – closes the NC program. All operations of that NC program are also closing up.
- **<Tune Postprocessor>** – calls the “Postprocessor tuning window”. After modification of postprocessor tunings the trajectory of all loaded programs, referring to that postprocessor tuning file will be automatically recalculated.
- **<Change Postprocessor>** – allows you to change postprocessor tuning file for interpretation of the current NC program. The trajectory of NC program recalculates automatically.
- **<Close Window>** – closes the NC program manager window.

2.1.10 Showing the current values of registers

The **Registers’ values** window displays the current values of the being tracked values during NC program execution. The window shows: name of register, its identifier, designation and the current value of the register. During NC program execution, in field “Value” current value is displayed

for the registers of the numerical type, and for the registers of enumerable type – explanation, corresponding to the current value. Selection of types of registers and its designations can be performed in the Machine information window. (Menu option: tune postprocessor).

Name	Id	Designation	Value
Nstr	---	String number	0
TotalX	X	Coordinate X (abs.)	0
TotalY	Y	Coordinate Y (abs.)	0
TotalZ	Z	Coordinate X (abs.)	0
MTime	---	Machining time	0
TotalL	---	Total length of movements	0
WorkL	---	length of work movements	0
RapidL	---	length of rapid movements	0
N	N	Block numeration	0
INTERP	G	Interpolation	Positioning
ABS_INC	G	Frame of reference	In absolute
G_FUNC	G	Function G	Dwell
PLANE	G	Active plane	XY plane
KORECV	G	Radius correction	Off
GPos	G	Basis point	Return to basis point checking
D	D	Corrector's number on diameter	0
DX	D	Unknown	0
X	X	X axis displacement	0
PAUSE	I	Dwell	0

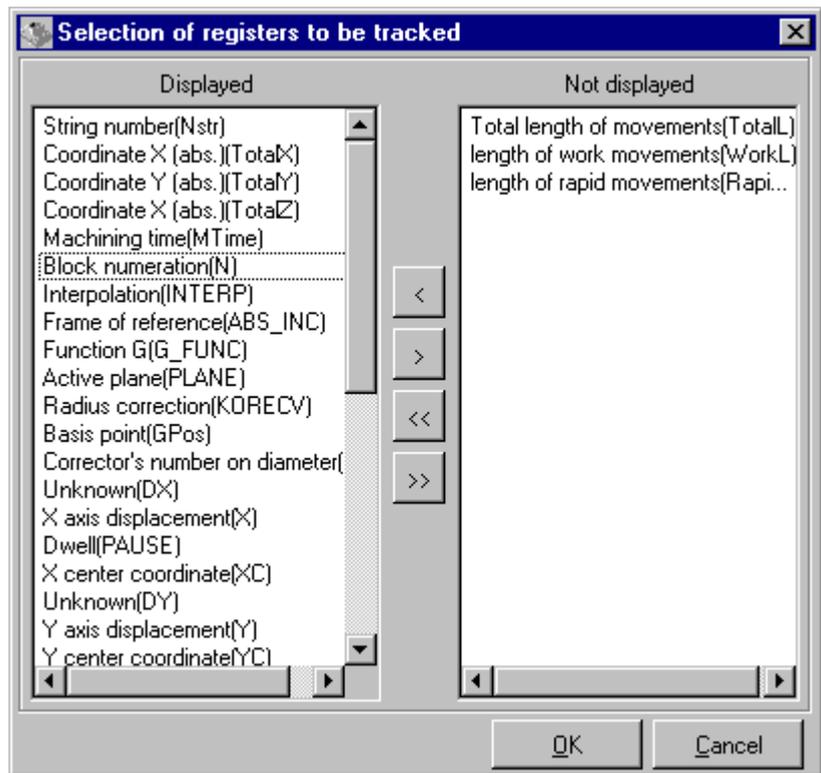
The set of registers corresponds to the list of registers in the current postprocessor tuning file. Apart from the postprocessor registers, the system track some general data, like: the number of the current string of NC, current tool coordinates (X, Y, Z), machining time, total length of movements, length of work and rapid tool movements. General data has no identifiers, for in general they have no direct application in a NC program and are calculated indirectly.

Tuning of the columns and strings can be performed from the pop-up menu, which appears upon clicking the right mouse button within the window area. The menu has the following options:

- **<Name>** – whether to show or not registers' names,
- **<Id>** – whether to show or not registers' identifiers,
- **<Designation>** – whether to show or not registers' designations,
- **<Tracking Parameters>** – opens the "Selection of the being tracked registers" window, where correction of the list of the being displayed registers is performed. By default all registers be tracked.
- **<Track All>** – show all registers in window.

2.1.11 Selection of registers to be tracked

To the left you can see the list of the being displayed in the window register values of statistical values and registers, and right – not displayed.



To move the selected register (statistical value) from one list to another use the following buttons:

< > – single add/remove in the list.

<< >> – add/ remove whole list.

2.2 STREAM EDITOR

Stream editor window can be opened by successively choosing items **<View> <Stream Editor Window>** in the main window. Additionally, the editor functions are available either from the main menu **<Edit> <Stream Editing>** or from the context menu of the NC program window.

Note:

If in the NC program, a fragment of a text is selected by block, then all actions will be applied to the block, otherwise – to the whole file.

2.2.1 Create/finish file macro

To change CNC system from one to another it might be enough to modify just a few parameters. This can be done successively performing functions from the stream editor window; however, with the greater number of modifications and the necessity to constantly perform always the same modifications with different files, an another decision is possible: create a file macro, containing a set of executable commands, which process the NC text. After creation of a file macro it will be offered to save it with a unique name. The file will have the *.MAC extension. If a file with this name already exists, then the older file will be renamed with *.~MAC extension.

Creation of a file macro can be achieved by selecting the Create macro menu item. After that all accomplished actions of the stream editor will be recorded into it. To finish recording the commands, choose the Finish macro menu item. The following execution of the commands recorded into the file macro can be done by selecting the Execute file macro menu item.

2.2.2 Execute file macro

Reads and performs the earlier created file macro. All the stream editing commands recorded into it are applied to the defined text (section of the text) of the current NC program.

2.2.3 Stream editor functions

■ Renumeration

Identifier:	N
Initial value:	1
Pace:	1

OK Cancel Help

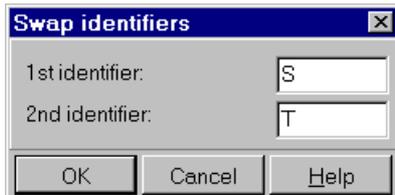
Allows you to renumerate NC blocks. Define the initial value of the numeration (1 by default) and its pace (1 by default) for the defined identifier (N by default). If the identifier is not defined, then the number inserts in the block beginning.

■ Insert identifier



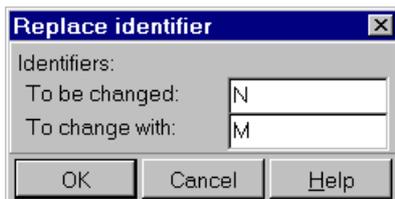
Inserts register identifiers into the end of every block, where such identifiers are missing, at that there can be several identifiers and they can be separated by spacing, any punctuation marks or written one after another. It is advisable to use for insertion of the block number, if before there has been no numeration. For regulation of words in a block it is necessary to use block pattern function.

■ Swap identifiers



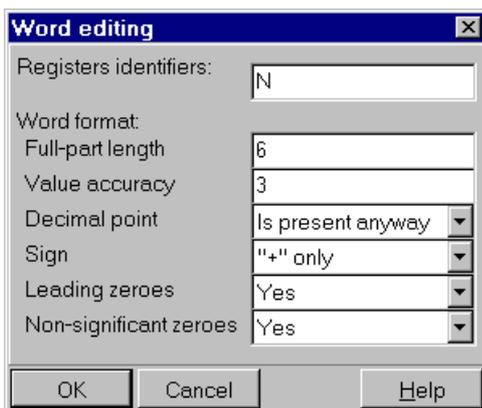
Swaps two identifiers (replaces only the identifiers, the values remain on the their places). Advisable to use for example if you need to swap X and Y planes. Processes only the first appearances of this identifiers in the block. To swap fragments of the block between each other it is necessary to type them in the corresponding fields.

■ Replace identifier



Replaces register identifiers in a word with another identifier. Processes only the first appearance of this register identifier in a block. To replace fragments of a block it is necessary to type them in the corresponding fields.

■ Edit words



Changes the format of the words, being defined by the register identifiers, at that there can be several identifiers and they can be separated by spacing, any punctuation marks or written one after another. The format of the word is defined by: the length of the whole part, tolerance of the value, presence of the point, sign, leading and non-significant zeroes.

Sign

The field defines the presence of the sign in the word:

- **always** "+"/"-" – sign "+" as well as "-" be obligatory.
- **No** – no signs at all.
- **"-" only (by default)** – no plus sign.
- **"+" only** – put "+" before the positive values.

Leading zeroes

Presence of the leading zeroes is defined from the condition whether to put them or not. The number of the being inserted leading zeroes is defined by the length of the whole part of the word.

Example:

N10 G1 X+001000 Y+002000 Z-010000

words X Y Z have leading zeroes in their format;

Non-significant zeroes

Presence of the non-significant zeroes is defined from the condition whether to put them or not. The number of the being inserted non-significant zeroes is defined by tolerance of the value. Presence of the point is also considered here.

Example:

N10 G1 X+001.000 Y+002.000 Z-010.000

words X Y Z have non-significant zeroes in their format;

Decimal point

- **is absent** – point is not used, and deletes where it was.
- **is present** - point is used, except for the places where was a comma.
- **is present anyway** - point is used always, even where was a comma.
- **comma is present** - comma is used, if there is no of it; and if there is point, then it is replaced with a comma.

Example: if point is missing then the expression looks as follows:

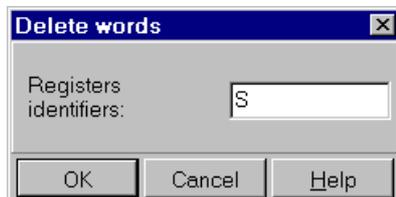
N10 G1 X+001000 Y+002000 Z-010000

otherwise:

N10 G1 X+001.000 Y+002.000 Z-010.000

During word formation with accuracy reduction you should remember that scaling is performed with error accumulation. In order to avoid losses of information at deletion of the decimal point, it is necessary to bring words to one standard format, i.e. first insert a decimal point to equalize word formats, and then if necessary put leading and non-significant zeroes.

■ Delete words



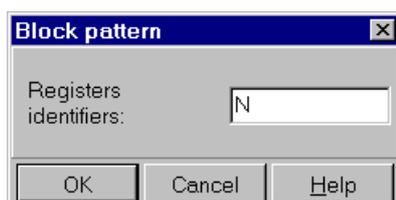
Allows you to delete words with the defined register identifiers from every block. (If there are several of these words then deletes them all) Type the register identifiers, which words should be deleted from every block.

■ Swap words



Swaps words (replaces the whole word; i.e. identifiers with their values). Processes only the first appearance of these words in a block. To swap block fragments between each other it is necessary to enter them into the corresponding fields.

■ Block pattern



Assigns the order of words in a block. You can define only the most important word identifiers, the remaining undefined words will be moved to the end of the block. Block pattern remains in effect until the end of the work seance.

■ Delete empty strings

Deletes empty strings; i.e. strings, containing nothing but blank space.

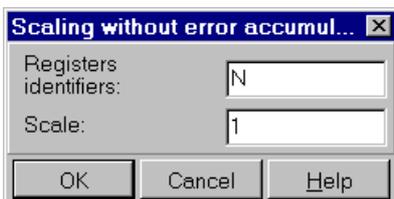
■ Put spacing

Puts spacing between words in a NC block.

■ Delete spacing

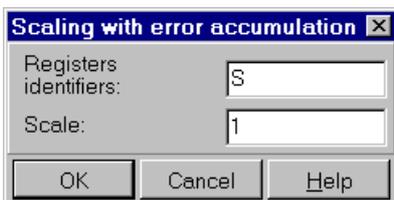
Deletes spacing between words.

■ Scaling without error accumulation



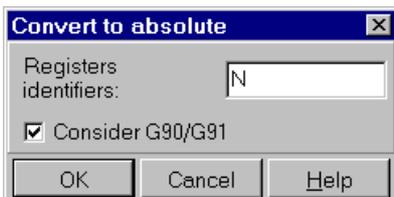
Increases/Decreases the value of the words with the defined register identifiers without accumulation of the scaling errors. The resulting accuracy of the scaled words will be equalized.

■ Scaling with error accumulation



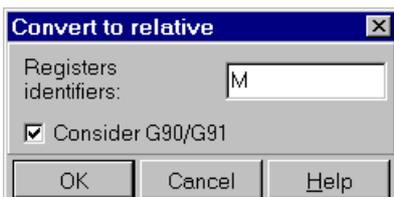
Increases/Decreases the value of the words with the defined register identifiers with accumulation of the scaling errors. The resulting accuracy of the scaled words will be equalized.

■ Convert to absolute



In the events when keys G90/G91 are not considered, it is meant that the frame of reference is relative. If keys G90/G91 are considered, then the frame of reference will be defined according to the keys G90 - absolute, G91 - relative. With the relative frame of reference, a translation from the relative to absolute coordinates is performed for the defined identifiers.

■ Convert to relative



In the events when keys G90/G91 are not considered, it is meant that the frame of reference is absolute. If keys G90/G91 are considered, then the frame of reference will be defined according to the keys G90 - absolute, G91 - relative. With the absolute frame of reference, a translation from the absolute to relative coordinates is performed for the defined identifiers.

■ Coordinates of arc center – in absolute

Translates coordinates of circle center (“I”, “J”, “K”) at circular interpolation from the relative coordinates to the absolute. It is meant that “I”, “J”, “K” are represented in the relative coordinate system.

■ Coordinates of arc center – in relative

Translates coordinates of circle center (“I”, “J”, “K”) at circular interpolation from the absolute coordinates to the relative. It is meant that “I”, “J”, “K” are represented in the absolute coordinate system.

■ Divide arcs into quarters

Divides arcs of a circle into quarters so that the central angle at this arc is not more than 90 degrees. It is used if CNC does not accept arcs, the angle at which is more than 90 degrees.

■ Upper case

Converts all letters from the lower case to the upper: small to capital.

■ Lower case

Converts all letters from the upper case to the lower: capital to small.

■ NC beginning and end creation

The dialog box is titled "NC beginning and end creation". It is divided into two main sections: "NC beginning" and "NC end".

NC beginning section:

- Text area: N10G90G17G49G40M5M9
N11Z50
- Radio buttons:
 - Instead of strings ## (value: 1-3)
 - Before string # (value: 1)
 - After string # (value: 1)

NC end section:

- Text area: N246G0Z5
N247M2M5M9
- Radio buttons:
 - Instead of strings ## (value: 1)
 - Before string # (value: 1)
 - After string # (value: 1)

Buttons: Ok, Cancel, Help

When editing a NC program a special attention is paid to the beginning and end of the program. With this function you can manually form a text of the beginning and/or end of a program and insert that text into a NC program. Where, you can insert the text instead of strings, before a string or after a string, numbers of which are defined in the corresponding fields. The text, which needs to be inserted into the beginning of a NC program, must be typed in the “NC beginning” window and the count of strings is performed from the beginning of the text; and text of NC program end – in the “NC end” window, the count of strings is performed from the end of the NC program text (from the last string upwards). Accordingly, for the beginning of NC “insert before string 1” means insertion before the first string of the program, and for the end “insert after string 1” means after the last string of NC program.

When creating NC beginning and end, it is sometimes necessary to output into a block the current values of registers, so that not to violate the program structure. For that reason there is a special syntax to use when describing the beginning and end of a NC program in the input fields **<NC beginning>** and **<NC end>**. If you define in the round braces the name of register in the typed-in text of NC beginning or end, then at inputting a fragment into a NC program the defined symbols will be replaced with the current value of the register. The value of the register will be outputted in accordance with its format (the format of registers can be defined in the **Editing tuning file to CNC system** window). If the defined in the round braces name of the register or the predefined value do not exist, then replacement will not be performed. The full list of names of registers and predefined names for the current postprocessor is available in the **Editing tuning file to CNC system** or **Selection of registers to be tracked** windows.

For example, if the input field will be outputted string:

N180 X{TotalX} Y{TotalY} F{Feed}

where: **TotalX** – predefined name of the current X coordinate of the tool;

TotalY – predefined name of the current Y coordinate of the tool;

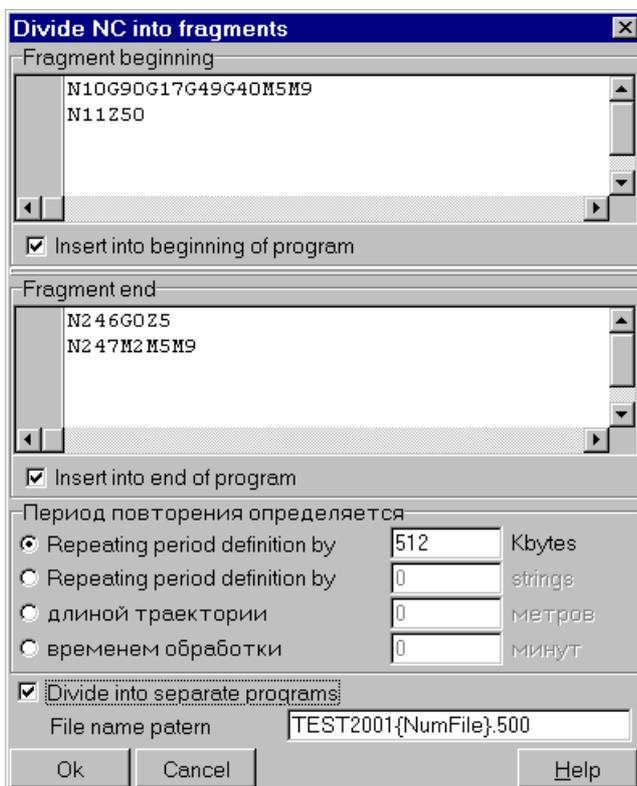
Feed – name of register with feedrate value.

then the following block will be inserted into the NC program

N180 X15 Y120.23 F200

if in the insertion place the current X and Y tool coordinates were equal to: accordingly 15 and 120.23, and feedrate was equal to 200.

■ NC dividing



If a NC program is slightly big you can divide it into several fragments.

In the “Fragment beginning” and “Fragment end” windows you can type the text which will be inserted in the beginning and end of the NC fragment respectively. The period of repetition of the insertion is defined either by the size of the NC program (in Kbytes), or the number of strings or the length of a trajectory (in meters) of time of machining (in minutes). Should it be necessary, the typed text can be inserted into the beginning and end of the being divided NC program if there are corresponding checkmark being put.

If there is a checkmark in the “divide into separate programs” field, then new windows with the corresponding parts of the NC program text will be created. The windows will be named according to the defined pattern, instead of {NumFile} a corresponding number of a window will be put.

Division into separate programs is recommended when for example the length of a NC program exceeds the allowed for the CNC. And usage of the function without dividing is advisable for instance for insertion of tool change commands, because of its deterioration, into a NC program.

When dividing NC into fragments, it is sometimes necessary to output into a block the current values of registers, so that not to violate the program structure. For that reason there is a special syntax to use when describing the beginning and end of a NC program in the input fields <**Fragment beginning**> and <**Fragment end**>. If you define in the round braces the name of register in the typed-in fragment, then at inputting a fragment into a NC program the defined symbols will be replaced with the current value of the register. The value of the register will be outputted in accordance with its format (the format of registers can be defined in the **Editing tuning file to CNC system** window). If the defined in the round braces name of the register or the predefined value do not exist, then replacement will not be performed. The full list of names of registers and predefined names for the current postprocessor is available in the **Editing tuning file to CNC system** or **Selection of registers to be tracked** windows.

For example, if the input field will be outputted string:

N180 X{TotalX} Y{TotalY} F{Feed}

where: **TotalX** – predefined name of the current X coordinate of the tool;

TotalY – predefined name of the current Y coordinate of the tool;

Feed – name of register with feedrate value.

then the following block will be inserted into the NC program

N180 X15 Y120.23 F200

if in the insertion place the current X and Y tool coordinates were equal to: accordingly 15 and 120.23, and feedrate was equal to 200.