



HEIDENHAIN







User's Manual
V600-02

MillPlus IT





NC Software
538 952-02
538 953-02
538 954-02
538 955-02
538 956-02

English (en)
6/2008









Controls on the visual display unit

	Select window
	User keys
	Info key
	Function soft keys Machine-function soft keys







Keys for main operating modes

	Manual operating mode
	Automatic operating mode
	Programming operating mode
	Control (Setup) operating mode












Keys for machine functions

		Key freely assignable via IPLC
		Key freely assignable via IPLC
		Key freely assignable via IPLC
		Key freely assignable via IPLC




Keys for spindle functions

	Spindle speed 100%
	Increase spindle speed
	Decrease spindle speed
	Spindle on, CCW
	Spindle stop
	Spindle on, CW


Manual operation

		Axis-direction keys for three main axes
		Axis-direction keys for further axes
		Axis-direction keys for 4th axis
		Rapid traverse
		Rapid traverse override/feed rate override
		Emergency stop
		NC on






Start / stop keys

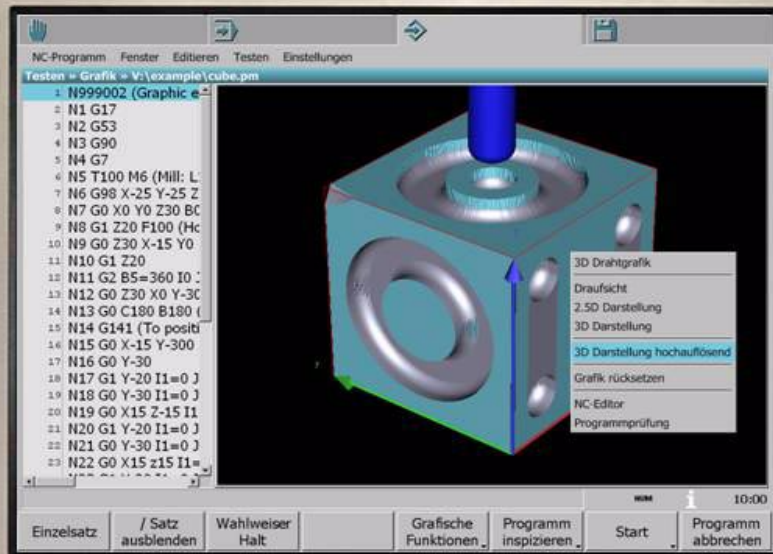
	START
	Feed rate STOP
	Feed rate and spindle STOP

Touchpad

	Positioning of the mouse cursor Selection and context buttons
------------------------------------------------------------------------------------	------------------------------------------------------------------

ASCII keyboard (NC functionality)

	Menu key: Open a menu from the menu row
	ALT key: Open a menu from the menu row
	Apps key: Open context menus
	Windows key: Switch to Windows applications
	Clear key: Clear error messages



HEIDENHAIN



MillPlus, Software and Features

This manual describes functions and features provided the MillPlus as of the following NC software numbers:

Processor type	NC software number
MillPlus IT V600	538 952-xx
MillPlus IT V600 Export	538 953-xx
MillPlus IT V600 DP*	538 954-xx
MillPlus IT V600 Export DP*	538 955-xx
MillPlus IT V600 Programming Station	538 956-xx

*DP= 2-processor system

The most important limitation of the export version of the MillPlus is that simultaneous linear motions are only possible in up to four axes.

The machine manufacturer adapts the features offered by the MillPlus to the capabilities of the specific machine via configuration settings. Some of the functions described in this manual may therefore not be among the features provided by the MillPlus on your machine tool.

MillPlus functions that may not be available on your machine include:

- Tool measurement with the laser measuring system
- Software option 1 (see below)

Please contact your machine tool builder to become familiar with the features of your machine.

In addition, the MillPlus also has further software functions that can be enabled by you or your machine tool builder.

Intended place of operation

The MillPlus complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

New Functions

In addition to the scope of functions offered by previous versions of the control, the MillPlus V600 offers several new functions.

Here is a brief overview of these functions:

- New and optimized motion control for shorter machining times
- Programming aids include dialog entry, Function Explorer, context-sensitive help for programming and operation
- Universal mouse operation (for single- and dual-processor systems)
- Clipboard function for data exchange within the control as well as with Windows applications (for dual-processor systems)
- Programming enhancements, such as limit plane and zoning functions, contour cycles with contour linking, syntax highlighting, higher-language elements, SQL table access
- Support for mid-program startup via the new program-structure function
- Setting of breakpoints without changing the program; helps for testing the program as well as for practice runs
- New tool and table editor. Dialog entry is also available for tools. The desired tools can also be selected via the tool-directory window.
- All tables are shown in a grid display. You can optimize each table for its task, by changing the sequence or width of columns, by showing or hiding them, or by sorting the contents.
- Improved simulation graphics with higher display resolution, block-number assignment in the line graphics, simple jumping between a block and the corresponding line in the graphics.
- Along with the 15.1" color TFT, a 19" version is also available. The 19" TFT has 10 instead of only 8 function keys.

MillPlus IT Website

For more information, go to **www.millplus.com** or **www.millplus.de**

Contents

Introduction	1
Manual Operating Mode	2
Automatic Operating Mode	3
Programming Operating Mode	4
Control (Setup) Operating Mode	5
Miscellaneous	6

1 Introduction 17

1.1 MillPlus IT 18	
Workshop oriented 18	
MillPlus IT with Windows 18	
Compatibility 19	
Instructions 19	
Programming station 19	
Remarks, notes and safety 20	
© Heidenhain Numeric B.V. Eindhoven, Netherlands 2008 20	
Legal information 20	
1.2 Visual Display Unit and Operating Panel 21	
Screen and NC keys 21	
Machine operating panel 21	
1.3 Operating Concept 22	
Main operating modes 22	
Menu bar 23	
Message line 25	
System bar 25	
Soft-key row: NC functions 26	
Soft-key row: Machine functions 26	
Application window 26	
1.4 Basic Operating Functions 27	
Input dialogs 27	
File manager 28	
Table editor 30	
NC editor 31	
Editing functions 33	
Shortcuts: CTRL+... 34	
Popup windows 34	
Context menu 34	
Machine status display (dashboard) 35	
1.5 Online Help and Calculator 37	
INFO functions 37	

1.6 User Soft Keys	39
Operation	39
Recording a user soft key	39
Text or image for user soft keys	40
Deleting a user soft key	40
1.7 Coordinates and Reference System	41
Axis configuration	41
Coordinate system	41
Defining coordinates	41
G7 coordinates	41
Datums	42
Machine-based reference point (R)	42

2 Manual Operating Mode 43

- 2.1 Introduction 44
 - Menu bar 44
- 2.2 General Functions 46
 - Switch-on 46
 - Crossing over the reference point 47
 - Switch-off 48
 - Moving the machine axes: Jogging 48
 - Spindle speed S, feed rate F, tool T and machine function M 50
 - Positioning manual axes 51
 - MDI: Free entry 52
 - Setting the zero point 54
 - Tool measurement 55
- 2.3 Workpiece Menu 56
 - Touch side 56
 - Determine center 57
 - Preset axes 58
- 2.4 Tool Menu 59
 - FST entry 59
 - Measure in jog 60
 - Set plane 60
- 2.5 MDI Menu 61
 - Free entry 61
- 2.6 Machine Tool Menu 62
 - Reference point 62
 - NC shutdown 62
- 2.7 Settings Menu 63
 - Jogging 63

3 Automatic Operating Mode 65

- 3.1 Introduction 66
 - Menu bar 66
- 3.2 General Functions 68
 - Program selection 68
 - Program execution 68
 - Interruption of program run 69
 - Moving the machine axes during an interruption 70
 - Intervening program run 70
 - Resuming program run after an interruption 71
 - Mid-program startup (block search) 72
 - Optional block skip 74
 - Selectable stop (Selective wait) 74
 - Program breakpoint 75
- 3.3 NC Program Menu 76
 - Select 76
 - Last selected programs 77
- 3.4 Execution Menu 78
 - Machining 78
- 3.5 Settings Menu 80
 - Execution 80

4 Programming Operating Mode 81

- 4.1 Introduction 82
 - Menu bar 82
- 4.2 General Functions 84
 - Creating/Editing a program 84
 - Testing a program 84
- 4.3 NC Program Menu 85
 - Open 85
 - New 85
 - Last selected programs 85
- 4.4 Window Menu 86
- 4.5 Edit Menu 87
 - NC editor 87
- 4.6 Test Menu 91
 - Simulation 91
- 4.7 Settings Menu 95
 - Edit 95
 - Test 95

5 Control (Setup) Operating Mode 97

5.1 Introduction	98
Menu bar	98
5.2 Menu: Tables	100
Tools	100
Tool magazine	107
Datums	108
Parameters	108
Points	108
5.3 Menu: NC-Explorer	109
File	109
5.4 Menu: Installation	110
Compensation	110
Configuration	110
Service	111
5.5 Menu: Settings	114
Tables	114

6 Miscellaneous 115

6.1 Accessories 116

3-D touch probes 116

The TS 640 triggering touch probe 116

HR electronic handwheels 116

1

Introduction

1.1 MillPlus *IT*

Workshop oriented

The MillPlus IT is designed for use with milling, drilling, boring and machining centers, as well as for use with mold machines. The MillPlus IT can also be traversed manually for simple machining operations.

Direct access to machining cycles shortens setup times and avoids unnecessary programming work.

Optimized motion control shortens the machining time while at the same time providing a well-machined workpiece surface. The various machine characteristics (swivel head, tilting table, additional axis, etc.) are specified in the kinematics model, which lets you concentrate on the machining process.

Different types of aid are available to the programmer: dialog entry, Function Explorer, context-sensitive online help, graphic simulation, etc.

The user-friendly operating concept, which is similar to Windows, can be adapted to the user's experience level.

MillPlus IT with Windows

The 2-processor version of the MillPlus IT uses the Windows operating system, providing a standard interface to further Windows applications. Data can easily be exchanged between the MillPlus and Windows applications via the Clipboard. Also, the control can easily be connected to your company network.

Other Windows oriented features available in both the single and double-processor versions:

- Complete PC keyboard with integrated touchpad
- USB ports, for easy attachment of standard PC peripherals, such as a mouse, CD-ROM drive, etc. to the control



Compatibility

The MillPlus V600 has many new functions compared to previous versions of the MillPlus. Operation of the control has been developed further, but you will nevertheless recognize the familiar operating concept of the MillPlus. The NC programming of the V600 is also based on its predecessors.

However, there are also significant differences in some cases. For example, machining programs written on older MillPlus controls must be checked. If an NC block is not accepted by the V600, a warning will be issued when checking the program. Also refer to the Programming Manual.

Instructions

These instructions are intended to support you while using the control.

Please read the information collected for you in this manual before you begin working with your machine. The User's Manual contains important information about operating the machine as well as operational safety, so that you may operate your machine in a safe and efficient manner.

Along with this User's Manual, there is also a Programming Manual, in which the G functions are described in detail.

The User's Manual and Programming Manual are also available online on the control.

Programming station

The MillPlus programming station, which runs on a PC with a Windows operating system, simulates the MillPlus control. Here you can create, test and optimize NC programs without needing a machine.

Remarks, notes and safety

We reserve the right to make changes to the features and accessories as part of further development. Therefore, no claims may be derived from the data, descriptions or images. Errors and omissions excepted.

- The information in this manual is absolutely essential for safe operation of the machine. Ensure that it is readily available at the machine.
- Regarding topics dealing with safety, please refer to the notes regarding proper use.
- The configuration of the control is modified appropriately to interface it with the machine. Some of these configuration settings can be accessed by the operator. Caution! Before changing any configuration settings, be sure that you understand the meanings and functions thereof. Otherwise please contact your machine supplier's service agency. Also refer to the Machine Manual for other information about your machine.
- We recommend backing up your program and sensitive data at regular intervals, in order to avoid such information being irretrievably lost should a system defect occur.

© Heidenhain Numeric B.V. Eindhoven, Netherlands 2008

The publisher shall not assume liability for any specifications based on information contained in this manual. Solely the data contained in the order and the corresponding technical specifications shall apply for the specifications of this numeric control.

All rights reserved. Reproduction of this material, in whole or in part, is not allowed without written permission from the copyright holder.

Legal information

This product uses open source software. Further information is available on the control under:

- Control operating mode
- Menu: Installation >> Diagnosis >> Software
- Legal information soft key

1.2 Visual Display Unit and Operating Panel

The MillPlus features a color flat-panel display and an operating panel:

- Soft keys are assigned to the color flat-panel display
- The operating panel consists of a standard ASCII keyboard (NC part) and a machine operating panel

Screen and NC keys

The following keys are located on the display unit, the keyboard and the machine operating panel:

- 1 Select window
- 2 User soft keys
- 3 Soft-key row (NC functions)
- 4 Soft-key row (machine functions)
- 5 Info key
- 6 ASCII keyboard
- 7 Menu key
- 8 ALT key
- 9 Windows key
- 10 Applications key
- 11 Touchpad with selection and context buttons

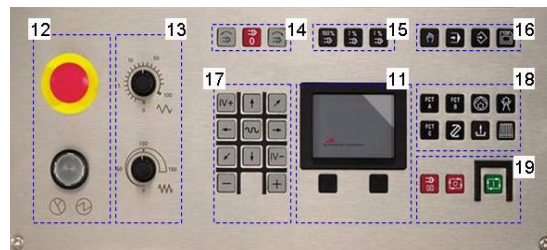
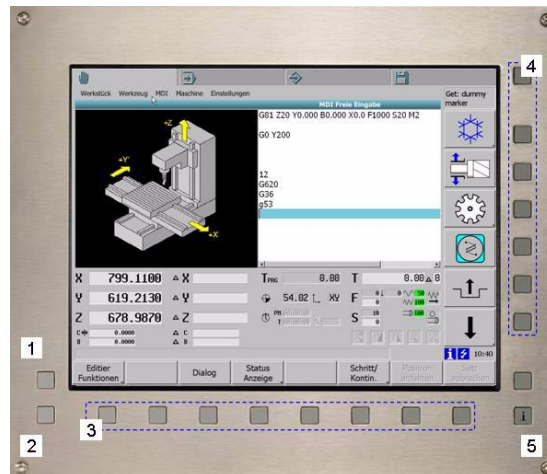
The functions of the individual keys are described on the inside front cover.

Machine operating panel

The machine keys are located on the machine operating panel:

- 12 NC switches: Emergency stop /NC on
- 13 Override knobs: Rapid traverse / Feed rate
- 14 Spindle control
- 15 Spindle-speed control
- 16 Main operating modes
- 17 Axis-direction keys
- 18 Keys for machine functions (PLC)
- 19 Start / stop keys

The functions of the individual keys are described on the inside front cover.

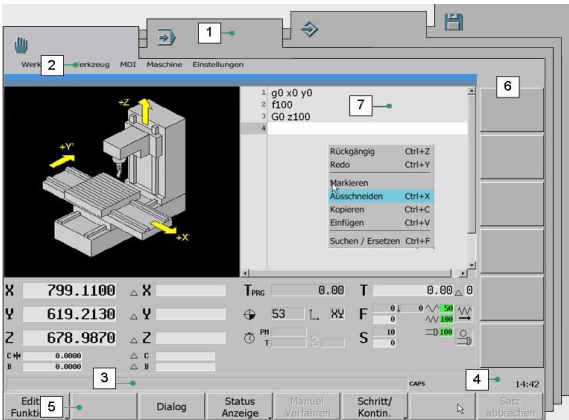


1.3 Operating Concept

The MillPlus has four main operating modes: Manual, Automatic, Programming and Control (Setup). A uniform manner of operating the MillPlus is made possible by the basic components of the user interface. This operation consists of navigation through windows using the touchpad, keys and soft keys.

Basic components

- 1 Main operating modes
- 2 Menu bar
- 3 Message line
- 4 System bar
- 5 Soft-key row (NC functions)
- 6 Soft-key row (machine functions)
- 7 Application window







Main operating modes

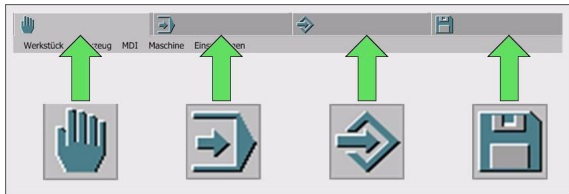
The four main operating modes are shown as tabs at the top of the screen. The active main operating mode is light gray. Switch between the main operating modes with the **Manual**, **Automatic**, **Programming**, and **Control** keys (on the operating panel and on the ASCII keyboard), or use the touchpad to click the main operating mode tabs on the screen.

You can switch between the main operating modes of the MillPlus at any time, since they operate parallel to each other. Each mode maintains its current status when you switch to a different mode. An informational window can be opened in addition to the four main operating modes.

Symbols








The main operating modes are indicated with symbols on the tabs.

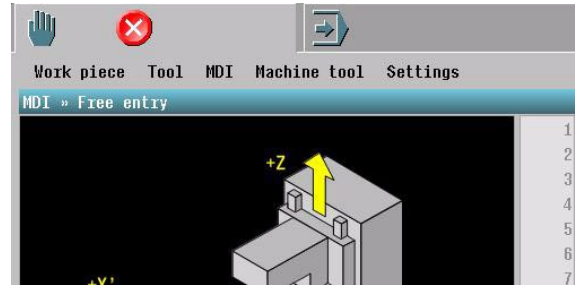
Symbol	Descriptions of the main operating modes
	Manual: Manual operation and automated sequences for setting up machines, workpieces and tools.
	Automatic: Breaking in workpiece programs, setting up the workpiece and automatic execution. The Manual operating mode is available during an interruption.
	Programming: Create, edit, simulate and manage NC programs.
	Control: Set up and manage the control, all control data, tool management, diagnostics and service functions.



Status display in the main operating mode tab

Symbols indicating the status of the respective main operating mode can appear in the main operating mode tab. These symbols are refreshed automatically.

Symbol	Description
	Program is running
	Program run is paused
	Program run is paused, and the program is open for editing
	An error has occurred
	Warning
	Display of dwell time or time remaining
	Program status, such as End of program

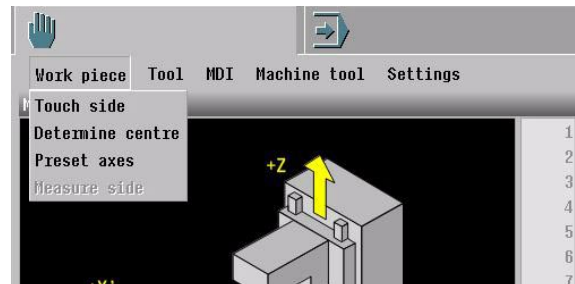


Menu bar

The menu bar gives an overview of the menus available in the respective main operating mode. The menus are drop-down menus, meaning that a small window appears with the appropriate menu items when the menu is selected.

This menu window shows the selectable functions and operating modes. Arrows in the menu window indicate that further window menus can be opened.

All soft keys are deactivated while a menu window is open. Only once a menu has been selected or the menu bar deactivated do the soft keys become active again.



Navigation

There are different methods for selecting the functions and operating modes of the menus:

- Touchpad
- Arrow keys
- Number keys

1 Touchpad



- ▶ Use the mouse pointer to select a menu from the menu bar. Click a menu item to open its menu window, and then move the mouse pointer to the desired function or operating mode. Click the function or operating mode to activate it.

2 Arrow keys



- ▶ Press the MENU key or



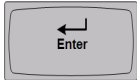
- ▶ the ALT key to open the menu



- ▶ Use the LEFT/RIGHT arrow keys to select a different menu from the menu bar



- ▶ Use the UP/DOWN arrow keys to select a function or operating mode from the menu window



- ▶ Activate the selected function or operating mode

3 Number keys



- ▶ Press the MENU key or



- ▶ the ALT key to open the menu



- ▶ Press the keys 0 - 9 in sequence to activate a function or operating mode from the menu

Example: Press 2 and then 2 again to activate the **Measure in jog** function.

Message line

Error message

The control displays current messages for the operator in the message line.

Messages	Description
Machine	Messages referring to the machine (PLC)
Operation	Messages about the sequence of operations
Programming	Programming errors or notes regarding the syntax

- The messages appear in the message line, with a yellow background
- A symbol may also be displayed in the main operating mode tab at the same time.



► Press CLEAR to remove the error message, or

Error info

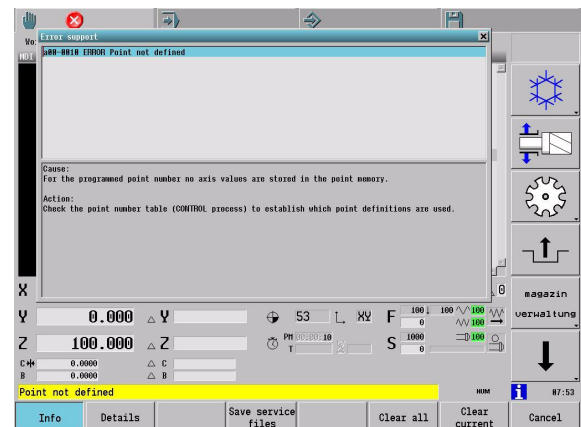
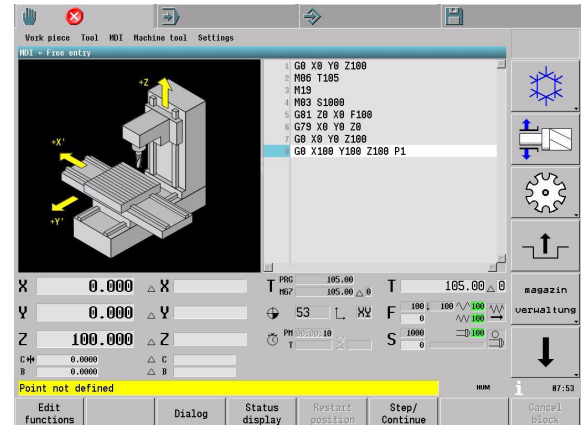


► Press INFO to open the online help (see "Online Help and Calculator" on page 37).

System bar

The following symbols are shown in the system bar:




Symbol	Description
	Info key
	User soft keys
	Recording
	Keyboard: Number lock on
	Keyboard: Caps lock on
	Time



Soft-key row: NC functions

Standard soft keys

Use the soft keys to directly activate a function permitted in the active operating mode. The soft keys can also be activated with the touchpad.

Symbol	Description
	Soft key inactive
	Soft key active
	Soft key with an arrow in the corner indicates another soft-key row

User soft keys

You can also define your own soft keys, in order to specify optimized sequences. The user soft keys are superimposed over the standard soft keys if the USER key (at the edge of the display unit) is pressed, or if the same main operating mode is selected twice in a row. Select the main operating mode again to display the standard soft keys again.

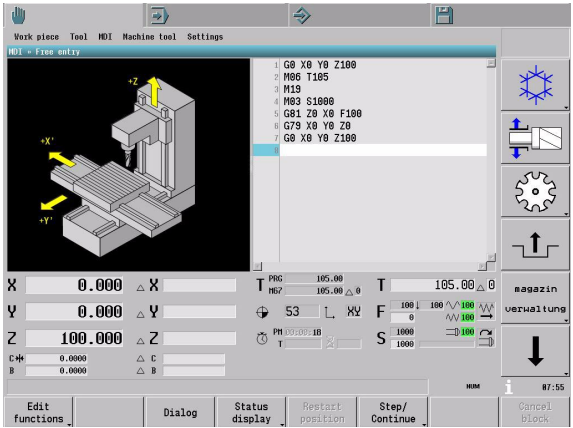
Soft-key row: Machine functions

The machine soft keys activate machine-specific functions. The functions depend on the machine, and are specified by the machine manufacturer via the PLC.

Application window

You can enter your data or influence the sequences of the selected program directly in the applications window, depending on the active operating mode. Some examples of application windows are:

- Manual: Next to the graphic of the machine is an input window, in which you can enter commands and then run them.
- Automatic: The folders and files are displayed for selection of the NC program. Once a program has been selected, it is shown over the entire width of the window.
- Programming: The program opened in the editor is shown over the entire width of the window. If the graphic simulation function is activated, then a part of the window is used for this.
- Control (Setup): You can see and edit the tool data in a tabular view.



1.4 Basic Operating Functions

Typical functions are available in multiple locations on the MillPlus.

The basic operating functions are:

- Input dialogs
- File manager
- Table editor
- NC editor
- Editing functions
- Shortcuts
- Popup windows
- Context menu
- Machine status display
- Online help
- User soft keys

Input dialogs

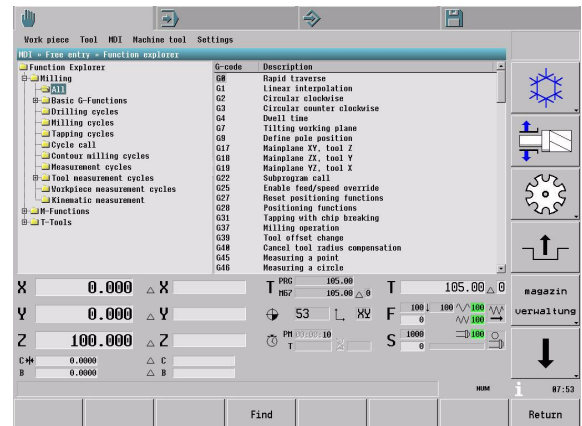
Support for data entry is available when creating a program in the program editor or with MDI, which helps you to structure the program block as needed. The level of support automatically adapts itself to the data entered in the block. The MillPlus offers a structured search view: the Function Explorer. The data-entry support is opened with the **Dialog** soft key.

Function Explorer

All functions and tools are collected by their functions into groups in the Function Explorer. This grouping makes it simple to find the function you are looking for. Press the ENTER key to open the selected function in the input dialog.

The main groups are:

- G functions
- M functions
- T tools



Support for data entry

The level of support adapts itself to the respective contents of the program block when the **Dialog** soft key is activated.

- Dialog**
 - ▶ Activates support for data entry. If a G function or an M function is entered (such as G81), then the input dialog window opens and the data can be entered via the keyboard. If only a G or an M is entered, or nothing at all, the Function Explorer opens automatically.
- Function explorer**
 - ▶ Activates the Function Explorer. Select the desired function, and open it with the ENTER key.
- Entry ready**
 - ▶ Saves the entered data in the program block. The input dialog window is closed.
- Cancel**
 - ▶ Data entry is cancelled in the dialog input window without saving the data in the program block.

File manager

To ensure that you can easily find your files, we recommend that you organize your hard disk into directories. You can divide a directory into further directories, which are called subdirectories. Use the file manager to select a file.

The file manager is available in the following operating modes:

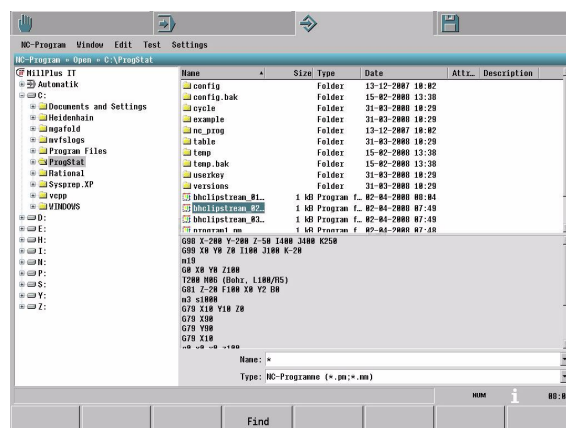
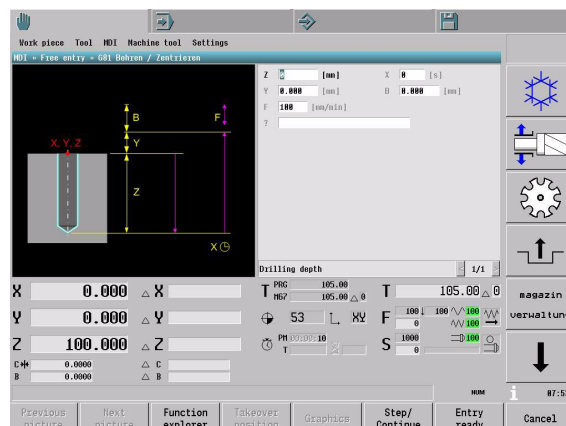
- Automatic, via the menu bar: **NC-Program > Select**
- Programming, via the menu bar: **NC-Program > Open** or via the context menu **Save as** while a program is open
- Control, via the menu bar: **NC-Explorer > File**

At the top of the list of drives you will find special drives with the opened programs in Automatic and Programming modes.

Screen layout

The file manager is divided into four window panes:

- 1 Explorer window with:
 - List of drives
 - Directories and subdirectories
- 2 Directory and file window with:
 - List of files in the selected directory
- 3 File preview window
 - The window shows a preview of the selected file
- 4 Filter window with:
 - File name or a part of it to be searched for
 - File type (*.pm, *.mm, etc.)



Names of files and directories

The name can consist of any letters and numbers.
The characters * \ / " ? < > are not permitted.

Sequence of operations

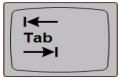
If you want to open a program, such as for program run in Automatic mode, first use the file manager to find the program.

Sequence of operations with the ASCII keyboard



- ▶ Open the menu bar and select the **NC-Program** menu item. In this menu, choose the **Select** or **Open** functions.

Navigation in the file manager



- ▶ Press TAB to switch the focus between the various panes.
- ▶ Use the arrow keys to open a drive or directory in the Explorer window [1].
 - Press the right arrow key again to switch the focus to the file window [2].
 - Press the left arrow key to switch the focus back to the Explorer window [1].
- ▶ Press the ENTER key to open a file.
- ▶ Press ALT+arrow key in the filter window [4] to select a previous filter setting or file name.

Table editor

All tables with data regarding tools, datum shifts, etc, are available in the Control (Setup) main operating mode.

Available tables:

- Tools
- Tool magazine
- Zero offsets
- Parameters
- Points

Editing a table

If a table is simply activated, then you can only view the information. The table itself has a gray background. If you want to edit a table (change the contents or add new data, for example), you must open the table by pressing the **Edit** soft key. You can make your entries in the table itself, or if the **Dialog** soft key is available, you can use the support for data entry.

Finding

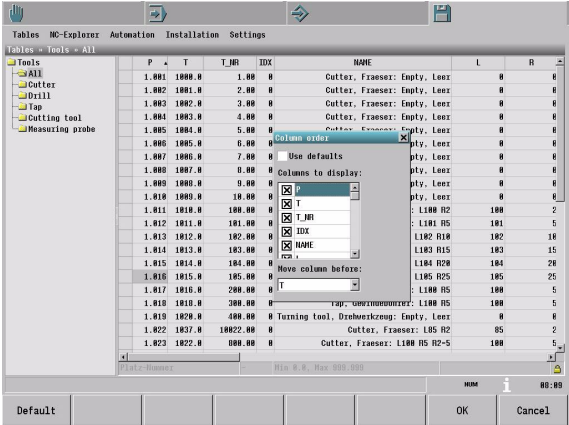
Find

► Enter the string you want to search for in the Find window, and press the **Find next** soft key or press ENTER. You can repeat the search by pressing these keys repeatedly.

Table settings

The table view can be configured, i.e. you can optimize it for your application. The following settings are available:

- Column assignment and order:
Position the mouse pointer on a column header and press the right mouse button to open the context menu. Select **Column order**. The **Column order** window shows the possibilities for configuration.
- Show columns: In the **Columns to display** window you can select which columns are to be shown.
- Column order: Indicate in **Move column before** what the order of the columns should be.
- Press **Default** to return to the default setting.
- Column width:
Position the mouse cursor to the left or right edge of a column header. Press the left mouse button and drag the column to its desired width.
- Sort data:
The data in a column is sorted in either ascending or descending order. An arrow in the column header shows the sort order. Click in the column header with the left mouse button to change the sort order.



NC editor

You can use the NC editor to create and edit NC programs. The program editor is available in the following main operating modes:

- Programming: **Edit > NC-Editor**
- Control: **NC-Explorer > File**

The NC editor builds the program line by line. Each line consists of the NC block and a line number. The line numbers are automatically generated in ascending order. Use the keyboard to enter the data for the program block.

Syntax highlighting

The program blocks consist of various elements, such as G functions, comments, FMST, etc. These elements can be shown in different colors in the block.

Select the **Settings > Edit** menu item to activate syntax highlighting and choose the colors.

Syntax checking


The correct syntax of a program can be checked while the blocks are being programmed.

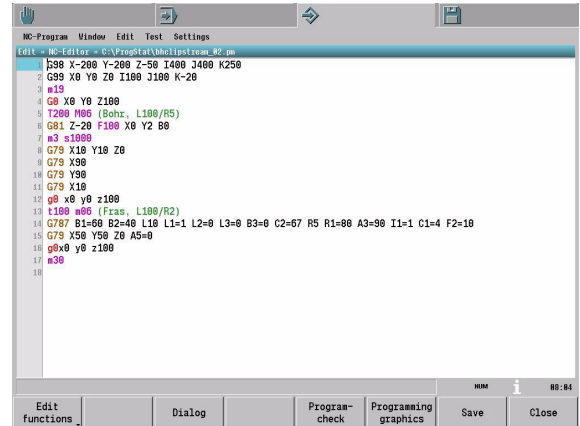
Select the **Settings > Edit** menu item to activate program checking.

Line number and status symbol

The gray column to the left of the program shows the line number, the line cursor, and a status symbol in front of the respective block.

Status symbol:

Symbol	Description
	Warning, syntax error in the block



Sequence of operations

1 Creating or editing a program



- ▶ In order to create an entirely new program, select the **NC-Program > New** menu item.
- ▶ In order to open an existing program, select the **NC-Program > Open** menu item.

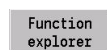


- ▶ If you want to save the new or edited program, press the **Save** soft key. A window appears in which you can enter the program name and type.

2 Support during programming



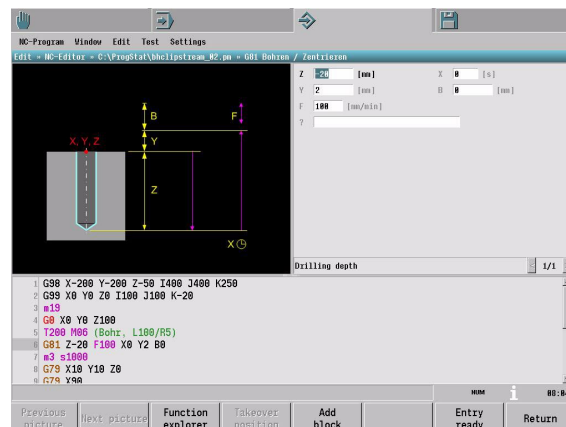
- ▶ Open the input dialog window during program entry in order to simplify the entry of data.



- ▶ The grouping of functions makes it simple to find the function you are looking for.



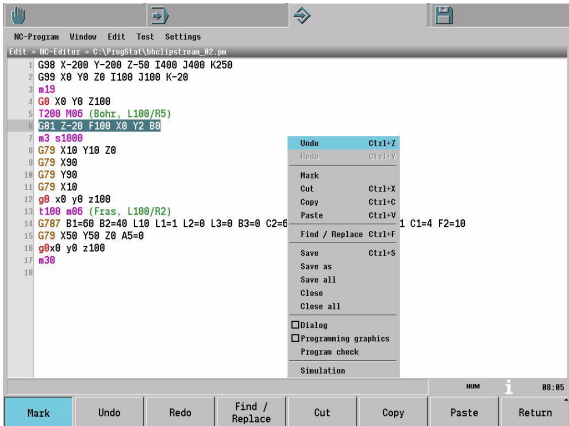
- ▶ Press the INFO key or icon to call the soft keys for the calculator and the online programming aids.



Editing functions

You can use the editing functions to change the contents of one or more line in the MDI list or in programs:

Soft key	Description
Mark	Mark this part of the block for cutting, copying or pasting
Undo	Undo an action (CTRL+Z)
Redo	Redo the undone action (CTRL+Y)
Find / Replace	Find any text within a program, and replace it with a new text if desired. A line number can also be searched for instead of a text. (CTRL+F)
Cut	Cut the marked block parts (CTRL+X)
Copy	Copy the marked block parts (CTRL+C)
Paste	Inserted the marked block parts after the cursor position (CTRL+V)



Shortcuts: CTRL+...

To speed operation, some functions can be activated via shortcuts. These functions are also available via soft key or over the context menu.

Some frequently-used shortcuts:

Shortcut	Description
CTRL+S	Save a program
CTRL+F	Find (replace)
CTRL+C	Copy
CTRL+V	Paste
CTL+Z	Undo

Popup windows

Sometimes a popup window for a temporary function (such as searching, saving, etc.) appears within the current application. This popup windows appears on top of the current application window.

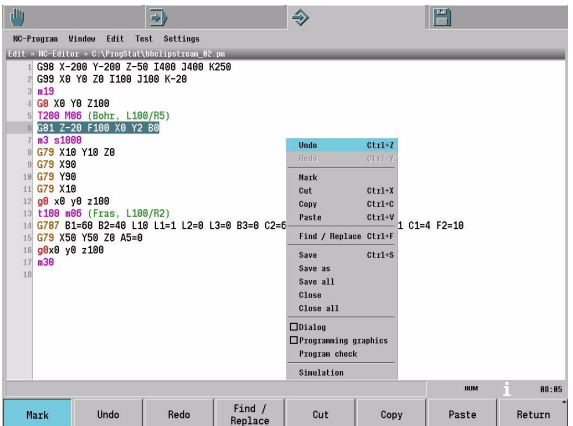
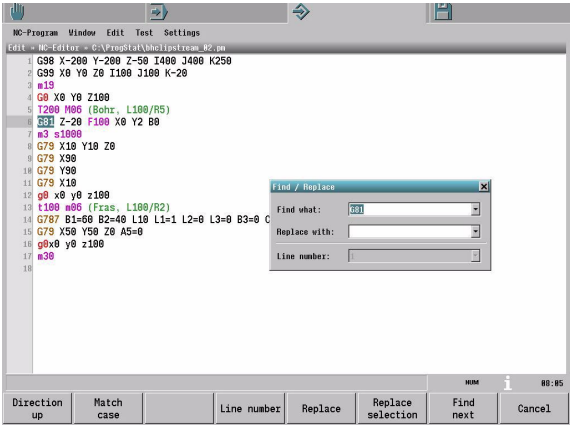
A soft-key row belonging to the popup window is shown. Press **ESC** to close the popup window.

In some cases you can switch the focus between the popup window and the application below it, such as the Find window in the NC editor. In these cases the MillPlus ensures that the correct soft-key row is always displayed.

The focus cannot be changed away from some popup windows. Such popup windows must be closed first.

Context menu

Call the context menu via the ASCII keyboard (applications key) or by clicking the right mouse button. The context menu shows you a list of functions available in the active operating mode. The opened context menu closes automatically if you press **ESC** or if you select a function.



Machine status display (dashboard)



The status display informs you of the current state of the machine tool. The status display appears automatically in the following operating modes:

- Manual
- Automatic

General status display

1 X, Y, Z, A, B, C: Axis positions

- Actual value
- Distance-to-go

Symbol	Description
	Axis clamping
	Axis transformation

2 T: Tool number



- Programmed tool
- Current tool
- Tool index

3 G53 - G59: Datum shift

4 G17 - G18: Machining plane

5 F: Feed rate

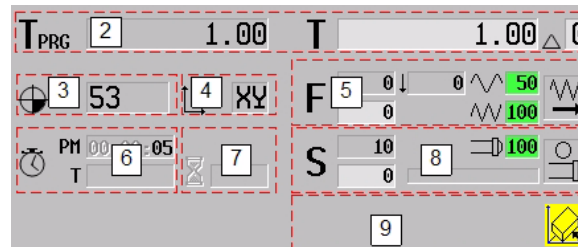
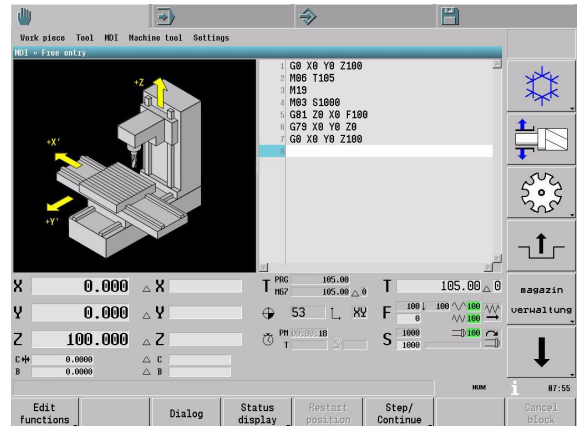
- Programmed feed rate
- Active feed rate
- Feed rate for plunging
- Override value for feed rate/rapid traverse

Symbol	Description
	G0: Rapid traverse
	G1: Feed rate

6 Program run time




- **PM:** Program
- **T:** Tool

7 G4: Dwell time







8 S: Spindle speed

- Programmed
- Spindle override

Symbol	Description
	M5: Spindle stop
	M3: Spindle left
	M4: Spindle right

9 Program status





Symbol	Description
	G73: Scale
	G73: Mirror
	G93: Rotate coordinate system
	G07: Tilt the machining plane

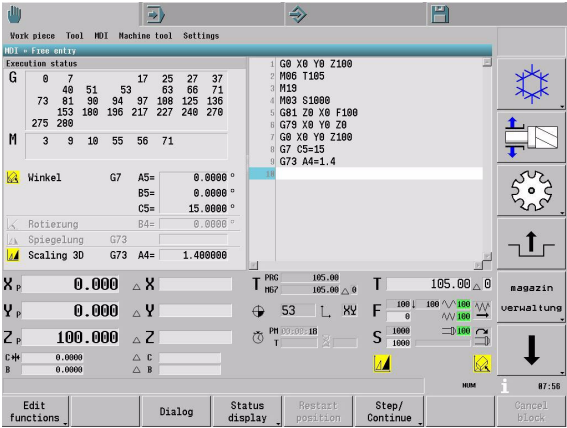
Additional status displays

Additional status displays, with detailed information about the program run, can be shown during program run and MDI.

To switch the status display:

- Press the **Status display** soft key to show a new soft-key row, with a selection of the various possible status displays.

Soft key	Description
	The machine picture is the default display
	Information about the execution status : <ul style="list-style-type: none"> - Active G functions - Active M functions - Angle G7 - Rotation G93 - Mirroring G72 - Scaling G73
	Information about offsets : <ul style="list-style-type: none"> - Axis positions G52, G54, G92+G93 - Tool
	Information about E parameters : <ul style="list-style-type: none"> - Parameters E+ES



1.5 Online Help and Calculator

INFO functions

At all times and in all main operating modes, you can call additional online help or a pocket calculator. Press the INFO key or icon, and then the appropriate soft key, to select one of these functions.



► Call the online help and calculator

Soft key	Description
Programming support	Call online programming support
Operating support	Call online operating support
Error support	Call explanations and fixes in case of error messages
Calculator	Call the calculator

The online programming and operating support appear in a popup window. You can use the mouse pointer to change the size and position of this window.

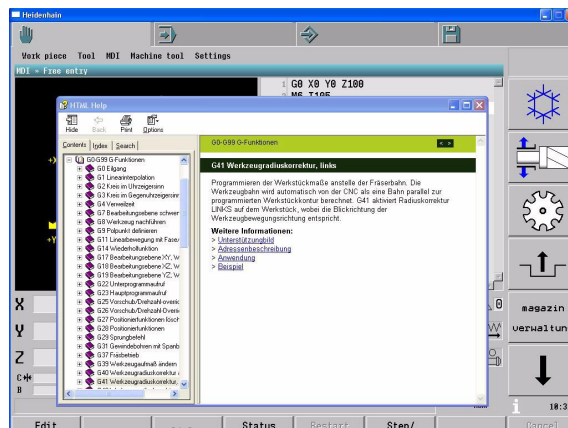
Programming support

Press INFO and then the **Programming support** soft key to find all programming functions, including the programming language, of the MillPlus online. This online support can be of assistance while you are entering a program.

The online help is context-sensitive: if a program is being entered, then the online help jumps directly to the detailed information about a function. If there is no programming context, then the online programming support opens with the overview of function groups.

Operating support

Press INFO and then the **Operating support** soft key to find all operating functions of the MillPlus online.



Error support

In case of an error, the MillPlus automatically shows a brief description of the error in the message line above the soft keys. Error messages occur if:

- Incorrect data was entered during programming
- Logical errors arise in the program during program testing or program run
- System errors during operation

If you require additional information about an error message, press INFO and then the **Error support** soft key.

A popup window appears:

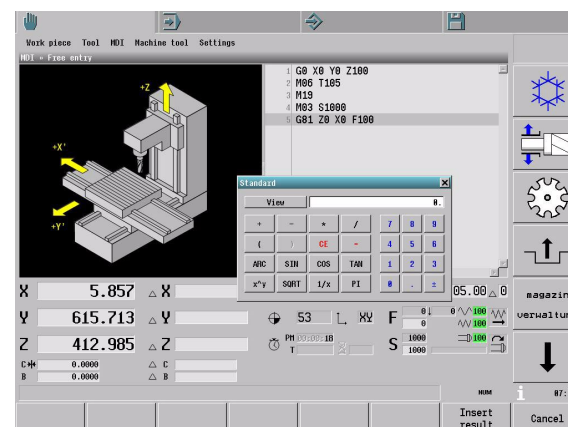
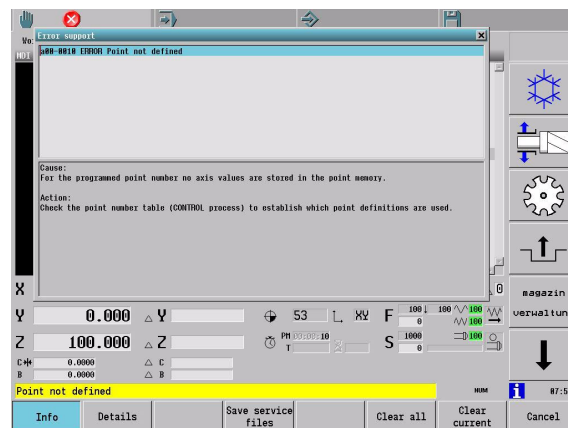
- List of error messages
- A description of the error cause and error fix for each error.

Calculator

The MillPlus features an integrated pocket calculator with the basic mathematical functions. The calculator can be called with INFO from any location.

You can use the calculator to calculate intermediate results during programming. These values can be sent directly to the cursor position in the program editor with the **Insert result** soft key.

Click the **View** button with the mouse to cycle through the three levels of calculator functions.



1.6 User Soft Keys

User soft keys, which you define yourself, are available in addition to the standard soft keys. These user soft keys enable quick and customized operation anywhere on the control.

Operation

User soft keys that are superimposed over the standard soft keys are available in all operating modes. Use this feature to create your own operating level, where you can have numerous operating steps or sequences be performed via a soft key.

Active the user soft keys by:

- Pressing the USER SOFT KEYS key at bottom left on the display unit, or by
- Pressing an operating mode key twice, or by
- Clicking an active operating mode tab with the mouse.



► The user soft keys symbol appears in the system bar.

The standard soft keys appear again after a user soft key has been activated, or after a main operating mode has been selected again.

Recording a user soft key

After placing the mouse pointer over a new user soft key that you want to define, open the context menu. Activate either the "Record UK" or "Cancel record UK" function.

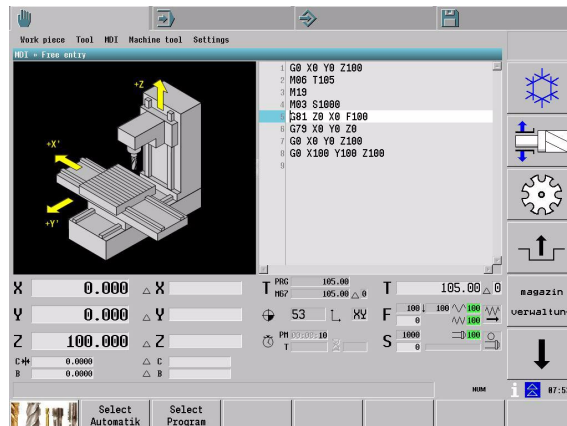


► The recording symbol appears in the system bar.

All keys pressed (on the ASCII keyboard as well as the operating panel, except for the Start and Stop keys) and all mouse clicks are recorded in sequence.

Stop recording by clicking the recording symbol.

You can then assign a text or an image to the user soft key.

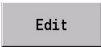


Text or image for user soft keys

You can choose to assign texts or images to user soft keys. The text or the image file name is assigned in a table in the Control (Setup) operating mode. The image file must be saved in the "Userkeys" directory on the "V" drive.



- ▶ Select the Control (Setup) main operating mode.
- ▶ Select the following function from the menu:
Installation >> Configuration >> User soft keys
- ▶ Open the table for editing.

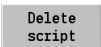
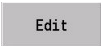


Deleting a user soft key

You can delete a user soft key from the table in the Control (Setup) main operating mode.



- ▶ Select the Control (Setup) main operating mode.
- ▶ Select the following function from the menu:
Installation >> Configuration >> User soft keys
- ▶ Open the table for editing. Remove the text and image name of the recording, and
- ▶ Delete the selected recording.



NAME	TEXT	IMAGE	SCRIPT
Manuell 1			
Manuell 2	Select Automatik		Yes
Manuell 3	Select Program		Yes
Manuell 4			Yes
Manuell 5			
Manuell 6			
Manuell 7			
Manuell 8			
Manuell 9			
Manuell 10			
Automatik 1		tool.bmp	Yes
Automatik 2			
Automatik 3	Select Program		Yes
Automatik 4			
Automatik 5			
Automatik 6			
Automatik 7			
Automatik 8			
Automatik 9			
Automatik 10			
Programmieren 1		tool.bmp	Yes
Programmieren 2	Select Automatik		Yes

1.7 Coordinates and Reference System

Axis configuration

A machine has three linear principal axes (X, Y Z), which are mutually perpendicular. The orientation of these axes is determined by the Z axis, which always runs parallel to the machine tool's spindle. The X axis is that with the greatest traverse path perpendicular to the Z axis. A rotary axis can be assigned to each principal axis. (top figure)

Coordinate system

The CNC can connect points by linear and circular paths of traverse (interpolations). Workpiece machining is programmed by entering the coordinates for a succession of points and connecting the points by linear or circular paths of traverse.

The positions of the CNC machine tool's axes are defined by the following standards: ISO 841, DIN 66217 and EIA RS-267-A. (center figure)

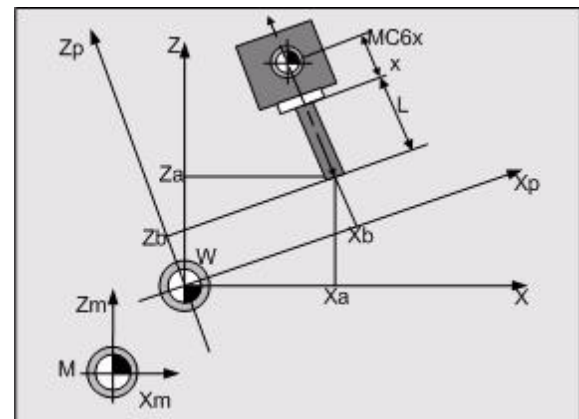
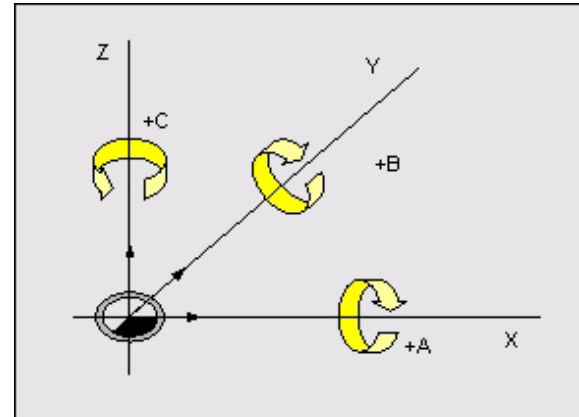
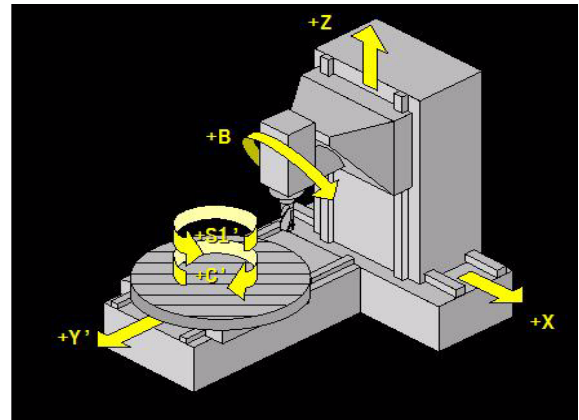
Defining coordinates

The coordinates of points in space (3-D) define traverse paths along the axes. The axis coordinates are in one of three planes (XY plane, ZX plane, YZ plane).

The coordinates can be entered using absolute, incremental, Cartesian or polar coordinates.

G7 coordinates

The position display on the screen can toggle between the position in the G7 plane (X_p , Z_p) or the machine coordinates (X, Y). Both refer to the active datum G52 + G54 + G92/G93. (bottom figure)



Datums

In order to determine the machine datum, a reference run must first be performed after switch-on. The machine datum is specified, since the offset data between the machine datum (M0) and the machine-based reference point (R) is saved via machine configuration data.

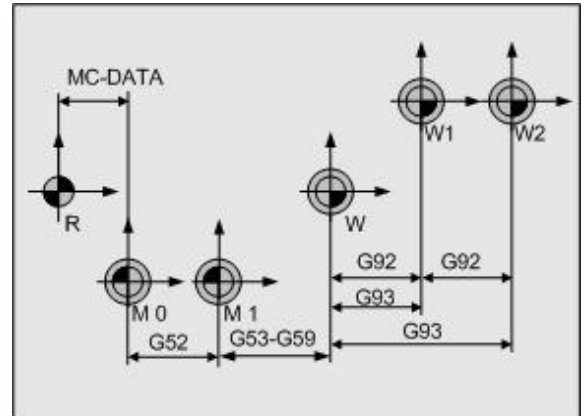
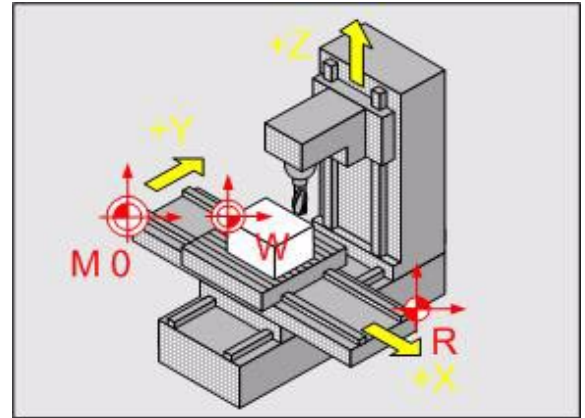
- R = machine reference point
- M0 = machine datum
- W = program datum

The operator specifies a program datum (W) for the workpiece, relative to which the workpiece dimensions are measured. This program datum must also refer to the machine datum, which can be specified with the G52 and/or G54...G59 functions.

Machine-based reference point (R)

Every traversable axis of a machine tool has a stationary point designated as the reference point. The reference points of all axes together form the machine-based reference point (R) (bottom figure).

During the reference run (see "Crossing over the reference point" on page 47), the tool traverses to the reference point of each respective axis. Once this point is reached, the CNC automatically zeroes the axis. The positions of the software limit switches are specified.



2

Manual Operating Mode

2.1 Introduction

The Manual operating mode provides functions for simple machining operations, for setting up the machine, and for setting up and pre-machining workpieces. In this operating mode, you operate and move the machine axes and the spindle manually.

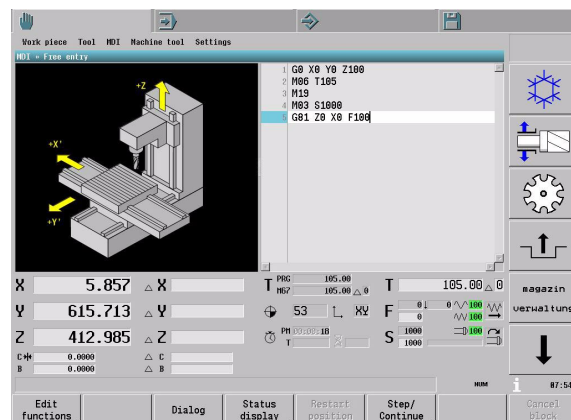
G-code commands for positioning the axes and machining simple workpieces as well as for machine and tool functions are entered and executed blockwise. You can intervene or interrupt program run, and then resume or cancel program run.

You can save the most important machine functions and machining operations in an MDI list and reuse the NC blocks from this list at a later time.

Menu bar

In the Manual operating mode, select subordinate operating modes or functions from the pull-down menus in the menu bar.

Menu	Description
Workpiece	Workpiece setup: Setting the zero points and workpiece measurement
Tool	Tool setup: Tool measurement, entry of tool technology data and setting the plane
MDI	Manual data input and execution of instruction sets
Machine tool	Machine-related functions: Reference-point traverse, axis diagnosis and shutdown of the MillPlus
Settings	Settings parameters for the functions in the Manual operating mode



Menu items



The menu items identified by an asterisk (*) are not supported by this software version.

The pull-down menus contain the following menu items providing the functions or operating modes described below:

Workpiece menu	Description
Touch side	Setting the workpiece zero point with a tool
Determine center	Setting the workpiece zero point in the center of a circle or rectangle by using the tool
Preset axes	Setting the current position as the workpiece zero point
Measure side *	Setting the workpiece zero point with a touch probe
Tool menu	Description
FST entry	Manual data input and execution of technological functions
Measure in jog	Tool measurement by touching the workpiece with the tool
Set plane	Activation of a working plane
MDI menu	Description
Free entry	Manual data input and execution of instruction sets
Machine tool menu	Description
Reference point	Determining the axis position through reference point traverse
NC shutdown	Shutting down the control in order to switch off the machine
Settings menu	Description
Jogging	Axis settings for jog operation

2.2 General Functions

Switch-on



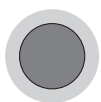
Switch-on and traversing the reference points can vary depending on the machine tool. Refer to your machine manual.

- Switch on the power supply of the machine and the MillPlus.

On a MillPlus with dual-processor system (Windows XP), the startup screen (**Log On to Windows**) appears after switch-on.

- Enter the appropriate data in **User name**, **Password** and **Log on to**. For the data to be entered, refer to your machine manual, the system administrator or your machine supplier.

Booting the MillPlus



- Release the EMERGENCY STOP switch.



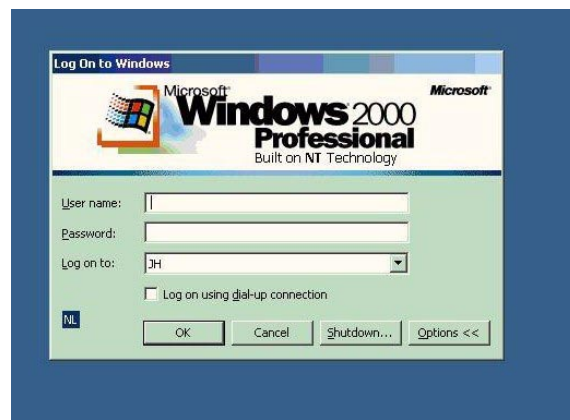
- Press and hold the MACHINE ON button.



- Press the CLEAR key.

The MillPlus starts up in the Manual operating mode and is now ready for operation. You can execute the following functions:

- Positioning the axes with the AXIS-DIRECTION KEYS
- Crossing over the reference points



Crossing over the reference point

After the MillPlus has started up, all axes must cross over the reference point. You can move the axes manually, but you cannot execute any commands in MDI or any programs in Automatic operating mode until the reference points have been crossed over.



Axes that are equipped with absolute encoders do not have to cross over the reference points. If all axes are equipped with absolute encoders, the MillPlus is ready for operation immediately after the machine control voltage is switched on.

Standard working plane

The reference run can be executed separately for each individual axis or for more than one axis simultaneously.

All axes

► Select all axes.

One axis

► Select one axis.



► After you have pressed the START key, the selected axes simultaneously move to the reference point at feed rate.

After the reference point has been crossed over, the MillPlus executes a machine-dependent rapid traverse movement to a fixed position.

The MillPlus is now ready for operation in the Manual operating mode.

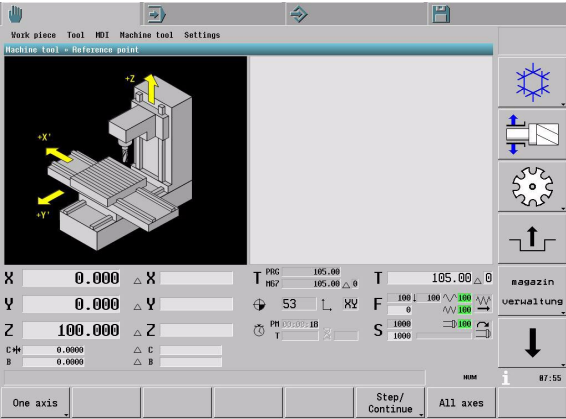


The reference points need only be crossed over if you want to traverse the machine axes. If you only want to edit or test programs, select the Programming operating mode after switching the machine control voltage on. You can then cross over the reference points later.

Traversing the reference point in a tilted working plane



The tilting of the plane is automatically cancelled when the reference point is traversed in an axis.



Switch-off

To prevent data from being lost at switch-off, the operating system of the MillPlus needs to be shut down properly:

- ▶ Select the Manual operating mode.
- ▶ Select **Machine tool > NC shutdown**.
- ▶ Shut down the MillPlus.



Inappropriate switch-off of the MillPlus can lead to data loss.

Moving the machine axes: Jogging



Manual traverse with the axis-direction keys can vary depending on the machine. The machine tool manual provides further information.

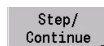
The axes are moved with the axis-direction keys:

- in continuous jog mode
- in incremental jog mode (0.001-1.000), or with a free increment
- at a predefined feed rate, or at a programmed feed rate

See "Jogging" on page 63.

Continuous jog

Use the axis-direction keys to move a single axis or more than one axis simultaneously.



- ▶ Press the **Step/Continue** soft key to select the continuous jog mode.



- ▶ Press the axis-direction key and hold it as long as you wish the axis to move.



- ▶ The axis feed rate is adjusted with FEED RATE OVERRIDE (0-100%).



- ▶ If you press an axis-direction key and the RAPID TRAVERSE key simultaneously, the axis moves at rapid traverse speed.

Incremental jog

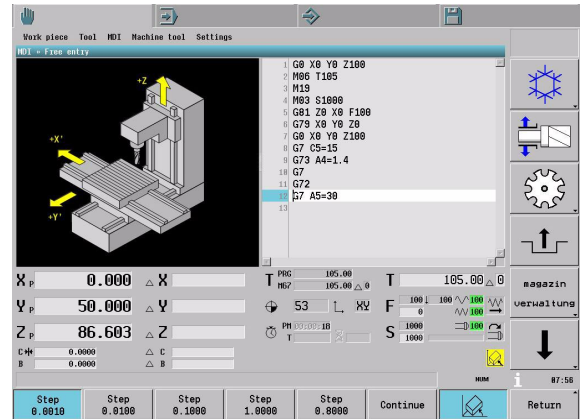
In incremental jog mode, you move a machine axis by a preset distance.

- ▶ Press the **Step/Continue** soft key to call the **Step** soft keys.
- ▶ Set the jog increment with the **Step 0.0010** soft key, for example.
- ▶ Press the axis-direction key. The axis moves in steps of 0.0010.

Moving the axes in a tilted plane

Using the axis-direction keys, you can also move the axes in a tilted plane. When the tilted plane is active, you must switch on jog movement in the tilted plane by pressing the soft key showing the tilted plane symbol. When movement in the tilted plane is active, "P" is displayed next to the axis designations, and the yellow symbol for tilted plane appears in the machine status display.

- ▶ Press the **Step/Continue** soft key in the Manual operating mode.
- ▶ Select tilted plane.



Traversing with the HR 410 electronic handwheel

The portable HR 410 handwheel is equipped with two permissive buttons. The permissive buttons are located below the star grip.

You can only move the machine axes when a permissive button is depressed (machine-dependent function).

The HR 410 handwheel features the following operating elements:

- 1 Emergency Stop button
- 2 Handwheel
- 3 Permissive buttons
- 4 Keys for axis selection
- 5 Keys for actual-position capture
- 6 Keys for defining the feed rate (slow, medium, fast; the feed rates are set by the machine tool builder)
- 7 Direction in which the MillPlus moves the selected axis
- 8 Machine functions (set by the machine tool builder)

The red indicator lights show the axis and feed rate you have selected.



Spindle speed S, feed rate F, tool T and machine function M

Application

In the Manual operating mode the values for tool T, machine function M, spindle speed S and feed rate F are entered with **FST entry** or **MDI Free Data Input**.



The MillPlus provides standard and machine-related machine functions M. The machine tool builder determines which machine functions M are available on your control.

Entering values with FST or MDI > Free entry

Enter the following values:

- 1 Tool call T with machine function M
- 2 Spindle speed S, alternatively with a machine function M
- 3 Feed rate F

Executing the values entered



- ▶ Start spindle rotation at the entered speed S, or start tool change with the entered tool number T.
- ▶ Execute feed rate F, and move the axis/axes at the entered feed rate if axis/axes are programmed.

Adjusting rapid traverse, feed rate and spindle speed

The override knobs are located on the machine operating panel.

With the override knobs, you can vary the spindle speed S and the feed rate F:



- ▶ Rapid traverse from 0% to 100%



- ▶ Feed rate F from 0% to 150%.

With the spindle keys, you can vary the value set for spindle speed S:



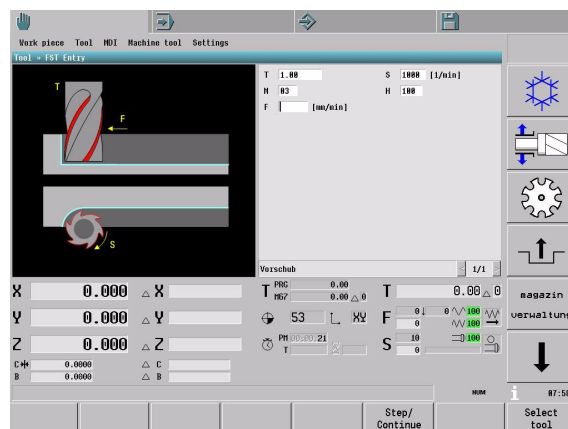
- ▶ Spindle speed S to 100%



- ▶ Increase spindle speed S (%)



- ▶ Reduce spindle speed S (%)



Positioning manual axes

Manual axes are rotary or linear axes that are equipped with a position measuring system, but they do not have a servo drive. In Manual operating mode, manual axes can be moved to the desired position at any time.

Programming manual axes

If manual axes are positioned via the program, the MillPlus will inform the machine operator about it. Program run stops and the following messages appear:

- The status bar displays the "Positioning manual axis" message.
- The distance-to-go display for the affected axes is highlighted in yellow.

The machine operator moves the manual axis/axes until the distance-to-go value is 0. Once the distance-to-go value is within a certain tolerance, the distance-to-go display turns green and program run can be resumed. If the manual axis is not within the tolerance when program run is restarted, the MillPlus issues the error message "Manual axis not in position".



- If the movement of a manual axis is smaller than the tolerance window, the MillPlus nevertheless stops program run and the distance-to-go display is highlighted in green.
- If the difference between the nominal position and the actual position is less than the programming format (0.001 or 0.0001 degrees), the deviation is not treated as a movement and does not lead to a program stop.
- It is not allowed to interpolate NC axes and manual axes. The MillPlus then displays the error message "Axis and manual axis not permitted."

MDI: Free entry

Manual data input and execution of instruction sets. This operating mode provides the following functions:

- 1 Entry and execution of individual blocks
- 2 The blocks entered are saved in the MDI list and can be reused later.

Programming and executing simple machining operations

The MDI operating mode enables you to set up the machine and to execute simple machining operations. The individual instructions (ISO or cycles) are entered blockwise in the MDI input window. Then press the START key to have the MillPlus execute the respective block. The block is saved in an MDI list (see "MDI Menu" on page 61).

The MDI list is saved in a file on the "Work" drive. The **\$MDI.pm** file can be opened in the program editor.



Limitations of MDI

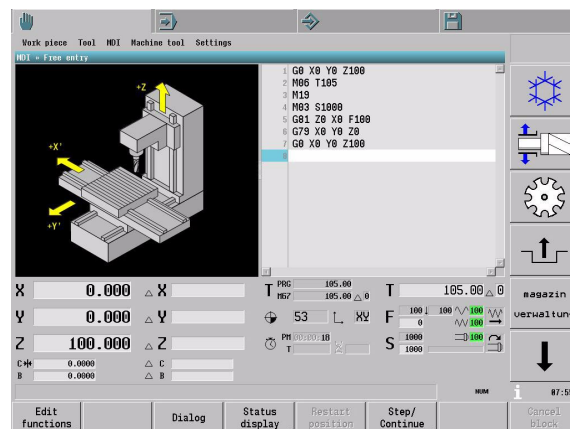
Contour programming, programming graphics, subprograms, program section repeats, and path compensation are not available.

Structure of the MDI list

The MDI list is a list of instruction sets. The blocks entered need not be related to each other. They may be listed in any sequence and do not make up an NC program.

Support for data entry

You can use the support for data entry, while you are entering data. The level of support adapts itself to the respective contents of the program block. Use the function explorer to select a function and enter the appropriate data in the input dialog. See "Function Explorer" on page 27 and "Input dialogs" on page 27.



Executing a block

Press the START key to execute the block you have entered, or a block you have selected from the MDI list. After the block has been executed, the cursor remains on the block.

Status information

During the execution of a block, the MillPlus provides detailed information on the current machining status. For example, the modal G functions and the current contents of E parameters are shown. See "Additional status displays" on page 36.

Edit functions

The Edit functions allow you to change the contents of one or more lines in the MDI list. The **Cut**, **Copy**, and **Paste** functions are available. You can also **find** a text in the MDI list. See "Editing functions" on page 33.

Setting the zero point

When the MillPlus is ready for operation and the reference points have been crossed, the zero points can be set.

You fix a zero point by setting the MillPlus display to the coordinates of a known position on the workpiece.

The measured zero point shift is saved as the active zero point G54-G59 or G54 I...

Preparation

- Clamp and align the workpiece.
- Insert a tool with known radius and length.
- Select the desired zero point G54-G59 or G54 I...

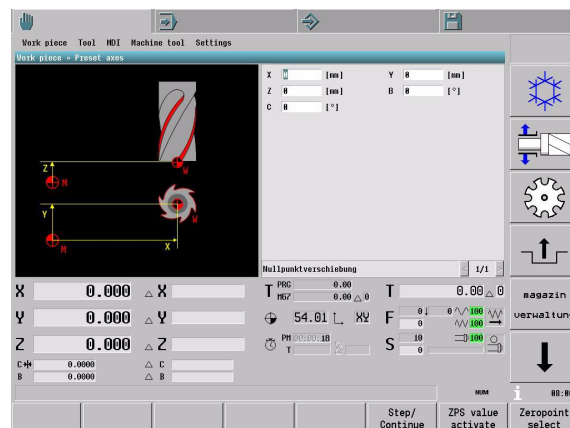
Zero point setting with the axis-direction keys



Protective measure

If the workpiece surface must not be scratched, you can lay a metal shim of known thickness d on it. Then enter a nominal value that is larger than the desired value by the value d .

In the Manual operating mode, the Workpiece menu offers various functions for setting the zero point. See "Workpiece Menu" on page 56.

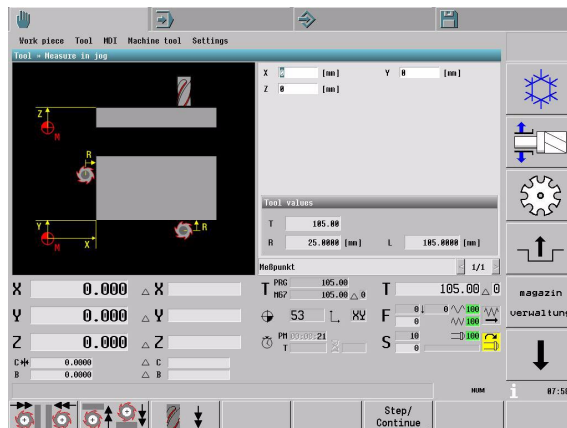


Tool measurement

Each tool is identified by a number. The associated tool data are saved in the tool table. When a tool is inserted from the tool magazine into the spindle with the **Txx** and **Mxx** commands, the tool data are assigned to the active tool.

The MillPlus can measure tool data

- with **Tool > Measure in jog** in the Manual operating mode (see “Tool Menu” on page 59).
- with **MDI > Free entry** in the Manual operating mode (see “MDI Menu” on page 61).
- during program run in the Automatic operating mode. See the tool measurement cycles in the “G600-G699 Measuring Cycles” chapter of the Programming manual.



2.3 Workpiece Menu

Workpiece setup: Zero point setting and workpiece measurement

The **Workpiece** menu contains the following menu items:

- Touch side
- Determine center
- Preset axes

Touch side

Setting the workpiece zero point with a tool.

You set the workpiece zero point by touching a workpiece edge with the tool. Enter the nominal value and confirm the axis traverse direction. The zero point is saved as the active **ZPS number**.

Activating the zero point

The zero point is saved as a **ZPS number**. In order to activate the zero point, the corresponding **ZPS number** must be activated in the zero point table.

Zeropoint
select

- Open the zero point table.

ZPS number
activate

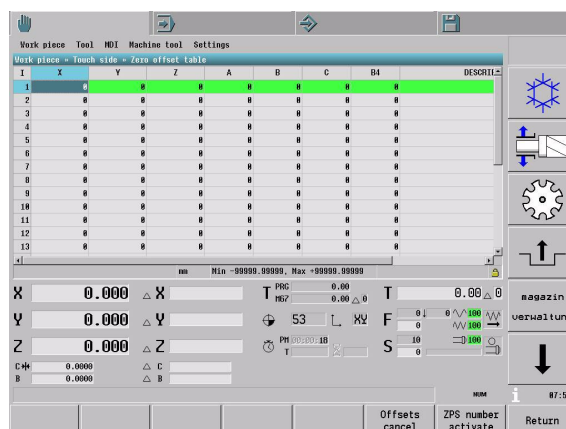
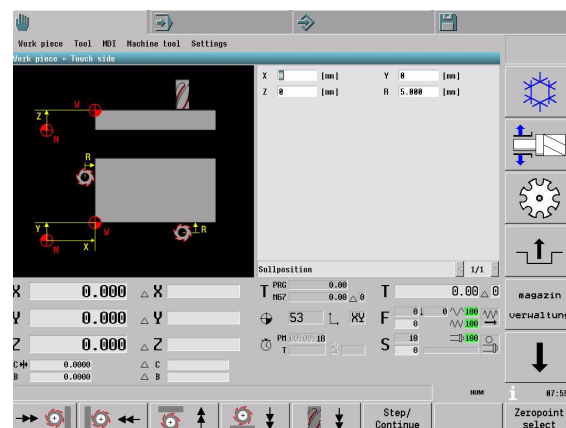
- Use the arrow keys to select the zero point number (e.g. G54 I1) and activate it. The zero point (G54 I1) is displayed in the machine display.

Touching a workpiece edge



- In the input window, enter the nominal value of the workpiece edge to be touched.
- Using the axis-direction keys, move the tool to the workpiece edge and touch (scratch) it with the tool.
- Confirm the appropriate axis traverse direction by soft key.

This saves the zero point of the respective axis as the active **ZPS number**. The other axis reference values are determined in the same manner.



Determine center





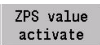
Setting the center of a workpiece with a tool.

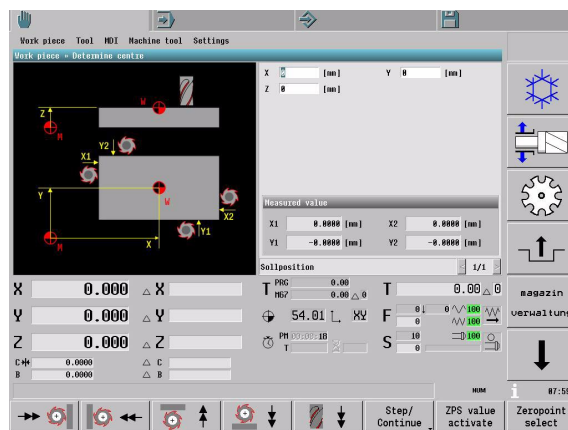
You determine the center of the workpiece as the workpiece zero point by touching two opposing workpiece edges with the tool. Enter the nominal values and confirm the axis traverse direction. The center is determined and saved as the active **ZPS number**.

Activating the zero point number

The zero point is saved as a **ZPS number**. In order to activate the zero point, the corresponding **ZPS number** must be activated in the zero point table (see "Touch side" on page 56).

Touching a workpiece edge

- 
 - Using the axis-direction keys, move the tool to the first workpiece edge and touch (scratch) it with the tool.
- 
 - In the input window, enter the nominal value of the probed workpiece edge.
- 
 - Move the tool axis to the opposite workpiece edge and touch (scratch) it with the tool.
- 
 - In the input window, enter the nominal value of the probed workpiece edge.
- 
 - Confirm the appropriate axis traverse direction by soft key.
- The measured center is saved as a **ZPS number**. The other workpiece centers are determined in the same manner.



Preset axes

Setting the current position with a tool.

To determine the actual position value, move the tool manually to any desired position. Enter the desired nominal position value. The value is saved as the active zero point.

Activating the zero point

The zero point is saved as active zero point G54-G59 or G54 I... . In order to activate it, the corresponding **ZPS number** must be activated in the zero point table (see “Touch side” on page 56).

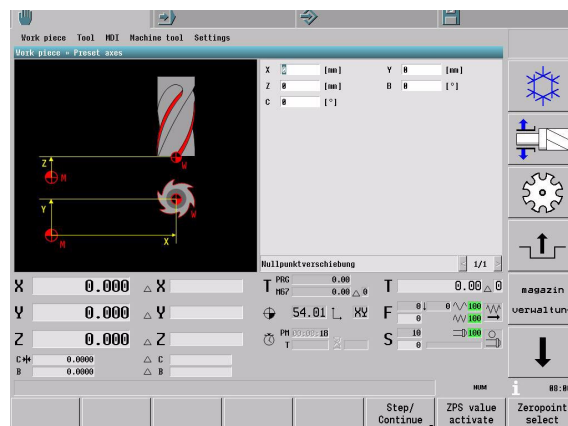
Moving to a position

- Enter the nominal position value in the input window.



ZPS value
activate

- Using the axis-direction keys, move the tool to any desired position.
- The zero point is set in the axes in which the nominal value is entered.



2.4 Tool Menu

Tool setup: Tool measurement, entry of tool technology data and setting the plane.

The **Tool** menu contains the following menu items:

- **FST entry**
- **Measure in jog**
- **Set plane**

FST entry

Entry and execution of technological machine functions.

You can enter machine functions, such as tool **Txx**, spindle speed **Sxxxx**, machine functions **Mxx** and feed rate **Fxxxx**. Press **START** to execute the functions.



Please keep in mind that when you start the functions,

- the spindle may start, and
- the axes will move to the tool change position if a tool is entered.

Entering and executing technological functions



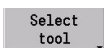
- ▶ Select **Tool** in the menu bar.
- ▶ Select **FST entry** and press **ENTER** to activate it.
- ▶ Enter the data for **T**, **M**, **S** and **F** in the input window.



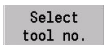
- ▶ After you have pressed the **START** key, the machine will execute the functions. The current values for **F**, **S** and **T** are displayed in the machine display.

Finding tool Txx

If you want to select the tool from the tool table, proceed as follows:

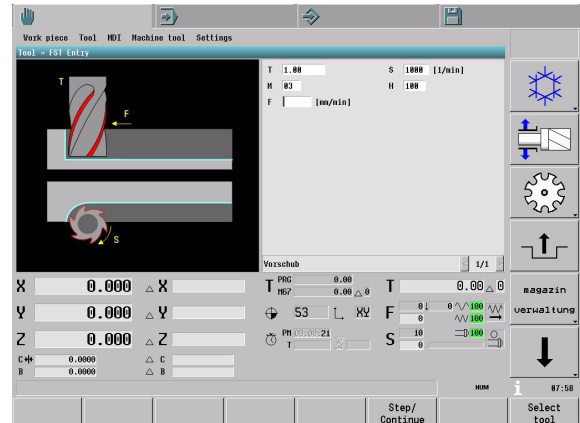


- ▶ Open the tool table.



- ▶ Select the tool number **Txx** with the arrow keys.

- ▶ The tool number from the tool table is transferred to the **FST entry** window.



Measure in jog

Tool measurement by touching the workpiece with the tool.

To measure a tool, touch the workpiece at a known dimension with the tool tip in the working plane or tool axis. When you press a soft key for the axis traverse direction, the MillPlus calculates the tool compensation values R or L from the nominal value entered.



You cannot measure the tool until you have inserted the tool into the spindle and activated the tool. The tool change function can vary depending on the individual machine tool. The machine tool manual provides further information.

Measuring the tool radius



- ▶ Select **Tool** in the menu bar.
- ▶ Select **Measure in jog** and press ENTER to activate it.
- ▶ In the input window, enter the nominal value of the workpiece edge to be touched.



- ▶ Using the axis-direction keys, move the tool to the workpiece edge and touch (scratch) it with the tool in the working plane.



- ▶ Press the soft key for the corresponding axis traverse direction (X or Y) in the working plane.

The MillPlus calculates the tool radius R for this axis and stores it in the tool memory. The compensation value for the measured tool is displayed in the **tool data** window.

To measure the tool length L, use the same procedure (touching the workpiece) in the tool axis direction.

Set plane

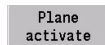
Activate the working plane.

The working plane is switched and activated with the **Plane nn** soft keys.

Setting the plane (e.g. XY)

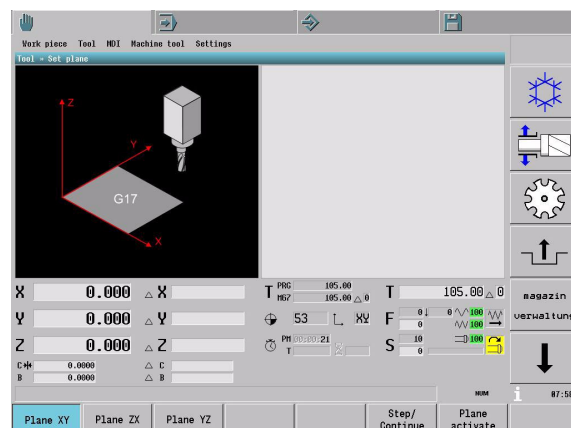
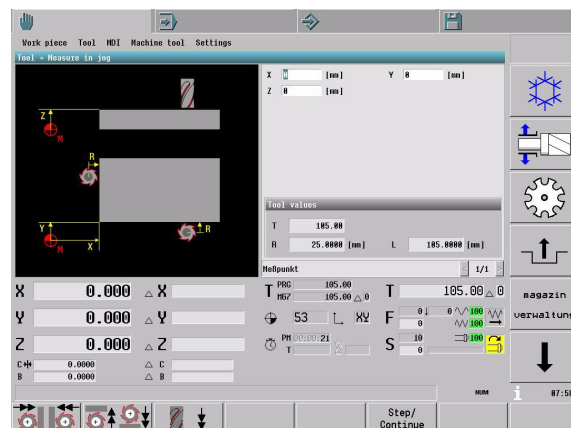


- ▶ Select the **Plane XY**.



- ▶ Activating the plane.

The machine display shows the current plane.



2.5 MDI Menu

Manual data input and execution of instruction sets

The **MDI** menu contains the following menu item:

■ Free entry

Free entry

Manual data input and execution of individual functions and machining operations.

In the **Free entry** window, you enter the individual instructions block by block. Then press the START key to have the MillPlus execute the respective block. The block is saved in an MDI list.. See "MDI: Free entry" on page 52.

Entry and execution



- ▶ Select **MDI** in the menu bar.
- ▶ Select **Free entry** and press ENTER to activate it.
- ▶ Use the keyboard to enter the data in a line of the MDI list.
- ▶ Press the START key to execute the block.



Support for data entry

You can use the support for data entry, while you are entering data. See also "Input dialogs" on page 27.

Dialog

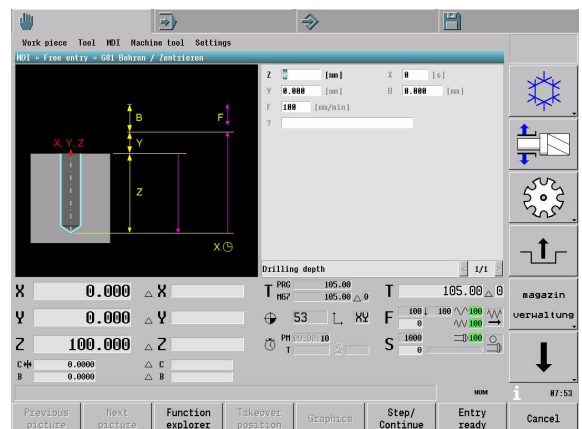
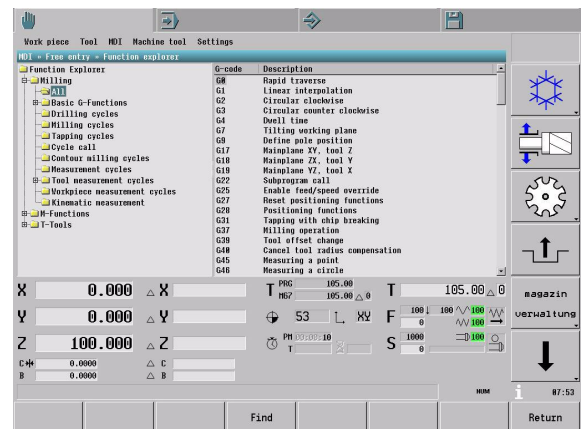
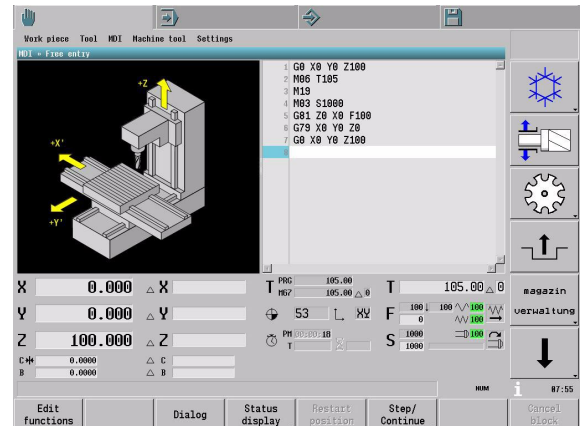
- ▶ Activate support for data entry.

The level of support adapts itself to the respective contents of the program block.

- 1 Entered data is known (e.g. G81): The support window shows the input window for the function (G81: Drilling/Centering)
- 2 Entered data is not known (e.g. Gxx or empty block): The **function explorer** displays an overview of all G code functions.

Entry ready

- ▶ After the entry has been concluded, the MillPlus returns to the **Free entry** window. The block is saved and ready for execution.



2.6 Machine Tool Menu

Machine-related functions: Reference point traverse, axis diagnosis and shutdown of the MillPlus.


The **Machine tool** menu contains the following menu items:

- **Reference point**
- **NC shutdown**

Reference point

Determining the axis position through reference point traverse.

After the MillPlus has started up, all axes must cross over the reference point (see "Crossing over the reference point" on page 47). If you want to repeat the reference run of the axes, proceed as follows:

- 

One axis

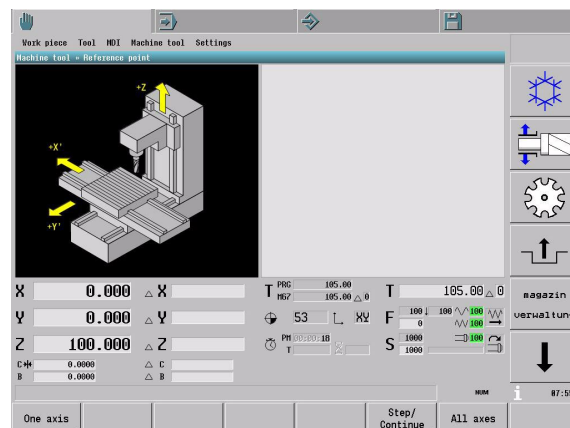
All axes

- ▶ Select **Machine tool** in the menu bar.
 - ▶ Select **Reference point** and press ENTER to activate it.
 - ▶ Select One axis or
 - ▶ All axes to traverse the reference point in one axis or in all axes.

After the axis/axes has/have crossed over the reference point, the current position will be displayed in the actual-value display.




Axes that are equipped with absolute encoders do not have to cross over the reference points. If all axes are equipped with absolute encoders, the MillPlus is ready for operation immediately after the machine control voltage is switched on.



NC shutdown

Shutting down the MillPlus properly.

- 

OFF

- ▶ Select **Machine tool** in the menu bar.
 - ▶ Select **NC shutdown** and press ENTER to activate it.
 - ▶ Press the **OFF** soft key to switch the MillPlus off.

2.7 Settings Menu

Settings parameters for the functions provided by the menus and menu items in the Manual operating mode.

The **Settings** menu contains the following menu items:

■ Jogging

Jogging

Axis settings for operation in jog mode.

The jog feed and the **Free step value** for the axes are defined in this window.

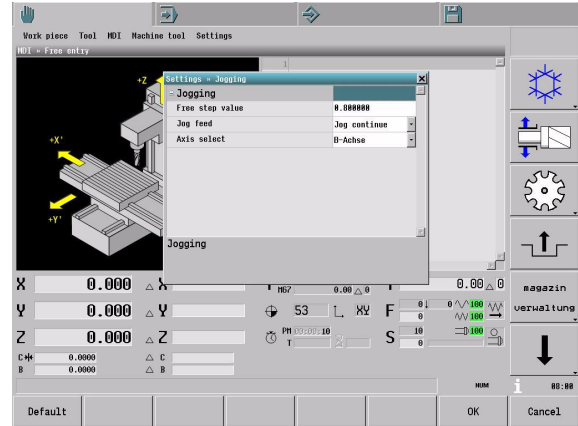


- ▶ Select **Settings** in the menu bar.
- ▶ Select **Jogging** and press ENTER to activate it. The MillPlus opens the **Settings >> Jogging** window.
- ▶ Enter the nominal value of the **Free step value** in the input window.
- ▶ Press the **Step/Continue** soft key to call the soft key on which the step size is displayed.



Possible settings in the **Settings >> Jogging** window:

- 1 Free step value
 - Set the free step value. The step size entered can be selected by pressing the **Step/Continue** soft key and then the corresponding soft key (F5).
- 2 Jog feed
 - Select a predefined or programmed feed rate for jogging.
- 3 Select axis
 - Assign the axis to the freely assignable axis-direction keys +/-.



3

**Automatic Operating
Mode**

3.1 Introduction

In the Automatic operating mode the MillPlus runs a part program in single-block or full-sequence mode to its end. For this the program needs to be selected from the work folder and be prepared by using the file manager.

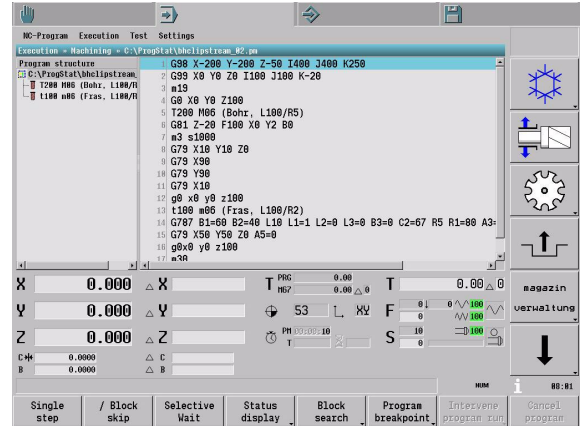
Programs that were created with the MillPlus or with an external CAD system ensure high-accuracy machining at high speed.

The **mid-program startup** function (block search) enables you to run a part program from any desired block. The MillPlus scans the program blocks up to that point.

Menu bar

In the Automatic operating mode you can select a subordinate operating mode or a function by using the menu bar or the menu items.

Menus	Description
NC Program	Select and activate part programs.
Execution	Run the active program in the Machining or Dry run mode of operation. The machine axes are moved.
Test	Functions for program testing before machining. The machine axes are not moved.
Settings	Settings for the Automatic operating mode.



Menu items



The menu items identified by an asterisk (*) are not supported by this software version.

The pull-down menus contain the following menu items providing the functions or operating modes described below:

NC Program	Description
Select	Select an NC part program by using the file manager.
Autostart *	Automatic start of an NC program at a programmed time.
Remote	After activating remote control, you can control all functions from a DNC computer. Functions, such as editing the tool memory, are deactivated at the operating panel of the control.
Ext. program call *	Select an NC program by using program call numbers that are entered in the CNC's memory and activated by the PLC.
Last selected programs	Quick selection of the four last NC programs that were selected.

Execution	Description
Machining	Machining of NC programs.
Dry run *	Test run of an NC program without machine movements.

Test	Description
Simulation *	Graphic testing of NC programs.
Program analysis *	Preliminary program analysis in order to avoid interruptions in the machining process.

Settings	Description
NC program *	Set the font size.
Execution	Program depiction: Show the program structure next to the program
Test *	Settings for the program test.

3.2 General Functions

Program selection

If you want to run a program on the machine, you must first select and activate the program. To select the desired program, select the

■ File manager

The file manager manages all programs stored on the drives and in the folders. Use the mouse or the ASCII keyboard to select and open a program.

■ Quick selection of the last 4 programs that were selected

When you select the Select menu item, the last programs that were selected are listed at the bottom of the window (including the path). You can open a program directly by selecting it from the list.

Program execution

The program you have selected is displayed in the program window. When the program activates subprograms during program run, the active subprogram is displayed in an additional window to the right of the main program. After the subprogram has been finished, the additional window is automatically closed.

Program structure

You can also display the program structure in addition to the selected program. Some important features of the main program and the subprograms, such as tool change, subprogram call, repetitions, etc. are listed in the program structure. This gives you a quick overview of the program structure, and you can see the program sequence during program run. You can also use the program structure for mid-program startup.

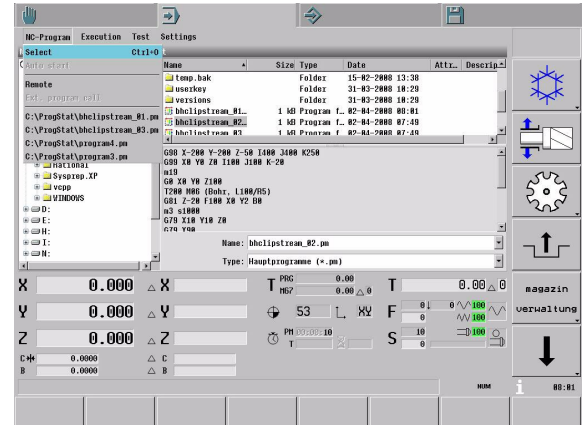
Select **Settings>> Execution** to activate the program structure.

Automatic program run (program run, full sequence)

To start the active program, press the START key. In the automatic program run mode, the MillPlus executes a part program continuously to its end. During program run, the block being executed is highlighted by the cursor. If the Program structure window is also displayed, the highlight is on the block with the active tool.

Program run, "Single step"

In the program run "Single step" mode, you must start each block separately by pressing the START key.



Interruption of program run

There are several ways to interrupt program run:

- Programmed interruption
- Interruption with feed rate STOP key, or feed rate + spindle STOP key
- Switching to program run "Single step"
- Error message

Programmed interruption

You can program interruptions directly in the part program. The MillPlus interrupts the program run at a block containing one of the following entries:

- Miscellaneous functions M0, M01 or M30 (program end)
- Miscellaneous function M06 (determined by the machine tool builder)

Interruption with feed rate STOP key, or feed rate + spindle STOP key

- Feed rate STOP key: The block that is currently being executed by the MillPlus is interrupted. The axis movements stop, and the spindle keeps rotating.
- Feed rate + spindle STOP key: The block that is currently being executed by the MillPlus is interrupted. The axis movements stop, and the spindle stops.

In either case, the "Program stopped" symbol appears in the tab showing the main operating mode. You can resume or cancel program run from this position.

Switching to program run "Single step"

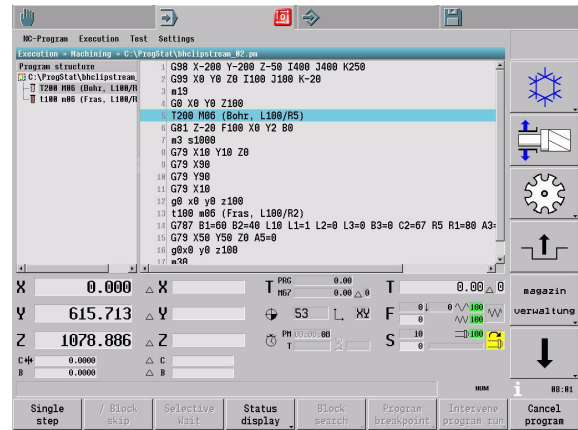
You can interrupt a program that is being run in the automatic program run mode of operation by switching to the program run **"Single step"** mode. The MillPlus interrupts the machining process at the end of the current block. Press the START key to execute the next block.

Breakpoints

You can use breakpoints to interrupt program run without changing the program. The program will then be executed up to the breakpoint and will not be continued until you restart it. See "Program breakpoint" on page 75.

Error message

The machining process may be interrupted, depending on the error message class, the machine error or the NC error. The axis movements are stopped, and the spindle stops. The "Program error" symbol appears in the tab showing the main operating mode, and an error message appears in the status bar.



Moving the machine axes during an interruption

You can retract the machine axes during an interruption in the same way as in the Manual operating mode.



Danger of collision

When retracting the axes, ensure that you press the correct axis direction keys that match the current coordinate system.

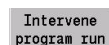
- The tool should be retracted first, and then the axes should be moved away from the workpiece. Please keep in mind the workpiece form and the workpiece position.
- In a tilted working plane (G7), you can use the "Jog in G7 plane" soft key to switch between the tilted working plane and the machine coordinates.

Intervening program run

It may sometimes be necessary to edit the machining parameters during program run. For example, changes can be made in the:

- MDI mode of operation
- Tool parameter table or workpiece zero point table
- Program editor

In order to edit a program or table, you must first interrupt the program run, and then activate the program by pressing the **Intervene program run** soft key.



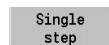
- ▶ Activate intervention.

You also have other possibilities to interrupt the program run:

1 STOP key: Program run is immediately interrupted



- ▶ Press the STOP key. (The current block will not be executed to its end).

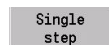


- ▶ Activate Single step mode.



- ▶ Press the START key to execute the interrupted block to its end.

2 Program run "Single step"



- ▶ Activate Single step mode. The current block will be executed to its end.

Resuming program run after an interruption

You can resume program run after an interruption. The possibilities to resume program run vary, depending on the point of interruption.

- 1 Resuming program run at the point of interruption in the block. For this, the block may consist of
 - One single instruction, such as a contour section, in the main program or subprogram.
 - A cycle, such as a drilling cycle, in the main program or subprogram.
- 2 Resuming program run at the beginning of the contour section block, the macro block or the cycle block.
 - This means that some machining operations will be repeated. Program run is therefore interrupted, and the Block search function is used to find the beginning of the block.

Resuming program run with the START key

Resume program

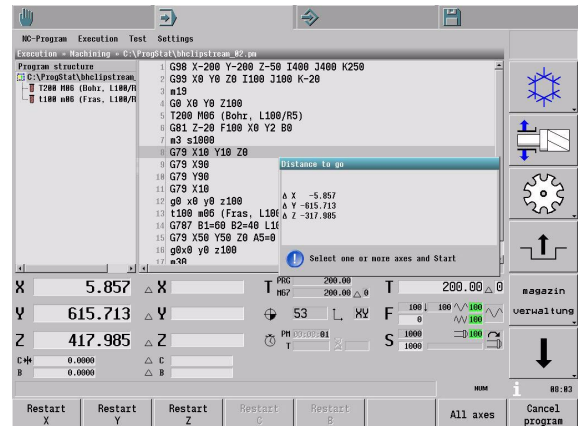
► **Resume program run** soft key: The **Distance to go** window shows the distance from the restart position:

- **All axes** soft key: Move the axes in the sequence that the MillPlus suggests. Press the START key.
- **Restart ?** soft keys: Move the axes in any sequence. Select the axis/axes by soft key and activate them with the START key.

You can resume program run by pressing the START key if the program was interrupted in one of the following ways:

- The STOP key was pressed.
- An interruption was programmed.

If you have moved the machine axes (in the Manual operating mode) during the interruption, you can resume program run (in the Automatic operating mode) in the following way:



Danger of collision

Move the axes in the sequence shown in the **Distance to go** window, and not according to the positioning logic.



► Press the START key to resume program run.

Mid-program startup (block search)

Startup point

The **mid-program startup** function (block search) enables you to run a part program from any desired block. The MillPlus scans the program blocks up to that point.

Block
search

- Activate mid-program startup.

There are several ways to select the startup point:

- Place the cursor on the desired block in the part program.
- Place the cursor on the desired instruction in the program structure, which compactly lists the important instructions. If subprograms are used, you can open a subprogram using the program structure, and then set the startup point in the subprogram.
- Use the **Find** soft key to search for a character string (instruction or text) or a line number in the program.
- You can also use the **Extended search** soft key to enter additional search criteria, such as the name of the program or subprogram, the desired line number or the number of repetitions at the startup point.

Mid-program startup (block search)

After selecting the startup point, you can choose between two possibilities of getting to the startup point:

Goto
cursor

- If you press the **Go to cursor** soft key, the MillPlus does not scan the program up to the selected block (without calculation). Program run starts directly from the selected block.

Search
to cursor

- If you press the **Search to cursor** soft key, the MillPlus scans the program up to the selected block (block search with calculation). After that, program run starts from the selected block.



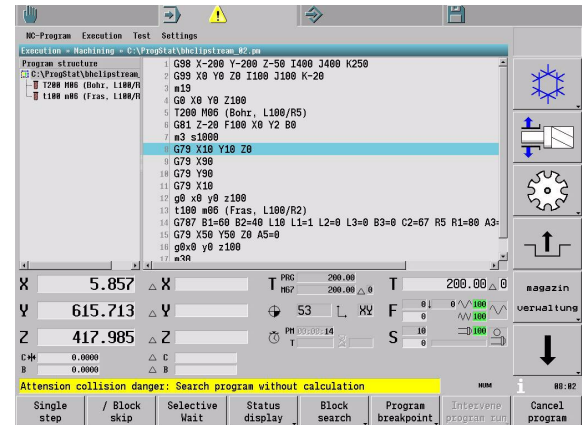
Please note that there is a danger of collision, especially when mid-program startup without calculation (**Go to cursor**) is used. The MillPlus therefore displays a warning.

Symbol

Description



Caution – Danger of collision: Search program without calculation



Starting program run

You cannot start program run from the selected block until the corresponding MSTH functions and the tool position have been reached. The MillPlus displays a suggestion in a pop-up window. You can accept the suggestion by soft key and execute it by pressing the START key. But you can also adapt the suggestion to your requirements.


- MSTH functions:** Execute the **MSTH functions** that have been determined through block search with calculation.

In the **MSTH function** window, select the **MSTH functions** to be executed:

All MSTH

Remove

Single MSTH




▶ Select all **MSTH functions** (accept the suggestion displayed by MillPlus).

▶ Remove an **MSTH function**. All other functions suggested by MillPlus are accepted.


▶ Select one single **MSTH function**.

▶ Press the **START** key to execute the selected **MSTH functions**.

After all selected MSTH functions have been executed, the **Distance to go** window appears.

- Distance to go:** Move the axes from the current position to the startup position. Select the axes:
 - All axes** soft key: Move the axes in the sequence suggested by the MillPlus in the window.
 - Restart ?** soft keys: Move the axes in any sequence.
- 

▶ Press the **START** key to move the selected axes to the startup position.




Danger of collision

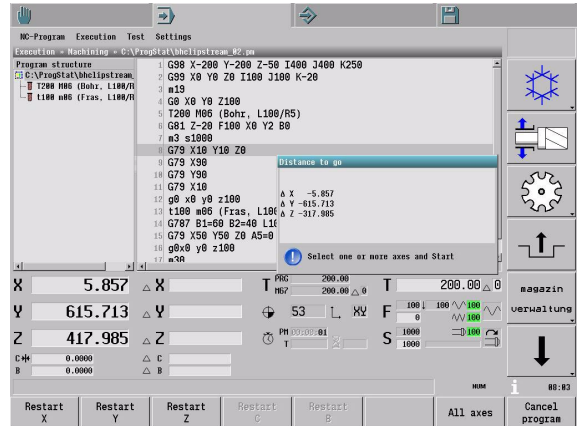
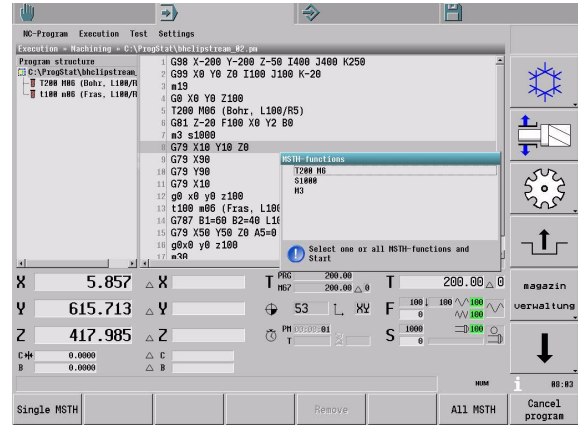
Move the axes in the sequence shown in the **Distance to go** window, and not according to the positioning logic.

After all axes have been moved to the startup position, you can start program execution.

Safety notes and general tips



With mid-program startup, program run is continued from the selected block. To be able to execute this block, the MillPlus may have to execute functions that cause collisions. These can be functions, such as tool change (approaching the tool change position), rotating the (swivel) head and (rotary) table, switching or tilting the working plane, approaching the workpiece on a straight path and continuing the machining process, etc.





- Before mid-program startup, in the Manual operating mode the (rotary) table and the (swivel) head with tool must be retracted to a position from which the above-mentioned functions can be executed without problems and without a danger of collision.
- It is recommended preparing the machine at the startup position in such a way that the program can be continued from the startup position without problems.

Optional block skip

In a test run or program run, the MillPlus can skip over blocks that begin with a slash (/):

Selectable stop (Selective wait)

The MillPlus optionally interrupts the program run or test run at blocks containing M01.

Program breakpoint

Use this function to set breakpoints at program blocks. The program will then be executed up to the breakpoint and will not be continued until you restart it. You can set multiple breakpoints in succession. Breakpoints are not saved in the program.

The breakpoint is identified by a red dot in front of the respective block. The red dot is displayed in a column to the left of the program. You can clear all breakpoints simultaneously, or you can clear an individual breakpoint. For this, you must first activate the program with the **Intervene program run** soft key.

Program
breakpoint

► Define the breakpoint.

There are several ways to find the block in which you want to set a breakpoint:

- Use the cursor, the PAGE UP/DOWN key or the mouse to navigate through the program.
- Use the program structure to navigate.
- Use the **Find** soft key to search for a character string or a line number.

3.3 NC Program Menu

Select and activate part programs. After selection, the MillPlus automatically switches to the active mode of operation (Machining, Dry run or Test). Default setting: Machining mode of operation.

The **NC Program** menu contains the following menu items:

- **Select**
- "Last selected programs"

Select

Select a part program from the program memory.

Use **Select** to select an existing part program in the file manager.

Opening the file manager



- Select **NC Program** in the menu bar.
- Select the **Select** menu item, select a program in the file manager and press ENTER to open it.

Use the **file manager** to find the desired program and press ENTER to activate it. For navigation, see "File manager" on page 28.

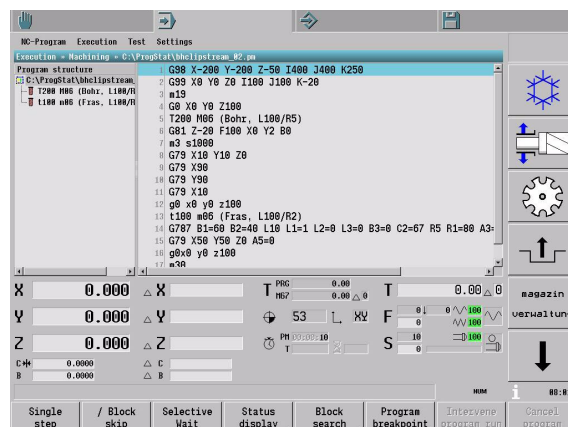
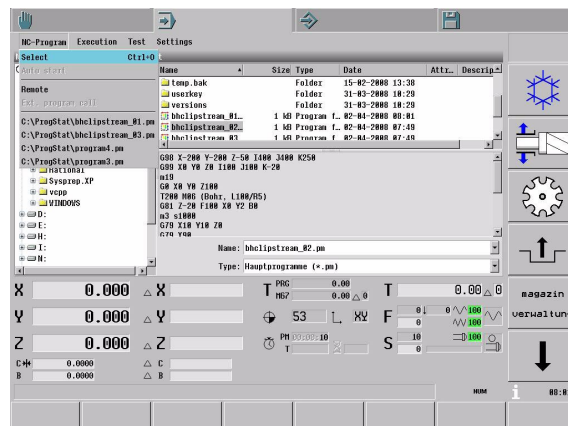
Running a part program

The selected part program is displayed in the program window. Depending on the operating mode (Machining, Dry run, Test) that was last active, you can start the part program.



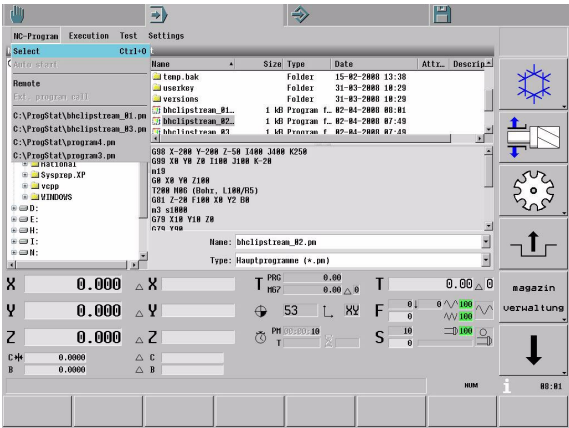
- Press the **START** key to run the part program in full sequence or single step mode.

For more information about executing a part program, see "Execution Menu" on page 78.



Last selected programs

The menu displays a list of the four last programs that were active in the Automatic mode of operation. The name and the path of the programs are saved. This enables you to select these programs with a few keystrokes.



3.3 NC Program Menu

3.4 Execution Menu

Execute the active program in the Machining or Dry run modes of operation. The machine axes are moved.

The **Execution** menu contains the following menu item:

■ Machining

Machining

Execution of part programs.

You can execute an existing part program in this mode of operation. You first have to select and activate the part program in the file manager (see "Select" on page 76).

Activating a part program



- ▶ Select **Execution** in the menu bar.
- ▶ Select **Machining** and activate it with the START key.

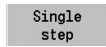
Starting and interrupting a part program

The selected part program is displayed in the program window. You can also activate the program structure by selecting **Settings > Program structure**:

To start program run:



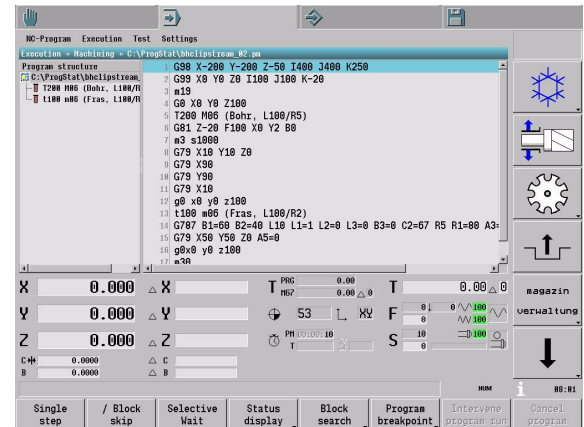
- ▶ Press the START key to run the program in full-sequence mode.



- ▶ Press the **Single step** soft key to run the program in single-block mode. You must restart program run after every end of block.



- ▶ You can interrupt program run at any time. During a program interruption you can:
 - Manually retract the tool from the workpiece and return.
 - Resume program run at the point of interruption or from the point of retraction by pressing the START key.
 - Cancel program run with the **Cancel program** soft key.



Status display

You can activate a status display during program run. The status display appears in a window above the machine status display.

Various status displays are available:

- Execution status
- Offsets
- OEM parameters
- E parameters
- Variables

Press the **Full** program soft key to deactivate the status display.

Block search (mid-program startup)

Use the **Block search** function to enter into a part program at a certain block and execute the program from this block. See "Mid-program startup (block search)" on page 72

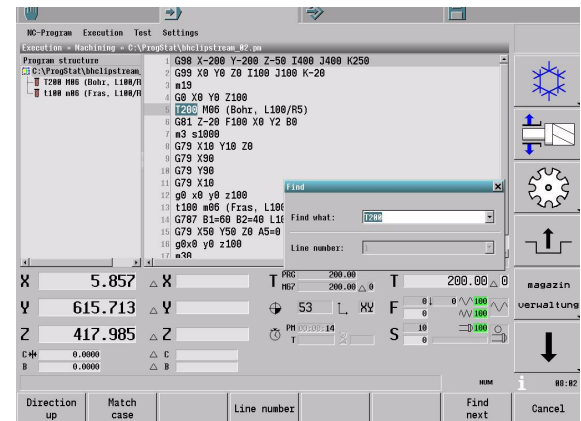
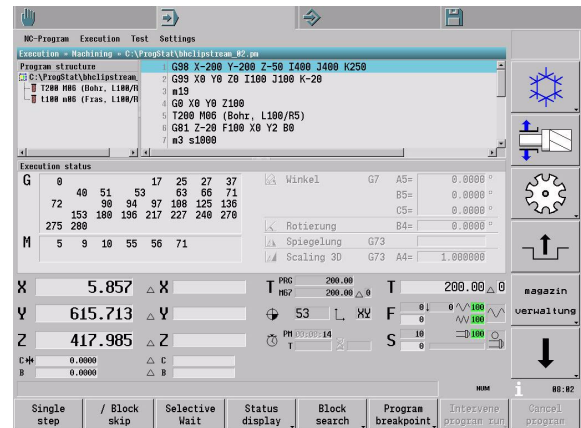
- Block search

▶ Select **Block search** to prepare mid-program startup.
- ↓

▶ Place the cursor on the block at which you want to enter into the program. You can also use the **Find** function to find the startup point.
- ↻

▶ Press the **Search to cursor** soft key if you want the MillPlus to scan the blocks up to the startup point (block search with calculation). Press the **Go to cursor** soft key if you do not want the MillPlus to scan the blocks up to the startup point (without calculation).
- ▶

▶ Press the **START** key to activate program run from the startup point.



The **Settings** menu contains the following menu item:

■ Execution

Execution

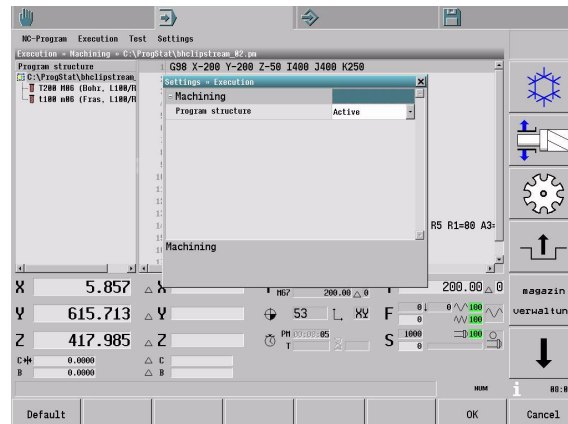
Select **Execution** to display the **Program structure**. The program structure lists some important features of the main program and the subprograms, such as tool change, subprogram call, repetitions, etc.

You can use the program structure to:

- Get a quick overview of the program structure.
- Monitor program run.
- Find a startup point in the program structure for mid-program startup.



- ▶ Select **Settings** in the menu bar.
- ▶ Select **Execution** and press ENTER to activate it. The **Settings >> Execution** window appears.
- ▶ Activate the **Program structure** function.



4

**Programming Operating
Mode**

4.1 Introduction

The Programming operating mode enables you to create and edit programs. New programs can be created immediately via the editor. Existing programs must first be found and opened via the file manager.

Support for data entry is available when entering data. The level of support is automatically adapted to the respective contents of the program blocks. Press the **Dialog > Function explorer** soft key to select the G, M or T function groups, and enter the required data into the dialog box that appears after selecting a function.

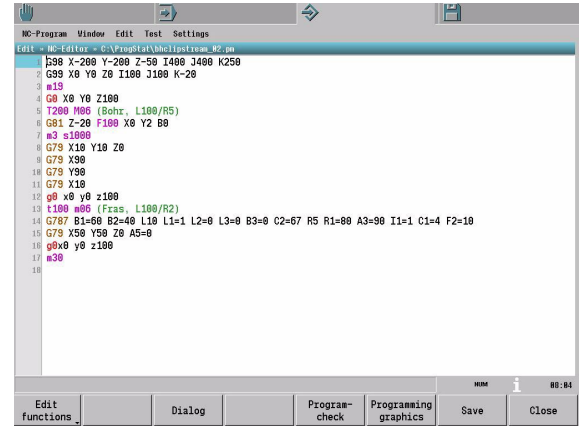
You can also graphically simulate the workpiece to check the programmed geometry. The MillPlus offers comprehensive graphical functions that provide various views for program run simulation.

Menu bar

In the Programming operating mode you can select a subordinate operating mode or a function by using the menu bar or the menu items.

Menu bar

Menu items	Description
NC program	Select a program, or create a new program. After selection, the MillPlus automatically switches to the active mode (editing or graphic simulation).
Window	The Window menu lists the programs that are open in the program editor. The Window menu can contain up to seven programs.
Edit	Program in the editing mode.
Test	Program in the testing mode.
Settings	Settings for the Programming operating mode.



Menu items



The menu items identified by an asterisk (*) are not supported by this software version.

The pull-down menus contain the following menu items providing the functions described below:

NC program	Description
Open	Open an NC part program by using the file manager.
New	Open the program editor for creating a new NC program.
Import from ... *	Import programs in .dxf file format.
Last selected programs	Quick selection of the four last NC programs that were selected.

Window	Description
Menu window	List of the programs that are open in the NC editor.

Edit	Description
NC editor	Open the NC editor to edit a program.

Test	Description
Simulation	The program is graphically simulated on the screen.

Settings	Description
Edit	Define the syntax highlighting, the syntax colors and the syntax check.
Test	Define the program analysis and the graphic simulation.

4.2 General Functions

Creating/Editing a program

Use the **NC editor** to create new part programs, or to edit existing programs. In the menu bar, select **NC program**. Then select Open or New to call the NC editor.

1 Open: Edit an existing program

Use the file manager to select a file. Use the NC editor to edit the file. After editing, you can **save** or **close** the program.

2 New: Create a new program

The program editor is started immediately and you can begin entering the program. After editing, you can **save** or **close** the program. When you save the program, the program is saved to a drive under a program name via the file manager. For more information, see “NC editor” on page 31.

Testing a program

New programs or edited programs can be tested before they are executed on the machine. The MillPlus offers various types of tests:

1 Program check: Syntax check of the complete program

To check the program, press the **Program check** soft key and then the **Start check** soft key. The complete program will be checked. If a syntax error is found, the MillPlus displays an error symbol in front of the incorrect block (the symbol is displayed in the column to the left of the program). All error messages are listed in the **Program check** window. Use the soft keys to navigate through the list or to call detailed information on an error message. You can immediately correct the incorrect block in the NC editor.

2 Simulation: Graphic simulation of the program in various views

The complete program can be simulated in various views. The graphic simulation is displayed next to the NC program. The program can be run in automatic mode or in single step mode. Breakpoints can be set in order to control the program run.

4.3 NC Program Menu

Select a program, or create a new program. After selection, the MillPlus automatically switches to the active mode (editing or graphic simulation).

The **NC Program** menu contains the following menu items:

- **Open**
- **New**
- **Last selected programs**

Open

Open an NC part program by using the file manager.

Select **Open** to activate the **file manager**, in which you select an existing program for editing.

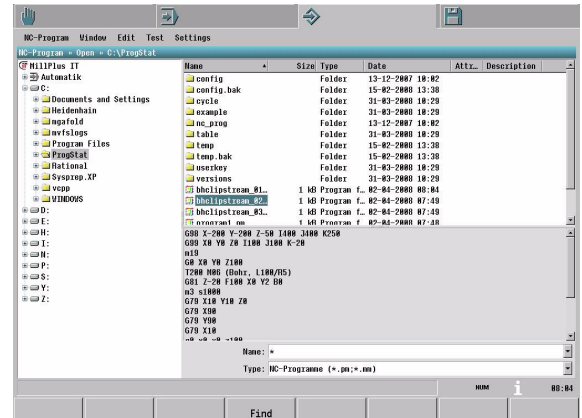
Opening the file manager

For navigating in the **file manager**, see “File manager” on page 28.



- ▶ Select **NC Program** in the menu bar.
- ▶ Select **Open** and press ENTER to open the **file manager**.

Use the **file manager** to find the desired program and press ENTER to activate it.



New

Create a new program with the **NC editor**.



- ▶ Select **NC Program** in the menu bar.
- ▶ Select **New** and press ENTER.

The **New** menu item opens a new, blank program in the NC program editor. The MillPlus automatically gives the program a name, which can be changed when you save the program.

For the editing and support features, see also “Open” on page 85.

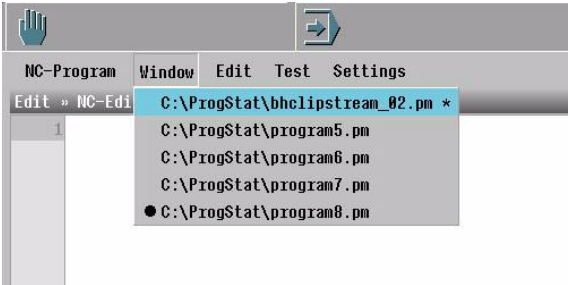
Last selected programs

The menu displays a list of the four last programs that were active in the Programming mode of operation. The name and the path of the programs are saved. This enables you to select these programs with a few keystrokes.

4.4 Window Menu

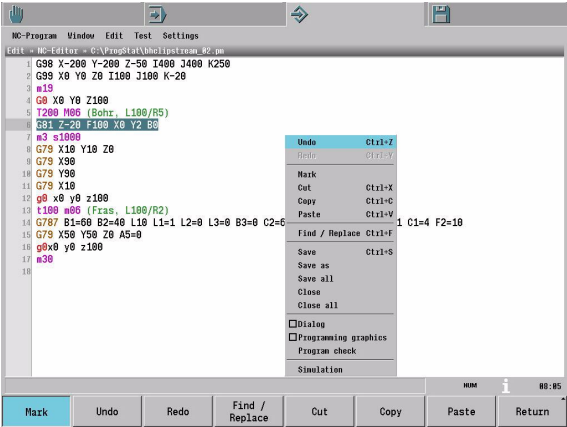
The Window menu lists the programs that are open in the **NC editor**. The Window menu can contain up to seven programs.

The program that is displayed in the editor is identified by a black dot in front of the program name. A program that has been edited is identified by an asterisk (*) after the program name.



Edit functions

The Edit functions allow you to change the contents of one or more lines in the program. The **Cut**, **Copy**, and **Paste** functions are available. You can also find and replace text in the program, or you can search for a line number. For a detailed description, see “Editing functions” on page 33.



Interactive programming graphics

While you are writing the part program, you can have the MillPlus display a wire plot of the programmed contour. The **programming graphics** is activated by soft key. The graphics shows the program section between the **Graphics from** and the **Graphics up to** marks.

While you are entering or editing the program lines, the MillPlus generates each path contour you program in the graphics window. If you do not want the MillPlus to modify graphics during programming, deactivate the **Redraw graphics after modification** function under **Settings, Edit, Programming graphics**.

Start
graphics

► Start the **programming graphics**.

If you select individual traverse paths in the program with the cursor, they will be displayed in orange in the **programming graphics**. If you select a traverse path in the graphics with the mouse, the cursor will jump to the associated block in the program.

Select the screen layout (position of program and graphics) in **Settings, Test, Program list** (no program list, or program list at bottom, right or left).

Graphic simulation

Use the **Graphic functions** soft key (or with CTRL + "+" key) to adjust the graphics. You can select the desired view (2-D or 3-D view) or adjust the graphics (the graphics can also be adjusted with CTRL + "+" key).

- Rotate (CTRL + right mouse-pad button)
- Magnify/Reduce (CTRL + "+" / CTRL + "-")
- Shift (CTRL + left mouse-pad button)

Change the program section displayed in the **programming graphics** by moving the program-end mark:



► From the NC editor, open the soft-key row containing the program-end mark (Graphics up to).

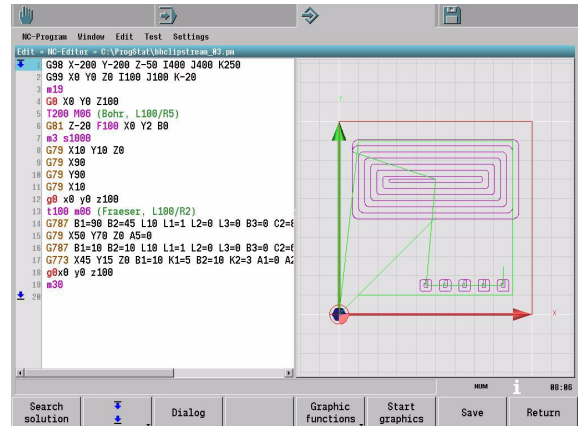
Graphics
up to

► Place the cursor on the desired block in the program and press the **Graphics up to** soft key. Press the **Start graphics** soft key to display the **programming graphics** up to the program-end mark.



Default

► Use this soft key to reset the mark to the actual program end (default setting).



Program check

The program that is active in the **NC editor** is checked for syntax errors block by block.

■ Program context: Active / Not active

If **Program context** is active, modal G functions are taken into account in the syntax check. If, for example, G7 is active, the MillPlus checks whether the subsequent blocks are allowed when G7 is active.

■ Configuration: Active / Not active

If **Configuration** is active, the MillPlus checks whether the blocks are allowed in the current axis configuration. For example, a block with A address is incorrect if the A axis is not configured for the machine.

■ Compatibility: Active / Not active

If **Compatibility** is active, the MillPlus checks whether the execution of the respective program is compatible with the previous version V5.xx.

If you want the MillPlus to check the program, you must start the program check:

Start check

- Press the **Start check** soft key. If the MillPlus finds syntax errors in the program, it displays the number of error message in the message line. All error messages are listed in the **Program check** window.

Message Info

- Use the soft keys to navigate through the error list. Detailed information on the error (cause and correction) can be called with the **Message info** soft key.

The functions can be activated and deactivated under **Settings >> Edit**.

Online help

You can call additional information about programming and operation at any time. See also "INFO functions" on page 37.

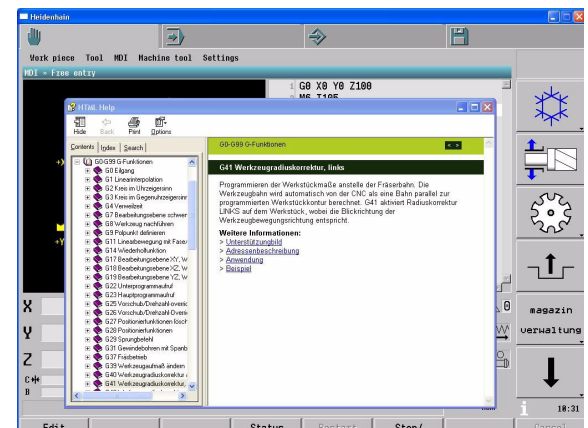
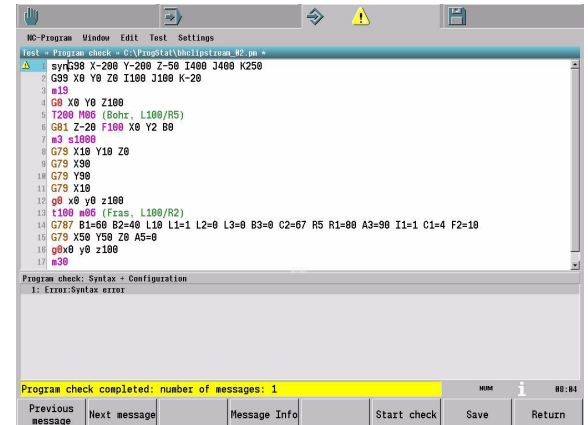


- To activate the online help, press the Info key on the screen.

The online help is displayed in window that pops up over the displayed program. The position and the dimensions of the window can be changed using the mouse pointer.

Use the following soft keys to select the desired information:

- Programming support
- Operating support
- Error support
- Calculator



4.6 Test Menu

Functions for program testing.

The **Test** menu contains the following menu item:

■ Simulation

Simulation

The graphic simulation depicts the workpiece as if it were being machined with a cylindrical tool or a ball-nose cutter. The graphic simulation can be run while the program is being executed in the Automatic operating mode.

- Select **Test** in the menu bar.
- Select **Simulation** and press ENTER to activate it.

The screen layout for program and graphic simulation varies depending on the settings defined in **Settings >> Test >> Simulation, Program list**. See "Settings Menu" on page 95.

Starting/stopping the graphic simulation




Use the **Graphics start** and **Stop** soft keys to start/stop the graphic simulation. The simulation can be run in single step mode or in automatic program run mode.

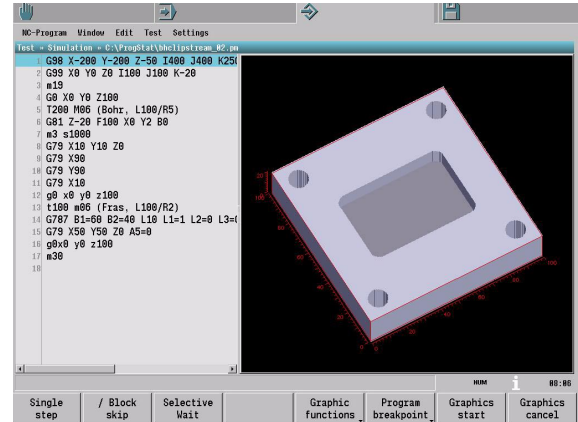
Start graphics ▶ Start graphic simulation.

During simulation you can use the **Selective wait** or **/Block skip** soft keys.

Graphic views and functions

You can choose between different views for the simulation. The MillPlus also provides various functions for modifying the view of the workpiece.

- Graphic functions** ▶ Select Graphic functions.
- View** ▶ Select the desired view: Wire plot, 2-D, 2.5-D, 3-D, high-resolution 3-D graphics
-  ▶ Rotate, magnify or reduce the workpiece.
-  ▶ Move the workpiece within the window.
-  ▶ Detail magnification of a part of the workpiece.






Views and associated functions


You can choose between different views for the graphic simulation. The MillPlus also provides graphical functions for optimizing the screen display.

If you have activated the corresponding function in **Settings >> Test >> Simulation**, the line numbers are displayed in the simulation, for example. See "Settings Menu" on page 95.




3-D wire plot

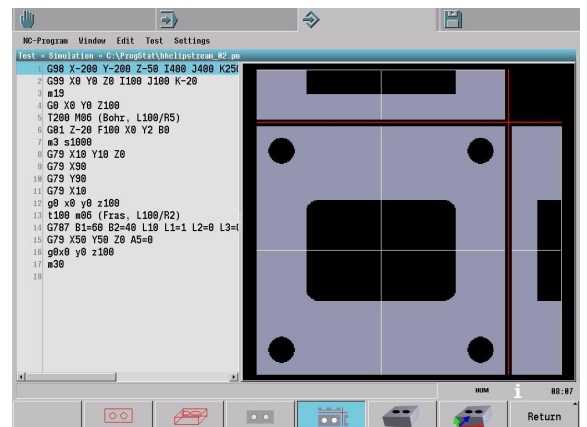
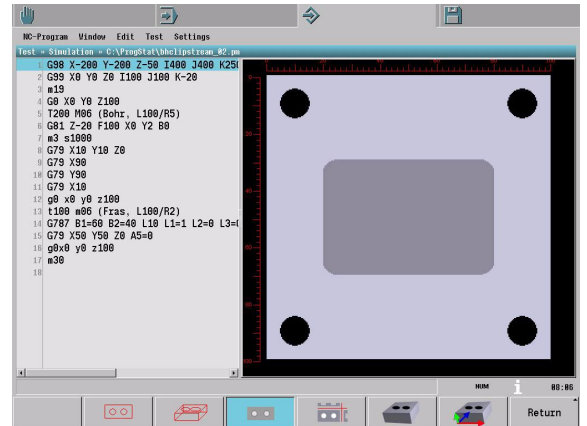
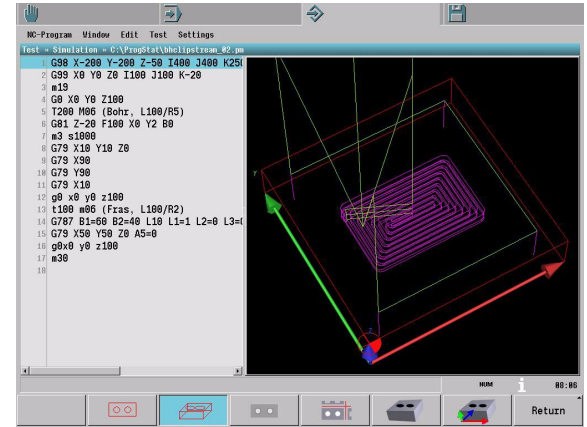
View	Function	Description
		The tool movements, rapid traverse and feed rate are displayed in different line colors.
		Functions for rotating and magnifying/reducing
		Functions for moving the image to the left/right or up/down

2-D view




View	Function	Description
		Top view of the workpiece. Regarding depth display, remember: The deeper the surface, the darker the shade. This is the fastest of the graphic display modes.

2.5-D view





View	Function	Description
		Top view with two sectional planes, similar to a workpiece drawing. In this view, functions for magnifying details are available and sectional planes can be set.
		Functions for moving the sectional plane to the left/right, to the front/back or up/down
		Magnify/Reduce the depiction of the workpiece

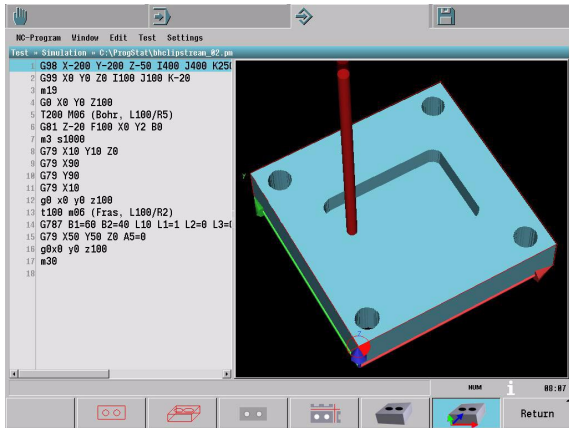
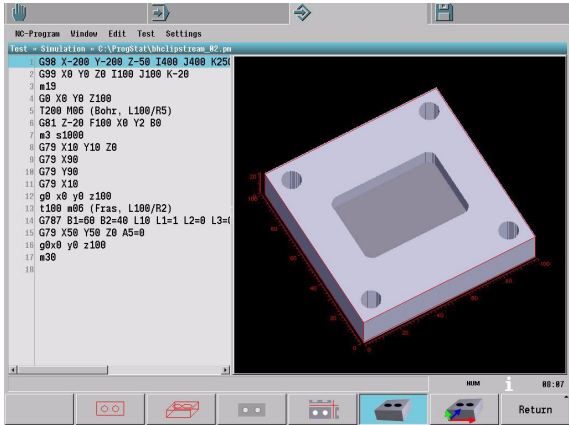


■ 3-D view

Views	Function	Description
		<p>The workpiece is displayed in three dimensions. You can rotate the 3-D display about the vertical and horizontal axes.</p> <p>You can also use the mouse pad to execute this function (hold down the CTRL key and drag the right mouse button).</p> <p>Functions for magnifying details are available in this view.</p>
		Functions for rotating and magnifying/reducing
		Magnify/Reduce the depiction of the workpiece

■ High-resolution 3-D view

Views	Function	Description
		<p>This view is only possible once the simulation has finished. The high-resolution graphics also displays machining operations in the tilted working plane.</p> <p>Functions for magnifying, rotating, shifting and magnifying details are available in this view.</p> <p>After the graphics has been restarted, the simulation automatically returns to the 3-D view. The high-resolution view can be selected once simulation has finished.</p>
		Functions for rotating and magnifying/reducing
		Functions for moving the image to the left/right or up/down
		Magnify/Reduce the depiction of the workpiece



Workpiece blank

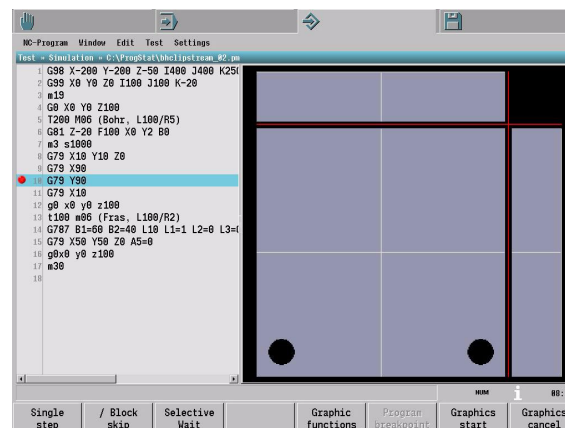
In the NC program, the workpiece blank must be defined with G99. A workpiece blank can only be defined as a quadrangle. G98 and G195 can also be used to define a workpiece blank.

Table values in the graphic simulation

A copy of the current tables (tools, E parameters, etc.) is used in the graphic simulation. The current table values are not changed by the simulation.

Program breakpoints

Use the **Program breakpoint** soft key to set breakpoints at the desired positions in the program. The MillPlus stops graphic simulation before the breakpoint. To continue the graphic simulation process, press the **Graphics start** soft key. See also "Program breakpoint" on page 75.



4.7 Settings Menu

Settings parameter for the functions provided by the menus and menu items in the Programming mode of operation.

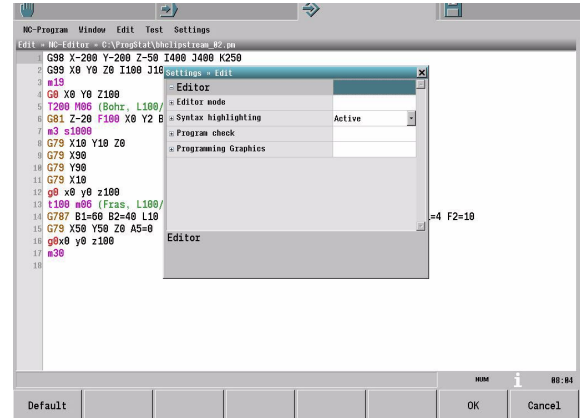
The **Settings** menu contains the following menu items:

- **Edit**
- **Test**

Edit

The following functions can be defined in the Edit menu item:

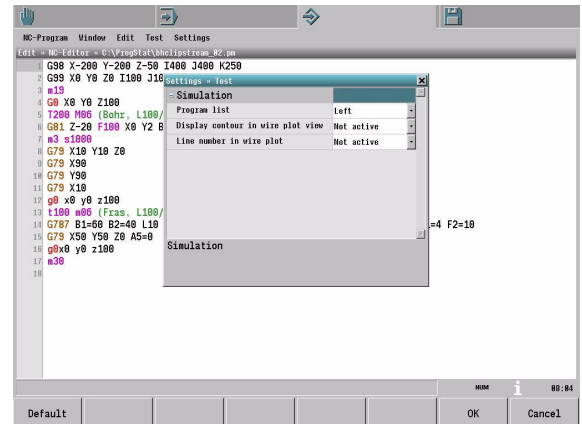
- **Editor mode:** Activate/Deactivate the syntax check and the Editor dialog mode.
- **Syntax highlighting:** Activate/deactivate syntax highlighting, and use the syntax colors to distinguish between the elements in a block. You can choose the colors for the following block elements from a palette:
Block ID, comment, character string, keyword, E parameter, contour, FMST, G0, G functions, address, invalid address.
- **Program check:** Activate/deactivate inspection of program context (including modal functions / only within a block), configuration (including axis configuration) and compatibility (including compatibility with V5xx).
- **Programming graphics:** Define whether the programming graphics is to be executed automatically or after a start command.



Test

The following functions can be defined in the Test menu item:

- **Simulation:** You can define three settings in the Simulation window.
 - Program list: The program list is displayed next to the graphic simulation (None, Right, Bottom or Left)
 - Display contour in wire plot view: Active / Not active
 - Line number in wire plot: Active / Not active



5

**Control (Setup)
Operating Mode**

5.1 Introduction

All data for tools, datums, etc., as well as data regarding the machine configuration is managed in the Control (Setup) operating mode. The other operating modes use this data.

The NC-Explorer is a file manager with which you can handle your files and directories.

There are management functions with which you can control your production tasks for purposes of automation.

Functions for diagnosing and optimizing the control and machine are available during commissioning as well as during operation of the MillPlus.

Menu bar

In the Control (Setup) operating mode you can select a subordinate operating mode or a function via the menu bar or the menu items.

Menu bar

Menu items	Description
Tables	The values for tools, E parameters and points are saved in tables. The tools are listed in groups of applications
NC-Explorer	Use the NC-Explorer to manage your files and directories
Automation	Management of automation tasks and pallets
Installation	Machine and system configuration data, as well as support functions for commissioning and maintenance
Settings	Settings specific to the Control operating mode

The screenshot shows the NC-Explorer software interface. The title bar includes 'Tables NC-Explorer', 'Automation', 'Installation', and 'Settings'. The main window displays a table with columns: P, T, T_NO, IDX, NAME, L, and R. The table lists various tools and their parameters. The 'Tools' menu is open, showing options like 'All', 'Cutter', 'Drill', 'Tap', 'Cutting tool', and 'Measuring probe'. The 'Cutter' option is selected, and the table displays data for various cutting tools, including 'Cutter, Frauser: Empty, Leer' and 'Turning tool, Drehwerkzeug: Empty, Leer'.

P	T	T_NO	IDX	NAME	L	R
1.001	1000.0	1.00	0	Cutter, Frauser: Empty, Leer	0	0
1.002	1001.0	2.00	0	Cutter, Frauser: Empty, Leer	0	0
1.003	1002.0	3.00	0	Cutter, Frauser: Empty, Leer	0	0
1.004	1003.0	4.00	0	Cutter, Frauser: Empty, Leer	0	0
1.005	1004.0	5.00	0	Cutter, Frauser: Empty, Leer	0	0
1.006	1005.0	6.00	0	Cutter, Frauser: Empty, Leer	0	0
1.007	1006.0	7.00	0	Cutter, Frauser: Empty, Leer	0	0
1.008	1007.0	8.00	0	Cutter, Frauser: Empty, Leer	0	0
1.009	1008.0	9.00	0	Turning tool, Drehwerkzeug: Empty, Leer	0	0
1.010	1009.0	10.00	0	Turning tool, Drehwerkzeug: Empty, Leer	0	0
1.011	1010.0	100.00	0	Cutter, Frauser: L100 R2	100	2
1.012	1011.0	101.00	0	Cutter, Frauser: L101 R5	101	5
1.013	1012.0	102.00	0	Cutter, Frauser: L102 R10	102	10
1.014	1013.0	103.00	0	Cutter, Frauser: L103 R15	103	15
1.015	1014.0	104.00	0	Cutter, Frauser: L104 R20	104	20
1.016	1015.0	105.00	0	Cutter, Frauser: L105 R25	105	25
1.017	1016.0	200.00	0	Drill, Bohrer: L100 R5	100	5
1.018	1017.0	300.00	0	Tap, Gewindebohrer: L100 R5	100	5
1.019	1018.0	400.00	0	Turning tool, Drehwerkzeug: Empty, Leer	0	0
1.022	1037.0	10022.00	0	Cutter, Frauser: L105 R2	05	2
1.023	1022.0	000.00	0	Cutter, Frauser: L100 R5 R2-5	100	5

Menu items



The menu items identified by an asterisk (*) are not supported by this software version.

The pull-down menus contain the following menu items providing the functions described below:

Tables	Description
Tools	Tool data of all tools, divided into groups
Tool magazine	Tools and their pocket numbers in the magazine
Zero points	Tool- and pallet-specific datums
Parameter	Overview of E and ES parameters
Points	Predefined points
NC-Explorer	Description
File	Management of files and directories
Automation	Description
Tool management *	Illustration of the tool magazine
Installation	Description
Diagnosis	Diagnostics functions of the control
Compensation	Configuration and axis compensation
Configuration	System configuration with machine-configuration data
System backup *	Data backup during installation
Service	Support functions, such as oscilloscope, axis values (OLM), PLC values, SIK, MOD, network, error display
Settings	Description
Tables	Setting for the tool-dialog mode

5.2 Menu: Tables

The **Tables** menu is used to view and edit the various tables for tools, parameters and points. The menu window shows an overview of the available tables. If there is an arrow after the table name, then further selection is possible.

The **Tables** menu contains the following items:

- **Tools**
- **Tool magazine**
- **Zero offsets**
- **Parameter**
- **Points**

Navigation is identical in all the tables. You must press the **Edit** soft key if you want to edit the table. Press the **Find** soft key to search for a value or text in the table.

Tools

The tool table contains a definition of each tool, as well as its data. Each tool is identified by a number. When working with tool tables, you can also enter tool names for each tool.

The tool number 0 is automatically defined as the zero tool with the length L=0 and the radius R=0. In tool tables, tool T0 should also be defined with L=0 and R=0.

See the Technology chapter of the Programming Manual for a complete overview of the tool-parameter definitions.

Opening and editing a tool table

The **Tools** table can be divided into the various tool types, based on the application groups. All tools are shown in the **All** group. In the other groups, only those tools assigned to the selected application group are shown.

Here are the tool application groups:

- Cutter
- Drill
- Tap
- Cutting tool
- Measuring probe

P	T	T_M0	IDX	NAME	L	R
1.001	1000.0	1.00	0	Cutter, Fraeser: Empty, Leer	0	0
1.002	1001.0	2.00	0	Cutter, Fraeser: Empty, Leer	0	0
1.003	1002.0	3.00	0	Cutter, Fraeser: Empty, Leer	0	0
1.004	1003.0	4.00	0	Cutter, Fraeser: Empty, Leer	0	0
1.005	1004.0	5.00	0	Cutter, Fraeser: Empty, Leer	0	0
1.006	1005.0	6.00	0	Cutter, Fraeser: Empty, Leer	0	0
1.007	1006.0	7.00	0	Cutter, Fraeser: Empty, Leer	0	0
1.008	1007.0	8.00	0	Cutter, Fraeser: Empty, Leer	0	0
1.009	1008.0	9.00	0	Turning Tool, Drehwerkzeug: Empty, Leer	0	0
1.010	1009.0	10.00	0	Turning Tool, Drehwerkzeug: Empty, Leer	0	0
1.011	1010.0	100.00	0	Cutter, Fraeser: L100 R2	100	2
1.012	1011.0	101.00	0	Cutter, Fraeser: L101 R5	101	5
1.013	1012.0	102.00	0	Cutter, Fraeser: L102 R10	102	10
1.014	1013.0	103.00	0	Cutter, Fraeser: L103 R15	103	15
1.015	1014.0	104.00	0	Cutter, Fraeser: L104 R20	104	20
1.016	1015.0	105.00	0	Cutter, Fraeser: L105 R25	105	25
1.017	1016.0	200.00	0	Drill, Bohrer: L100 R5	100	5
1.018	1018.0	300.00	0	Tap, Gewindebohrer: L100 R5	100	5
1.019	1020.0	400.00	0	Turning Tool, Drehwerkzeug: Empty, Leer	0	0
1.022	1027.0	10022.00	0	Cutter, Fraeser: L05 R2	05	2
1.023	1022.0	000.00	0	Cutter, Fraeser: L100 R5 R2-5	100	5

Tool table: Standard tool data

Name	Inputs
P	Magazine and pocket identification: The magazine number is before the decimal point (optional), and the pocket number after the decimal point
T_NR	Tool number (8.2): Number by which the tool is called in the program. The code for the associated replacement tool is after the decimal point
IDX	Index number, programmed T2= value in the NC program (maximum: 9)
NAME	Name of the tool (max. 40 characters)
L	Tool length L
R	Tool radius R
R2	Tool radius R2 for toroid cutters (only for 3-D radius compensation or graphical representation of a machining operation with corner rounding cutters)
DL	Oversize tool length L
DR	Oversize tool radius R
DR2	Oversize tool radius R2
O	Tool orientation, orientation for corner radius for turning tools
TL	Set tool lock (0=not locked, 1=locked)
TS	Tool status 0 = Not defined 1 = Measured and OK 2 = Out of tolerance 3 = Broken 4 = Tooltime expired
TIME1	Maximum tool life in minutes. This function can vary depending on the individual machine tool. Your machine manual provides more information on TIME1.
TIME2	Maximum tool life in minutes during an M6: If the current tool age exceeds this value, the MillPlus changes the tool during the next M6 call of the replacement tool (see also CUR.TIME) Required tool life in NC language programmed in T3=
CUR.TIME	Current tool age of the tool in minutes: The MillPlus automatically keeps track of the current tool age. A starting value can be entered for used tools.
TYPE	Tool type for defining the tool shape: UNDEFINED = not defined MILL = milling cutter DRILL = drilling tool TAP = tapping tool TURN = turning tool TCHP = touch probe
DOC	Comment on tool (max. 40 characters)
PTYP	Tool type for evaluation in the tool magazine table (this address must be entered if Tin-Tout is used)

Name	Inputs
PLC	Information on this tool that is to be sent to the PLC
LCUTS	Tooth length of the tool
ANGLE	Maximum plunge angle of the tool for reciprocating plunge-cut in cycles. If this value is 90, then it is assumed that the cutter can plunge into the material vertically.
NMAX	Limits the spindle speed for this tool. The programmed value is monitored (error message) as well as an increase in the shaft speed via the potentiometer.
LIFTOFF	Definition of whether the MillPlus should retract the tool in the direction of the positive tool axis at an NC stop in order to avoid leaving dwell marks on the contour.
P1, P2, P3	Machine-dependent function: Transfer of a value to the PLC. Refer to your machine manual.
T-ANGLE	Point angle of the tool. Is used by the Centering cycle in order to calculate the centering depth from the diameter entry (currently without function).
PITCH	Thread pitch of the tool (currently without function)

Tool table: For tool measurement

Name	Inputs
CUT	Number of teeth (20 teeth maximum)
LTOL	Wear tolerance of the tool length L. If the entered value is exceeded, the MillPlus locks the tool (status L). Input range: 0 to 0.9999 mm
RTOL	Wear tolerance of the tool radius R. If the entered value is exceeded, the MillPlus locks the tool (status L). Input range: 0 to 0.9999 mm
DIRECT.	Cutting direction of the tool for measuring the tool during rotation (-1=M3, +1=M4). If this value is not entered, M3 is assumed.
R_OFFS	For tool length measurement: Tool offset between stylus center and tool center. Preset value: tool radius R
L_OFFS	Tool radius measurement: Additional tool offset between upper surface of stylus and lower surface of tool. Default: 0
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the MillPlus locks the tool (status L). Input range: 0 to 0.9999 mm
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the MillPlus locks the tool (status L). Input range: 0 to 0.9999 mm

Tool table: For 3-D touch probes

Name	Inputs
PROBE	Touch probe type
CAL_OF1	During calibration, the MillPlus stores in this column the center misalignment in the reference axis of the 3-D probe, if a tool number is indicated in the calibration menu.
CAL_OF2	During calibration, the MillPlus stores in this column the center misalignment in the minor axis of the 3-D probe, if a tool number is indicated in the calibration menu.
CAL_ANG.	During calibration, the MillPlus stores in this column the spindle angle at which the 3-D probe was calibrated, if a tool number is indicated in the calibration menu.
RAPID_FEED	Rapid traverse for probing
PROBE_FEED	Measuring feed rate
MEAS_RANGE	Maximum measuring range
SAFETY_DIST	Safety clearance
ALLOW_FF	Pre-positioning at rapid traverse
POS_PROBE	Oriented spindle stop

Replacement tool

A replacement tool is inserted when the tool life of the current machining tool expires, or, if monitoring of the tool-cutting force is active, when the lowest permissible level of performance cannot be attained.

The replacement tool is identified by the last two digits of the tool number. This means that up to 99 replacement tools can be assigned to a tool.

Tool oversize (DL, DR, DR2)

These parameters serve to define the oversizes of the tool shape. The actual length of the tool is the length **(L)** plus the length oversize **(DL)**. The actual radius of the tool is the radius **(R)** plus the radius oversize **(DR)**. The length and radius compensation functions use these length and radius values. If nothing is entered, the value zero is assumed. The actual corner radius of the tool is the radius **(R2)** plus the corner radius oversize **(DR2)**.

Tool length	$L + DL$
Tool radius	$R + DR$
Corner radius	$R2 + DR2$



The length is used for length compensation, the radius for radius compensation (G41-G44 or G141), and the corner radius for 3-D radius compensation (G141).

Monitoring of tool age (TIME1, TIME2, CUR.TIME)

TIME1 indicates whether the tool age of a specified tool is to be monitored.

TIME1=0 Tool age not monitored.

TIME1= >0 Specification of tool age, which is monitored.

TIME1= assigns a tool life in minutes to the tool. The range for the tool life is 1 to 9999 minutes.

Whenever a tool is in use, its CUR.TIME in the memory is increased by the cutting time.

Once the tool life has expired, a warning is output and the tool is locked with TL=1.

If the same tool is called again, a replacement tool is inserted, if there is one.

Tool magazine

The tool places are defined in the **Tool magazine** table, and the corresponding tool numbers are indicated. A complete overview of the tool parameters is given in the section on the tool table.

For a tool magazine, a tool table stating the places in the magazine for each tool as well as the other relevant data must be created before the first program is run.

In the event of a tool change (M6), the programmed tool is extracted from the magazine.



The machine tool builder adapts the functional range of the magazine table to the requirements of your machine. Refer to your machine manual.

Tool magazine parameters

Name	Inputs
P	Identification of the magazine and place. Magazine number before the decimal point (optional), place number after the decimal point
T	Identification in the tool table without index after the decimal point
PNAME	Description of the place name (max. 40 characters)
RSV	Reserved pocket. Place is reserved, e.g. for the tool in the spindle
ST	Special tool, places directly before and after this place are locked in column L
F	Fixed pocket. The tool is always returned to this place
L	Place locked. Display: Yellow
PLC	Information on this tool that is to be sent to the PLC
P1, P2, P3, P4, P5	Value 1 to 5. Machine-dependent function: Transfer of a value to the PLC. Refer to your machine manual.
PTYP	Tool type for evaluation in the place table, same as PTYP of the tool
LOCKED_ABOVE	Locks the place above. For magazine rules
LOCKED_BELOW	Locks the place below. For magazine rules
LOCKED_LEFT	Locks the place at left. For magazine rules
LOCKED_RIGHT	Locks the place at right. For magazine rules

Oversize tools (ST)

The **ST** word indicates whether the tool only occupies one place in the tool magazine, or if it is an oversize tool occupying three places, i.e. the place in which the tool is located as well as the two empty places to the left and right of the tool, for example. Such a tool must be returned to the same place in the magazine after use.

Datums

The **Zero offsets** table has definitions of datums, such as those that you have defined with the **Manual >> Workpiece >> Touch side** function.

There are datums for:

- Workpieces
- Pallets

Parameters

The values for E parameters and ES parameters are saved in the **Parameter** table.

There are parameter tables for:

- E parameters
- ES parameters

Points

The axis coordinates are specified in the **Points** table under the point numbers.

I	X	Y	Z	A	B	C	D4	DESCRIPTION
1	0	0	0	0	0	0	0	
2	0	0	0	0	0	0	0	
3	0	0	0	0	0	0	0	
4	0	0	0	0	0	0	0	
5	0	0	0	0	0	0	0	
6	0	0	0	0	0	0	0	
7	0	0	0	0	0	0	0	
8	0	0	0	0	0	0	0	
9	0	0	0	0	0	0	0	
10	0	0	0	0	0	0	0	
11	0	0	0	0	0	0	0	
12	0	0	0	0	0	0	0	
13	0	0	0	0	0	0	0	
14	0	0	0	0	0	0	0	
15	0	0	0	0	0	0	0	
16	0	0	0	0	0	0	0	
17	0	0	0	0	0	0	0	
18	0	0	0	0	0	0	0	
19	0	0	0	0	0	0	0	
20	0	0	0	0	0	0	0	
21	0	0	0	0	0	0	0	

5.3 Menu: NC-Explorer

The NC-Explorer gives you an overview of the available files and the structure of the directories.

The **NC-Explorer** menu contains the following item:







- **File**

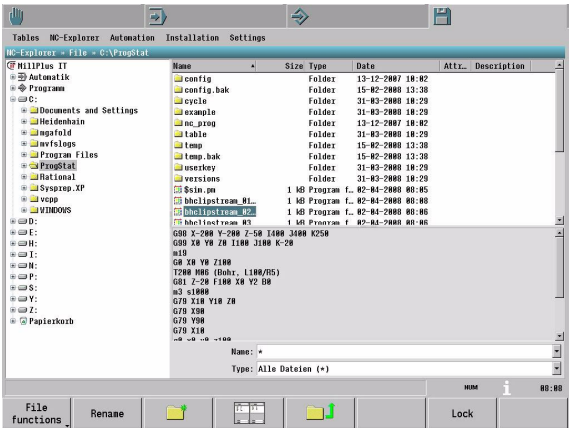
File

Use this function to manage files and directories via the file manager. This includes functions like cutting, copying and pasting files, renaming files and directories, and adding new directories.

Functions

When the NC-Explorer is open, a soft-key row with the following functions is shown:

Soft keys	Description
	File functions (copying, cutting, pasting, searching)
	Change the name of a file or directory
	Create a new directory
	Split the screen into one or two file-manager windows
	Go up one level in the directory structure
	Write-protect a file. The attribute (r) is set for the file.



5.4 Menu: Installation

Use the **Installation** functions to view and edit the MillPlus configuration data.

The **Installation** menu contains the following items:

- **Diagnosis**
- **Compensation**
- **Configuration**
- **Service**



Please note that only trained personnel may change the configuration or diagnose the MillPlus.

The functions listed below are described in detail in the Technical Manual.

We strongly recommend contacting your machine tool builder before you change these settings on the MillPlus.

Compensation

Configuration

Use the **Compensation >> Configuration** function to specify the axis assignments via the configuration table.

Axes

Use the **Compensation >> Axes** function to view and edit the various axis-compensation tables.

Configuration

System

Use the **Configuration >> System** function to view and edit various types of configuration data (machine parameters).

User soft keys

Use the **Configuration >> User soft keys** function to assign names and images to user soft keys. The table also indicates whether a script is available for the respective soft keys (see “User Soft Keys” on page 39).

Tables NC-Explorer Automation Installation Settings			
Installation - User softkeys - C:\ProgStat\userkey\userkey.ukt			
NAME	TEXT	IMAGE	SCRIPT
Manuell 1		tool.bmp	Yes
Manuell 2	Select Automatik		Yes
Manuell 3	Select Program		Yes
Manuell 4			Yes
Manuell 5			
Manuell 6			
Manuell 7			
Manuell 8			
Manuell 9			
Manuell 10			
Automatik 1		tool.bmp	Yes
Automatik 2			
Automatik 3	Select Program		Yes
Automatik 4			
Automatik 5			
Automatik 6			
Automatik 7			
Automatik 8			
Automatik 9			
Automatik 10		tool.bmp	Yes
Programmieren 1			
Programmieren 2	Select Automatik		Yes
Anwender-Softkey-Text - Text width 25 C:\ProgStat\userkey.ukt			
<div> <div>Find</div> <div>Edit</div> <div>Delete script</div> </div>			

Service

The **Service** function is intended particularly for commissioning and troubleshooting of the MillPlus by specialists. Numerous basic settings for the MillPlus can be entered, as well as changes made. This function is also well-suited to diagnosing and optimizing the MillPlus for the machine tool.

Oscilloscope

The control features an integrated oscilloscope. This oscilloscope features six channels for recording analog signals and 16 channels for recording digital signals.

Axes (OLM)

The OLM (online monitor) supports the commissioning and diagnosis of control components by displaying variables for axes and channels, as well as by displaying hardware signals.

Various trace functions can be used and commands activated.

PLC

The integrated PLC of the control contains its own text editor for creating the PLC program. This service function provides the Service department with more information about the PLC.

SIK

On the standard version of the MillPlus, four control loops (3 axes + 1 spindle) are enabled. Further control loops can be enabled by the machine tool manufacturer. You can enable further control options by entering a code number.

If you wish to enable additional software options, please contact your machine tool manufacturer for the code number. You will be given the code number after stating your SIK number.

MOD

Most machine-configuration data is specified by the machine tool manufacturer. You can change some configuration data, i.e. the user parameters, yourself. Enter the code number 123 in the **MOD** mode.

User parameters available for you to change:

- DisplaySetting
- ProgrammingSettings
- CfgUserPath
- CfgSystemTime
- CfgEditorSettings
- NCchannel

Network

If the MillPlus is connected to a network, it also displays the connected network drives in the directory window. All functions (selecting a drive, copying files, etc.) also apply to network drives, provided that you have been granted the corresponding rights.

The USB interface of the control

Backing up data from or loading onto the MillPlus is especially easy with USB devices. The MillPlus supports the following USB block devices:

- Floppy disk drives with FAT/VFAT file system
- Memory sticks with the FAT/VFAT file system
- Hard disks with the FAT/VFAT file system
- CD-ROM drives with the Joliet (ISO 9660) file system

The MillPlus automatically detects these types of USB devices when connected. The MillPlus does not support USB devices with other file systems (such as NTFS).

The USB devices appear as separate drives in the file manager, so you can use the file-management functions correspondingly.



Please ensure that the correct procedure is maintained when removing a USB device.

Error display

The MillPlus generates error messages when it detects problems such as:

- Incorrect data was entered during programming
- Logical errors arise in the program during program testing or program run
- System errors during operation

If an error occurs, a symbol appears in the main operating mode tab, and an error message appears in the message line. Select the **Service >> Error view** menu item to open a window with additional information about this error. See also "Error support" on page 38.

5.5 Menu: Settings

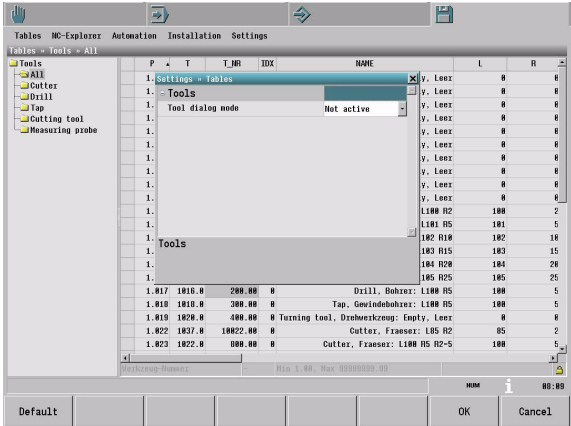
Settings parameters for the menu functions in the Control (Setup) operating mode.

The **Settings** menu contains the following item:

- **Tables**

Tables

You can set the tool dialog mode under the Tables menu item.



6

Miscellaneous

6.1 Accessories

3-D touch probes

With the various HEIDENHAIN 3-D touch probe systems you can:

- Automatically align workpieces
- Quickly and precisely set datums
- Measure the workpiece during program run

The TS 640 triggering touch probe

This touch probe is particularly effective for automatic workpiece alignment, datum setting and workpiece measurement.

The TS 640 touch probe features infrared transmission of the triggering signal to the control. This makes it highly convenient for use on machines with automatic tool changers.

Principle of operation: HEIDENHAIN triggering touch probes feature a wear-resistant optical switch that generates an electrical signal as soon as the stylus is deflected. This signal is transmitted to the TNC, which stores the current position of the stylus as an actual value.



HR electronic handwheels

Electronic handwheels facilitate moving the axis slides precisely by hand. A wide range of traverses per handwheel revolution is available. Along with the HR 130 and HR 150 panel-mounted handwheels, HEIDENHAIN also offers the HR 410 portable handwheel.



SYMBOLS

3-D touch probes ... 116

A

Application window ... 26
 ASCII keyboard ... 29
 Automatic mode of operation >
 Execution ... 78
 >Machining ... 78
 Automatic mode of operation > NC
 Program ... 76
 >Last selected programs ... 77
 >Select ... 76
 Automatic mode of operation >
 Settings ... 80
 > Execution ... 80
 Automatic operating mode ... 66
 Axes (OLM) ... 111
 Axis configuration ... 41
 Axis-direction keys
 Tilted plane ... 49

B

Block search (mid-program
 startup) ... 72

C

Calculator ... 37, 38
 Compatibility ... 19
 Compensation ... 110
 Configuration ... 110
 Context menu ... 34
 Continuous jog ... 48
 Control (Setup) operating mode ... 98
 Control > Installation ... 110
 Control > NC-Explorer ... 109
 >File ... 109
 Control > Settings ... 114
 >Tables ... 114
 Control > Tables ... 100
 >Parameters ... 108
 >Points ... 108
 >Tool magazine ... 107
 >Tools ... 100
 >Zero offsets ... 108
 Coordinate system ... 41
 Coordinates ... 41

D

Data, finding ... 30
 Datums ... 42

E

Editing functions ... 33
 Error display ... 113
 Error messages ... 113
 Error support ... 38

F

Feed rate STOP ... 69
 File manager ... 28
 FSTM entry ... 50
 Function Explorer ... 27

G

G7 coordinates ... 41
 General status display ... 35
 Graphics ... 91
 2.5-D ... 92
 3-D ... 93
 3-D, high-resolution ... 93
 Plan view ... 92
 Wire plot ... 92

H

HR handwheel ... 116

I

Incremental jog ... 49
 INFO functions ... 37
 Input dialogs ... 27
 Instructions ... 19
 Interruption ... 70
 Interruption of program run ... 69, 71

J

Jogging ... 48

L

Line number and status symbol ... 31

M

Machine operating panel ... 21
 Main operating modes ... 22
 Manual axes, positioning ... 51
 Manual operating mode ... 44
 Manual operating mode > Machine
 tool ... 62
 > NC shutdown ... 62
 > Reference point ... 62
 Manual operating mode > MDI ... 61
 > Free entry ... 61
 Manual operating mode >
 Settings ... 63
 > Jogging ... 63
 Manual operating mode > Tool ... 59
 > FST entry ... 59
 > Measure in jog ... 60
 > Set plane ... 60
 Manual operating mode >
 Workpiece ... 56
 > Determine center ... 57
 > Preset axes ... 58
 > Touch side ... 56
 MDI
 Free entry ... 52
 MDI.pm ... 52
 Menu bar ... 23
 Automatic operating mode ... 66
 Control (Setup) Operating
 Mode ... 98
 Manual operating mode ... 44
 Programming operating mode ... 82
 Menu items
 Automatic operating mode ... 67
 Manual operating mode ... 45
 Message line ... 25
 Mid-program startup ... 72
 Mid-program startup with Go to
 cursor ... 72
 Mid-program startup with Search to
 cursor ... 72
 MillPlus IT with Windows ... 18
 MOD ... 112
 Moving the machine axes ... 48

N

Names of files and directories ... 29
 NC editor ... 31, 32
 Network ... 112
 New functions ... 6

O

Online help ... 37
 Operating concept ... 22
 Operating functions ... 27
 Operating support ... 37
 Optional block skip ... 74
 Oscilloscope ... 111
 Override knobs ... 50

P

PLC ... 111
 Popup windows ... 34
 Program breakpoint ... 75
 Program check ... 90
 Program execution ... 68
 Program run "Single step" ... 69
 Program run correction ... 70
 Program selection ... 68
 Program structure ... 68, 80
 Program, closing ... 87
 Program, creating/editing ... 84
 Program, editing ... 87
 Program, saving ... 87
 Program, testing ... 84
 Programming > Editing ... 87
 > NC editor ... 87
 > Test ... 91
 Programming > NC Program ... 85
 > Last selected programs ... 85
 > New ... 85
 > Open ... 85
 Programming > Settings ... 95
 > Edit ... 95
 > Test ... 95
 Programming > Window ... 86
 Programming operating mode ... 82
 Programming station ... 19
 Programming support ... 37

R

Reference point ... 42, 47
 Reference point, crossing over ... 47
 Reference system ... 41
 Resuming program run ... 71

S

Safety ... 20
 Screen and NC keys ... 21
 Screen layout ... 28
 Selectable stop (Selective wait) ... 74
 Sequence of operations ... 29
 Service ... 111
 Shortcuts
 CTRL+... ... 34
 SIK ... 111
 Soft-key row
 Machine functions ... 26
 NC functions ... 26
 Spindle STOP ... 69
 Startup point ... 72
 Status display ... 23, 35
 Status displays ... 36
 Support for data entry ... 28
 Switch-off ... 48
 Switch-on ... 46
 Symbols ... 22
 Syntax checking ... 31
 Syntax highlighting ... 31
 System bar ... 25

T

Table editor ... 30
 Table settings ... 30
 Table, editing ... 30
 Tool measurement ... 55
 Tool, editing ... 101
 Traversing with the HR 410
 handwheel ... 49

U

USB ... 112

V

Visual display unit and operating
 panel ... 21

W

Workshop oriented ... 18

Z

Zero point, setting... ... 54

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 (8669) 31-0

FAX +49 (8669) 5061

E-Mail: info@heidenhain.de

Technical support FAX +49 (8669) 32-1000

Measuring systems ☎ +49 (8669) 31-3104

E-Mail: service.ms-support@heidenhain.de

TNC support ☎ +49 (8669) 31-3101

E-Mail: service.nc-support@heidenhain.de

NC programming ☎ +49 (8669) 31-3103

E-Mail: service.nc-pgm@heidenhain.de

PLC programming ☎ +49 (8669) 31-3102

E-Mail: service.plc@heidenhain.de

Lathe controls ☎ +49 (8669) 31-3105

E-Mail: service.lathe-support@heidenhain.de

www.heidenhain.de

