

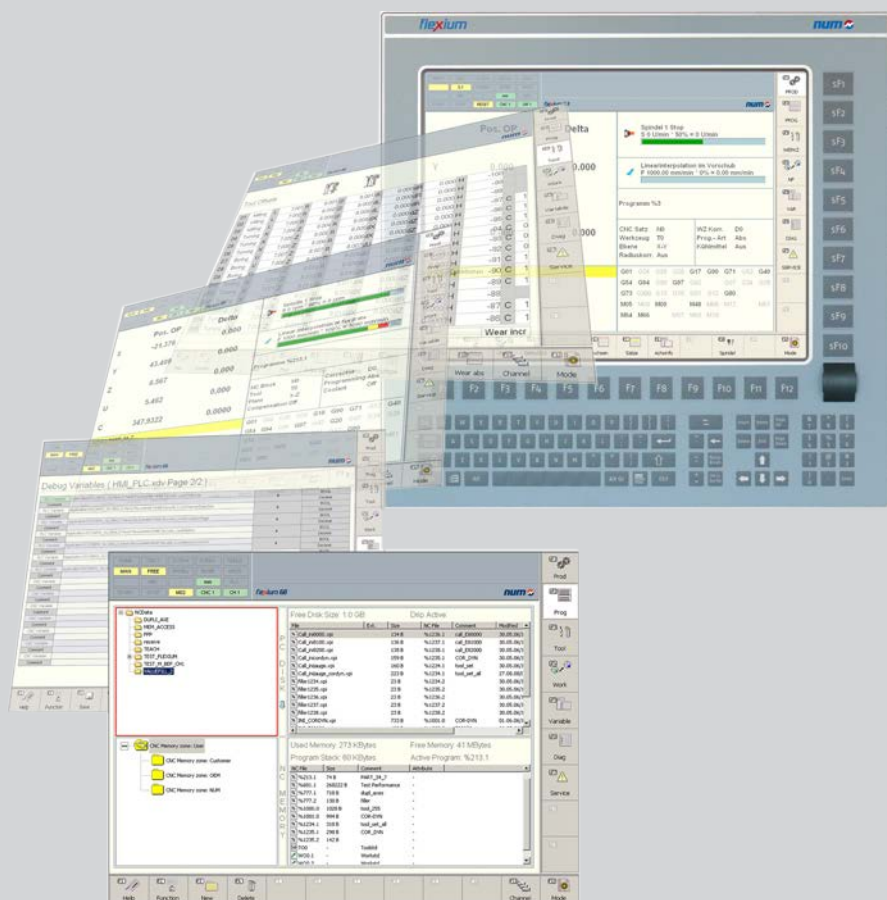
Flexium HMI

Operator Manual

(software release 4.0.20.10)

M00017EN-05

Edition 05.2016



Despite the care taken in the preparation of this document, NUM cannot guarantee the accuracy of the information it contains and cannot be held responsible for any errors therein, nor for any damage which might result from the use or application of the document.

The physical, technical and functional characteristics of the hardware and software products and the services described in this document are subject to modification and cannot under any circumstances be regarded as contractual.

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

© Copyright NUM 2008

All rights reserved. No part of this manual may be copied or reproduced in any form or by any means whatsoever, including photographic or magnetic processes. The transcription on an electronic machine of all or part of the contents is forbidden.

© Copyright NUM 2008 software NUM Flexium family

This software is the property of NUM. Each memorized copy of this software sold confers upon the purchaser a non-exclusive licence strictly limited to the use of the said copy. No copy or other form of duplication of this product is authorized.

Summary

1	Foreword	11
1.1	Important notice	11
1.1.1	Use of this manual	11
1.1.2	CNC mode selection by the PLC	11
1.2	Use of the Flexium HMI Operator manual	12
1.2.1	Operations	12
1.2.2	Possible action during an operation	12
1.3	Document revisions	13
1.4	Use of symbology in the manual.....	14
1.5	NUM Agencies.....	14
1.6	Index	14
2	Product presentation	19
2.1	Turning the System on and off	19
2.1.1	Turning on the System	19
2.1.2	Turning off the System	20
3	Help Window.....	25
3.1	The Help Window.....	25
3.2	Access to the Help Window	25
4	General	31
4.1	Context.....	31
4.2	General information on the HMI start-up page	32
4.3	Status Windows	34
4.4	Spindle and feed rate information.....	36
4.4.1	Display with spindle speed and override	36
4.4.2	Display with spindle speed and spindle load	37
5	Context “Production”	41
5.1	Access to the Context “Production”	41
5.1.1	Context “Production” display when starting the HMI	41
5.2	Channel selection	43
5.3	Spindle monitor	43
5.4	Duplicated axes	44
5.5	Axes information	45
5.6	CNC Blocks Page	46
5.7	Axes Position Page.....	47
5.7.1	Axis travel limits	48
5.8	CNC functions.....	49
5.9	Dialog field	50
5.9.1	Data input into the dialog field	50
5.9.2	Deleting the dialog list	51

6	Context “Programming”	55
6.1	Access to the Context “Programming”	55
6.1.1	Representation of data locations in your system	55
6.1.2	Definition of the files existing on PC disk and in CNC memory	55
6.1.3	Start-up window of the context “Programming”	56
6.2	Moving between Windows	57
6.3	Selecting Directory and Folder on the PC disk	57
6.4	Selecting a File on PC disk	58
6.5	Creating a new part program on PC disk	60
6.5.1	Hidden programs and subroutines	61
6.5.2	Part programs created with another editor or on an external system	61
6.5.3	Format of an ISO part program	61
6.6	Creating a new Compact part program on PC disk	62
6.6.1	Compact programs created with another editor or on an external system	62
6.6.2	Format of a Compact part program	63
6.7	Creating a new “Usinage” Program on PC disk	64
6.7.1	“Usinage” programs created with another editor or on an external system	64
6.7.2	Format of a “Usinage” program	65
6.7.3	The include function	65
6.7.4	The Loop and Include functions	65
6.8	Creating a new tool offset file on PC disk	66
6.8.1	Tool offset files created with another editor or on an external system	67
6.9	Creating a new work offset file on PC	68
6.9.1	Work offset files created with another editor or on an external system	69
6.10	Editing a file on PC disk	70
6.10.1	Keyboard functions	71
6.11	Editing a “Usinage” file on PC disk	72
6.11.1	Keyboard functions	73
6.12	Including a Part Program in an “Usinage” File	73
6.13	Creating a Loop in an “Usinage” File	73
6.14	Editing NUM CNC Cycles	74
6.15	Drip Feed Mode	75
6.15.1	Principle of drip feed mode	75
6.15.2	Drip feed on machines with several axes channels	76
6.15.3	Block search with drip feed mode	77
6.16	Loading a file into CNC memory	78
6.16.1	Load one file only	78
6.16.2	Load several successive files	78
6.16.3	Load several non successive files	78
6.17	Protected Memory Area Management	79
6.17.1	Selecting CNC Memory Zones	80
6.17.2	Closing the memory zones	80
6.18	NC Memory	81
6.19	Selecting the Active Part Program	82
6.20	Saving Files to PC Disk	83
6.20.1	Save one file only	83
6.20.2	Save several successive files	83
6.20.3	Save several non successive files	83
6.21	Editing a Program directly in NC Memory	84
6.22	Deleting Files in PC Disk or NC Memory	86
6.22.1	Delete one file only	86
6.22.2	Delete several successive files	86
6.22.3	Delete several non successive files	86

7	Context “Tools”	91
7.1	Access to the Context “Tools”	91
7.2	Entering tool dimensions and miscellaneous values	92
7.3	Selecting the Tool Type	92
7.4	Resetting the whole Tool Offset table	93
7.5	Teach-in of the Tool Dimensions	94
7.5.1	Milling tool setting page	94
7.5.2	Turning tool setting page	95
7.5.3	Boring tool setting page	95
7.5.4	Procedure	96
8	Context “Work Offset”	101
8.1	Access to the Context “Work Offset”	101
8.1.1	Milling display	101
8.1.2	Turning display	102
8.2	Entering Offset Values	103
8.3	Entering Scaling Factor	103
8.4	Entering Axes Assignment	103
8.5	Setting the Work Offset (DAT1) by Teach-in for Milling	104
8.6	Setting the Work Offset (DAT1) by Teach-in for Turning	106
9	Context “Variables”	111
9.1	Access to the Context “Variables”	111
9.1.1	Variables context window	112
9.2	Entering variables and comments	113
9.3	CNC variables and comments	113
9.3.1	Display of Symbolic variables	113
9.4	PLC variables and comments	114
9.5	Modification of variables values	115
9.6	Creating a list of consecutive variables	116
10	Context “Diagnostic”	121
10.1	Access to the Context “Diagnostic”	121
10.1.1	Diagnostic context window	121
10.2	Extended Help	122
10.2.1	Example of a help page	123
10.3	Alarm History Page	124
10.3.1	History Page and Filters	125
10.4	Exception Page	127
10.5	Test Point Monitor	128
10.6	Safety module diagnostic	129
10.7	CNC and DISC NT Configuration	130
10.7.1	System Functions Page	131
10.7.2	Device List	132
11	Context “Service”	137
11.1	Access to the Context “Service”	137
11.1.1	Start window of the context “Service”	137
11.2	General Settings Page	138
11.2.1	Setting selection	138
11.2.2	Explanation of the setting options	139
11.2.3	Box PC Access	141
11.2.4	Password Settings	142

11.2.5	Custom tools	143
11.3	System Time and Date Settings	144
11.3.1	Setting the system time and date	144
11.4	Location Settings	145
11.4.1	Keyboard functions	145
11.5	Password Settings	146
11.6	Diagnostic Settings	146
11.6.1	System messages.....	147
11.6.2	PLC messages.....	147
11.7	Feed Rate Settings	148
11.7.1	Explanation of the setting options.....	149
11.8	Spindle Settings.....	150
11.8.1	Explanation of the setting options.....	150
11.8.2	History	151
11.9	Management Settings.....	152
11.9.1	Explanation of the setting options.....	153
11.9.2	Password management	154
11.10	Machine Data.....	155
11.10.1	NCK parameters	156
11.10.2	Drive parameters	158
11.10.3	Restart	161
11.10.4	Change Password.....	161
12	CopyExplorer.....	165
12.1	Access to CopyExplorer	165
12.2	Save	166
12.2.1	Save one file only.....	166
12.2.2	Save several successive files	166
12.2.3	Save several non successive files	166
12.2.4	Save all files	166
12.3	Restore.....	166
13	The Virtual Keyboard	171
13.1	Use of the Keyboard	171
13.1.1	Activating the keyboard.....	171
13.1.2	Closing the keyboard display	171
14	Review	175
14.1	System Overview	175
14.1.1	Operating Mode	175
14.1.2	Defining a Program.....	175
14.1.3	Creating a Program.....	176
14.2	Review of Program Coordinate Systems.....	177
14.2.1	Review of Axis Definition and Direction.....	177
14.3	Machine Overview	178
14.3.1	General	178
14.3.2	Turning and Milling Machines	178
14.4	Turning Machine - Specific Features of Front and Rear Turrets.....	179
14.4.1	View of a Turning Machine with Rear Turret.....	179
14.4.2	View of a Turning Machine with Front Turret	179
14.5	Definition of Travels and Origins.....	180
14.5.1	Homing with Incremental or Combined Encoders Using Absolute Dimensions.....	180
14.5.2	Schematic Representation of Travels (Turning and Milling machine).....	181
14.6	Offset Definitions	182
14.7	Turning Machine - Offsets Represented on the Z and X Axes.....	183
14.7.1	Offsets on the Z Axis (Solution 1 with DAT2)	183

14.7.2	Offsets on the Z Axis (Solution 2 with DAT2)	183
14.7.3	Offsets on the X Axis (Solution 1 with DAT2)	184
14.7.4	Offsets on the X Axis (Solution 2 without DAT2)	184
14.7.5	Schematic Representation of the Offsets on X and Z	185
14.8	Milling machine - Offsets Represented on the Z, X and Y Axes	186
14.8.1	Offsets on the Z axis	186
14.8.2	Offsets on the X axis	186
14.8.3	Offsets on the Y axis	187
14.8.4	Schematic Representation of All Offsets	188
14.8.5	Special Case of Milling Machines Equipped with Rotary Tables	189
14.9	Tool Axis Orientation	191
14.9.1	Using the G16 P+ function on turning or milling machine	192
14.10	Definition of Tool Dimensions and Offsets	193
14.10.1	Turning Machine - Dimensions, Tool Tip Radius and Orientation	193
14.10.2	Tool Tip Radius and Orientation	194
14.10.3	Milling Machine - Definition of Tool Dimensions	195
14.11	Definition of Tool Wear	196
14.11.1	Turning Machine	196
14.11.2	Turning Machine	197
15	CNC Operation	201
15.1	Declaration of Axis Assignment	201
15.2	Execution of a Program	202
15.2.1	Program Test	202
15.3	Execution on the machine	203
15.3.1	Optional Stop and Block Skip Enable	203
15.3.2	Program Execution - General	203
15.3.3	Execution of a Program Step by Step	205
15.3.4	High Speed Execution of a Program	206
15.3.5	Execution of a Program in Automatic Mode	206
15.4	Operator Interventions	207
15.4.1	Unplanned Interventions	207
15.4.2	Interruption of Machining	207
15.4.3	Retracting the Axes	208
15.4.4	Manual Axis Recall	208
15.4.5	Automatic Axis Recall	209
15.4.6	Sequence Number Search	211
15.4.7	Extended possibilities in Search mode	213
15.4.8	Backtrack along Path	214
15.4.9	Return along Path	217
15.4.10	Emergency Retraction	218
15.4.11	Program Resumption after a Following Error	218
15.5	Programmed Interventions	219
15.5.1	Machining Stop Function M00 (or M01 if enabled)	219
15.5.2	Forcing an intervention by M12	219

Page deliberately empty.

Summary

1	Foreword	11
1.1	Important notice	11
1.1.1	Use of this manual	11
1.1.2	CNC mode selection by the PLC	11
1.2	Use of the Flexium HMI Operator manual	12
1.2.1	Operations	12
1.2.2	Possible action during an operation	12
1.3	Document revisions	13
1.4	Use of symbology in the manual.....	14
1.5	NUM Agencies.....	14
1.6	Index	14

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

Page deliberately empty.

1 Foreword

1.1 Important notice

1.1.1 Use of this manual



Depending on the functions present on your system, some pages of the HMI may contain additional features which are not described in this manual. They are documented in the operator manual 'Flexium HMI Additional Functions'.

1.1.2 CNC mode selection by the PLC



To avoid conflicts with the mode select functions of the Flexium HMI, some modes must not be selected by the PLC.

The selection of the following modes by the PLC is not allowed:

- Origin shifts
- Automatic tool setting
- Edit.

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

1.2 Use of the Flexium HMI Operator manual

This manual is intended to provide the user with all the information required to use Flexium HMI, the **H**uman **M**achine **I**nterface (called **HMI** below) for Flexium systems.

1.2.1 Operations

The present manual contains operations. Actions to carry out are presented as follows:

Select the desired channel	 F11
For some actions, you must press 2 keys at the same time. Simultaneous press on 2 keys is shown as follows: Ctrl + F4 .	

1.2.2 Possible action during an operation

Press F9 to scroll right in the contents window
Press F8 to scroll left in the contents window
Press F7 to scroll the contents window one page down

1.3 Document revisions

Date	Index	Reason for revision
09 - 2008	00	Document creation
06 - 2010	01	Revised to conform to software version 3.1.0.10
11 - 2010	02	Revised to conform to software version 3.2.0.0 New chapters: - 9.3.1 Particularities of Symbolic variables - 10.5 Test Point Monitor - 10.6 SAM Diagnostic - 10.7.2 Device List Modified chapters: - 10.1.1 Access to the Test Point Monitor and SAM diagnostic pages - 10.7 Access to the Device List - 11.7.1 Real speed display and override by PLC - 12.1 Management of Device List files.
05 - 2011	03	Revised to conform to software version 3.3.0.0 New chapters: - 4.4.2 Spindle load display - 15.4.7 Extended possibilities in Search mode Modified chapters: - 4.2 Function information - 4.3 Status window, nano-interpolation - 4.4 Spindle and feed rate information - 6.1.3 Active program number - 6.10.1 Keyboard functions - 9.1 PLC variables.
03 - 2012	04	Revised to conform to software version 3.4.0.0 New chapters: - 5.7.1 Axis Travel Limits - 6.14 Editing NUM CNC Cycles - 6.20 Editing a Program directly in NC Memory - 10.3.1 History Page and Filters. Modified chapters: - 5.7 Axes Position Page - 5.9.1 Data input into the dialog field - 6.4 Access to Flexium 3D - 6.17 NC Memory - 10.3 Alarm History page - 11.2.2 Setting Options, Prod Page, mode bar on top - 11.2.2 Setting Options, Edit CNC cycles - 11.7.1 Override by PLC - 11.9.1 CNC editor: autoscroll.
05 - 2016	06	Revised to conform to software version 4.0.20.10 New chapters: - 11.2.3 Box PC Access - 11.2.4 Password Settings - 11.2.5 Customer tools. - 11.6.1 System messages. - 11.8.2 History. - 11.9.2 Password management. - 11.10 Machine data. Modified chapters: - 2 Product presentation - 4.1 Context - 11.1 Start window of context "Service" - 11.2 General Settings Page. - 11.2.2 Explanation of the setting options. - 11.2.6 Diagnostic Setting. - 11.6.2 PLC messages. - 11.9 Management Settings.

1.4 Use of symbology in the manual

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

1.5 NUM Agencies

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

1.6 Index

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

Page deliberately empty.

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

Page deliberately empty.

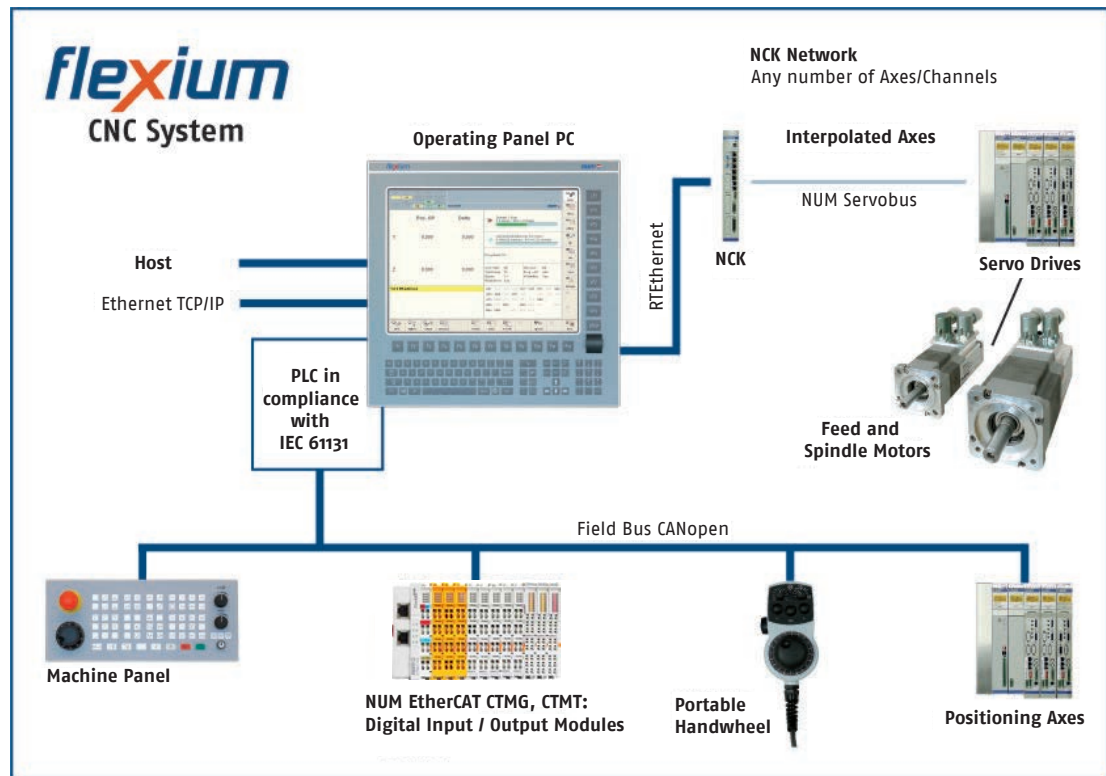
Summary

2	Product presentation	19
2.1	Turning the System on and off	19
2.1.1	Turning on the System	19
2.1.2	Turning off the System	20

Page deliberately empty.

2 Product presentation

The figure below is a general view of the CNC system.



2.1 Turning the System on and off

2.1.1 Turning on the System

Power up the NCK and the PC.

2.1.2 Turning off the System

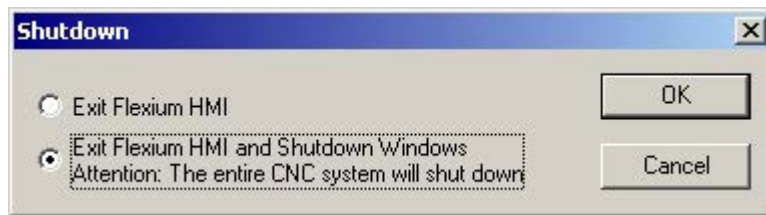


The HMI is a PC based system.
Do not switch off the power before closing the HMI as described below!

To exit the HMI from any HMI context

+ or +

The shutdown dialog shows two ways to exit the system:



Exit Flexium HMI:

Selecting this option only closes the HMI, leaving the CNC system running.



Exit HMI and Shutdown Windows:

This option shuts down the HMI and the CNC system.
Make sure the machine has been stopped normally before selecting this option, as the machining could be stopped in an uncontrolled way.

Page deliberately empty.

1
2
3
4
5
6
7
8
9
10
11
12
13
14
15

Page deliberately empty.

Summary

3	Help Window.....	25
3.1	The Help Window.....	25
3.2	Access to the Help Window.....	25

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

Page deliberately empty.

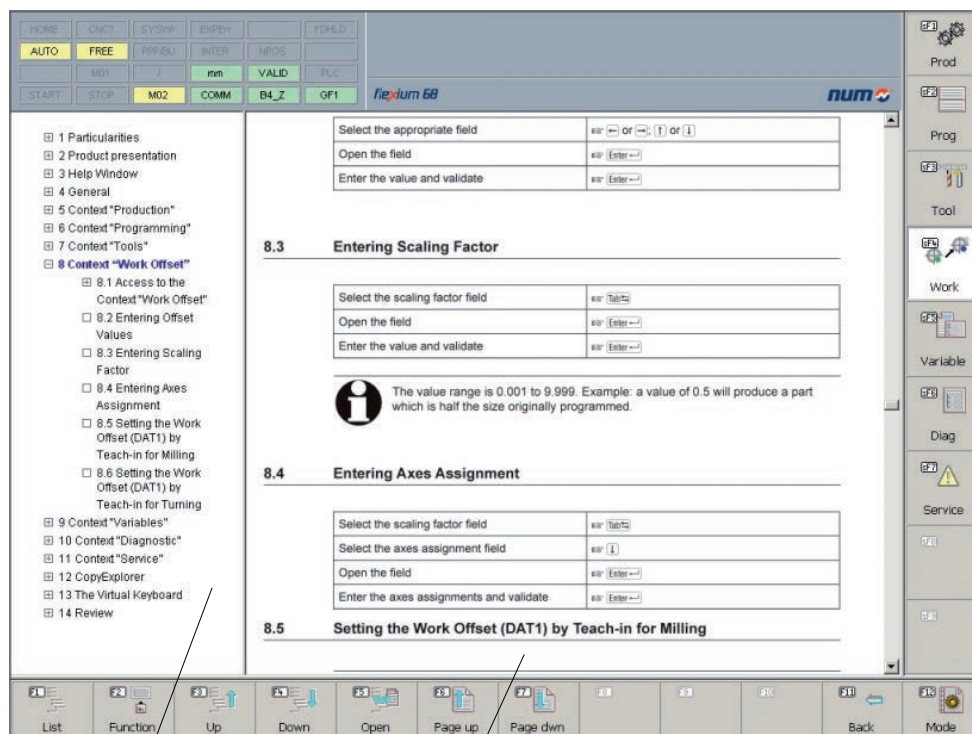
3 Help Window

3.1 The Help Window

The Help Window is available in every context. It offers all information contained in the conventional user manual. Topics or chapters can be directly selected by clicking on the desired item in the navigation list.

3.2 Access to the Help Window

To access the Help Window, activate the sub-context bar	 F12
To open the Help Window	 F1



Navigation list

Contents window

To exit the Help window	 F11
-------------------------	---

Possible actions

Press F7 to scroll the contents window one page down
Press F6 to scroll the contents window one page up
Press F5 to open the contents of the chapter selected in the navigation list
Press F4 to scroll down in the navigation list
Press F3 to scroll up in the navigation list
Press F2 to access all existing functions. Select a function with ↑ or ↓ and Enter ↵
Press F1 to open or close the navigation list



The information represented in the help window depends on the last selected page in a given context.
If “Help” is activated in a basic page of the HMI, information contained in the Basic Operator Manual will be displayed.
Accessing “Help” from a page being part of an optional feature will open the Additional Feature Manual.

Page deliberately empty.

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

Page deliberately empty.

Summary

4	General	31
4.1	Context.....	31
4.2	General information on the HMI start-up page.....	32
4.3	Status Windows	34
4.4	Spindle and feed rate information.....	36
4.4.1	Display with spindle speed and override	36
4.4.2	Display with spindle speed and spindle load	37

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

Page deliberately empty.

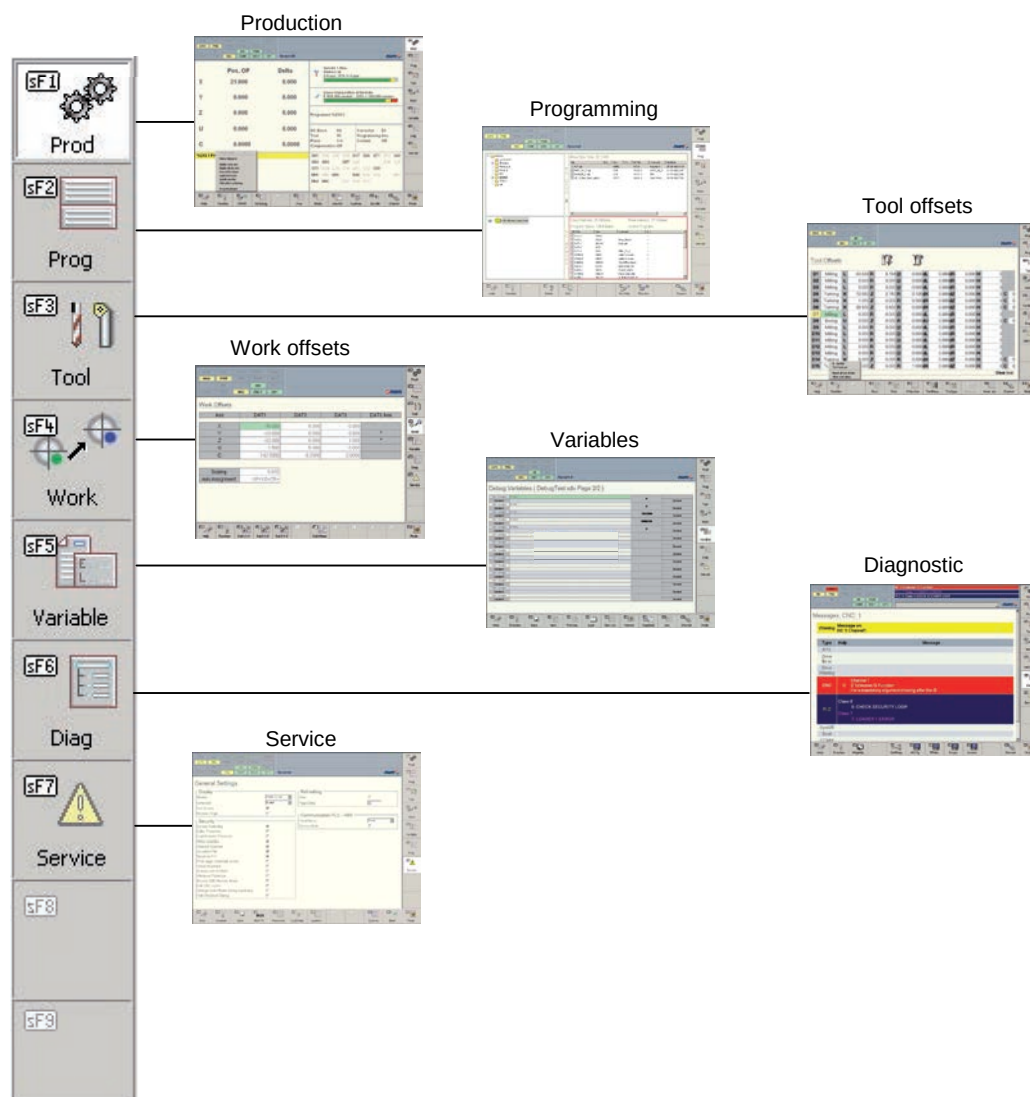
4 General

This manual should be considered as a supplement to the Flexium Programming manual.
(Reference : M00018)

It supplies additional information concerning the multi-context Flexium Human Machine Interface (HMI).

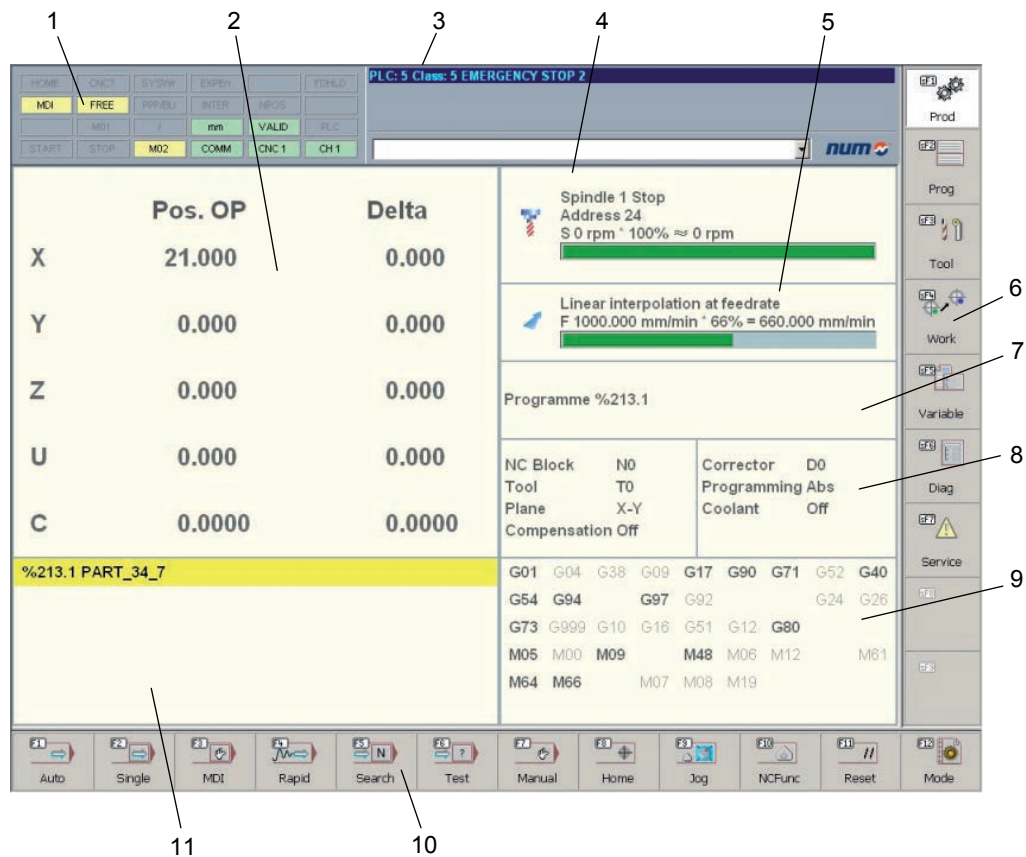
4.1 Context

A context is a work area where similar tasks belonging to a same group may be performed.
Seven contexts are provided:



4.2 General information on the HMI start-up page

Display of the HMI start-up page



- 1 - CNC status window
- 2 - Axis position information (axes which are not homed are shown in red)
- 3 - Messages (CNC, PLC, system and \$ messages)
- 4 - Dialog field to be used in MDI, SEARCH and HOME modes
It is also used in the case of a message display with wait for an operator response.



To input data, the CNC function bar must be active.

- 5 - Override information for spindle and feed rate
- 6 - Main context bar
- 7 - Active program
- 8 - General information
- 9 - Function information
- 10 - CNC functions and sub-context bar
- 11 - Actual CNC block display
(a block containing an error is shown on a red colour background).

Detailed function information

G01	G04	G38	G09	G17	G90	G71	G52	G40	} G functions
G54	G94		G97	G92			G24	G26	
G73	G999	G10	G16	G51	G12	G80			
M05	M00	M09		M48	M06	M12		M61	} M functions
M64	M66		M07	M08	M19	M13	M200		

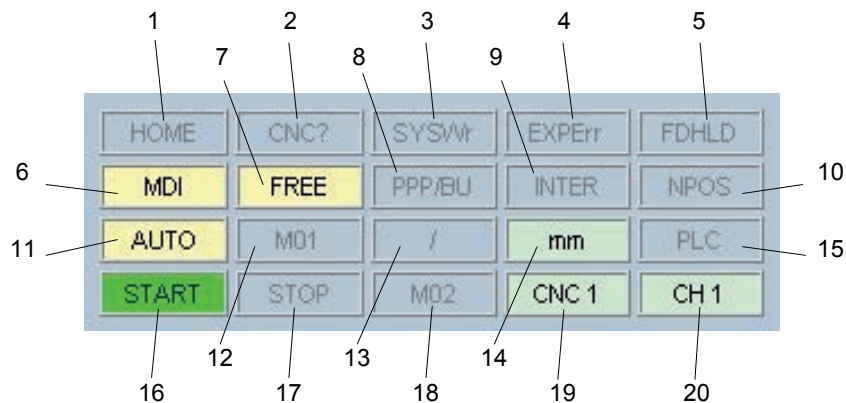
Last active M function with acknowledge

Last active M function without acknowledge

4.3 Status Windows

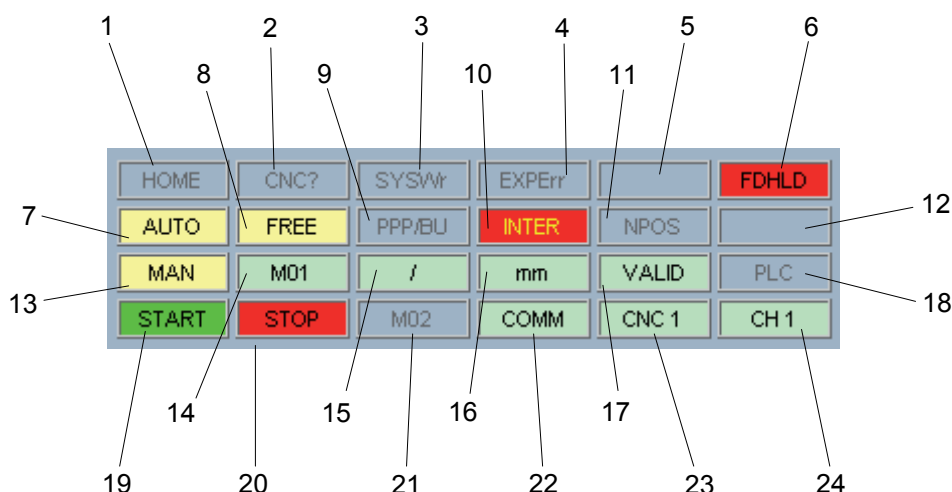
The Status Window shows the status of the CNC at any given time via indicators displayed in 20 fields. **When no indicators are required in a field, it is not displayed.**

The description below applies to the Flexium HMI up to version 3.1.0.10



- 1 - Homing not performed (HOME flashing)
- 2 - CNC error (CNC ? flashing)
- 3 - System warnings
(see Flexium Programming manual reference : M00018)
- 4 - Exception error has occurred
- 5 - Feed interruption (FDHLD)
- 6 - Current mode
- 7 - Selected JOG increment (.001 to 10000, FREE or HANDWH)
- 8 - Drip feed mode active.
(with the function BackupAgent: data backup or restore in progress)
- 9 - Axis recall (AXRCL flashing) or manual override (INTER flashing)
- 10 - Axis not in position (N_POS)
- 11 - Mode waiting (selected, but not yet active)
- 12 - Optional stop (M01) enabled
- 13 - Block skip (/) enabled
- 14 - Display unit: millimetres (mm) or inches (inch)
If the system is set to nano-interpolation, a small "n" is displayed beside the display unit.
- 15 - PLC state:
PLC: PLC in run
STOP: PLC in stop
EXPerr: Exception error
BrkPnt: Stopped on Breakpoint
DbStep: Debug step
1Cycle: One cycle mode
- 16 - In cycle (START)
- 17 - Machining stop (STOP)
- 18 - End of part program (M02), programmed stop (M00) or reset
(momentary display of RESET)
- 19 - CNC address
- 20 - Channel number.

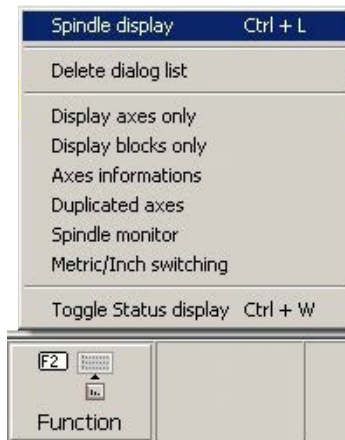
From the Flexium HMI version 3.2.0.0 on, some new indicators have been added:



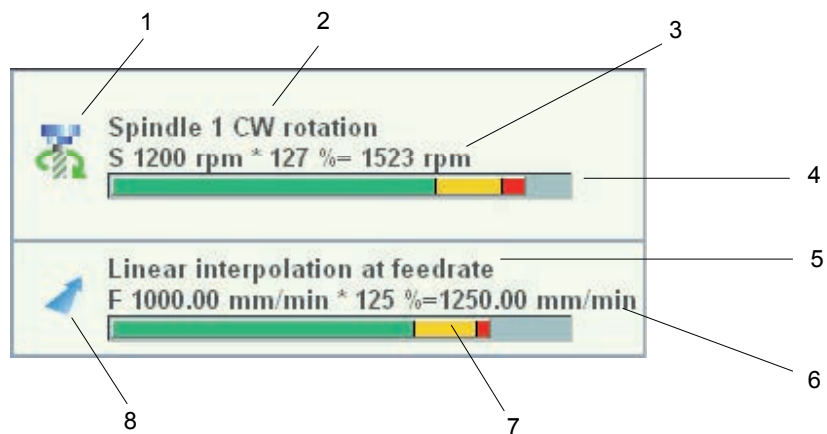
- 1 - Homing not performed (HOME flashing)
- 2 - CNC error (CNC ? flashing)
- 3 - System warnings
(see Flexium Programming manual Reference : M00018)
- 4 - Exception error has occurred
- 5 - Not used
- 6 - Feed interruption (FDHLD)
This is a combination of both PLC signals General feed enable and Channel specific feed enable
- 7 - Current mode
- 8 - Selected JOG increment (.001 to 10000, FREE or HANDWH)
- 9 - Drip feed mode active
(with the function BackupAgent: data backup or restore in progress)
- 10 - Axis recall (AXRCL flashing) or manual override (INTER flashing)
- 11 - Axis not in position (N_POS)
- 12 - Not used
- 13 - Mode waiting (selected, but not yet active)
- 14 - Optional stop (M01) enabled
- 15 - Block skip (/) enabled
- 16 - Display unit: millimetres (mm) or inches (inch)
- 17 - Axes channel enabled
- 18 - PLC state:
PLC: PLC in run
STOP: PLC in stop
EXPErr: Exception error
BrkPnt: Stopped on Breakpoint
DbStep: Debug step
1Cycle: One cycle mode
- 19 - In cycle (START)
- 20 - Machining stop (STOP)
- 21 - End of part program (M02), programmed stop (M00) or reset (momentary display of RESET)
- 22 - Axes channel state: Common, Independent or Auxiliary channel
- 23 - CNC address
- 24 - Channel number.

4.4 Spindle and feed rate information

This window may show various information. To switch between displays, use the function “Spindle Display” in the “Production” context.

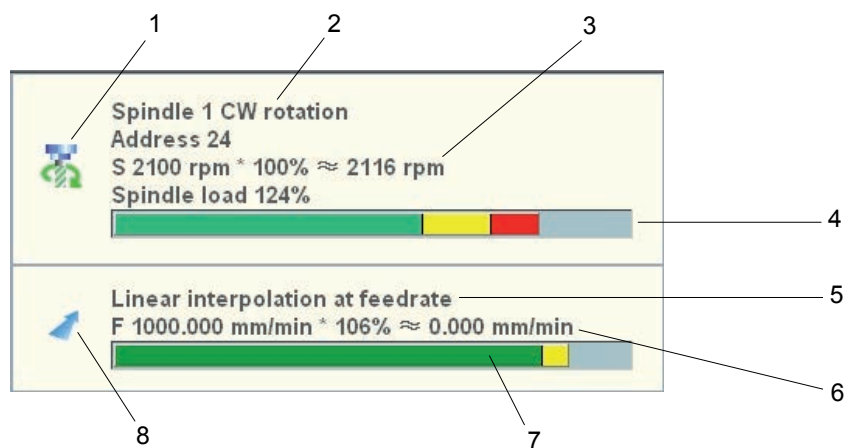


4.4.1 Display with spindle speed and override



- 1 - Spindle state icon
- 2 - Spindle state description
- 3 - Programmed spindle speed, override value and real spindle speed
- 4 - Spindle override graph bar
- 5 - Interpolation mode description
- 6 - Programmed feed rate, override value and real feed rate
- 7 - Feed rate override graph bar
- 8 - Interpolation mode icon.

4.4.2 Display with spindle speed and spindle load



- 1 - Spindle state icon
- 2 - Spindle state description and spindle address
- 3 - Programmed spindle speed, override value and real spindle speed
- 4 - Spindle load value and graph bar
- 5 - Interpolation mode description
- 6 - Programmed feed rate, override value and real feed rate
- 7 - Feed rate override graph bar
- 8 - Interpolation mode icon.

Page deliberately empty.

Summary

5	Context "Production"	41
5.1	Access to the Context "Production"	41
5.1.1	Context "Production" display when starting the HMI	41
5.2	Channel selection	43
5.3	Spindle monitor	43
5.4	Duplicated axes	44
5.5	Axes information	45
5.6	CNC Blocks Page	46
5.7	Axes Position Page.....	47
5.7.1	Axis travel limits	48
5.8	CNC functions.....	49
5.9	Dialog field	50
5.9.1	Data input into the dialog field	50
5.9.2	Deleting the dialog list	51

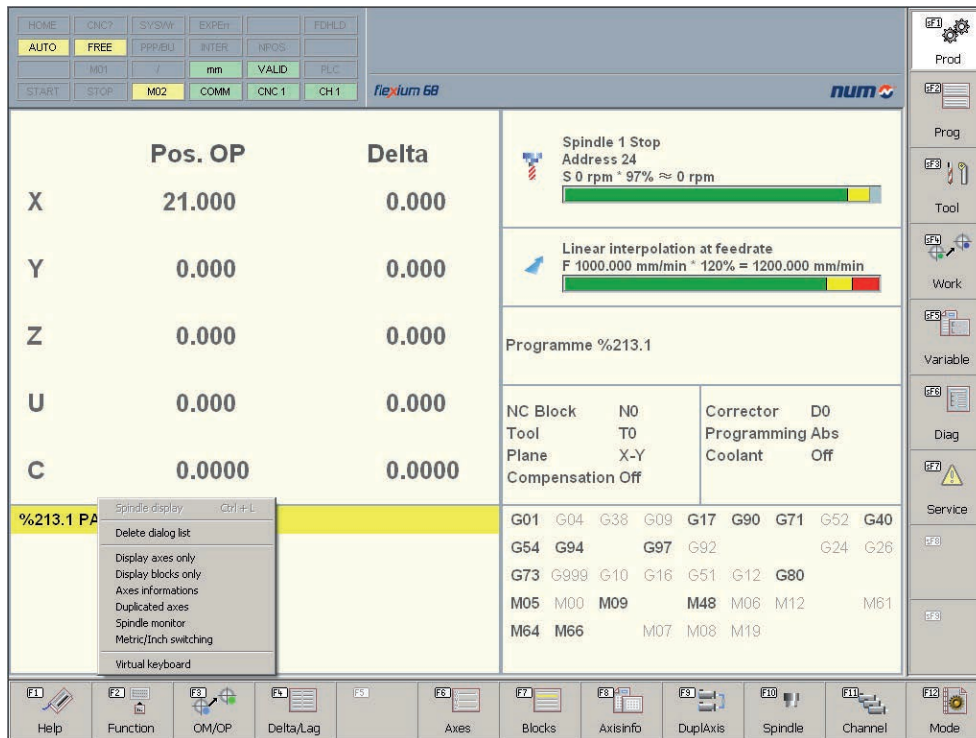
Page deliberately empty.

5 Context "Production"

5.1 Access to the Context "Production"

To access the context "Production", press	 Shift + F1
To close pop-up menus and windows, press	 Esc

5.1.1 Context "Production" display when starting the HMI



Possible actions

Press F12 to toggle between sub-contexts and CNC functions. This action is available in every context.	
Press F11 to select the axes channel (multi-channel machines only)	
Press F10 to access the spindle monitor page	
Press F9 to access the duplicated axes page	
Press F8 to access the axes information page	
Press F7 to access the CNC blocks page	
Press F6 to access the axes positions page	
Press F4 to toggle between Delta and Lag display	
Press F3 to toggle between OP, OM and Relative display	
	The values of the Relative display are reset as soon as this mode is selected. The description of the relative measurement of distances can be found in chapter '5.7 Axes Positions Page'. The Relative display must be activated in the context "Service". For detailed information, refer to chapter '11.2 General Settings Page' in this manual.
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter↵	

5.2 Channel selection



This feature is only relevant on multi-channel machines. Depending on the settings in the service context, it may not be available on your machine.



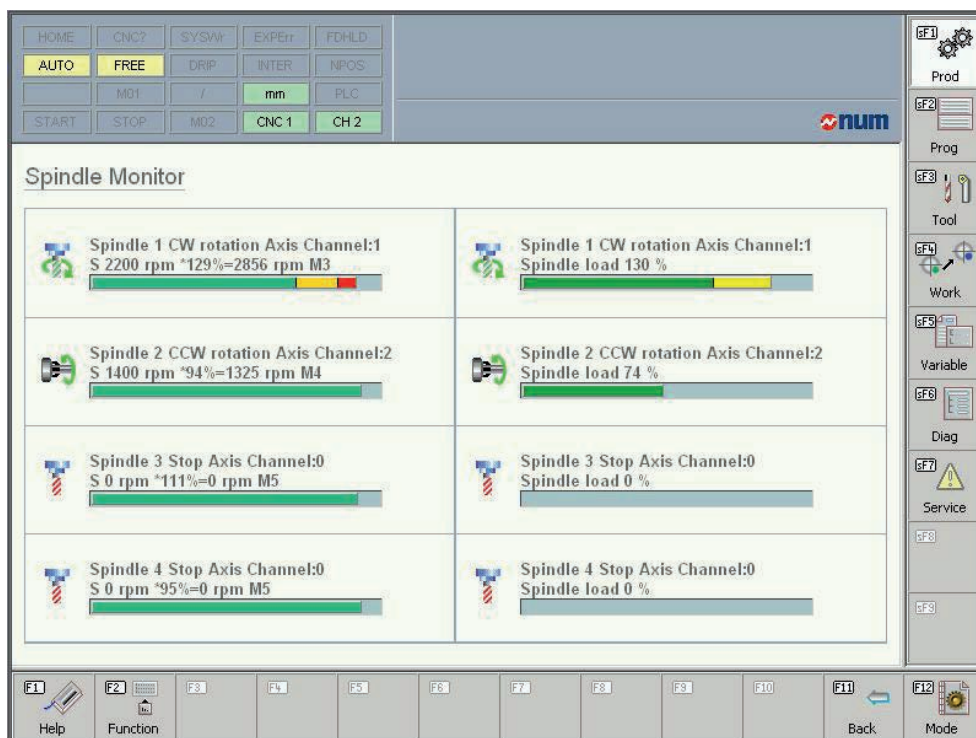
Open the channel selection	F11
Select the desired channel	↑ or ↓
Validate	Enter ↵

5.3 Spindle monitor

This page shows the state of the spindles in two windows.

- Left hand window: programmed speed and override value.
- Right hand window: spindle load.

To access the spindle monitor page	F10
------------------------------------	-----



To exit the spindle monitor page	F11
----------------------------------	-----

5.4 Duplicated axes

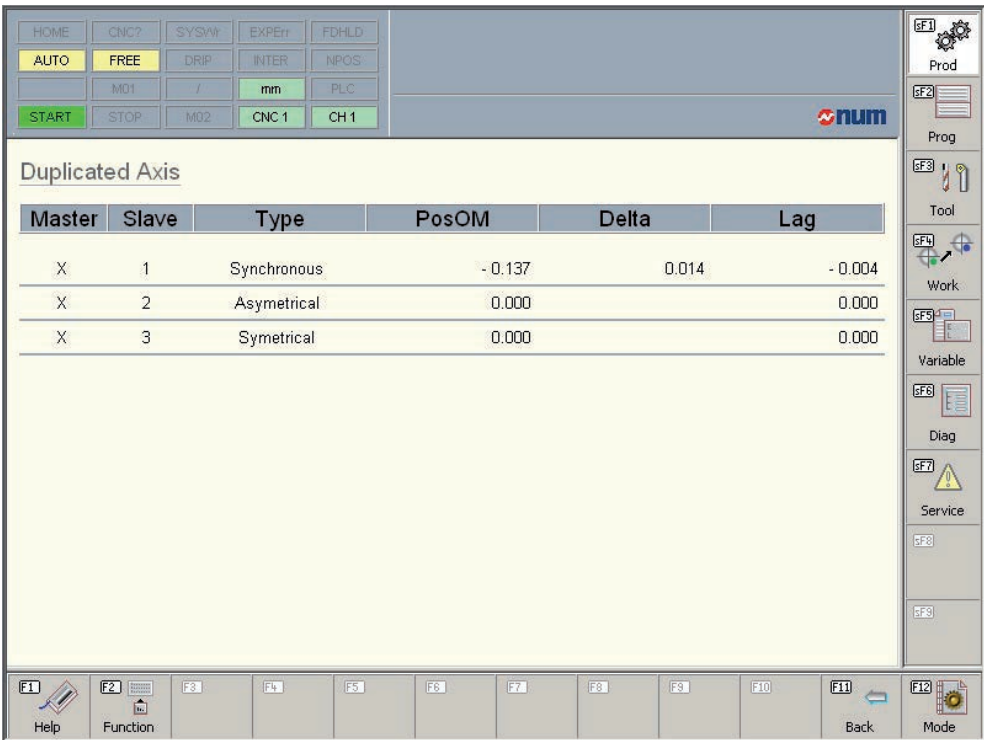
This page gives information on the coupling, duplication and synchronization of axes.
For detailed information about axes duplication and synchronization, refer to the Flexium Programming manual (reference: M00018).



This function is only available if the function “Duplicated and synchronized axes” is present.

To access the Duplicated Axis page

 F9



Master	Slave	Type	PosOM	Delta	Lag
X	1	Synchronous	- 0.137	0.014	- 0.004
X	2	Asymetrical	0.000		0.000
X	3	Symetrical	0.000		0.000

To exit the Duplicated Axis page

 F11

5.5 Axes information

This page gives supplementary information on axes and program states.

To access the axes information page

 F8

HOME		CNC?		SYS SW		EXPER		FDHLD	
AUTO		FREE		DRIP		INTER		NPOS	
		MO1		/		mm		PLC	
START		STOP		MO2		CNC 1		CH 1	

*** DRAWING 1 ***

num

Program:	%13 N170	Axis Address Table	
NC Block:	N160		
G77 Repetition:	0 / 0		
Programmed position	X:37.628 Y:-40.145 Z:-32.000 U:0.000 C:0.0000		
Programmed Work Offset:	X 0.000 Y 0.000 Z 0.000 U 0.000 C 0.0000		
Mirror Axis:	X:0 Y:0 Z:0 U:0 C:0		
Synchro Point:			
Tool Direction:	R+	Scaling:	1
ED:	0	Grinder Inclination:	0
EG:	100	Axis Inclination:	0
Assigned Axis:	XP+YQ+ZR+		

F1	F2	F3	F4	F5	F6	F7	F8	F9	F10	F11	F12
Help	Function									Back	Mode

F1  Prod

F2  Prog

F3  Tool

F4  Work

F5  Variable

F6  Diag

F7  Service

F8

F9

To exit the axes information page

 F11

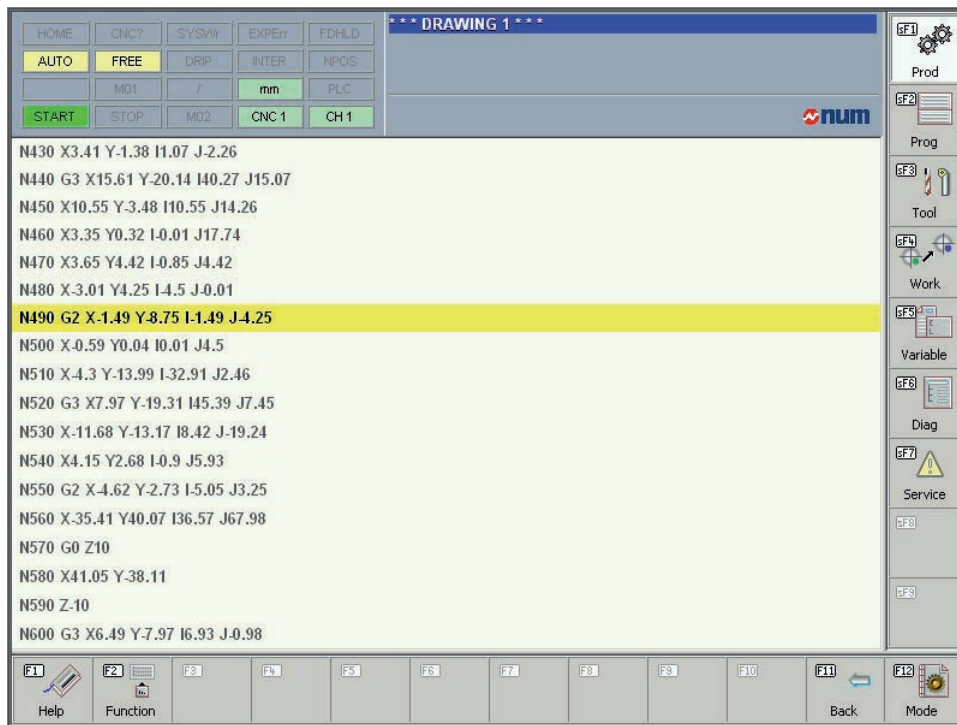
5.6 CNC Blocks Page

This page displays the CNC blocks in full-screen mode and allows to follow easily the course of machining.

The upper part shows the blocks already executed.

The actual block is marked with a yellow bar and the lower part of the screen shows the blocks still to be executed.

To access the CNC Blocks page



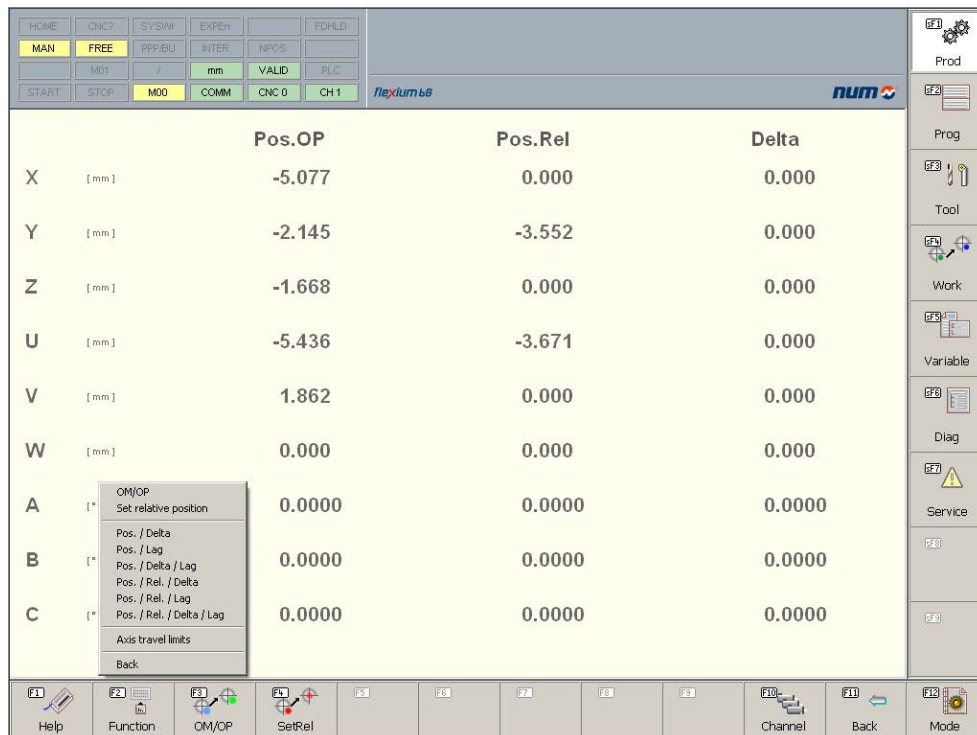
5.7 Axes Position Page

This page shows all axes values and provides a precise control of the axes positions. It offers the possibility to display relative positions, allowing measurement of distances.

The relative measurement can be used in all CNC modes in which axes motions are possible. The relative position feature in this page can be enabled in the context "Service, General settings".

To access the axes position page

 F6



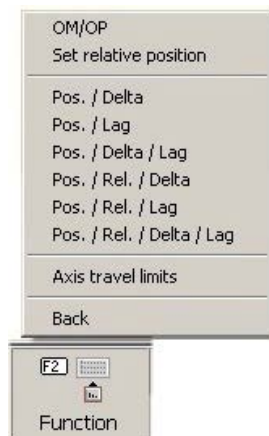
Possible actions

Press **F4** to reset the relative axes positions display

Press **F3** to toggle between OM and OP display

Press **F2** to access all existing functions. Select the function with **↑** or **↓** and **Enter↵**

The following display modes may be selected:



5.7.1 Axis travel limits

For machine setup, knowing the actual axis position and the static as well as the dynamic limits can be useful.

To access the axis travel limits page, select “Axis travel limits” in the function menu.

	Stat.max	Dyn.max	Pos.OM	Dyn.min	Stat.min
X [mm]	1000.000	100.000	-7.780	-100.000	-1000.000
Y [mm]	1000.001	101.111	-5.935	-101.111	-1000.001
Z [mm]	1000.002	102.222	-1.668	-102.222	-1000.002
U [mm]	1000.003	103.333	-3.495	-103.333	-1000.003
V [mm]	1000.004	104.444	1.862	-104.444	-1000.004
W [mm]	1000.005	105.555	0.000	-105.555	-1000.005
A [*]			0.0000		
B [*]			0.0000		
C [*]			0.0000		

OM/OP:
Set relative position
Pos. / Delta
Pos. / Lag
Pos. / Delta / Lag
Pos. / Rel. / Delta
Pos. / Rel. / Lag
Pos. / Rel. / Delta / Lag
Axis travel limits
Back

5.8 CNC functions



The CNC function bar can be activated in every available context.



Navigation between windows and fields in a given context (mostly with Tab, ↑, ↓, ←, →) can only be accomplished when the CNC function bar is not activated!

For further information about CNC functions, refer to chapter 15 of this manual.

To activate the CNC function bar	F12
----------------------------------	-----



Possible actions

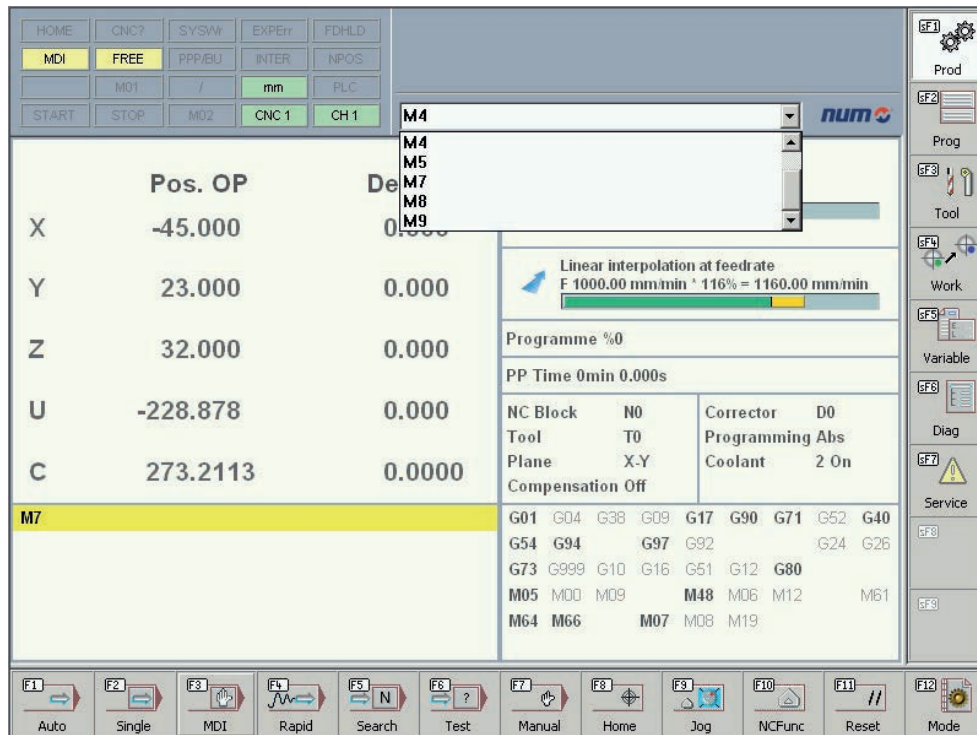
Press F12 to toggle between CNC function bar and sub-contexts display
Press F11 to activate the CNC reset
Press F10 to activate or deactivate the M01 and block skip functions. Select the function with ↑ or ↓ and Enter
Press F9 to access the jog increment selection. Select the increment with ↑ or ↓ and Enter
Press F8 to select the home mode
Press F7 to select the manual mode
Press F6 to select the test mode
Press F5 to select the search mode
Press F4 to select the rapid mode
Press F3 to select the MDI mode
Press F2 to select the single mode
Press F1 to select the automatic mode

5.9 Dialog field

The dialogue box is present only in certain modes (and sometimes only under certain conditions). It is used to enter data in the MDI, SEARCH and HOME modes. It is also used in the case of a message display with wait for an operator response.



To input data, the CNC function bar must be active. If the search mode is selected, the dialog field opens only after the axes have been homed.



5.9.1 Data input into the dialog field

Enter data and validate	Enter
The last entries are stored in a dialog list. It is possible to recall, modify and execute them again	
To open the dialog list and select an entry	↑ or ↓
If needed, the selected entry may be edited	← or →
Validate	Enter
To close the entry list	Esc



If the dialog list is already open, entering the first character of data sets the focus to the first similar value contained in the list.

5.9.2 Deleting the dialog list

The entire dialog list can be deleted if needed.



This function is only available in the context "Production".

Activate the sub-context bar

 F12

Select function

 F2

Reset machining time
 Spindle display Ctrl + L
Delete dialog list
 Display axes only
 Display blocks only
 Axes informations
 Duplicated axes
 Spindle monitor

Select "Delete dialog list"

 ↑ or ↓

Validate

 Enter ↵

Page deliberately empty.

Summary

6	Context "Programming"	55
6.1	Access to the Context "Programming"	55
6.1.1	Representation of data locations in your system	55
6.1.2	Definition of the files existing on PC disk and in CNC memory	55
6.1.3	Start-up window of the context "Programming"	56
6.2	Moving between Windows	57
6.3	Selecting Directory and Folder on the PC disk	57
6.4	Selecting a File on PC disk	58
6.5	Creating a new part program on PC disk	60
6.5.1	Hidden programs and subroutines	61
6.5.2	Part programs created with another editor or on an external system	61
6.5.3	Format of an ISO part program	61
6.6	Creating a new Compact part program on PC disk	62
6.6.1	Compact programs created with another editor or on an external system	62
6.6.2	Format of a Compact part program	63
6.7	Creating a new "Usinage" Program on PC disk	64
6.7.1	"Usinage" programs created with another editor or on an external system	64
6.7.2	Format of a "Usinage" program	65
6.7.3	The include function	65
6.7.4	The Loop and Include functions	65
6.8	Creating a new tool offset file on PC disk	66
6.8.1	Tool offset files created with another editor or on an external system	67
6.9	Creating a new work offset file on PC	68
6.9.1	Work offset files created with another editor or on an external system	69
6.10	Editing a file on PC disk	70
6.10.1	Keyboard functions	71
6.11	Editing a "Usinage" file on PC disk	72
6.11.1	Keyboard functions	73
6.12	Including a Part Program in an "Usinage" File	73
6.13	Creating a Loop in an "Usinage" File	73
6.14	Editing NUM CNC Cycles	74
6.15	Drip Feed Mode	75
6.15.1	Principle of drip feed mode	75
6.15.2	Drip feed on machines with several axes channels	76
6.15.3	Block search with drip feed mode	77
6.16	Loading a file into CNC memory	78
6.16.1	Load one file only	78
6.16.2	Load several successive files	78
6.16.3	Load several non successive files	78
6.17	Protected Memory Area Management	79
6.17.1	Selecting CNC Memory Zones	80
6.17.2	Closing the memory zones	80
6.18	NC Memory	81
6.19	Selecting the Active Part Program	82
6.20	Saving Files to PC Disk	83
6.20.1	Save one file only	83
6.20.2	Save several successive files	83

6.20.3	Save several non successive files	83
6.21	Editing a Program directly in NC Memory	84
6.22	Deleting Files in PC Disk or NC Memory.....	86
6.22.1	Delete one file only	86
6.22.2	Delete several successive files.....	86
6.22.3	Delete several non successive files.....	86

6 Context "Programming"

6.1 Access to the Context "Programming"

To access the context "Programming"	  + 
To close pop-up menus and windows, press	 

The context "Programming" is used to select the active program and edit and save your data.

6.1.1 Representation of data locations in your system

Operating Panel PC



Programs, work and tool offset files, stored on PC disk

NCK



Disk to CNC Memory

CNC Memory to Disk

Programs, work and tool offset files, ready for machining in CNC memory

6.1.2 Definition of the files existing on PC disk and in CNC memory

Part programs *.xpi

A part program *.xpi can be executed in two different manners:

- Directly from the CNC memory,
- From the PC disk in drip feed mode. This mode is mainly used to handle large data volume programs such as those created by CAD systems. See "Restrictions related to drip feed mode" in the Flexium Programming manual (Reference : M00018).

Compact part programs *.xpc

A compact part program may contain several individual ISO part programs.

During a transfer operation disk > CNC, these are loaded altogether into the CNC memory.

Usage programs *.usi

A usage program can only be executed in drip feed mode.

The two functions Include and Loop allow specific programming, subroutine calls and repetition of the files* xpi.

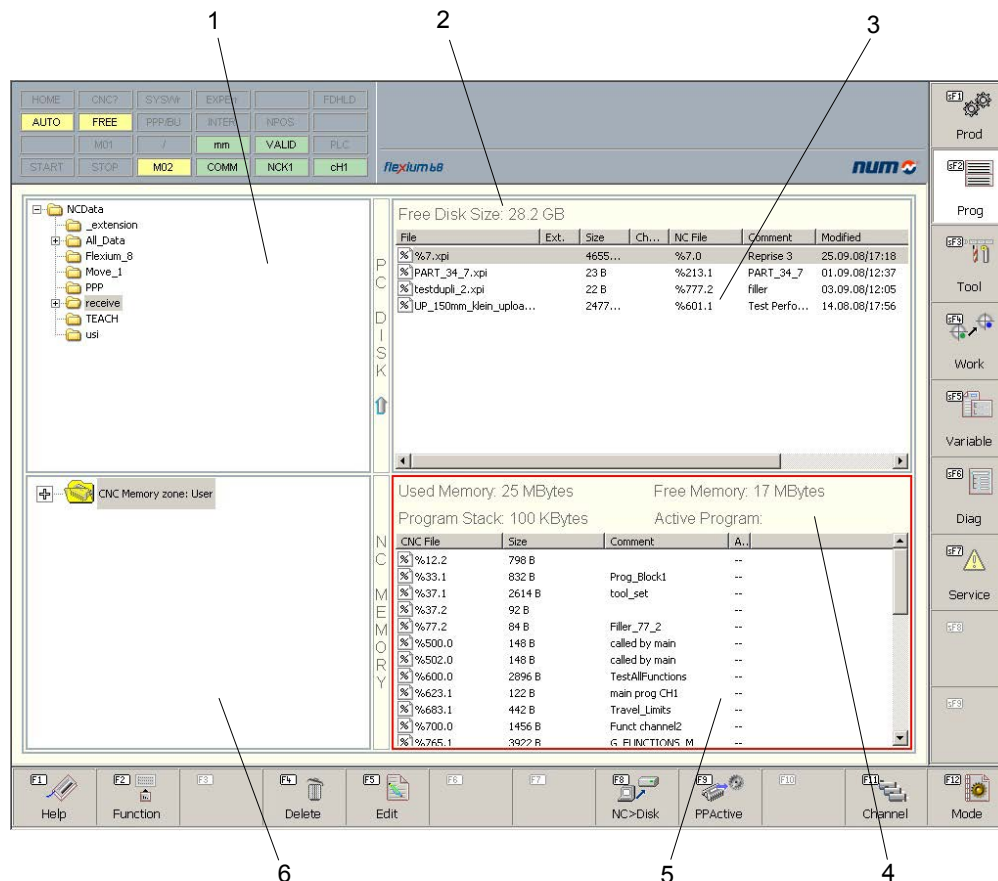
Tool offset files *.jau

Contain all tool offset data. They are resident in CNC memory and must be loaded according to the tools being used.

Work offset files *.dat

Contain all work offset data. They are resident in CNC memory and must be loaded according to the part being machined.

6.1.3 Start-up window of the context “Programming”



- 1 - Directory and folder window on PC disk
- 2 - Free disk size and active drip file display
- 3 - File list window on PC disk (the field lengths can be adjusted with a mouse, these settings will be saved automatically)
Files can be sorted by clicking on the desired column title.
Sorting may be done by:
file name, file extension, size, CNC file name, comment, modification date or attribute
- 4 - CNC memory information and active program
(only the number of the main program is displayed)
- 5 - CNC memory file list window (the field lengths can be adjusted with a mouse, these settings will be saved automatically)
Files can be sorted by clicking on the desired column title
Sorting may be done by: file name, size, comment or attribute
- 6 - CNC memory zone select window.

6.2 Moving between Windows

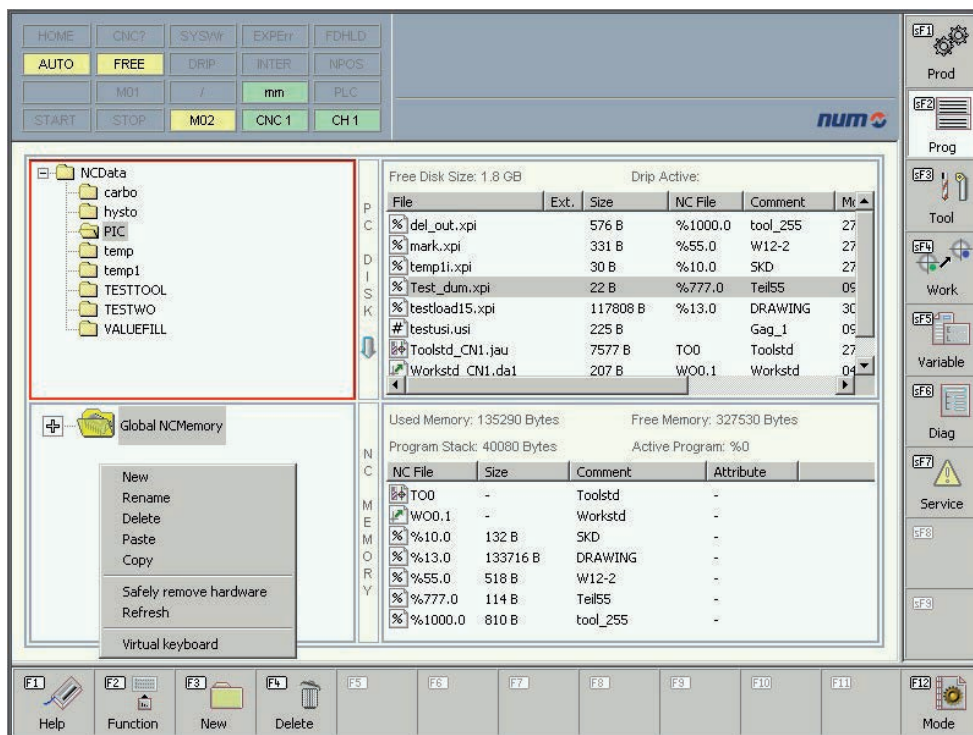
Select the window	Tab
-------------------	-----

6.3 Selecting Directory and Folder on the PC disk

Select the window	Tab ...
Select the desired directory or folder	
Validate	Enter



It is possible to use external data storage media like USB sticks, etc. Before removing such devices, they should be logged out of the system. To do this: **F2**, select "Safely remove hardware" in the pop-up menu.

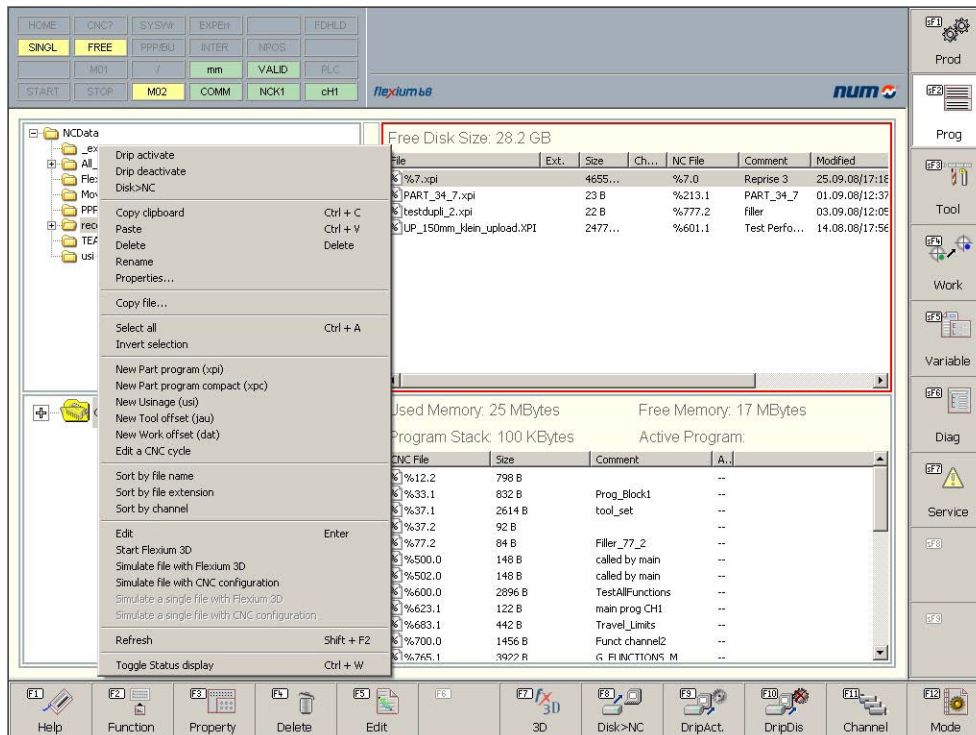


Possible actions

Press F4 or Delete to delete a folder
Press F3 to create a folder
Press F2 to copy, paste, delete, rename, create a folder
If the editor protection has been activated in the context "Service" or by the PLC, the following will not be available: F4 Delete. In the F2 pop-up menu: Delete and Rename.

6.4 Selecting a File on PC disk

Select the right hand window on the PC disk	Tab...
Select the desired file	↑ or ↓
To move the cursor one window field at a time	PgUp or PgDn
To move directly to the top or the bottom of the list	Home or End
To view all data on a line	← or →



Possible actions

Press F10 to deactivate drip feed mode
Press F9 to activate drip feed mode (CNC must be in automatic, single, rapid, search or test mode)
Press F8 to transfer a file from PC disk to CNC memory
Press F7 to access Flexium 3D. For further information, refer to the Flexium 3D manual, reference M00029.
Press F5 to edit a file
Press F4 or Delete to delete a file
Press F3 to change the properties of a file
Press F2 to create a new file or to access all existing functions. Select the function with ↑ or ↓ and Enter
 If the editor protection has been activated in the context "Service" or by the PLC, the following will not be available: F8 Disk>NC F5 Edit F4 Delete F3 Property In the F2 Function menu: Delete, Rename and Property.

6.5 Creating a new part program on PC disk

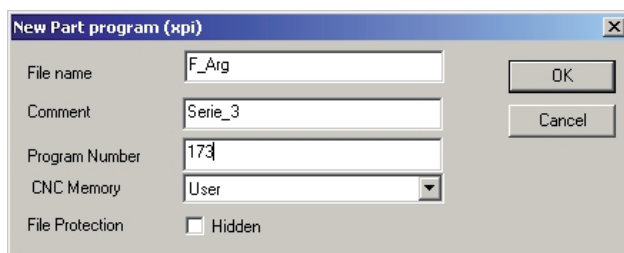
Select the right hand window on the PC disk	 Tab ...
Select Function	 F2



A pop-up menu shows the possible file types:

- Part program (*.xpi)
- Compact part program (*.xpc)
- Usinage (*.usi)
- Tool offset (*.jau)
- Work offset (*.dat).

Select New Part program (*.xpi)	 ↑ or ↓ ...
Validate	 Enter ↵



Select the appropriate field in the pop-up window	 Tab ...
- Enter file name, comment and program number. - Program number: On a single axes channel machine, enter a whole number. On a machine having several axes channels, enter the program number followed by the channel number. Example: %123.1 is the part program for axes channel 1 %123.2 is the part program for axes channel 2.	
 Comments must begin with a letter. Depending on the settings in the Service context, the next field, CNC Memory, may not be available on your machine.	

The CNC memory zone or area chosen as default in this field is the global memory area "User". No change in the setting is needed if the program is to be loaded in this global area. If the program is to be loaded in a protected memory area, proceed as follows:

Select a CNC memory zone	 ↑ or ↓ ...
Validate	 Enter ↵

6.5.1 Hidden programs and subroutines

A suppression mark may be added to the program number to inhibit the display of programs or subroutines being executed. Such files are given the attribute "Hidden" which is displayed as an "h" in the Attribute column of PC disk and NC memory.

This feature can be activated by pressing  Space or  Enter ↵ in the "Hidden" box.

After having entered all necessary properties of the file, press  Enter ↵
The new part program appears in the right hand window of PC disk and may now be edited.

6.5.2 Part programs created with another editor or on an external system

Before using them, properties must first be given to these files.	
Select the program (See '6.4 Selecting a File on the PC disk')	
Select "Properties" Enter the properties as described above	 F3

6.5.3 Format of an ISO part program

Refer to the Flexium Programming manual (Reference: M00018).

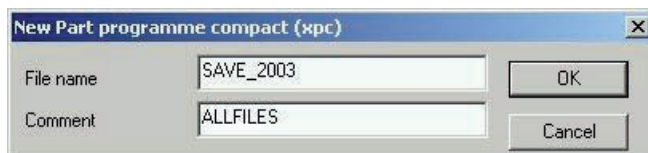
6.6 Creating a new Compact part program on PC disk

Select the right hand window on the PC disk	 Tab
Select Function	 F2

A pop-up menu shows the possible file types:

- Part program (*.xpi)
- Compact part program (*.xpc)
- Usinage (*.usi)
- Tool offset (*.jau)
- Work offset (*.dat).

Select New Compact part program (.xpc)	 ↑ or ↓
Validate	 Enter



Select the appropriate field in the pop-up window	 Tab ...
Enter file name and comment	
Validate	 Enter
 Comments must begin with a letter. The new Compact file appears in the right hand window of PC disk and may now be edited.	

6.6.1 Compact programs created with another editor or on an external system

Before using them, properties must first be given to these files.

Select the program (See '6.4 Selecting a File on the PC disk')	
Select Properties Enter the properties as described above	 F3

6.6.2 Format of a Compact part program

A Compact part program may contain several individual ISO part programs which can be loaded altogether into the CNC memory in one operation.

```
%100  
  
N10 G52 G0 X100 Y100 Z100  
N20 M64 M3 S5000  
...  
...  
M2  
%235  
N10 T12 D16  
N20 G1 X200 Y200 F2000  
N50 .....  
...  
...  
N99 M2  
%562  
...  
...  
M2
```

The main use of Compact files is to load programs into the CNC which have been unloaded using the CNC mode "Unload, All programs" of a CNC of the range NUM 10xx, Axiom or Axiom Power.

To be able to use such files with the HMI, they have to be renamed with the file extension *.xpc.

6.7 Creating a new “Usage” Program on PC disk

Select the right hand window on the PC disk	 Tab...
Select Function	 F2

A pop-up menu shows the possible file types

- Part program (*.xpi)
- Compact part program (*.xpc)
- Usage (*.usi)
- Tool offset (*.jau)
- Work offset (*.dat).

Select new Usage (.usi)	 ↑ or ↓
Validate	 Enter↵



Select the appropriate field in the pop-up window	 Tab...
Enter filename and comment	
Validate	 Enter↵



Comments must begin with a letter.

The new Usage file appears in the right hand window of PC disk and may now be edited.

6.7.1 “Usage” programs created with another editor or on an external system

Before using them, properties must first be given to these files.

Select the program (See ‘6.4 Selecting a File on the PC disk’)	
Select Properties Enter the properties as described above	 F3



“Usage” program can only be executed in drip feed mode.

6.7.2 Format of a "Usage" program

The programming format is the same as an ISO part program, but the usage program offers some new possibilities

Normal ISO code:

```
%100 (Usage with ISO functions only)
N10 G52 G0 X100 Y100 Z100
N20 M64 M3 S5000
N30 T12 D16
N40 G1 X200 Y200 F2000
N50 .....
N99 M2
```

6.7.3 The include function

The ISO program nc100k.xpi included will be executed once.



The end of program function (M02) is not allowed in an included program. There is no block number allowed before #include and #loop commands.

```
%100 (Usage with ISO and Include command)
N10 G52 G0 X100 Y100 Z100
N20 M64 M3 S5000
N30 T12 D16
N40 G1 X200 Y200 F2000
#include c:\ncdata\nc100k.xpi...
N99 M2
```

Only the ISO program nc100k.xpi included will be executed once.

```
%100 (Usage with Include command only)
#include c:\ncdata\nc100k.xpi
N99 M2
```

6.7.4 The Loop and Include functions

The ISO program nc100k.xpi included will be executed 100 times.

```
%100 (usage with Include and Loop)
N10 G52 G0 X100 Y100 Z100
N20 M64 M3 S5000
N30 T12 D16
N40 G1 X200 Y200 F2000
#LOOP[100]
#include c:\ncdata\nc100k.xpi...
#LOOP END
N99 M2
```

6.8 Creating a new tool offset file on PC disk



At the first start of the HMI after its installation, the CNC memory already contains a tool offset file with values of zero for all tool dimensions. This CNC file named TO0 with the comment Toolstd can be used and modified as described in the chapter “Tool Context”.

Select the right hand window on the PC disk	Tab...
Select Function	F2

A pop-up menu shows the possible file types:

- Part program (*.xpi)
- Compact part program (*.xpc)
- Usinage (*.usi)
- Tool offset (*.jau)
- Work offset (*.dat).

Select New Tool offset (.jau)	↑ or ↓
Validate	Enter↵

Select the appropriate field in the pop-up window	Tab...
Enter filename, file number and comment.	



Comments must begin with a letter.

Dimension of the tool offsets (mm or inch) must also be chosen. This is important for later value input, as it gives the possibility to enter the value directly in the chosen dimension.

Select a dimension button	Tab...
Toggle between “mm” and “inch”	← or →
To validate,	Enter↵

The new Tool offset file appears in the right hand window of PC disk and may now be edited.

6.8.1 Tool offset files created with another editor or on an external system

Before using them, properties must first be given to these files.

Select the file (See '6.4 Selecting a File on the PC disk')	
Select "Properties" Enter the properties as described above	F3

Format of a tool offset file

T7(part 072) D1 L29.455 R3.002 @0.250 H12.695 D2 X97.450 Z61.110 R11.370 @0.100 H0.000 C3 D3 U35.750 Z5.480 R0.200 H0.000 C7 D4 L0.000 R0.000 @0.000 H0.000	Properties of the file Address format for a milling tool Address format for a turning tool Address format for a boring tool
---	--



All addresses for every tool type must be entered even if they are not used.
Addresses with a null value must be followed by a zero.

6.9 Creating a new work offset file on PC



At the first start of the HMI after its installation, the CNC memory already contains a work offset file with values of zero for all DATs . This CNC file named WO0.x (x standing for the axes channel number) with the comment Workstd can be used and modified as described in the chapter “Work offset Context”.

Select the right hand window on the PC disk	Tab...
Select Function	F2

A pop-up menu shows the possible file types:

- Part program (*.xpi)
- Compact part program (*.xpc)
- Usinage (*.usi)
- Tool offset (*.jau)
- Work offset (*.dat).

Select New Work offset (*.dat)	↑ or ↓
Validate	Enter ↵

Select the appropriate field in the pop-up window	Tab...
Enter filename, file number and comment	



Comments must begin with a letter.

Dimension of the work offsets (mm or inch) must also be chosen. This is important for later value input, as it gives the possibility to enter the value directly in the chosen dimension.

Select a dimension button	Tab...
Toggle between “mm” and “inch”	← or →
In multi-channel configuration, the channel number corresponding to the work offset file must be entered. After having entered all necessary properties of the file, press Enter ↵	
The new Work offset file appears in the right hand window of PC disk and may now be edited.	

6.9.1 Work offset files created with another editor or on an external system

Before using them, properties must first be given to these files.

Select the file (See '6.4 Selecting a File on the PC disk')	
Select Properties	 F3
Enter the properties as described above	

Format of a work offset file

W0(Workstd) DAT1X0.015 DAT2X0.000 DAT3X0.000 DAT1Y2.000 DAT2Y5.000 DAT3Y7.000 DAT1Z0.000 DAT2Z0.000 DAT3Z8.000 DAT1U0.000 DAT2U0.000 DAT3U0.000 DAT1V0.000 DAT2V0.000 DAT3V0.000 DAT1W0.000 DAT2W0.000 DAT3W0.000 DAT1A114.000 DAT2A44.000 DAT3A0.0000 DAT1B221.0000 DAT2B99.0000 DAT3B0.0000 DAT1C0.0000 DAT2C0.0000 DAT3C0.0000	Properties of the file Example of file for axes channel with 9 axes maximum (X, Y, Z, U, V, W, A, B and C)
--	---



All addresses for every work offset must be entered even if they are not used. Addresses with a null value must be followed by a zero.

6.10 Editing a file on PC disk

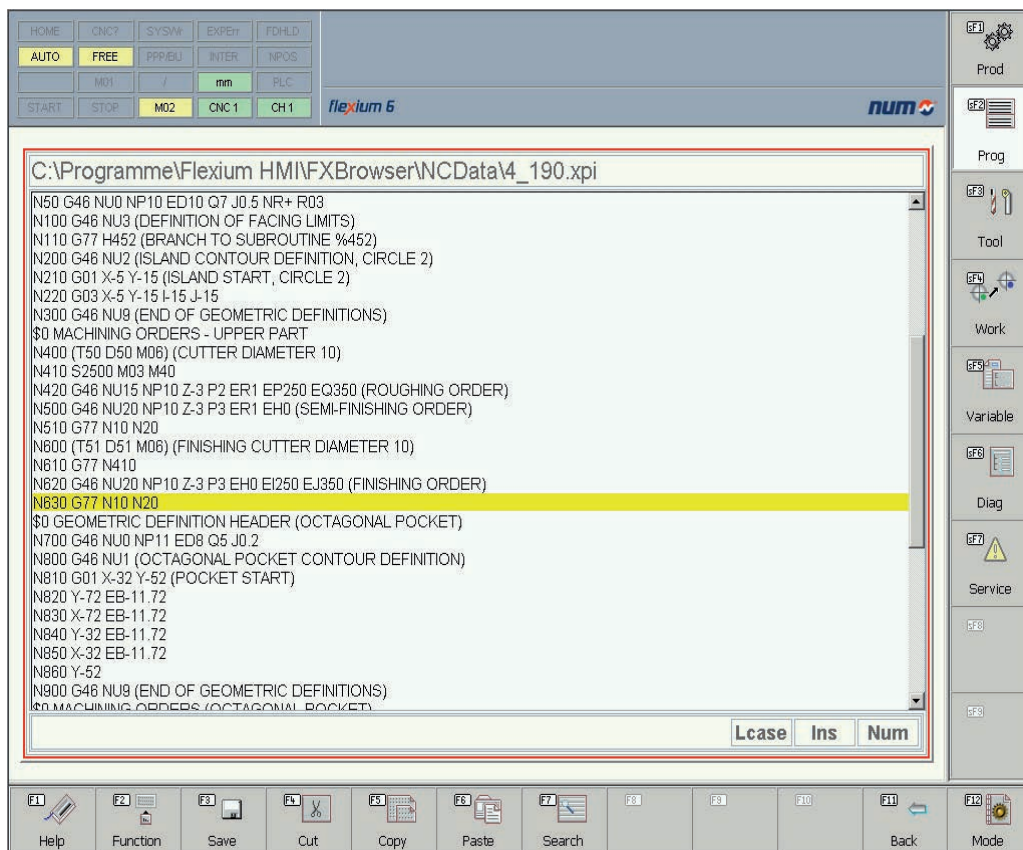
The editor is used to modify, change or improve an existing file or program.
All file types available on PC disk may be edited. In the case of a "Usage" file, the special "Usage editor" will be opened automatically.



The CNC does not recognise functions and commands entered in lower case. Be sure to enter upper case letters only.

Select the right hand window on the PC disk	Tab...
Select the desired file	↑ or ↓
To move the cursor one window field at a time	PgUp or PgDn
To move directly to the top or the bottom of the list	Home or End
Press	F5

This opens the full screen editor:



Possible actions

Press F11 to leave the editor
Press F7 to search for a given string (can also be performed with Ctrl + F)
Press F6 to paste
Press F5 to copy
Press F4 to cut
Press F3 to save the file
Press F2 to undo the previous action or to access all existing functions Select the function with ↑ or ↓ and Enter↵

6.10.1 Keyboard functions

Move cursor up or down: ↑ or ↓
Move cursor left or right: ← or →
Move cursor to end of line: End
Move cursor to beginning of line: Home
Delete character on the left: ← Bksp
Delete character on the right: Del
Mark from cursor position to the right: ⇧ Shift + →
Mark from cursor position to the left: ⇧ Shift + ←
Mark actual line and down (cursor must be at beginning of line): ⇧ Shift + ↓
Mark next line up (cursor must be at beginning of line): ⇧ Shift + ↑
Mark to begin of the file : Ctrl + ⇧ Shift + Home
Mark to end of the file : Ctrl + ⇧ Shift + End
Copy: Ctrl + C
Cut: Ctrl + X
Paste: Ctrl + V
Delete: Del
Undo: Ctrl + Z
Toggle between insert and overwrite: Insert
Toggle between upper and lower case: Ctrl + "S" or CapsLock
To exit edit mode
 F11

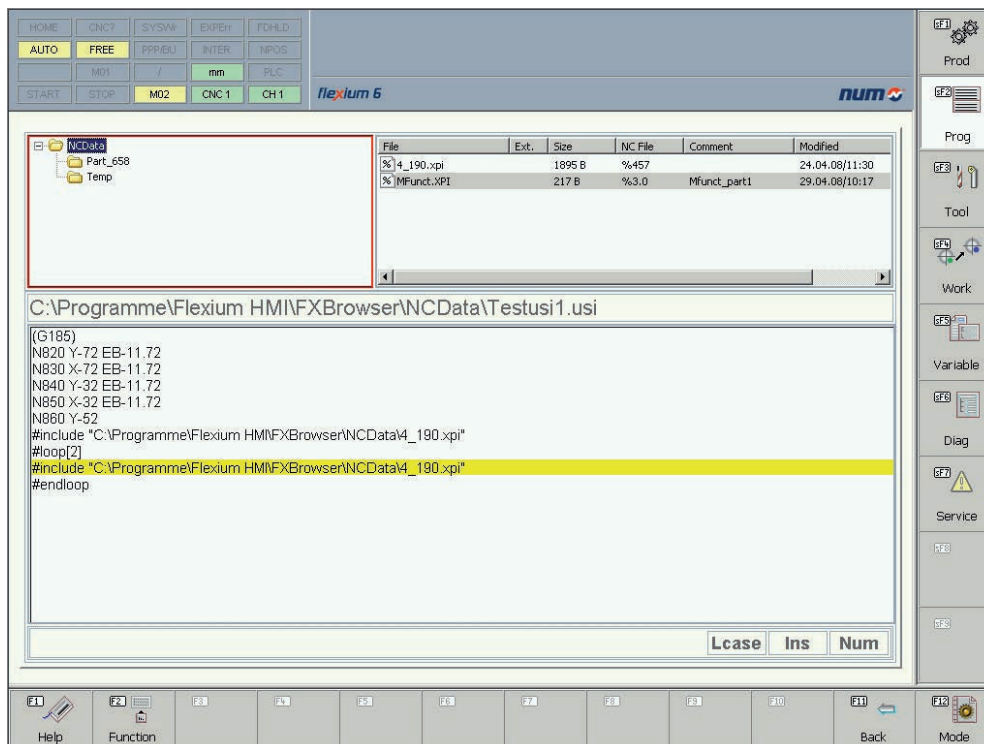
6.11 Editing a "Usage" file on PC disk



The CNC does not recognise functions and commands entered in lower case. Be sure to enter upper case letters only for CNC functions.

Select the right hand window on the PC disk	Tab
Select the desired "Usage File"	↑ or ↓
To move the cursor one window field at a time	PgUp or PgDn
To move directly to the top or the bottom of the list	Home or End
Press	F5

This opens the "Usage" editor:



Possible actions

Press F10 to insert a loop command
Press F9 to include a selected program
Press F7 to search for a string
Press F6 to paste
Press F5 to copy
Press F4 to cut
Press F3 to save the "Usage" file
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter

6.11.1 Keyboard functions

They are the same as for the file editor.

To enter a tabulator: **Ctrl** + **Tab**

6.12 Including a Part Program in an "Usage" File



An included part program must not contain the M02 function.

Select the right hand window on the PC disk	Tab ...
Select the part program to be included	↑ or ↓
To move the cursor one window field at a time	PgUp or PgDn
To move directly to the top or the bottom of the list	Home or End
Select the include function	F9

6.13 Creating a Loop in an "Usage" File

Select the loop command	F10
Include the part program to be executed in the loop between #loop[x] and #endloop	

6.14 Editing NUM CNC Cycles

From the Flexium HMI 3.4.0.0 on, it is possible to edit NUM CNC cycles to adapt them to specific requirements.



This function has to be enabled in the context “Service, general settings”.

In the context “PROG”	
Select the right hand window of PC disk	Tab...
Open the function list	F2
In the list, select “Edit a CNC Cycle”	
To select a setting	Tab...
To open a combination box	Enter↵
To select an item in an open combination box	↑ or ↓ and Enter↵

Select the desired cycle, enter a file name, select the memory area and validate with OK.
The cycle will be copied to the selected folder under the name entered before.

The number of proposed cycles depends on the licenses present in your system.

The homing cycles are always available:
%9990 / %10199 / %9990.9

These cycles are always available:
%10080 / %11019

These cycles are available if the license “Turning cycles” is present:
%10033 / %10063 / %10063.2 / %10063.4 / %10064 / %10064.1 / %10065 / %10066
%10081 / %10082 / %10083 / %10084 / %10085 / %10087 / %10089

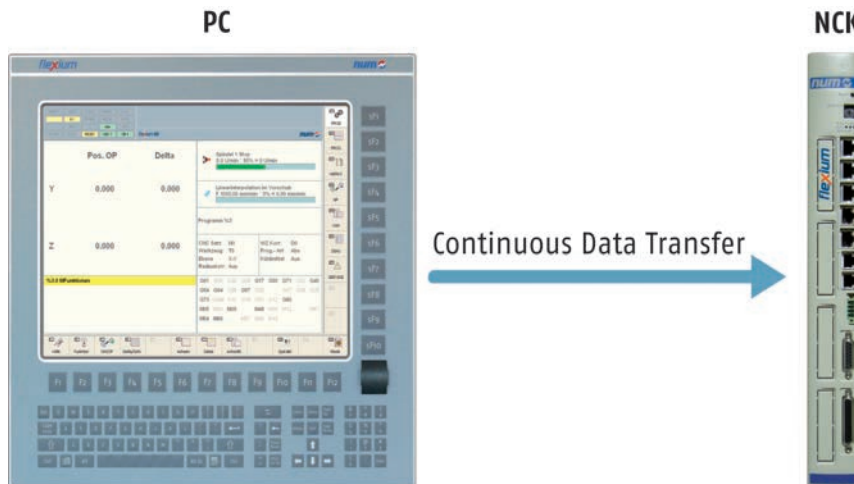
These cycles are available if the license “Milling cycles and standard pocket cycles” is present:
%10031 / %10045 / %10086 / %10088
%10081 / %10082 / %10083 / %10084 / %10085 / %10087 / %10089

6.15 Drip Feed Mode



CNC must be in automatic, single, rapid, search or test mode and in the RESET state (M02 displayed). The program to be executed in drip feed mode must not be present in CNC memory.

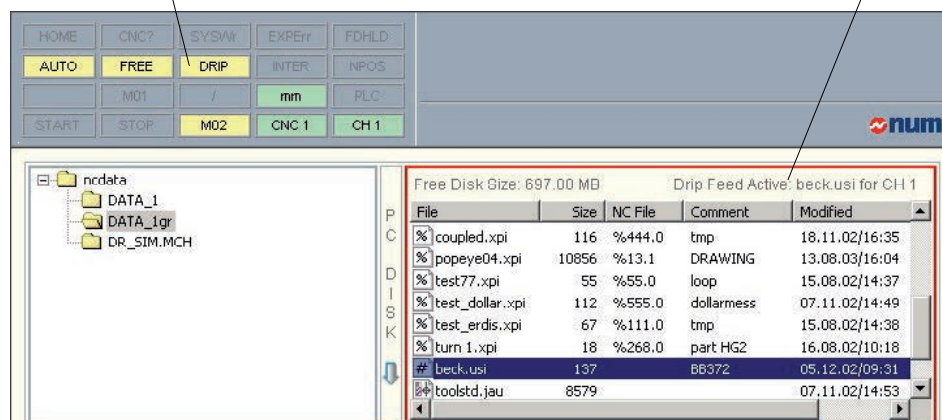
6.15.1 Principle of drip feed mode



Select the right hand window on the PC disk	☞ Tab ...
Select the desired part or "Usage" program	☞ ↑ or ↓
To move the cursor one window field at a time	☞ PgUp or PgDn
To move directly to the top or the bottom of the list	☞ Home or End
Activate the drip feed mode	☞ F9

Drip feed mode active (DRIP)

Active program for drip feed



Press the "CYCLE" key on the machine panel to execute the program.



Drip feed programs are always loaded into the CNC as %0.

Once the program has been activated and executed once, it is possible to repeat this program without need to reactivate it. The program will be started again only by pressing the “CYCLE” key on the operator panel.

Modifications cannot be made to a program during its execution in drip feed mode. All operations which could change the contents or the properties of the file (edit, delete, rename, properties) are not allowed.

Modifications can only be made after having deactivated the drip feed program with **[F10]**. This deactivation is only possible in the state RESET / M02.

However, a program activated for drip feed can be edited during the execution of the original file. After the modification, this program must be saved under another file name.

The modified program has to be reactivated with **[F9]** before starting again.

6.15.2 Drip feed on machines with several axes channels

It is possible to use the drip feed mode on machines with several axes channels.



Drip feed is only available on common channels.

Operation:

- Load a program %0.n in CNC memory for each existing axes channel except the channel in which the drip feed will be executed.

Example: machine with 3 axes channels, drip feed in channel 1:

- The programs %0.2 and %0.3 must be loaded in CNC memory.
- Select the axes channel in the HMI in which the drip feed program is to be executed (in this case : channel 1).

Activate the part or “Usinage” program for drip feed
--

[F9]



Check all settings and validate with OK

Press the “CYCLE” key on the machine panel to execute the program.
--

To exit drip feed mode: select an active part program.
(See chapter 6.18 Selecting the active part program).

6.15.3 Block search with drip feed mode

The SEARCH mode enables to start a program in drip feed mode from a determined block. The operation is similar to the one used in SEARCH mode for search in a part program resident in CNC memory.

For detailed information about the function "Sequence Number Search", refer to chapter 15 of this manual.

Requirements:

- HOME made on all axes
- No program running
- Program selected for drip feed mode.

Select the program for drip feed mode and activate it	 F9
The drip feed mode is activated and the DRIP indicator in the status window turns to yellow	
Select SEARCH mode	 F5
Enter the block number and validate	 Enter ↵
Press the Cycle Start button	

From here on, the operations to be executed correspond to the ones described in chapter 15 "Sequence Number Search" in this manual.

6.16 Loading a file into CNC memory

6.16.1 Load one file only

Select the right hand window on the PC disk	 Tab ...
Select the desired file	 ↑ or ↓
To move the cursor one window field at a time	 PgUp or PgDn
To move directly to the top or the bottom of the list	 Home or End
Load the file into CNC memory It is possible to choose if the file has to be loaded or loaded and activated (for part programs only)	 F8



The activation of part programs is not enabled on all systems. This function is dependent on the setting made in the “Service” context. For the description of this setting, refer to chapter ‘11.2 General Settings Page’.

6.16.2 Load several successive files

Select the desired file	 ↑ or ↓
To move the cursor one window field at a time	 PgUp or PgDn
To move directly to the top or the bottom of the list	 Home or End
Mark the files to load	 ⇧ Shift + ↑ or ↓
Load the files into CNC memory	 F8

6.16.3 Load several non successive files

Select the first file	 ↑ or ↓
To move the cursor one window field at a time	 PgUp or PgDn
To move directly to the top or the bottom of the list	 Home or End
A) Enter select mode	 Ctrl + ↑ or ↓
B) Mark the next file to load	 Space
Return to A) to select further files	
Load the files into CNC memory	 F8



To load Tool offsets and Work offsets (TO and WO files), the CNC must be in Reset / M02 state.

6.17 Protected Memory Area Management

The purpose of protected memory areas, also called zones, is to prohibit access to some programs (fixed subroutines) which may be subject to copyright, and to guarantee correct operation of the machine.

Programs in these areas can be protected by a password and may not be modified.

To achieve this, the RAM memory is divided into four areas or zones:

- area 0: free access User area,
- area 1: Customer area,
- area 2: OEM area,
- area 3: NUM area.

Areas 1 to 3 are called protected and their access may require the entry of a password.

Programs with the same numbers can exist in several memory areas. In this case, the program in the area with the lowest number is executed (for instance, if subroutine %8567 is called and this subroutine exists in areas 1 and 3, the subroutine in area 1 is executed).

Protected areas are divided into two categories:

- No password protection: free access to all programs in this specific area
- Password protection: all programs in this area can be executed, but they cannot be copied to PC Disk or deleted without entering the correct password.

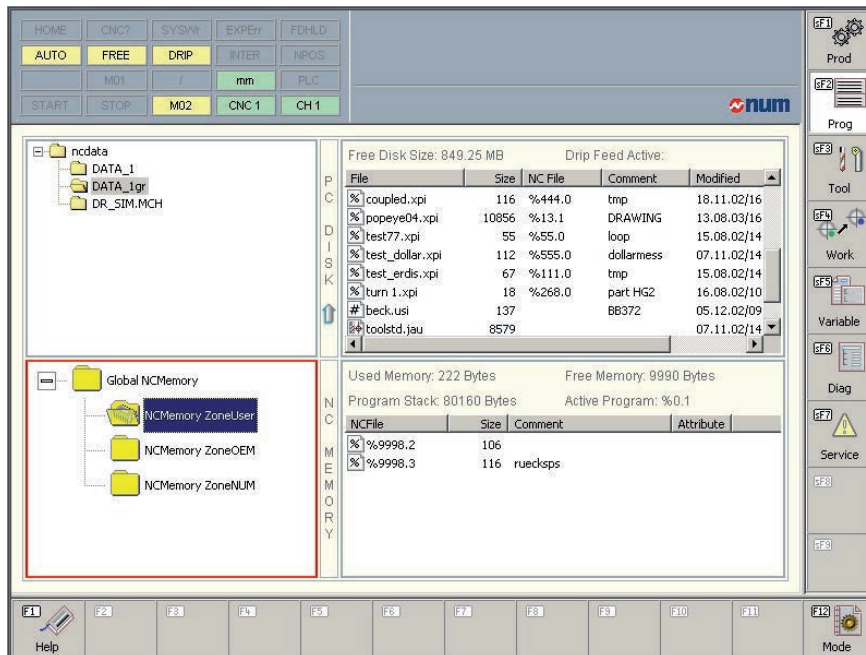


Depending on the settings in the Service context, access to the protected memory areas may not be enabled on your machine.

6.17.1 Selecting CNC Memory Zones

Select the left hand window in the NC memory area	 Tab ...
Open the global memory	
Select the desired zone	 ↑ or ↓ 

The files contained in the chosen zone are shown in the right hand window of NC memory.



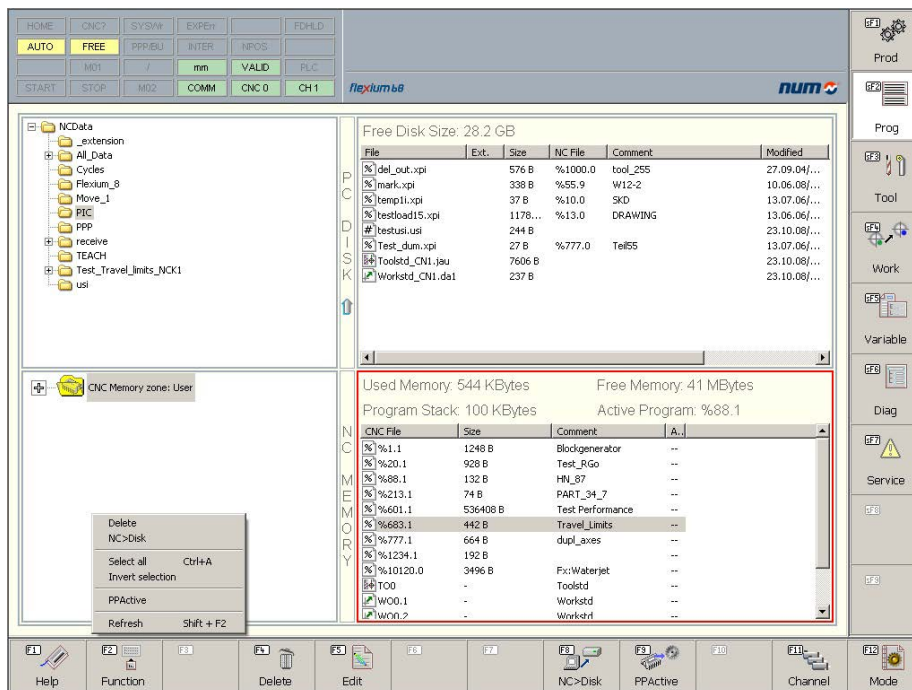
6.17.2 Closing the memory zones

Select the left hand window in the NC memory	 Tab ...
Close the memory zones	

6.18 NC Memory

Select the right hand window in the NC memory area

Tab...



Possible actions

Press **F9** to activate a part program

Press **F8** to save a file to PC disk

Press **F5** to open and edit the selected program

Press **F4** or **Delete** to delete a file

Press **F2** to access all existing functions. Select the function with **↑** or **↓** and **Enter**



If the editor protection has been activated in the "Service" context, the following will not be available:

F8 NC>Disk.

F5 Edit.

F4 Delete.

In the **F2** pop-up menu: Delete, NC>Disk.

6.19 Selecting the Active Part Program

Select the right hand window in NC memory	  ...
Select the desired part program	  or 
To move the cursor one window field at a time	  or 
To move directly to the top or the bottom of the list	  or 
Activate the part program	 

6.20 Saving Files to PC Disk

6.20.1 Save one file only

Select the desired file	 ↑ or ↓
To move the cursor one window field at a time	 PgUp or PgDn
To move directly to the top or the bottom of the list	 Home or End
Transfer the file from NC memory to PC disk	 F8

6.20.2 Save several successive files

Select the first file	 ↑ or ↓
To move the cursor one window field at a time	 PgUp or PgDn
To move directly to the top or the bottom of the list	 Home or End
Mark the files to save	 ⇧ Shift + ↑ or ↓
Save the files to PC disk	 F8

6.20.3 Save several non successive files

Select the first file	 ↑ or ↓
To move the cursor one window field at a time	 PgUp or PgDn
To move directly to the top or the bottom of the list	 Home or End
A) Enter select mode	 Ctrl + ↑ or ↓
B) Mark the next file to save	 Space
Return to A) to select further files	
Save the files to PC disk	 F8

6.21 Editing a Program directly in NC Memory

From the Flexium HMI 3.4.0.0 on, it is possible to edit a part program directly in NC memory, even during execution.



The access to the editor is only enabled if:

- the memory area "User" (area 0) is selected
- the selected file is a part program
- the editor is not protected by settings in the context "Service" or by the PLC.

Procedure

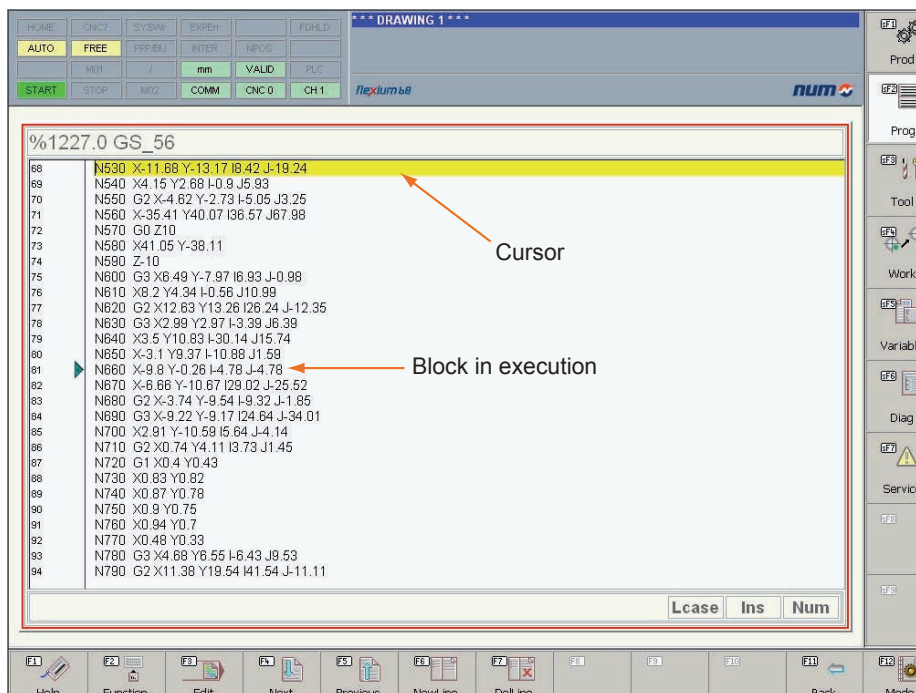
During part program execution, select the context "PROG" and memory area "User"

Select the right hand window of NC memory	Tab...
Select the desired file	↑ or ↓
To move the cursor one window field at a time	PgUp or PgDn
To move directly to the top or the bottom of the list	Home or End
Press Edit	F5



At this stage we do not switch yet to EDIT mode, but request access to the editor.

An editor will open with a pointer on the block in execution (if any):



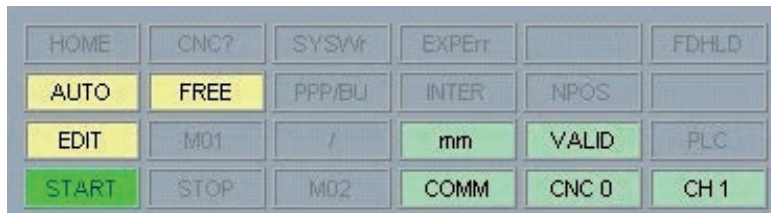
Possible actions

Press [F11] to leave the editor
Press [F7] to delete the line on which the cursor is located
Press [F6] to insert a new line after the cursor location
Press [F5] to display the previous page of the list
Press [F4] to display the next page of the list
Press [F3] to access the editor functions
Press [F2] to access all existing functions. Select the function with [↑] or [↓] and [Enter↵]



To enable an automatic scrolling between pages during program execution, the checkbox "CNC editor: autoscroll" in the context "sF7 Service, Management" must be activated.

It is now possible to enter EDIT mode	[F3]
The EDIT mode will be in standby until the current block is fully executed	



Once EDIT mode becomes active, you can browse through the program, Add, Remove or Change lines.



This is a line editor.

Each modification on a line should be validated with **[Enter↵]**.

Pressing **[Enter↵]** does not create a new line, rather this is performed by the function key **[F6]**.



The main characteristics of this editor are:

- Line numbers are displayed
- Cut, Copy, Paste, Search are not possible
- A Save key is not needed. By pressing ENTER, the new or modified line is automatically written to the NCK memory.
- If the feature Teach-in is available, the Teach-in Editor can also be used

After editing, switch back to AUTO or SEQ mode and resume execution.

6.22 Deleting Files in PC Disk or NC Memory



It is not possible to delete work offset or tool offset files in NC memory. These files may only be modified by transferring them from PC disk to NC memory.

6.22.1 Delete one file only

Select the right hand window in NC memory or PC disk	Tab ...
Select the file	↑ or ↓
To move the cursor one window field at a time	PgUp or PgDn
To move directly to the top or the bottom of the list	Home or End
Delete the file	F4

6.22.2 Delete several successive files

Select the file	↑ or ↓
To move the cursor one window field at a time	PgUp or PgDn
To move directly to the top or the bottom of the list	Home or End
Mark the files to delete	⇧ Shift + ↑ or ↓
Delete the files	F4

6.22.3 Delete several non successive files

Select the file	↑ or ↓
To move the cursor one window field at a time	PgUp or PgDn
To move directly to the top or the bottom of the list	Home or End
A) Enter select mode	Ctrl + ↑ or ↓
B) Mark the next file to delete	Space
Return to A) to select further files	
Delete the files	F4

Page deliberately empty.

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

Page deliberately empty.

Summary

7	Context "Tools"	91
7.1	Access to the Context "Tools"	91
7.2	Entering tool dimensions and miscellaneous values	92
7.3	Selecting the Tool Type.....	92
7.4	Resetting the whole Tool Offset table.....	93
7.5	Teach-in of the Tool Dimensions	94
7.5.1	Milling tool setting page	94
7.5.2	Turning tool setting page	95
7.5.3	Boring tool setting page	95
7.5.4	Procedure	96

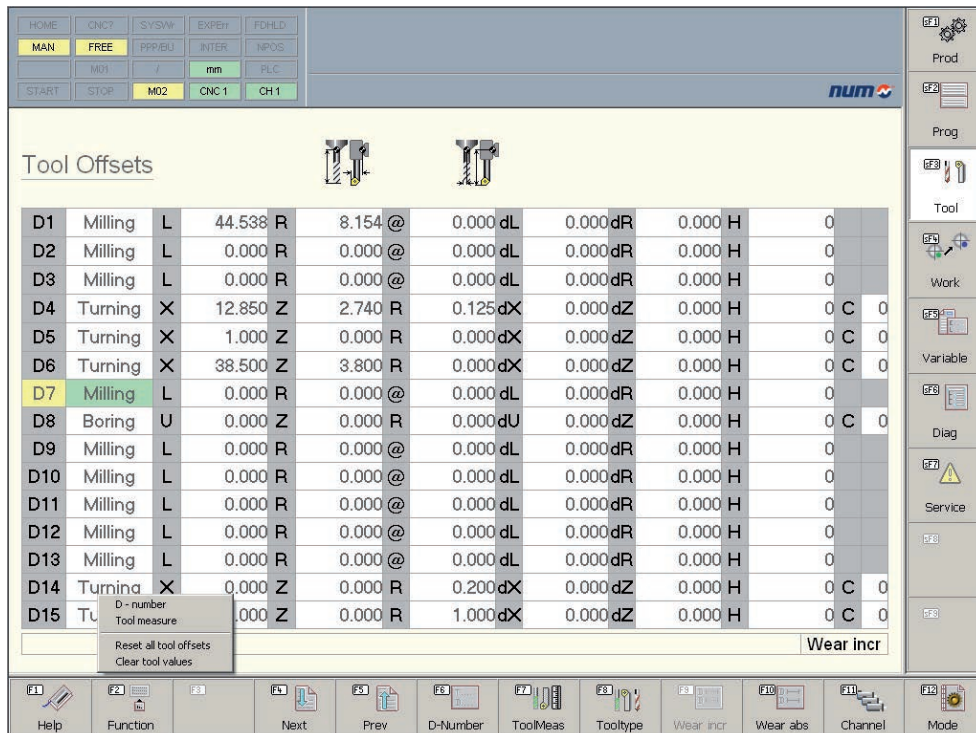
Page deliberately empty.

7 Context "Tools"

7.1 Access to the Context "Tools"

To access the context "Tools"	Shift + F3
To close pop-up menus and windows	Esc

This context is used to enter and modify tool dimensions and to measure the tools.
For detailed information about tool settings, refer to chapter '14 Review' in this manual.



Possible actions

Press F10 to enter absolute tool wear offsets
Press F9 to enter incremental tool wear offsets
Press F8 to choose the tool type. This function is only available if the cursor is in the left column
Press F7 to access tool measure
Press F6 to position the cursor directly on the desired tool offset number
Press F5 to select the previous page of the list
Press F4 to select the next page of the list
Press F2 to access all existing functions. Select the function with or and Enter

7.2 Entering tool dimensions and miscellaneous values

Select a tool using number D, enter the appropriate tool number and validate or Select a tool using the list	 F6  Enter  F4 , F5 , ↑ , ↓ or PgUp , PgDn
Select the required field	 ← or →
Open the field	 Enter ↵
Enter the desired value and validate	 Enter ↵



The cursor can only be moved between white fields. When entering values in the C field for turning or boring tools, it may be necessary to back off the cursor to the left before moving it up or down.

7.3 Selecting the Tool Type



The tool type can only be modified with **F8** if the cursor is in the left column.

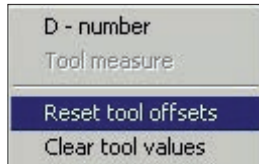
The following tool types may be chosen: milling, turning and boring.

Select a tool using D number, enter the appropriate tool number and validate or Select a tool using the list	 F6 ,  Enter  F4 , F5 , ↑ , ↓ or PgUp , PgDn
Position the cursor on the left hand field	 ←
Select the desired tool type	 F8

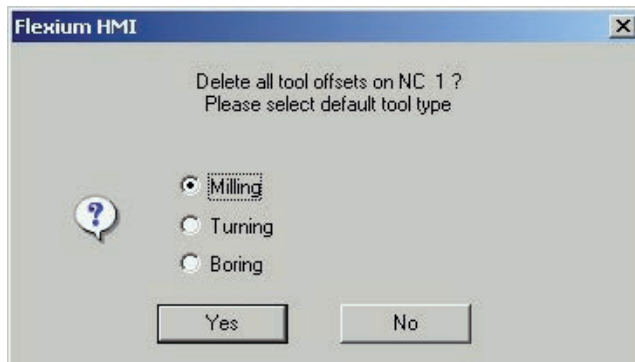
7.4 Resetting the whole Tool Offset table

If needed, the whole Tool Offset table can be reset.
This procedure allows to define a default type for all tools.

Activate the sub-context bar	 F12
Select function	 F2



Select "Reset tool offsets"	 ↑ or ↓
Validate	 Enter ↵



Select the default tool type	 ↑ or ↓
Validate	 Enter ↵

7.5 Teach-in of the Tool Dimensions



The axes must have been homed and the CNC must be in manual mode before this function can be accessed.

Select tool measure	F7
---------------------	----

7.5.1 Milling tool setting page

The screenshot shows the 'Tool Measuring' screen. At the top, there's a status bar with buttons like HOME, CNC?, SYSW, EXPR, FDHLD, MAN, FREE, PP/BU, INTER, INFO, and a 'num' logo. Below this is a table for tool dimensions:

	Pos. OP	D7	Milling	L	52.986	R	7.974	@	0.000
X	7.974								
Y	23.000								
Z	52.986								
U	0.000								
C	0.0000								

To the right of the table is a diagram of a milling tool with two dropdown menus: 'Direction' set to 'R+' and 'Axis' set to 'Z'. Below the 'Axis' dropdown is a green bar with the value '0.000'. On the far right, there's a vertical toolbar with icons for Prod, Prog, Tool, Work, Variable, Diag, Service, and Mode. At the bottom, there's a horizontal toolbar with icons for Help, Function, ToolPar, Valid, D-Number, Tooltype, Channel, Back, and Mode.

Possible actions

Press F8 to choose the tool type
Press F6 to select the tool offset number
Press F5 to validate the setting
Press F4 to select tool direction, axis and for turning and boring tools the tip orientation
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter

7.5.3 Boring tool setting page

HOME		CNC?		SYSM		EXPER		FDHLD		
MAN		FREE		PPF/EU		INTER		NPOS		
		MDI		/		mm		PLC		
START		STOP		M02		CNC 1		CH 2		

Tool Measuring

Pos. OP	D7	Turning	X	52.986	Z	7.974	R	0.000	C	0
----------------	----	---------	---	--------	---	-------	---	-------	---	---

X	0.000
Z	0.000
C	0.0000

Direction

P+

Axis

X

0.000

Tip

C0

1 Help	2 Function	3 ToolPar	4 Valid	5 D-Number	6	7 Tooltype	8	9 Channel	10 Back	11 Mode
--------	------------	-----------	---------	------------	---	------------	---	-----------	---------	---------

11

12

13

14

15

16

17

18

19

20

21

22

23

24

25

26

27

28

29

30

31

32

33

34

35

36

37

38

39

40

41

42

43

44

45

46

47

48

49

50

51

52

53

54

55

56

57

58

59

60

61

62

63

64

65

66

67

68

69

70

71

72

73

74

75

76

77

78

79

80

81

82

83

84

85

86

87

88

89

90

91

92

93

94

95

96

97

98

99

100

7.5.4 Procedure

Select the tool offset number	 F6
Set the focus on the tool type field	 ← or →
Select the tool type	 F8
Set the focus on the R or @ field	 →
Open the field	 Enter ↵
Enter the value for R or @ and validate	 Enter ↵
Set the focus on the tool direction field	 Tab ⇥
Select the desired tool direction	 ↑ or ↓
Or	 F4 and ↑ or ↓ and Enter ↵
Set the focus on the axis field	 Tab ⇥
Select the desired axis	 ↑ or ↓
Or	 F4 and ↑ or ↓ and Enter ↵
Set the focus on the workpiece dimension field	 Tab ⇥
Open the field	 Enter ↵
Enter the workpiece dimension and validate	 Enter ↵
Move the axis to position the tool head in contact with the workpiece	
Validate the setting	 F5
To exit tool measuring mode	 F11



Workpiece and setting piece dimensions are stored and can be used again for a further measurement.

Page deliberately empty.

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

Page deliberately empty.

Summary

8	Context "Work Offset"	101
8.1	Access to the Context "Work Offset"	101
8.1.1	Milling display	101
8.1.2	Turning display	102
8.2	Entering Offset Values	103
8.3	Entering Scaling Factor	103
8.4	Entering Axes Assignment	103
8.5	Setting the Work Offset (DAT1) by Teach-in for Milling	104
8.6	Setting the Work Offset (DAT1) by Teach-in for Turning	106

Page deliberately empty.

8 Context "Work Offset"

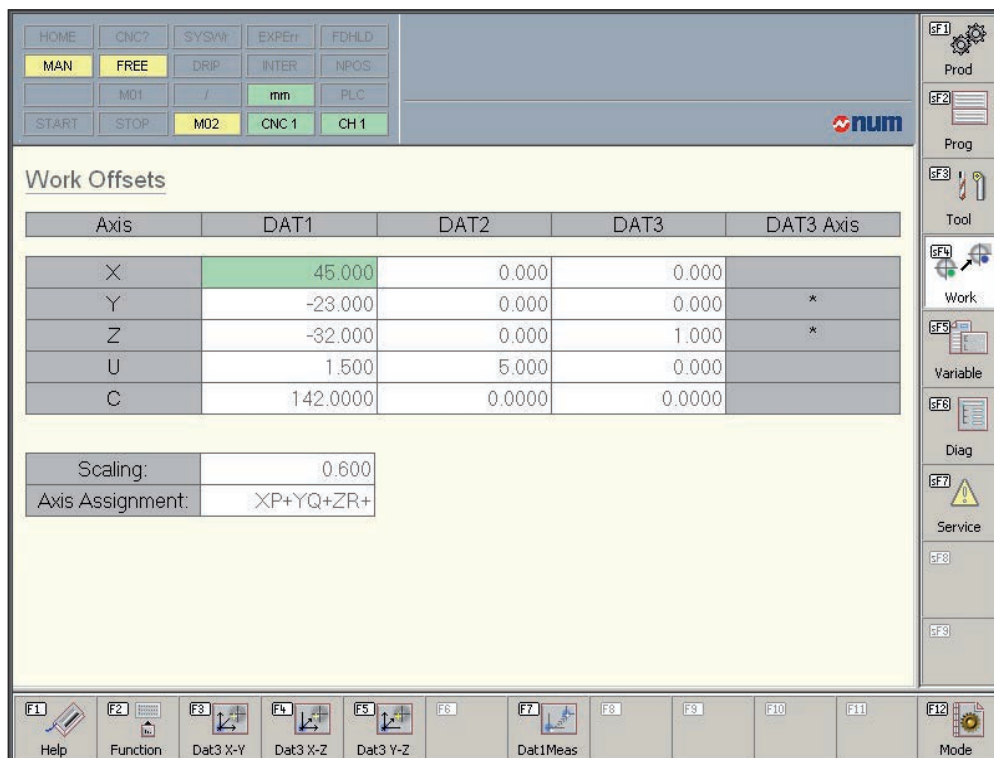
8.1 Access to the Context "Work Offset"

To access the context "Tools"	  + 
To close pop-up menus and windows	 

This context is used to enter and modify work offsets, scaling and axis assignment values manually. It also offers the possibility to measure the work offset automatically.

For detailed information about work offsets, refer to chapter '14 Review' in this manual. The display differs according to the type of machine (milling or turning).

8.1.1 Milling display



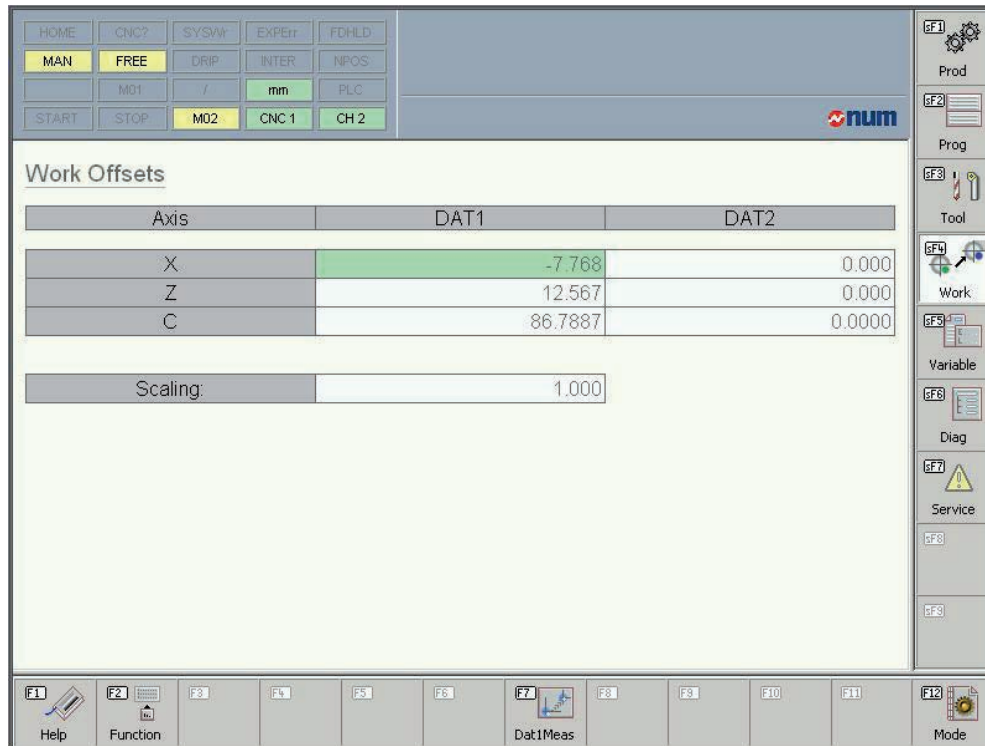
Possible actions

Press  to access DAT1 measuring
Press  to set the DAT3 for Y and Z
Press  to set the DAT3 for X and Z
Press  to set the DAT3 for X and Y
Press  to access all existing functions. Select the function with  or  and 



The DAT3 setting functions    are only displayed if the CNC is in the reset state.

8.1.2 Turning display



Possible actions

Press **F7** to access DAT1 measuring

Press **F2** to access all existing functions. Select the function with **↑** or **↓** and **Enter↵**

8.2 Entering Offset Values

Select the appropriate field	 ← or →; ↑ or ↓
Open the field	 Enter↵
Enter the value and validate	 Enter↵

8.3 Entering Scaling Factor

Select the scaling factor field	 Tab
Open the field	 Enter↵
Enter the value and validate	 Enter↵



The value range is 0.001 to 9.999. Example: a value of 0.5 will produce a part which is half the size originally programmed.

8.4 Entering Axes Assignment

Select the scaling factor field	 Tab
Select the axes assignment field	 ↓
Open the field	 Enter↵
Enter the axes assignments and validate	 Enter↵



For further information about Axes Assignment, refer to chapter '15 CNC Operation' of this manual.

8.5 Setting the Work Offset (DAT1) by Teach-in for Milling



The axes must have been homed and the CNC must be in manual mode before this function can be accessed.

Select DAT1 measuring

F7

HOME	CNC?	SYS	EXPER	FDHLD
MAN	FREE	DRIP	INTER	NPOS
	M01	/	mm	PLC
START	STOP	M02	CNC 1	CH 1

Pos.	OP	DAT1	DAT2
X	-45.000	45.000	0.000
Y	23.000	-23.000	0.000
Z	32.000	-32.000	0.000
U	-6.500	1.500	5.000
C	218.0000	142.0000	0.0000

DAT1 Measuring

DAT1 Axis

X

5.000

20.000

0

F1 Help

F2 Function

F3 OM/OP

F4 DAT1Axis

F5 Valid

F6 Dir

F7 Dir

F8

F9

F10

F11 Back

F12 Mode

GF1 Prod

GF2 Prog

GF3 Tool

GF4 Work

GF5 Variable

GF6 Diag

GF7 Service

GF8

GF9

Possible actions

Press F7 to select negative direction for teach-in
Press F6 to select positive direction for teach-in
Press F5 to validate the values
Press F4 to select the axis
Press F3 to toggle the axes position display between OP and OM
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter↵

Set the focus on the DAT1 Axis field	 Tab
Select the desired axis	 ↑ or ↓
Or	 F4 and ↑ or ↓ and Enter↵
Set the focus on the dimension input fields	 Tab
Open the field	 Enter↵
Enter the value and validate	 Enter↵
Select positive or negative move direction	 F6 or F7
Move the axis to position the tool head in contact with the setting tool	
Validate the setting	 F5
The measured value is now displayed in the DAT1 column in the left hand window	
To exit DAT1 measuring mode	 F11



Workpiece and setting piece dimensions are stored and can be used again for a further measurement.

8.6 **Setting the Work Offset (DAT1) by Teach-in for Turning**



The axes must have been homed and the CNC must be in manual mode before this function can be accessed.

Select DAT1 measuring

F7

HOME CNC? SYSM EXPERT FDHLD

MAN FREE DRIP INTER NPOS

MO1 / mm PLC

START STOP M02 CNC 1 CH 2

num

	Pos. OP	DAT1	DAT2
X	-4.444	2.222	0.000
Z	-0.222	0.222	0.000
C	358.7438	0.0222	1.2340

DAT1 Measuring

DAT1 Axis

X

5.000

24.600

F1 Help

F2 Function

F3 OM/OP

F4 DAT1Axis

F5 Valid

F6 Dir

F7 Dir

F8

F9

F10

F11 Back

F12 Mode

F1 Prod

F2 Prog

F3 Tool

F4 Work

F5 Variable

F6 Diag

F7 Service

F8

F9

Possible actions

Press F5 to validate the values
Press F4 to select the axis
Press F3 to toggle the axes position display between OP and OM
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter↵

Set the focus on the DAT1 Axis field	 Tab
Select the desired axis	 ↑ or ↓
Or	 F4 and ↑ or ↓ and Enter↵
Set the focus on the dimension input fields	 Tab
Open the field	 Enter↵
Enter the value and validate	 Enter↵
Move the axis to position the tool head in contact with the setting tool	
Validate the setting	 F5
The measured value is now displayed in the DAT1 column in the left hand window.	
To exit DAT1 measuring mode	 F11



Workpiece and setting piece dimensions are stored and can be used again for a further measurement.

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

Page deliberately empty.

Summary

9	Context "Variables"	111
9.1	Access to the Context "Variables"	111
9.1.1	Variables context window	112
9.2	Entering variables and comments	113
9.3	CNC variables and comments	113
9.3.1	Display of Symbolic variables	113
9.4	PLC variables and comments	114
9.5	Modification of variables values	115
9.6	Creating a list of consecutive variables	116

Page deliberately empty.

9 Context "Variables"

9.1 Access to the Context "Variables"

To access the context "Variables"	  + 
To close pop-up menus and windows	 

The Variable context is intended as a control and debugging tool.
It is used to display, save and load variable lists described in the debug files *.xdv.

The variables that may be displayed are:

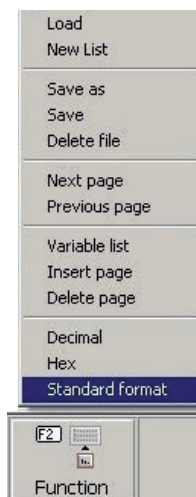
- External parameters E
- L variables
- Symbolic CNC program variables
- PLC variables: according to the PLC project.

In PLC programming, the whole set of variable data types according to IEC 61131-3 is supported.

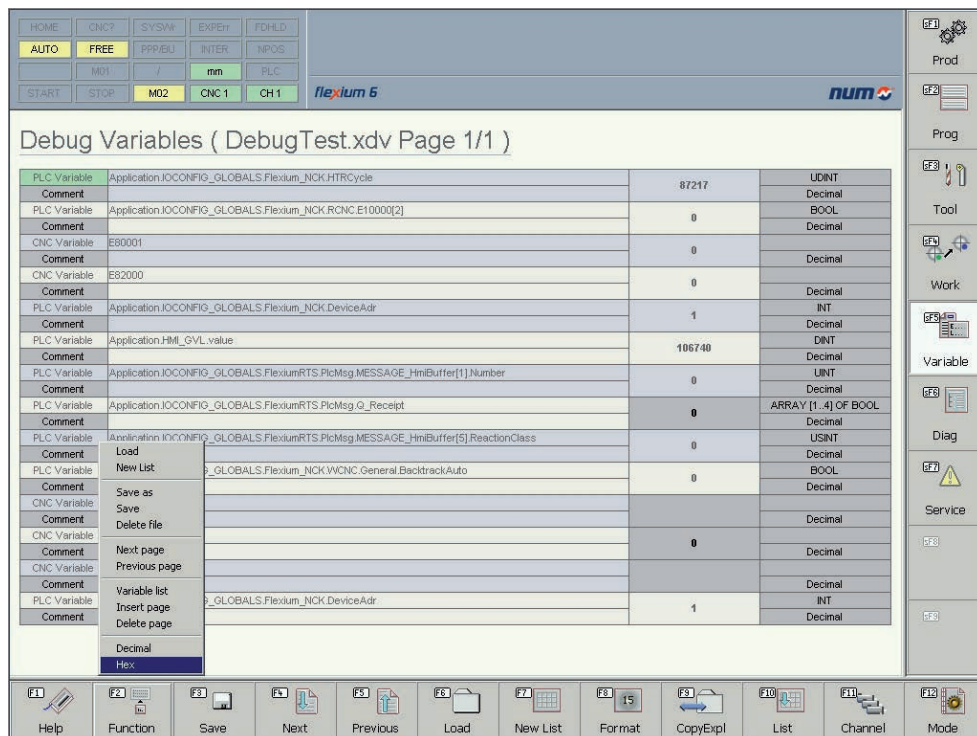
Their formats are:

BOOL, BYTE, WORD, DWORD, LWORD, SINT, USINT, INT, UINT, DINT, UDINT, LINT, ULINT, REAL, LREAL, STRING, WSTRING, TIME, TIME_OF_DAY, TOD, DT, DATE_AND_TIME, DATE.

To set the standard display format for non numeric variables:



9.1.1 Variables context window



Possible actions

Press **[F10]** to create a list of consecutive variables

Press **[F9]** to access CopyExplorer

For the description of CopyExplorer, refer to chapter '12 CopyExplorer' in this manual

Press **[F8]** to select the type of display (hexadecimal and decimal)

To choose a format : **[↑]** or **[↓]** and **[Enter ↵]**



L variables can only be displayed in decimal mode.

Press **[F7]** to create a new list and erase the contents of the present list on the screen

Press **[F6]** to load an existing list from PC disk

Press **[F5]** to display the previous page of the list

Press **[F4]** to display the next page of the list

Press **[F3]** to save the actual list

Press **[F2]** to access all existing functions. Select the function with **[↑]** or **[↓]** and **[Enter ↵]**

9.2 Entering variables and comments

If needed, the variable type has to be modified first:

Select the field CNC or PLC variable	← or →; ↑ or ↓
Open the field	Enter ↵
Select CNC, PLC or CNC symbolic variable	↑ or ↓
Validate the variable type	Enter ↵

9.3 CNC variables and comments

Select the appropriate field	← or →; ↑ or ↓
Open the field	Enter ↵
Enter the variable name and validate. The comment field may be left blank.	Enter ↵

9.3.1 Display of Symbolic variables

Symbolic CNC variables can be handled like any other CNC variables. However, the display of their values is slightly different.

- Symbolic variables are selected, but not yet declared.
The message "Invalid variable" is displayed:

CNC Symbolic	TEST	Invalid variable	LREAL
Comment			Decimal
CNC Symbolic	TAB(2,2)	Invalid variable	LREAL
Comment			Decimal
CNC Symbolic	TAB(2,3)	Invalid variable	LREAL
Comment			Decimal

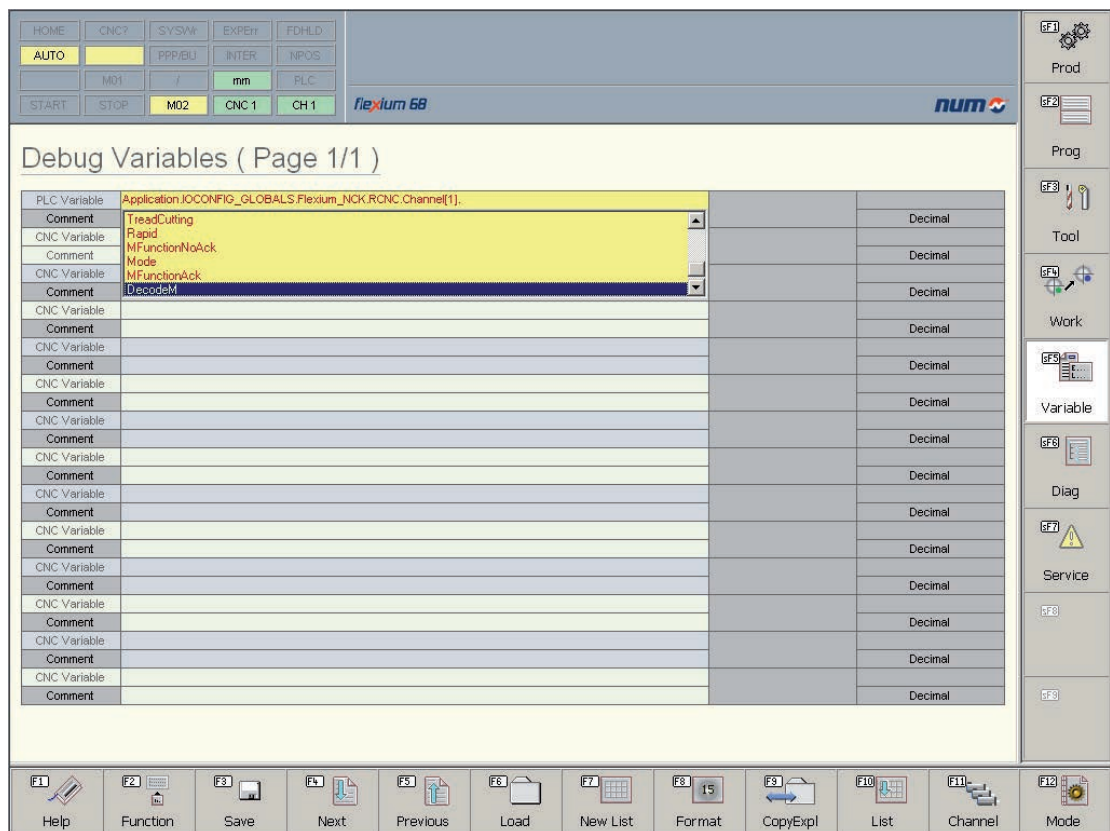
- The part program is started and the variables are declared.

```
%7.1()()
VAR
[TAB(5,5)]
[TEST]=500
ENDV
[TAB(2,3)]=12
M00
```

- The actual values of the selected variables are displayed as long as the program is running and until the occurrence of M02 and Reset:

CNC Symbolic	TEST	500	LREAL
Comment			Decimal
CNC Symbolic	TAB(2,2)	0	LREAL
Comment			Decimal
CNC Symbolic	TAB(2,3)	12	LREAL
Comment			Decimal

9.4 PLC variables and comments



The variable context provides an automatic help for the selection of PLC variables.
The variable name is the same as the one used in the PLC application project.

As an example, if you wish to select the variable for the decoded M function M40:
Application.IOCONFIG_GLOBALS.Flexium_NCK.RCNC.Channel[1].DecodeM.M40

Select the appropriate field	← or →; ↑ or ↓
Open the field	Enter↵
The first part of the possible variable name appears in a drop-down list	
Select the first part of the name	↑ or ↓
Validate	Enter↵
Enter a full stop (period)	
The next part of the possible variable name appears in a drop-down list	
Continue as described above until the full variable name has been entered	Enter↵

9.5 Modification of variables values

The values of variables can be modified. This is only possible if the box "Write variables" in the Context "Service" has been checked. and if writing is enabled by the PLC.

Only variables having a white or light grey value field can be modified.

Select the field "Value"	 ← or →; ↑ or ↓
Open the field	 Enter ↵
Enter the new value and validate	 Enter ↵

1

2

3

4

5

6

7

8

9

10

11

12

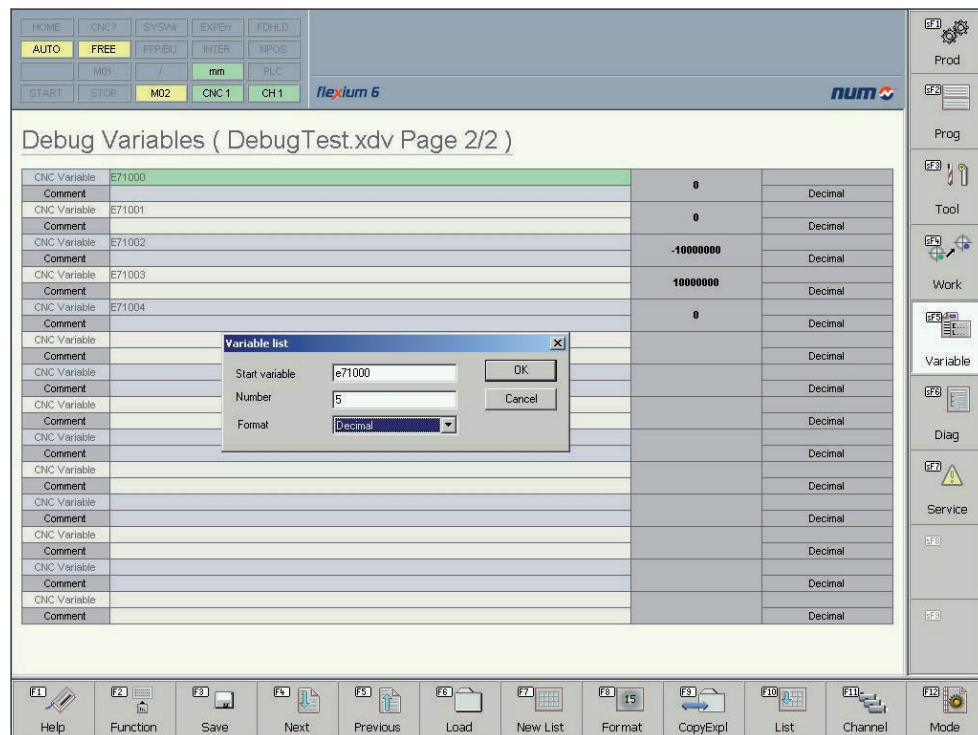
13

14

15

9.6 Creating a list of consecutive variables

This function provides an easy way to enter a list of consecutive variables.
This function is only available for CNC variables.



Select the desired start field in the column "Variable"	or ; or
Open the entry window	F10
Select the appropriate field	Tab
Enter the start variable name and the number of variables	
Select the display format	or
Validate	Enter

Page deliberately empty.

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

Page deliberately empty.

Summary

10	Context "Diagnostic"	121
10.1	Access to the Context "Diagnostic"	121
10.1.1	Diagnostic context window	121
10.2	Extended Help	122
10.2.1	Example of a help page	123
10.3	Alarm History Page.....	124
10.3.1	History Page and Filters	125
10.4	Exception Page.....	127
10.5	Test Point Monitor	128
10.6	Safety module diagnostic.....	129
10.7	CNC and DISC NT Configuration	130
10.7.1	System Functions Page	131
10.7.2	Device List	132

Page deliberately empty.

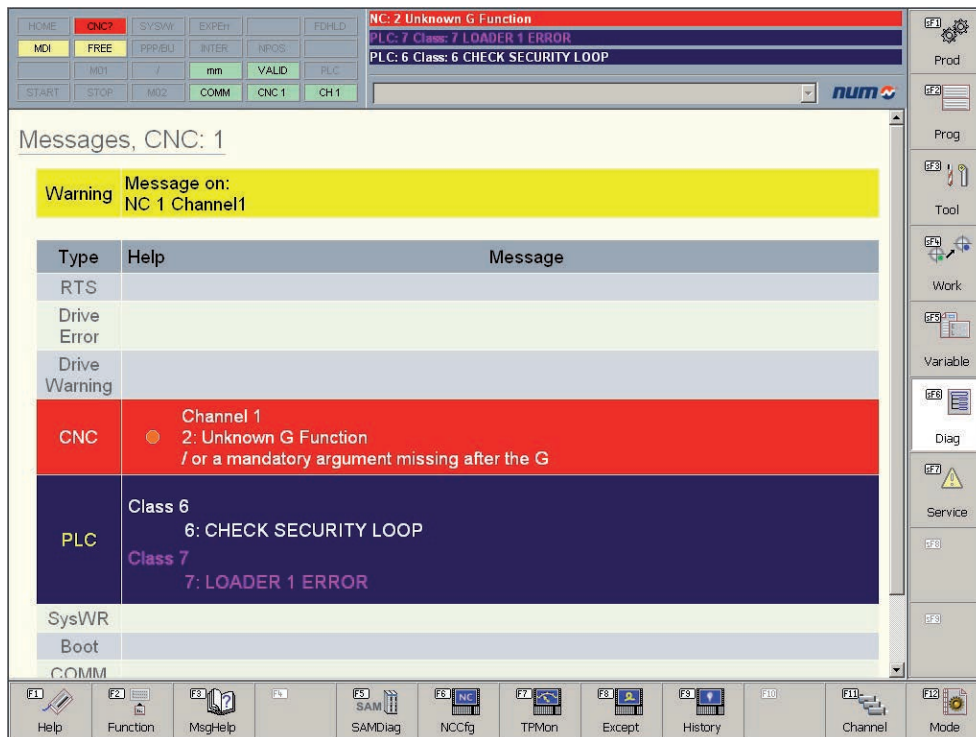
10 Context "Diagnostic"

10.1 Access to the Context "Diagnostic"

To access the context "Diagnostic"	 Shift + F6
To close pop-up menus and windows	 Esc

The initial page shows all actual messages which could have been generated by the RTS (Flexium Run Time System), Drives, CNC, PLC, as well as system warnings, CNC boot messages, communication errors, HMI and User messages.

10.1.1 Diagnostic context window



Possible actions

Press F9 to access the alarm history page
Press F8 to access the exception records
Press F7 to access the test point monitor page
Press F6 to display the CNC and DISCNT configuration data
Press F5 to access the SAM diagnostic page
Press F3 to open the extended help page
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter ↵

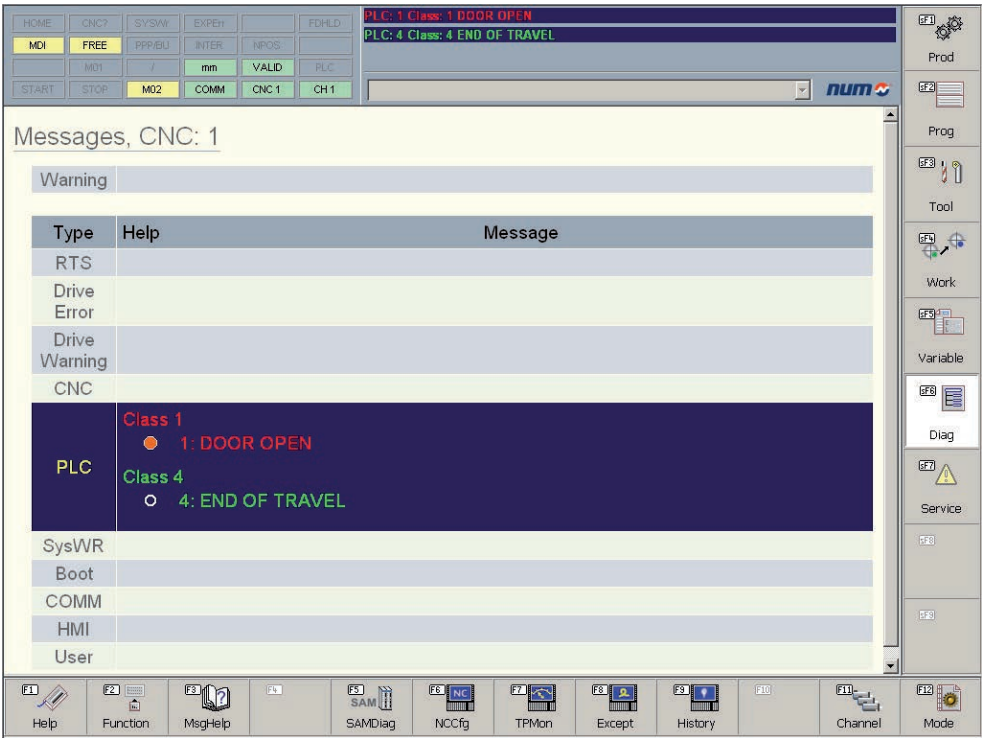
10.2 Extended Help

The HMI offers the possibility to display an extended help for every message. This help can be made up of text or pictures.



The extended help for CNC messages is already integrated. However, the help files provided can be adapted and extended.

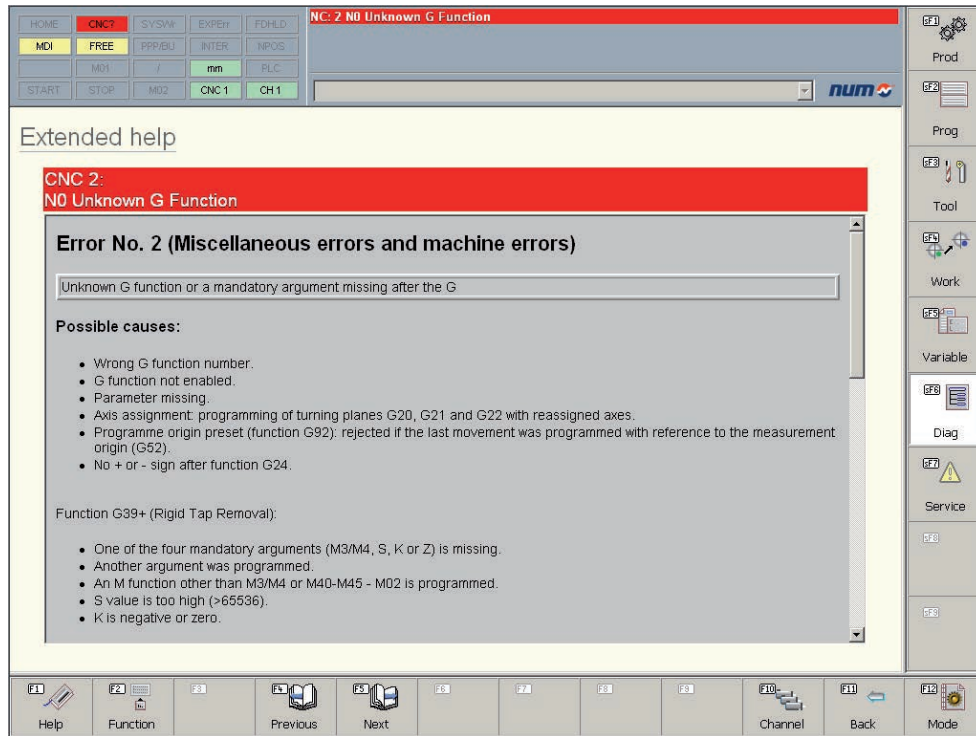
Once a message has been configured with an extended help, it will be displayed with a radio button and shown as follows:



Possible actions

To select a message	 ↑ or ↓
To open the extended help page	 F3

10.2.1 Example of a help page



Possible actions

Press **F5** to display the help page of the next message

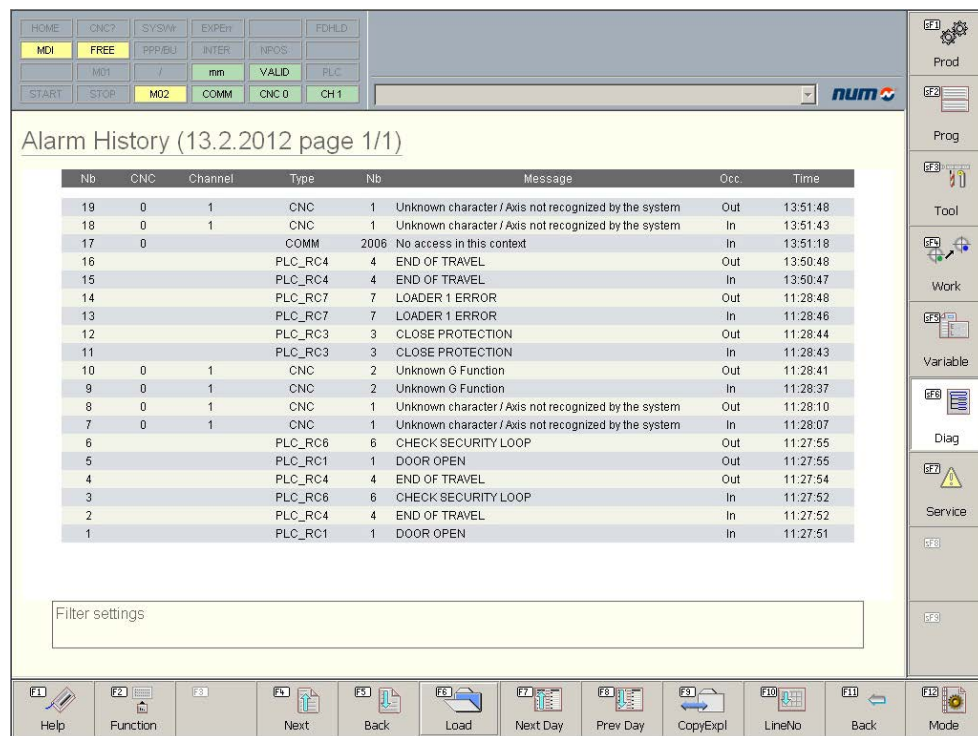
Press **F4** to display the help page of the previous message

10.3 Alarm History Page

The alarm history page contains all alarms and messages that have been recorded on a particular day.

Depending on the settings made in the “Service” context, this feature is used to trace the system state during a period of up to 365 days.

Select alarm history
 F9



Alarm History (13.2.2012 page 1/1)

Nb	CNC	Channel	Type	Nb	Message	Occ.	Time
19	0	1	CNC	1	Unknown character / Axis not recognized by the system	Out	13:51:48
18	0	1	CNC	1	Unknown character / Axis not recognized by the system	In	13:51:43
17	0		COMM	2006	No access in this context	In	13:51:18
16			PLC_RC4	4	END OF TRAVEL	Out	13:50:48
15			PLC_RC4	4	END OF TRAVEL	In	13:50:47
14			PLC_RC7	7	LOADER 1 ERROR	Out	11:28:48
13			PLC_RC7	7	LOADER 1 ERROR	In	11:28:46
12			PLC_RC3	3	CLOSE PROTECTION	Out	11:28:44
11			PLC_RC3	3	CLOSE PROTECTION	In	11:28:43
10	0	1	CNC	2	Unknown G Function	Out	11:28:41
9	0	1	CNC	2	Unknown G Function	In	11:28:37
8	0	1	CNC	1	Unknown character / Axis not recognized by the system	Out	11:28:10
7	0	1	CNC	1	Unknown character / Axis not recognized by the system	In	11:28:07
6			PLC_RC6	6	CHECK SECURITY LOOP	Out	11:27:55
5			PLC_RC1	1	DOOR OPEN	Out	11:27:55
4			PLC_RC4	4	END OF TRAVEL	Out	11:27:54
3			PLC_RC6	6	CHECK SECURITY LOOP	In	11:27:52
2			PLC_RC4	4	END OF TRAVEL	In	11:27:52
1			PLC_RC1	1	DOOR OPEN	In	11:27:51

Filter settings

Possible actions

Press F10 to display messages starting at a specific line number
Press F9 to access CopyExplorer. For the description of CopyExplorer, refer to chapter '12 CopyExplorer' in this manual
Press F8 to display the history file of the previous day
Press F7 to display the history file of the next day
Press F6 to load a history file
Press F5 to display the previous page
Press F4 to display the next page
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter

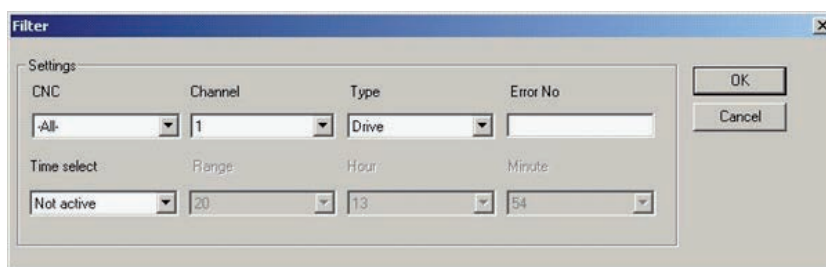
10.3.1 History Page and Filters

From the Flexium HMI 3.4.0.0 on, it is possible to activate several filters to make the selection of message types easier.

When opening a history file, no filter is activated	
To enable filtering, open the function list	 F2
In the list, select "Filter"	



This opens the filter selection window:



You can now select the various filters and their combination	
To select a setting	 Tab
To open a combination box	 Enter
To select an item in an open combination box	 ↑ or ↓ and Enter

HOME OK? STOP OPEN F2HLD

MC1 FREE F2HLD ENTER MPOS

START STOP M02 COMM CNC 0 CH 1

num

Selected entries 13.2.2012 (1/1)

Nb	CNC	Channel	Type	Nb	Message	Occ	Time
19	0	1	CNC	1	Unknown character / Axis not recognized by the system	Out	13:51:48
18	0	1	CNC	1	Unknown character / Axis not recognized by the system	In	13:51:43
17	0		COMM	2006	No access in this context	In	13:51:18
16			PLC_RC4	4	END OF TRAVEL	Out	13:50:48
15			PLC_RC4	4	END OF TRAVEL	In	13:50:47
14			PLC_RC7	7	LOADER 1 ERROR	Out	11:28:48
13			PLC_RC7	7	LOADER 1 ERROR	In	11:28:46
12			PLC_RC3	3	CLOSE PROTECTION	Out	11:28:44
11			PLC_RC3	3	CLOSE PROTECTION	In	11:28:43
10	0	1	CNC	2	Unknown G Function	Out	11:28:41
9	0	1	CNC	2	Unknown G Function	In	11:28:37
8	0	1	CNC	1	Unknown character / Axis not recognized by the system	Out	11:28:10
7	0	1	CNC	1	Unknown character / Axis not recognized by the system	In	11:28:07
6			PLC_RC6	6	CHECK SECURITY LOOP	Out	11:27:55
5			PLC_RC1	1	DOOR OPEN	Out	11:27:55
4			PLC_RC4	4	END OF TRAVEL	Out	11:27:54
3			PLC_RC6	6	CHECK SECURITY LOOP	In	11:27:52
2			PLC_RC4	4	END OF TRAVEL	In	11:27:52
1			PLC_RC1	1	DOOR OPEN	In	11:27:51

Filter settings: CNC (-All-) Channel (-All-) Type (-All-) Error No ()
Time select (Not active) Time (13:56)

Help Function Next Back Load Next Day Prev Day CopyExpl LineNo Back Mode

Prod Prog Tool Work Variable Diag Service



Be aware that a multiple selection of filters can lock out messages which are present, but do not match with the chosen selection. To reload the full history list and cancel the filter selections, select "Load without filter" in the function list.

10.4 Exception Page

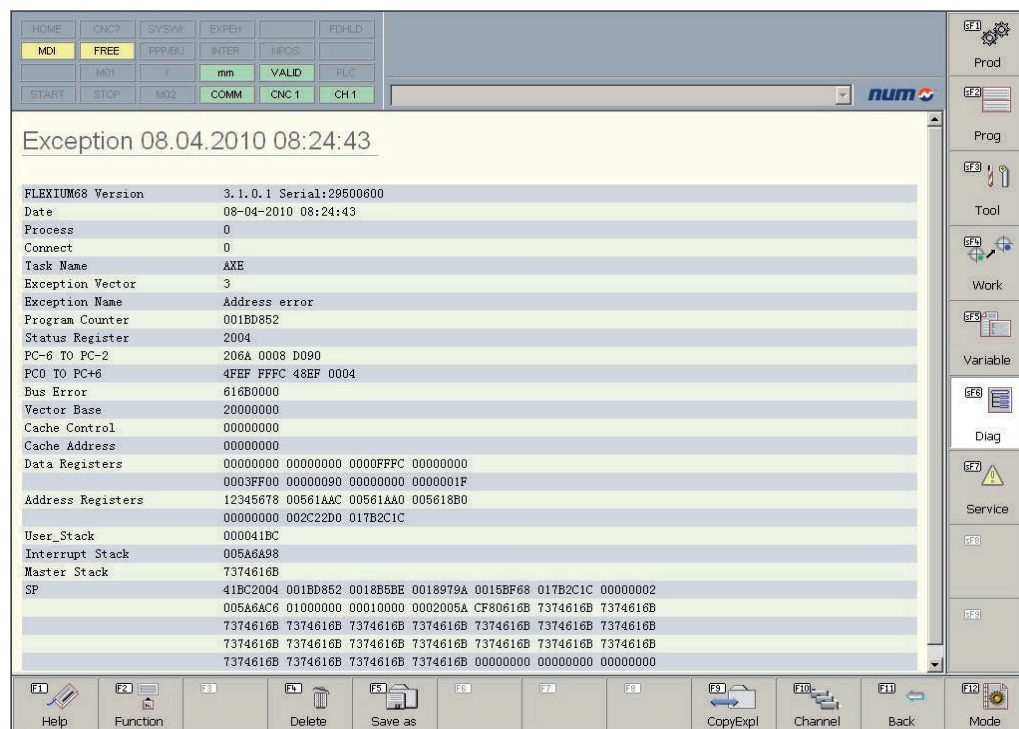
An exception file contains the data of a CNC system error caused by hardware or software failure or by a wrong CNC configuration. This data can only be interpreted by NUM.

An exception error is signalled by the EXPERR field in the status window. This field will stay activated until the exception data has been loaded into the exception page.

If an exception error occurs, please save it on PC disk for further analysis.

Select the exception page	F8
---------------------------	----

After selecting this page, the last exception that occurred is loaded and displayed.



Possible actions

Press F9 to access CopyExplorer	
For the description of CopyExplorer, refer to chapter '12 CopyExplorer' in this manual. Press F5 to save the exception file	
Press F4 to delete an exception file	
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter	
To scroll up or down within a file, use ↑, ↓ or PgUp, PgDn	
To exit the exception page	F11

10.5 Test Point Monitor

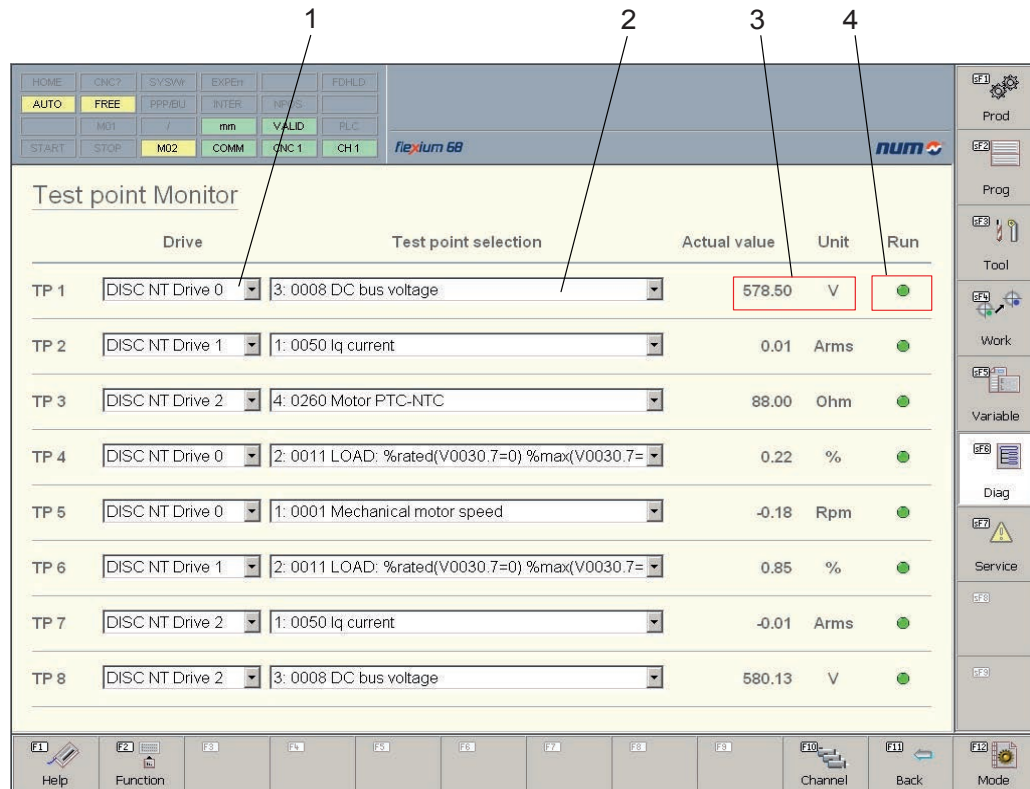
More than 100 measuring points are available on every DISC NT drive.

Four of these measuring points on each drive can be assigned to test points that may be displayed in the HMI.

For further information on measuring point assignment, refer to the Flexium Commissioning Manual (Reference M00010).

The Test Point Monitor page displays up to eight of these test points.

They can be selected among any four test points of any drive.



- 1 Drive selection
- 2 Test point selection
- 3 Actual value and unit of the test point
- 4 Operating state of the drive
Green = Run, gray = Halt.

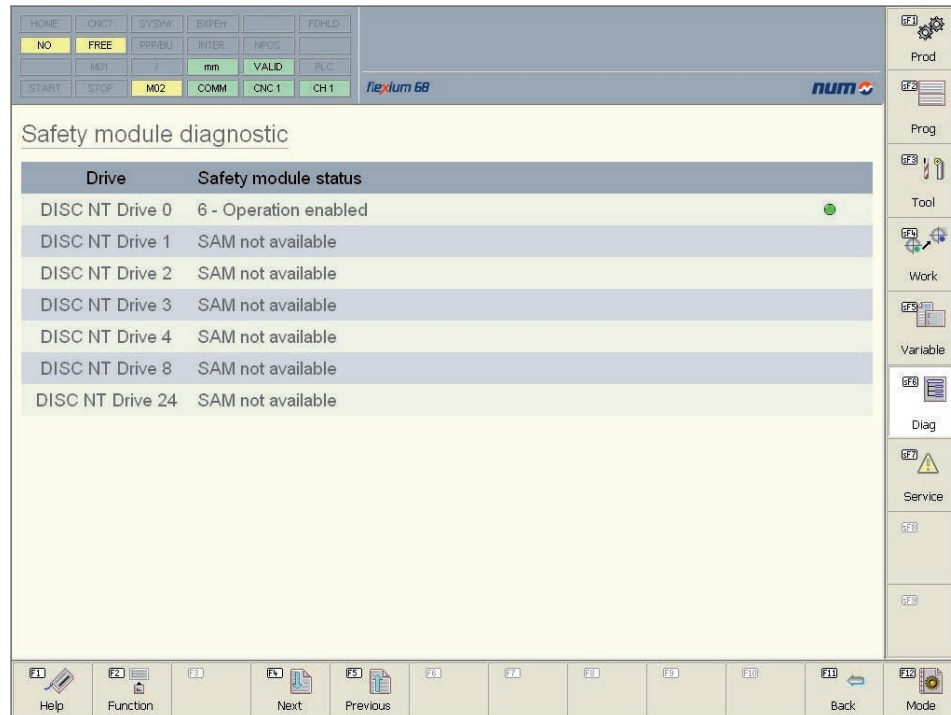
Navigation and selection

Select a field	Tab
Open a list or an entry field	Enter
Select an entry in an open list	↑ or ↓
Validate a selection or an entry	Enter

10.6 Safety module diagnostic

The SAM-Num safety module can be used to integrate safety functions for personal protection against dangerous movements into the MDLU3 drive positioning controllers.

The states of the SAM modules are displayed as decimal codes and plain text descriptions. In addition, a color LED indicator facilitates the identification of the various states.



State		
1	"Start"	LED gray
2	"Not ready to switch on"	LED gray
3	"Switch on disabled"	LED orange
4	"Ready to switch on"	LED orange
6	"Operation enabled"	LED green
7	"Quick stop"	LED red
8	"Fault reaction active"	LED red
9	"Fault"	LED red flashing
x	"Undefined state"	LED gray

10.7 CNC and DISC NT Configuration

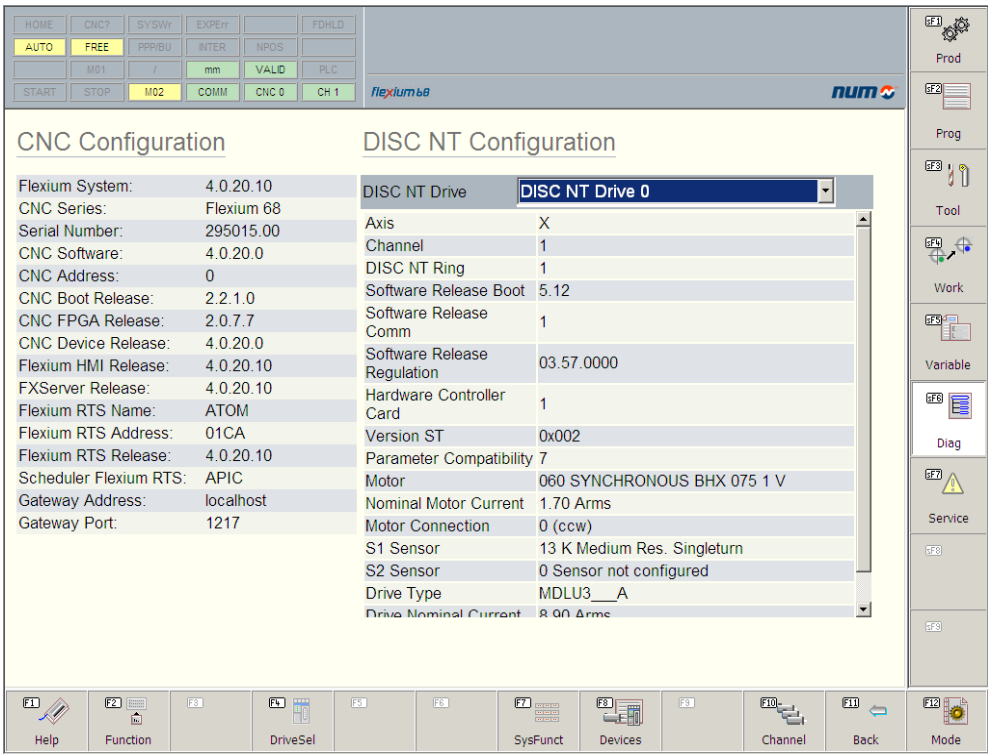
This page shows information about the CNC, software versions and DISC NT configuration.



The right hand part of the page contains information only if DISC NT drives are used in your system.

Select the configuration page

 F6

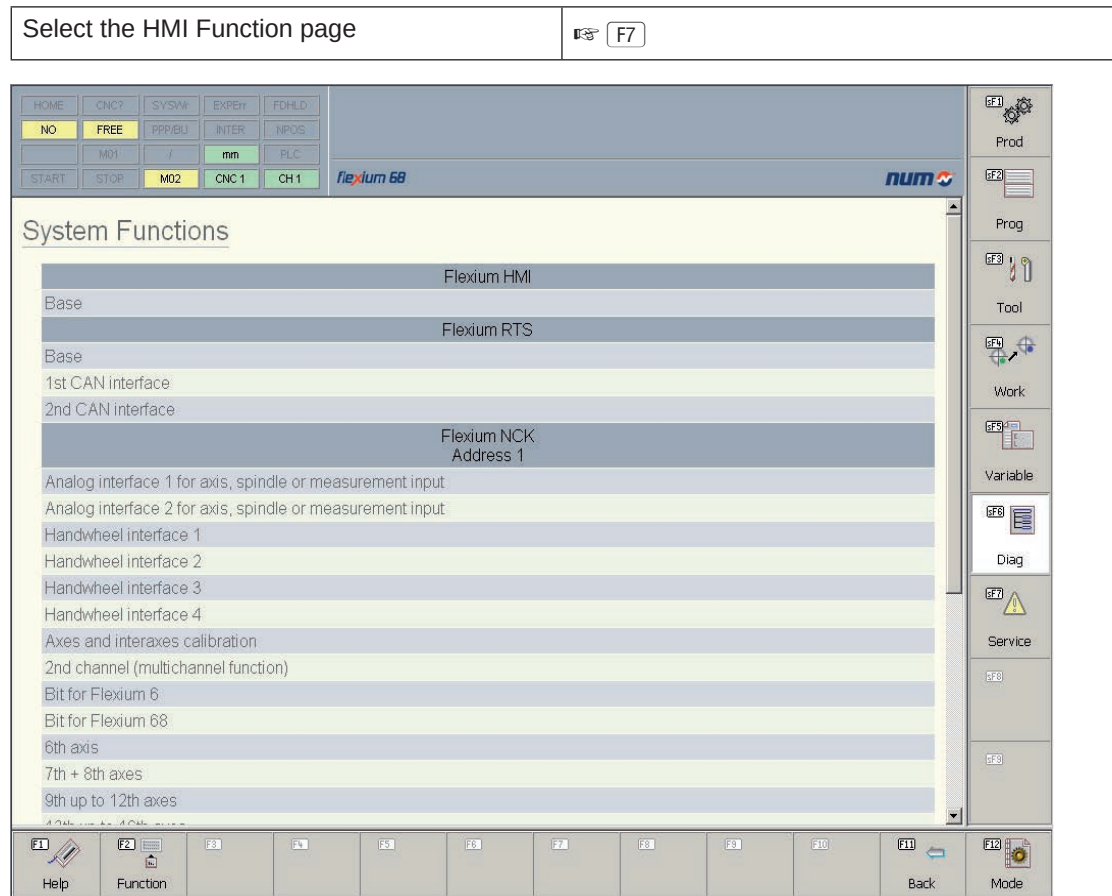


Possible actions

Press  to access the Device List page	
Press  to open the HMI options page	
Press  to select the DISC NT drive information	
Press  to access all existing functions. Select the function with  or  and 	
To open the drive select box	 
To select the drive	  or  and 
To exit the configuration page	 

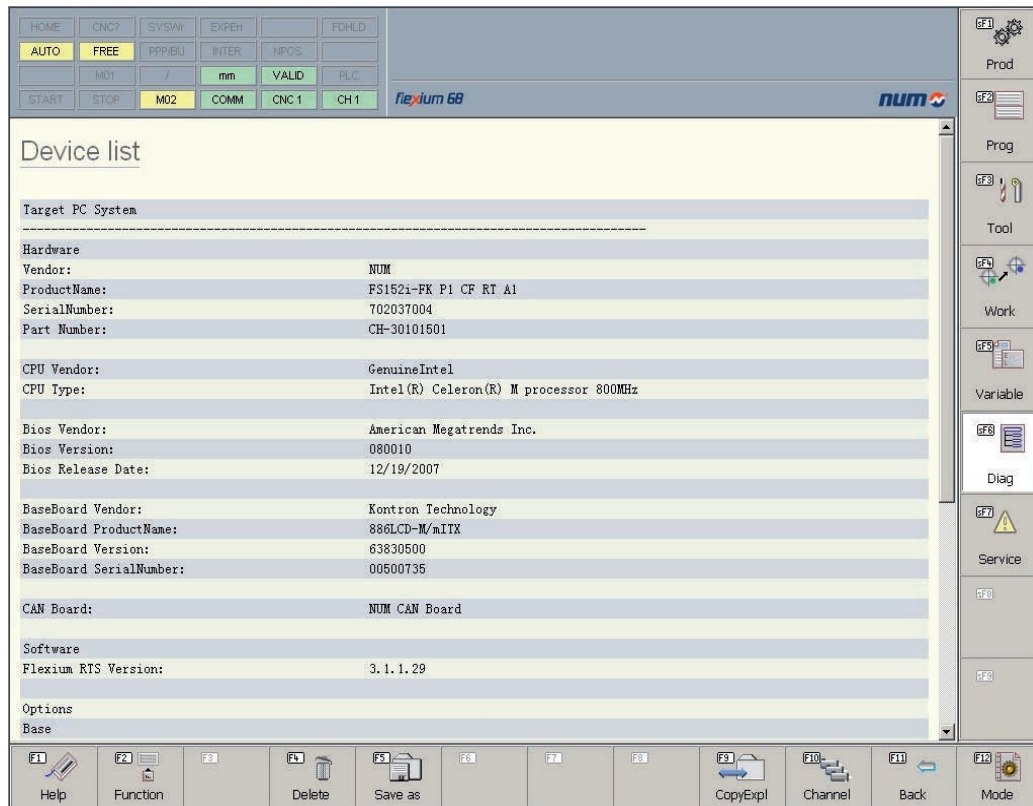
10.7.1 System Functions Page

This page contains a list of all available functions enabled in the Flexium system.



10.7.2 Device List

The Device List page shows all available information about the Flexium and PC system. This information can be gathered in a file which can be used for archive purposes or as basis for ordering spare parts etc.



Possible actions

Press **[F9]** to access CopyExplorer

Press **[F5]** to generate and save a device list file

Press **[F4]** to delete a device list file



Files generated using the function **[F5]** will be saved in
C:\Program Files\NUM\Flexium HMI\FXBrowser\Data\
with the extension .xsk

If you wish to have a more flexible access to these files, it is recommended
to use the functions of CopyExplorer.

Page deliberately empty.

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

Page deliberately empty.

Summary

11	Context "Service"	137
11.1	Access to the Context "Service"	137
11.1.1	Start window of the context "Service"	137
11.2	General Settings Page.....	138
11.2.1	Setting selection	138
11.2.2	Explanation of the setting options.....	139
11.2.3	Box PC Access	141
11.2.4	Password Settings	142
11.2.5	Custom tools	143
11.3	System Time and Date Settings	144
11.3.1	Setting the system time and date	144
11.4	Location Settings	145
11.4.1	Keyboard functions	145
11.5	Password Settings	146
11.6	Diagnostic Settings.....	146
11.6.1	System messages.....	147
11.6.2	PLC messages.....	147
11.7	Feed Rate Settings	148
11.7.1	Explanation of the setting options.....	149
11.8	Spindle Settings.....	150
11.8.1	Explanation of the setting options.....	150
11.8.2	History	151
11.9	Management Settings.....	152
11.9.1	Explanation of the setting options.....	153
11.9.2	Password management	154
11.10	Machine Data.....	155
11.10.1	NCK parameters	156
11.10.2	Drive parameters	158
11.10.3	Restart	161
11.10.4	Change Password.....	161

Page deliberately empty.

11 Context "Service"

11.1 Access to the Context "Service"

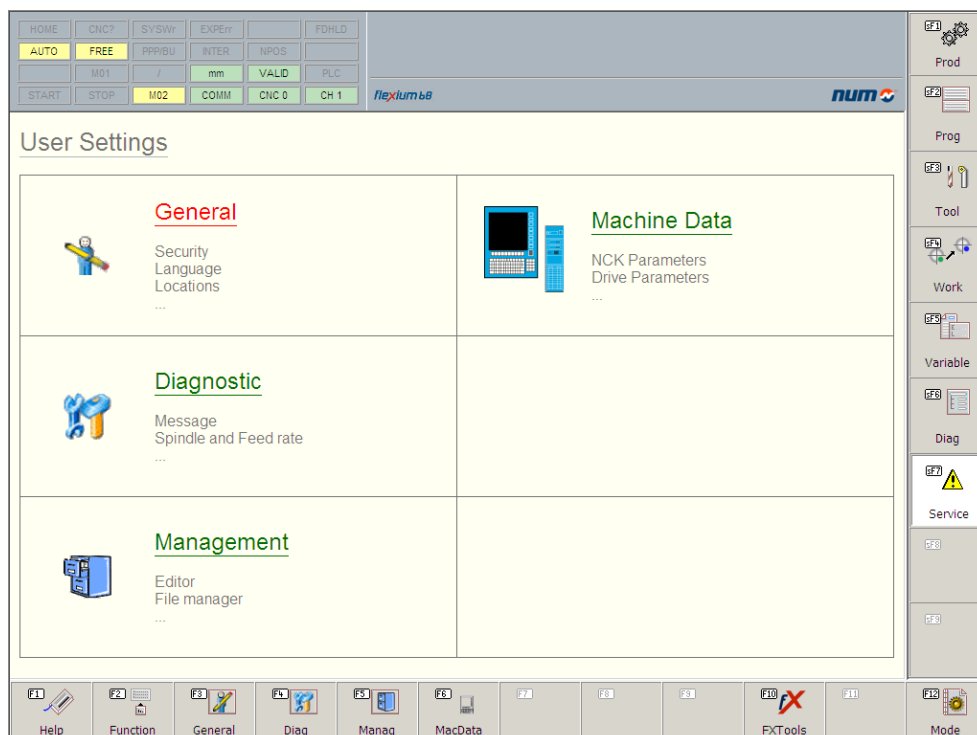


For authorised personnel only !

This context gives access to all settings of the HMI and to the CNC system time and date management.

This context is protected. It opens only after you have entered the correct password. The standard password at delivery is "OEM"
For security reasons, please change this password as soon as you have configured the HMI.

11.1.1 Start window of the context "Service"



Possible actions

Press **[F10]** to access Flexium Tools.

This will launch Flexium Tools. This software is used to configure and program the whole machine. Access to this tool may require a password initially communicated to the machine builder.



The use of Flexium Tools by unauthorized personnel can have very serious consequences for machine operation and safety. Flexium Tools should be used only by specifically trained persons. If started accidentally, close the program immediately with **[Alt] + [F4]**.

Press **[F6]** to access the Machine Data

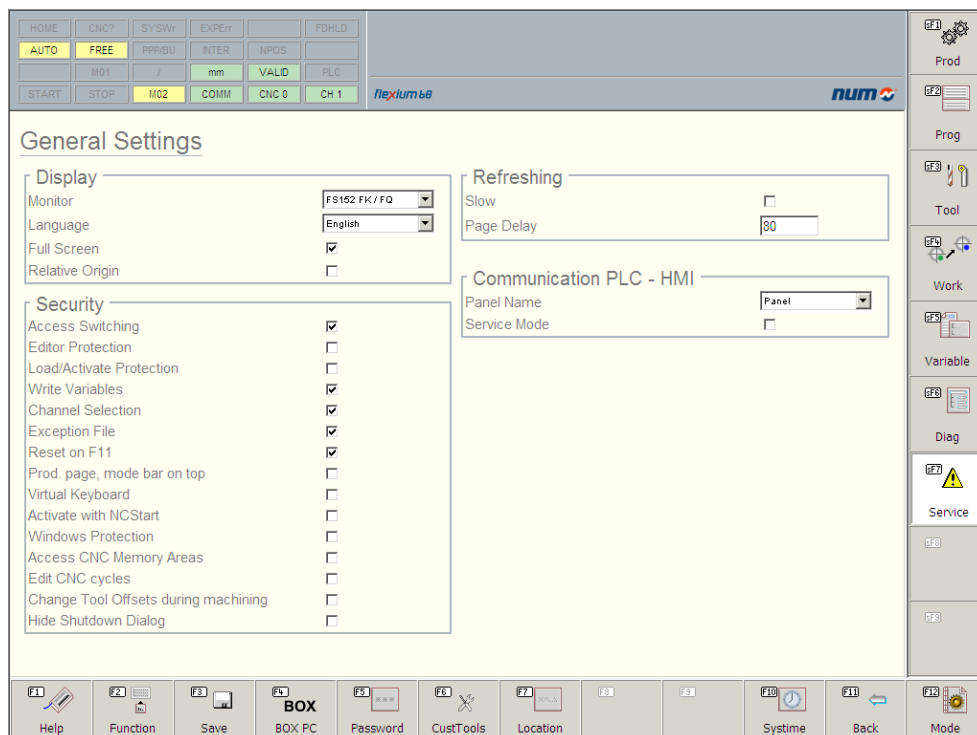
Press **[F5]** to access the management settings page

Press **[F4]** to access the diagnostic settings page

Press **[F3]** to access the general settings page

Press **[F2]** to access all existing functions. Select the function with **[↑]** or **[↓]** and **[Enter]**

11.2 General Settings Page



Possible actions

Press [F10] to access the system time and date settings
Press [F7] to access the data and file location settings
Press [F6] to Customer Tools Access
Press [F5] to access the password settings
Press [F4] to BOX PC Access
Press [F3] to save the settings
Press [F2] to access all existing functions. Select the function with [↑] or [↓] and [Enter↵]

11.2.1 Setting selection

To select a setting	[Tab⇧]
To open a combination box	[Enter↵]
To select an item in an open combination box	[↑] or [↓] and [Enter↵]
To activate or deactivate a setting box	[Space] or [Enter↵]

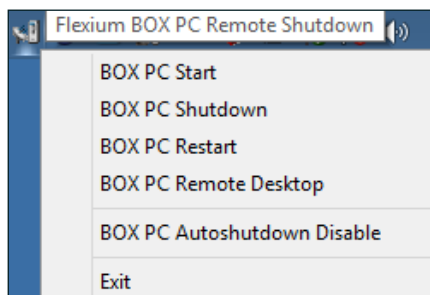
11.2.2 Explanation of the setting options

Display	
Monitor	To select the monitor type used.
Language	To select the HMI language.
Full screen	Activate this box to start the HMI in full-screen mode.
Relative origin	Allows the activation of the relative axes positions display in the context "Production".
Security	
Access switching	This setting is not used in the basic version of the HMI. However, additional functions may modify the aspect of the key symbols sF1 to sF9 as well as the pages they display when selected. The activation of this setting allows switching between standard and modified display. The switching itself is performed in any context by pressing  Shift + .
Editor protection	If this box is activated, there is no possibility to access the editor in the Programming context. No file on PC disk can be modified or deleted by the HMI.
Load/activate protection	Usually, the functions "Save, load and activate" of the editor as well as "Load and activate" when loading a part program into the CNC are enabled. These functions can be disabled by checking the box.
Write variables	If this box is activated, the modification of variables values in the context "Variables" is allowed.
Channel selection	For multi-channel machines only: If this box is activated, the channel data displayed can be selected by the HMI. If this box is not active, channel selection must be done by an external switch via the PLC. Caution: the PLC signal overrides the HMI channel selection function, so be sure they do not interfere.
Exception file	An exception file contains the data of a CNC system error caused by hardware or software failure or by a wrong CNC configuration. This data can only be interpreted by NUM.
Reset on F11	Activate this box to display the "Reset" key in the CNC function bar.
Prod. page, mode bar on top	On entering the production page (sF1), the CNC function bar will be selected by default.
Virtual keyboard	Allows to call up a virtual keyboard with the key F2 "Function" in all pages providing a data input possibility.
Activate with NC start	When working with several programs, it is possible that the HMI gets switched to the background and is no more visible on the screen. Activating this box causes the HMI to be displayed again as soon as the CNC Start button is pressed.
Windows protection	Activating this option prevents a direct access to Windows. Full functionality of the HMI remains available.  Some Windows functions may not be accessed anymore without entering a password.

Access CNC memory areas	Activate this box to allow loading and saving files to protected memory areas or zones. The memory is divided into four areas: - area 0 : free access User area - area 1: Customer area - area 2: OEM area - area 3: NUM area. Areas 1 and 2 are called protected and their access may require entry of a password. For the settings and modification of passwords, refer to the context "Service", Management settings. Programs with the same numbers can exist in several memory areas. In this case, the program in the area with the lowest number is executed (for instance, if subroutine %8567 is called and this subroutine exists in areas 0 and 3, the subroutine in area 0 is executed). Area 3 is reserved for NUM applications and cannot be modified. The size of each protected memory area is specified by machine parameter P95.
Edit CNC cycles	Allow access to NUM general and machining cycles in the context "PROG". These cycles may be modified and adapted to specific customer requirements.
Change Tool Offsets during machining	Change Tool Offsets during Machining. Checking this box allows for changing the tool length values during a stop.
Hide Shutdown dialog	Checking this box hides the shutdown dialog when pressing Alt + F4. The HMI will be closed immediately and the system stays under power.
Refreshing	
Slow	Activating this box slows down the refreshing of the HMI display. This makes more computing time available for other applications.
Page delay	This value determines the delay between the fast activation of the function keys sF1 to sF9 and the set-up of the corresponding page. The default value is 80ms.
Communication PLC-HMI	
Panel name	Selection of the panel name given in the application project to allow PLC control of the HMI functions. (e.g. lock of context or function selection)
Service mode	Activate to allow the connection of another panel or PC without panel name or to override the context or function lock. PLC control of the HMI is not enabled in Service Mode.

11.2.3 Box PC Access

Box PC access	Select to open the Box PC setting box and activate the access keys to the Box PC. Refer to the Commissioning Manual for information on Box PC connections and settings.
Shut down Box PC with Windows	If the HMI PC is shutdown, the Box PC shutdown too.
IP Address	IP Address of the Box PC.
User Name	User name of the Account of the DPLC / Box PC.
Wake Up over LAN Access (additional settings required)	Remote start up of the Box PC via the main PC tray icon (see image below).



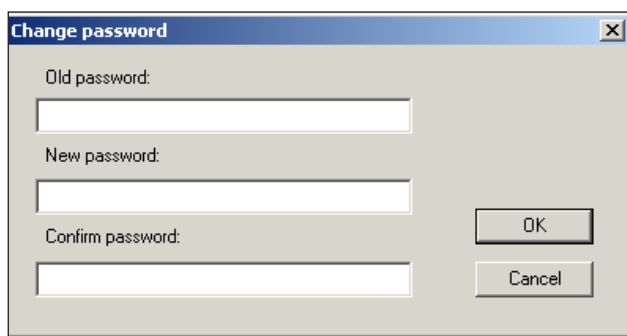
11.2.4 Password Settings

This page permits to change the main password of the 'Service' context.

As said in the chapter 11.1 it is important to change this password after commissioning the machine to prevent unauthorized access.

The prompt to change the password is obtained by pressing **F5** in the *General Settings* page of the 'Service' context.

Open the password entry field	 F5
Enter the old password	
Enter the new password and validate	 Enter 

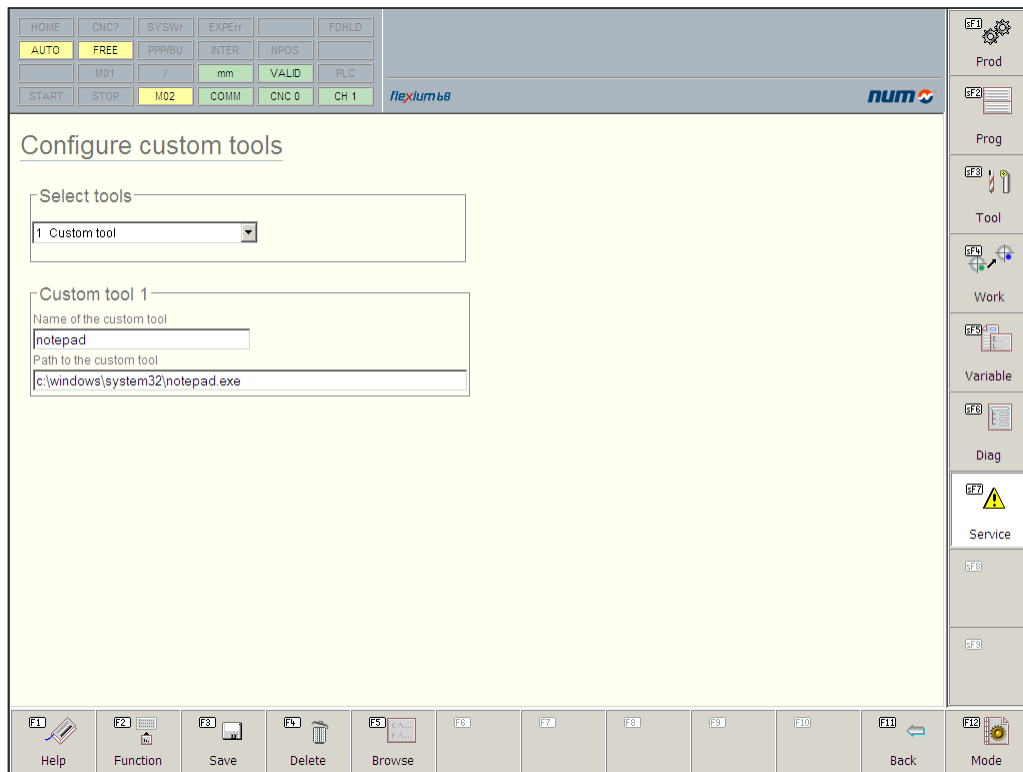


A screenshot of a 'Change password' dialog box. The dialog has a title bar with the text 'Change password' and a close button (X). Inside the dialog, there are three text input fields labeled 'Old password:', 'New password:', and 'Confirm password:'. To the right of the input fields are two buttons: 'OK' and 'Cancel'.

11.2.5 Custom tools

This page permits to define up to 6 different windows programs (e.g. Text editor) that can be called directly within the HMI.

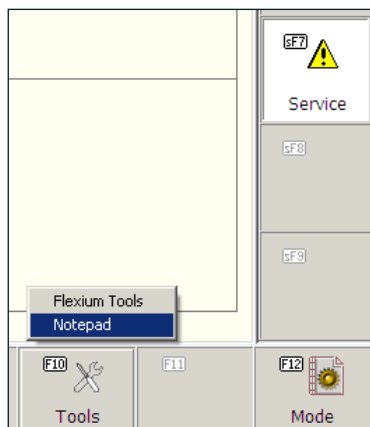
To access this page, press **[F6]** in the *General Settings* page of the 'Service' context



Possible actions

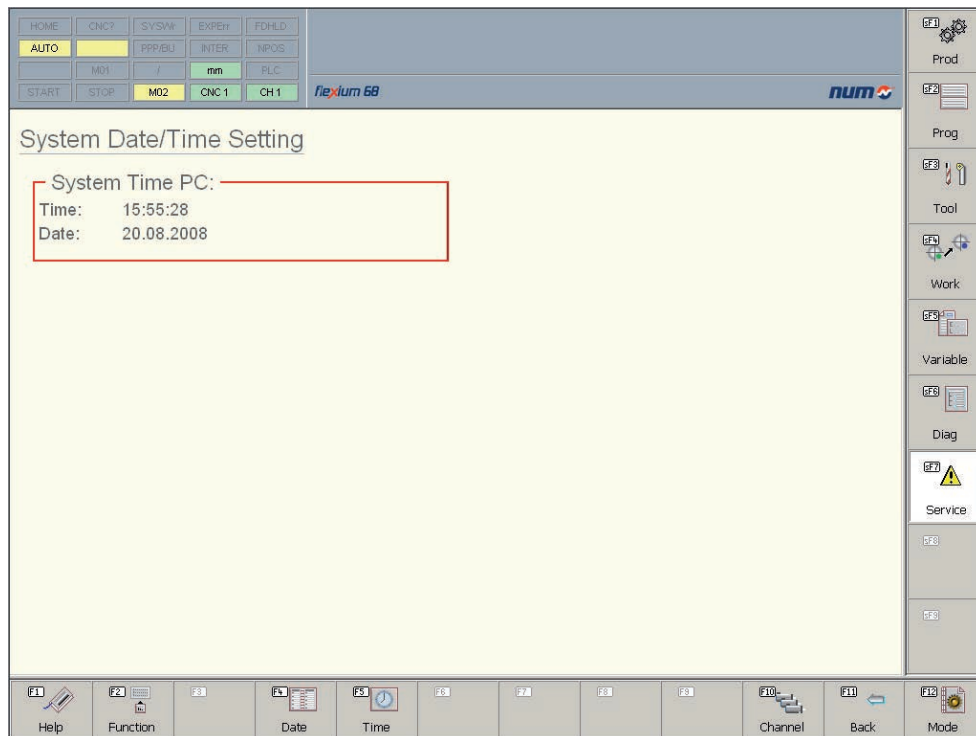
Select the custom tool you want to personalize
Enter a name for this tool (freely selected)
Enter the path or use the Browse ([F5]) button the select the program
Press Save

Access to this program will be found on the main "Service" page.



11.3 System Time and Date Settings

This page allows to set the system date and time.



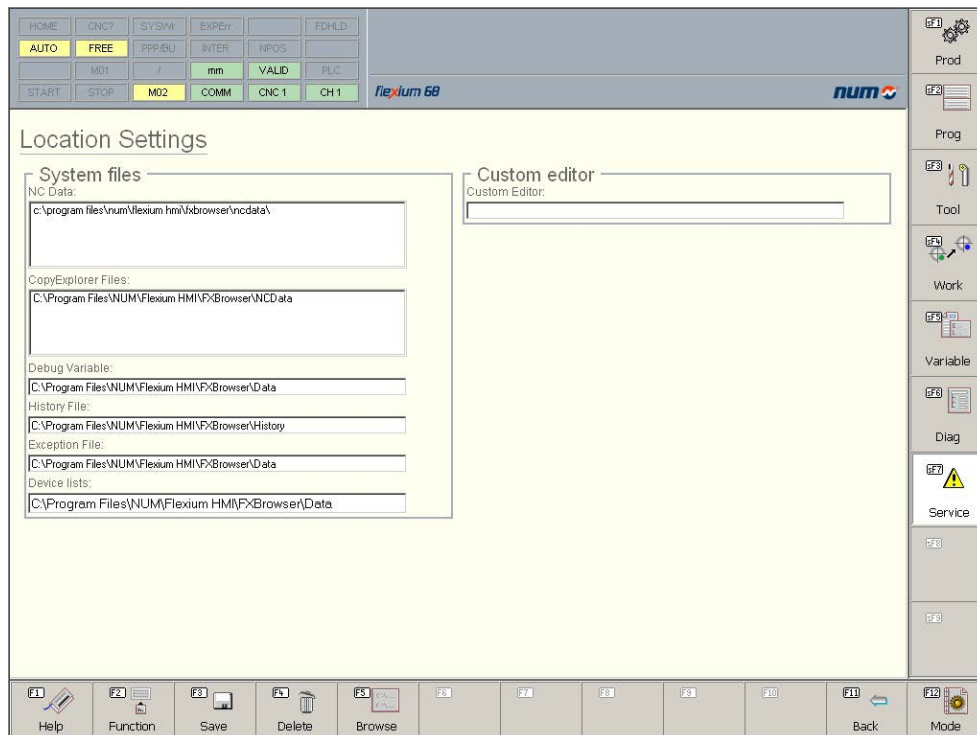
Possible actions

Press F5 to set the system time
Press F4 to set the system date
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter

11.3.1 Setting the system time and date

Select time or date settings	F5 or F4
Select a list in the setting window	Tab
Open the selected setting list	Enter
Select a setting	↑ or ↓
Select the next setting	Tab
Validate	Enter

11.4 Location Settings



Uncontrolled changes can cause system dysfunction.

Possible actions

Press F5 to open the directory list
Press F4 to delete an entry from a list
Press F3 to save the settings
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter

11.4.1 Keyboard functions

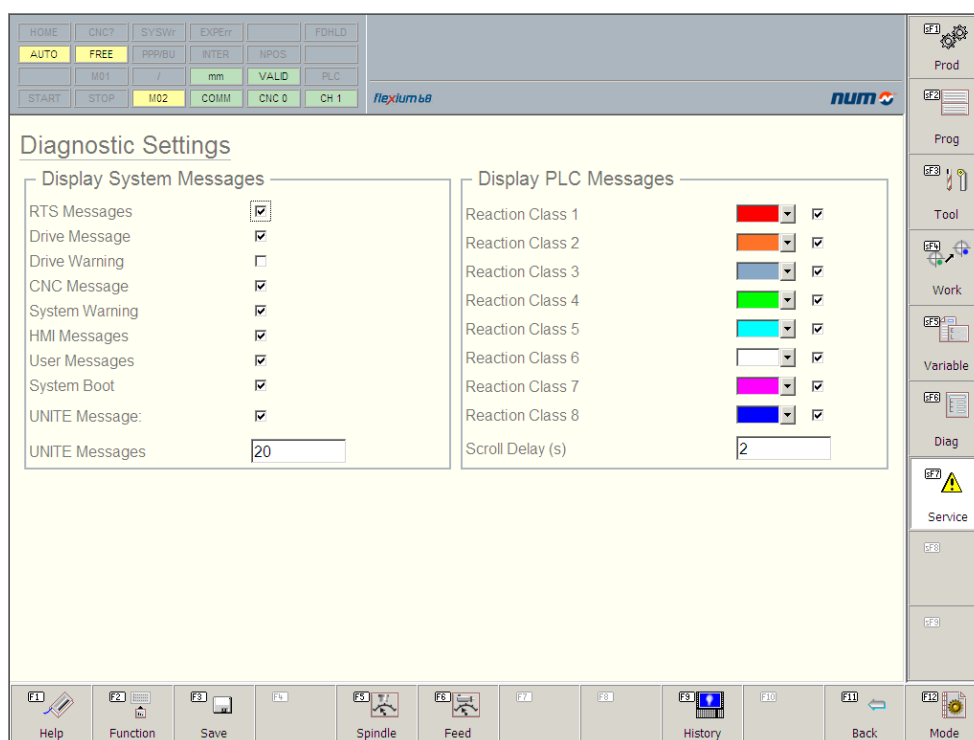
To move between the fields Tab
To open the directory list F5
To select a directory ↑ or ↓
To open a directory →
To close a directory ←

11.5 Password Settings

Open the password entry field	 F5
Enter the old password	
Enter the new password and validate	 Enter↵

11.6 Diagnostic Settings

This page allows to define the events which are to be recorded in the history file. It also contains the settings for the PLC message management.



Possible actions

Press  F9 to access the History setting page
Press  F6 to access the feed rate setting page
Press  F5 to access the spindle setting page
Press  F3 to save the settings
Press  F2 to access all existing functions. Select the function with  or  and  Enter↵

11.6.1 System messages

The check box in front of each category defines if the type of message should be displayed or not.

RTS Messages	Messages issues by the RTS system.
Drive message	Messages of the digital drives.
Drive warning	Warnings of the digital drives.
CNC message	CNC messages such as programming errors.
System warning	CNC system warnings.
HMI messages	Messages issued by the HMI.
User messages	Application specific messages, e.g. from additional HMI pages.
System boot	These are messages generated by the CNC at boot time.
UNITE message	UNITE messages are displayed a short time in the messege field every time a communication error with the CNC is detected.
UNITE message time	The default display time is set at 10 seconds. You may modify it in a range from 1 to 60 seconds.

11.6.2 PLC messages

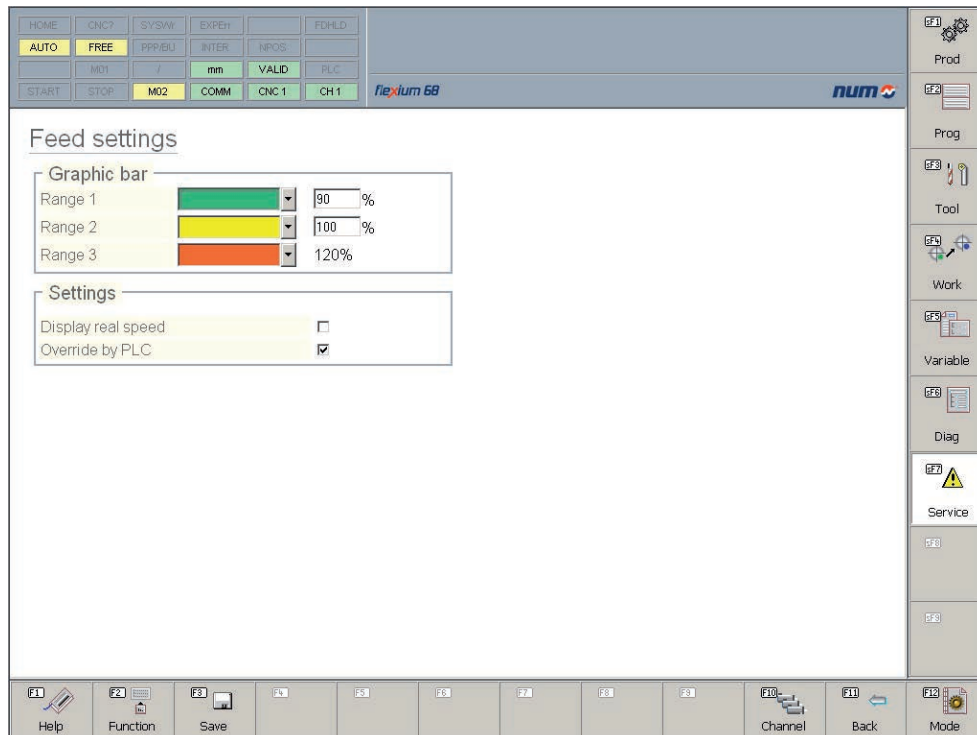
Reaction class Three settings are available for each reaction class:

- Selection of the display colour
Two settings are available for each reaction class:
 - Defines if this class of message should be displayed.
(for additional information on colour selection, refer to chapter 'Feed rate settings').
- Selection of the display colour. (for additional information on colour selection, refer to chapter 'Feed rate settings')
- In: record the event in the history log file at its appearance.
- Out: record the event in the history log file at the time it disappears.

Scroll delay Display of the messages in the message field.
If more than 3 messages are pending, the display will be scrolled in a predefined time to allow the next messages to be displayed.

11.7 Feed Rate Settings

This page is used to set the ranges of the feed rate graphic display.



Possible actions

Press **F3** to save the settings

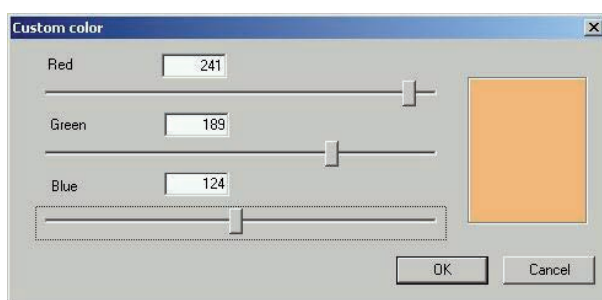
Press **F2** to access all existing functions. Select the function with **↑** or **↓** and **Enter**

11.7.1 Explanation of the setting options

The entire range of the feed rate override can be divided into three sub-ranges. An own colour may be assigned to every sub-range.

Select a setting	 Tab
For the % fields: enter the maximum value of the range and validate	 Enter
For the colour fields: open the colour palette	 Enter
Pick the desired colour	 or  ;  or 
Validate	 Enter

If the desired colour cannot be found in the standard colour palette, select the field "Others". This opens the custom colour window. It allows to set a custom colour using a combination of red, green and blue.



Select a colour	 Tab
Enter the colour value in % and validate	 Enter

Miscellaneous settings

Display real speed

Check this box to display the real axes speed in the axes position pages. (for linear axes only)

Override by PLC

If this box is checked, the displayed feed override value will be the one given by the PLC only.

This prevents erratic values to be displayed in case the override is continuously modified by Dynamic Operators (e.g. waterjet cutting, glue deposit etc).

11.8 Spindle Settings

This page is used to set the ranges of the spindle graphic displays.
This includes the graphic bars for override as well as for spindle load.
(from Flexium HMI version 3.2.0.0 on)

	Override settings			Spindle load		
Spindle 1	Range 1		79 %	Range 1		100 %
	Range 2		95 %	Range 2		130 %
	Range 3		100 %	Range 3		150 %
Spindle 2	Range 1		80 %	Range 1		100 %
	Range 2		90 %	Range 2		130 %
	Range 3		100 %	Range 3		150 %
Spindle 3	Range 1		60 %	Range 1		100 %
	Range 2		90 %	Range 2		130 %
	Range 3		100 %	Range 3		150 %
Spindle 4	Range 1		30 %	Range 1		100 %
	Range 2		70 %	Range 2		130 %
	Range 3		0 %	Range 3		150 %

Possible actions

Press **F3** to save the settings

Press **F2** to access all existing functions. Select the function with **↑** or **↓** and **Enter**

11.8.1 Explanation of the setting options

Setting options for spindle override	The entire range of the spindle override can be divided into three sub-ranges. An own colour may be assigned to every sub-range. For explanations of the settings, refer to chapter '11.7 Feed Rate Settings' in this manual.
Setting options for spindle load	The entire range of the spindle load can be divided into three sub-ranges. An own colour may be assigned to every sub-range.

11.8.2 History

This page is intended to define which message according to their type are logged in an history file and if they are logged, when they appear and/or when they are cleared.

For the different types of messages refer to the paragraph 'Diagnostic settings'.

To access this page, press **F9** in the 'Diagnostic Settings' of the 'Service' context.

11.8.2.1 History setting

History Settings

Log System Messages

History Log:	☒	
History Logfiles:	20	
RTS Messages	☒ In	
Drive Message:	☒ In	☒ Out
Drive Warning:	☒ In	☒ Out
CNC Message:	☒ In	☒ Out
System Warning:	☒ In	☒ Out
HMI Message:	☒ In	☒ Out
User Message:	☒ In	☒ Out
System Init:	☒ In	
Exception Message	☒ In	
UNITE Message:	☒ In	

Log PLC Messages

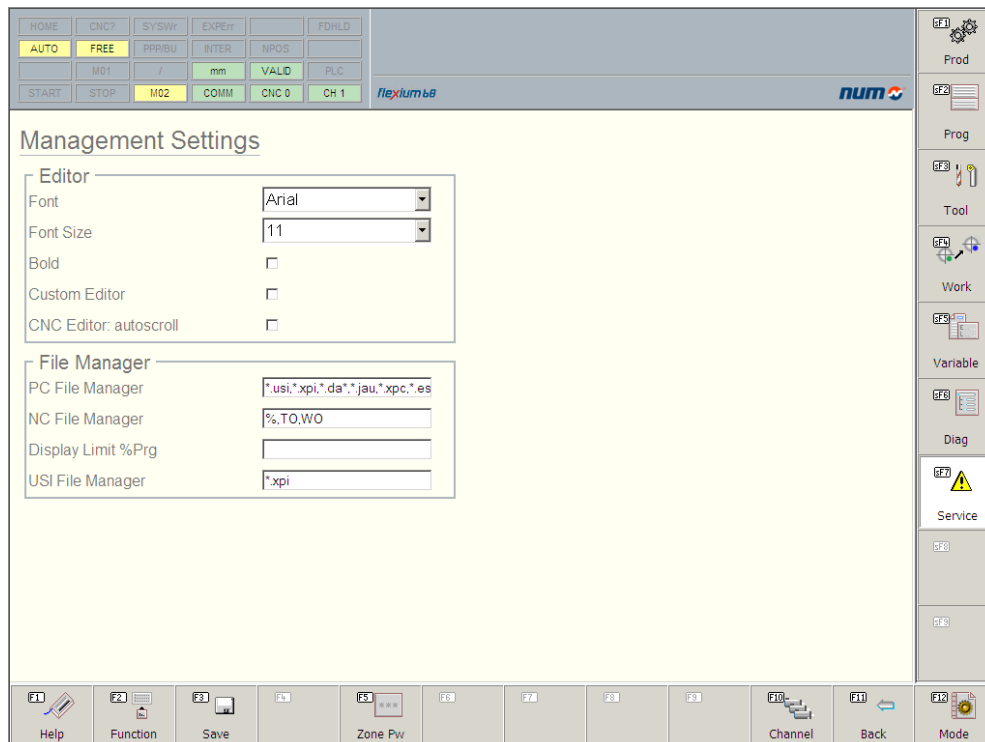
Reaction Class 1	☒ In	☒ Out
Reaction Class 2	☒ In	☒ Out
Reaction Class 3	☒ In	☒ Out
Reaction Class 4	☒ In	☒ Out
Reaction Class 5	☒ In	☒ Out
Reaction Class 6	☒ In	☒ Out
Reaction Class 7	☒ In	☒ Out
Reaction Class 8	☒ In	☒ Out

Top Navigation Bar: HOME, CNC?, SYSWt, EXPerr, FDHLD, AUTO, FREE, PPR/BU, INTER, NPOS, M01, /, mm, VALID, PLC, START, STOP, M02, COMM, CNC 0, CH 1, flexium 68, num.

Right Sidebar: Prod, Prog, Tool, Work, Variable, Diag, Service.

Bottom Toolbar: F1 Help, F2 Function, F3 Save, F4, F5, F6, F7, F8, F9, F10, F11 Back, F12 Mode.

11.9 Management Settings



Possible actions

Press **F5** to enter or modify the memory zones password

Press **F3** to save the settings

Press **F2** to access all existing functions. Select the function with **↑** or **↓** and **Enter**

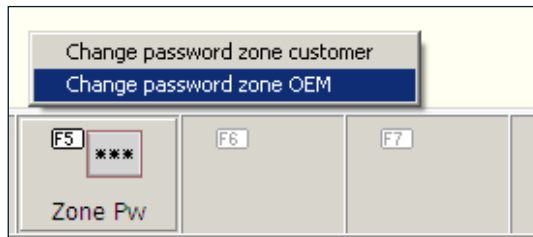
11.9.1 Explanation of the setting options

Editor font	The editor font can be Arial or Courier.
Font size	The editor font size can be chosen from 10 to 16 dots.
Bold	To select bold characters.
Custom editor	Activate to use a custom editor. Refer to chapter '11.5 Data and File Location Settings' in this manual for details on how to set the path to this editor.
CNC editor: autoscroll	In the CNC editor page: enable automatic scrolling between pages during program execution.
PC file manager	List of the file extensions managed by the HMI. These files will be displayed in the right hand window of PC Disk in the "Programming" context.  Only files in NUM format (DIN 66 025) can be loaded in the CNC
NC file manager	List of the file types displayed in the CNC memory window in the context "Programming".
Display limit	Allows the limitation of the display of part programs in the CNC memory window in the context "Programming". Example: if the value %584 has been set in the field "Display limit", only part programs with a number up to %583 will be displayed.
USI file manager	List of the file extensions which can be processed by the USI editor.

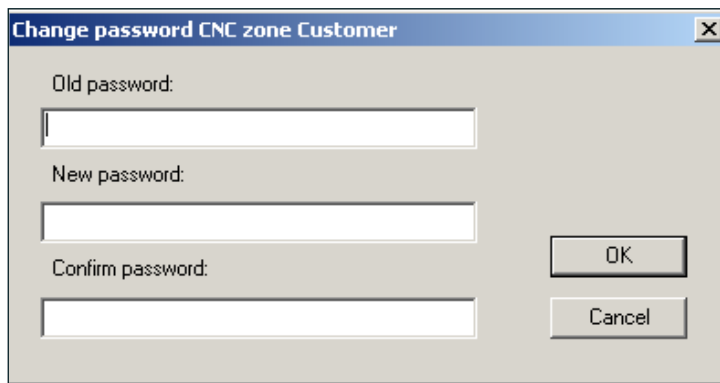
11.9.2 Password management

This page is intended to set or change the password to access part program memory zones

Pressing **[F5]** in the *Management Settings* page of the 'Service Context' shows the following menu.



Select the zone password to set or change, the following prompt will be displayed.

A screenshot of a dialog box titled "Change password CNC zone Customer". It contains three input fields: "Old password:", "New password:", and "Confirm password:". To the right of the input fields are two buttons: "OK" and "Cancel".

11.10 Machine Data

This page is intended to modify NCK and Drives parameters.

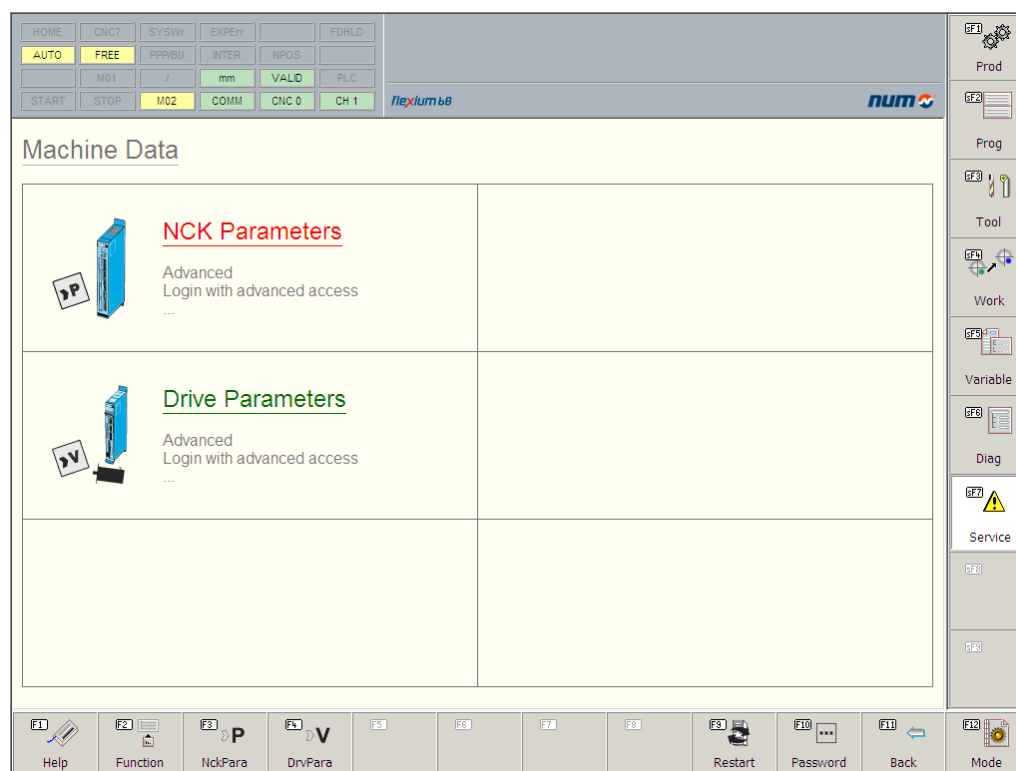
Two levels are authorized:

- **Standard access:** (e.g. Backlash, Home position).
Can be modified by maintenance personnel with enough knowledge about parameters functions.
- **Advanced access:**
For all NCK and Drives parameters. Requires an additional password entry.
Should only be modified by specialists!

To access this page, press **F6** in the main page of the 'Service' context.

Possible actions

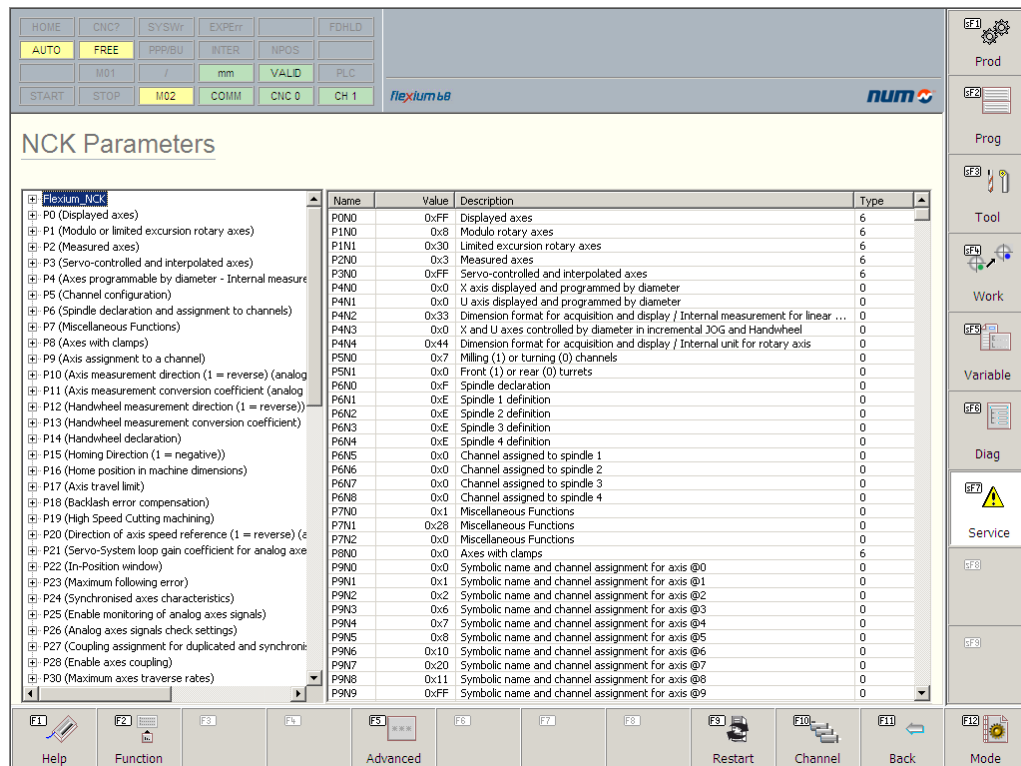
Press F10 to modify the advanced access Password
Press F9 to reboot the system (advanced Password required)
Press F4 to access the Drives parameters
Press F3 to access the NCK parameters



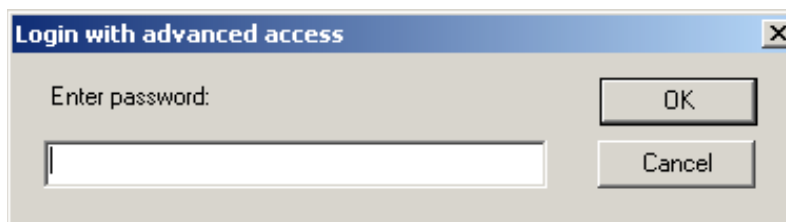
11.10.1 NCK parameters

When accessing this page all NCK parameters are read and displayed.

To access this page, press **F3** in the 'Machine Data' page of the 'Service' context.



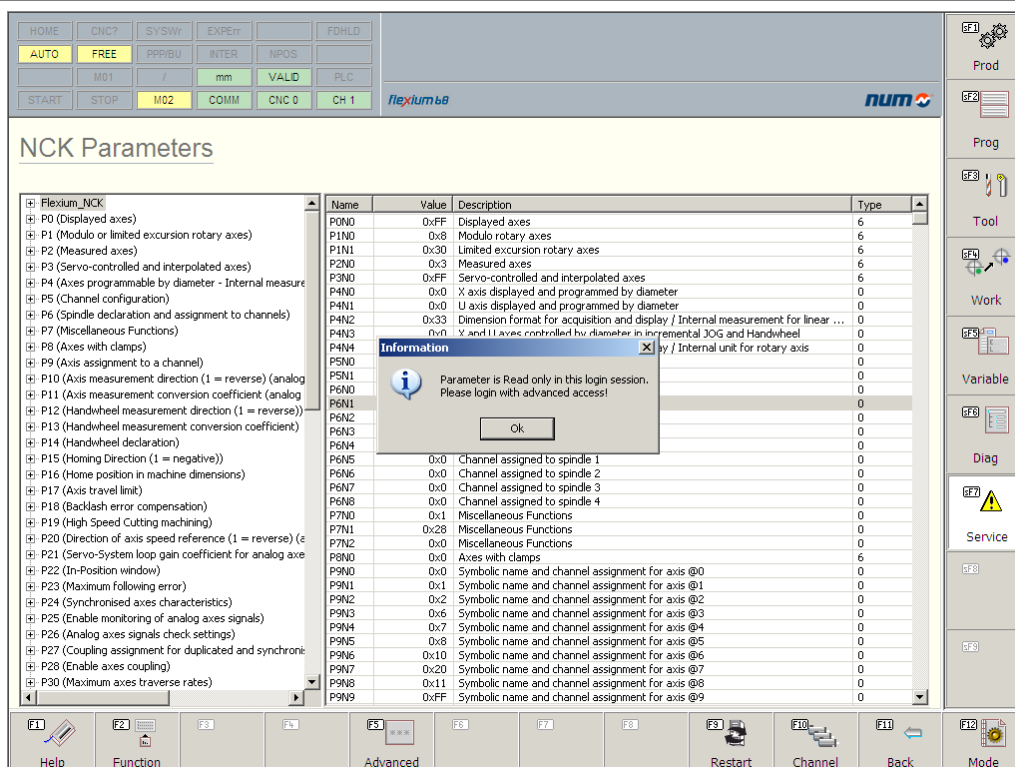
Click on a parameter to get access to its value. If the parameter is with advanced access you will not be able to continue until the advanced password is given.



The default password is *EXTOEM*.

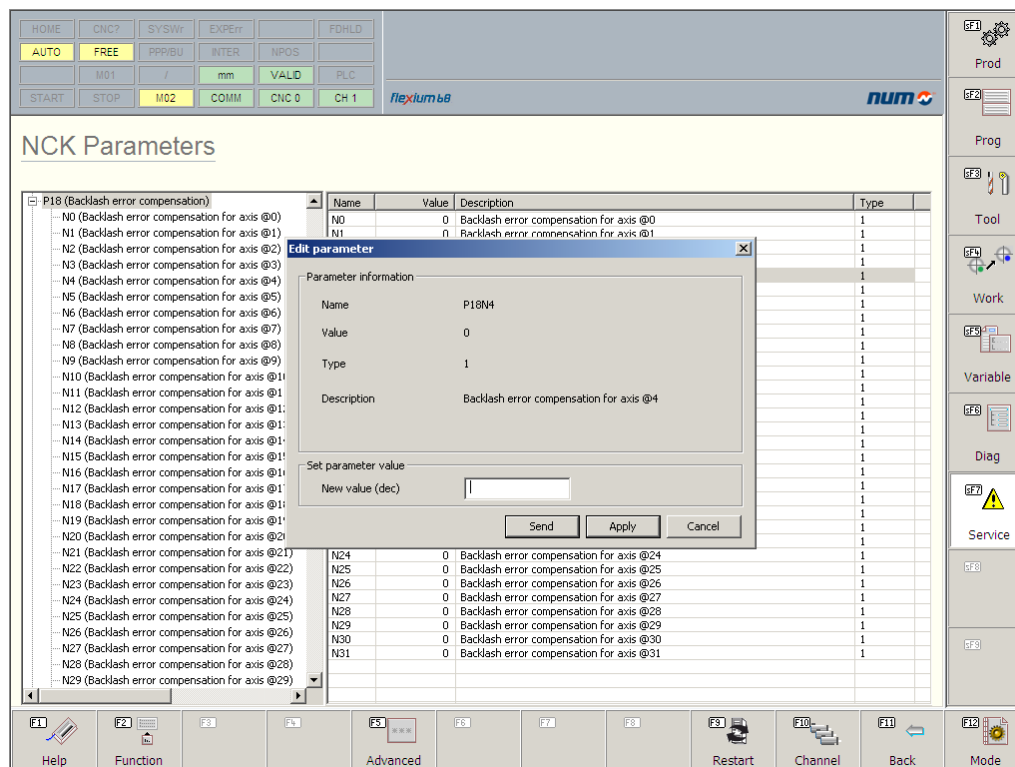


It is strongly suggested to modify this password before putting the machine in production.



When access is granted the following windows is displayed:

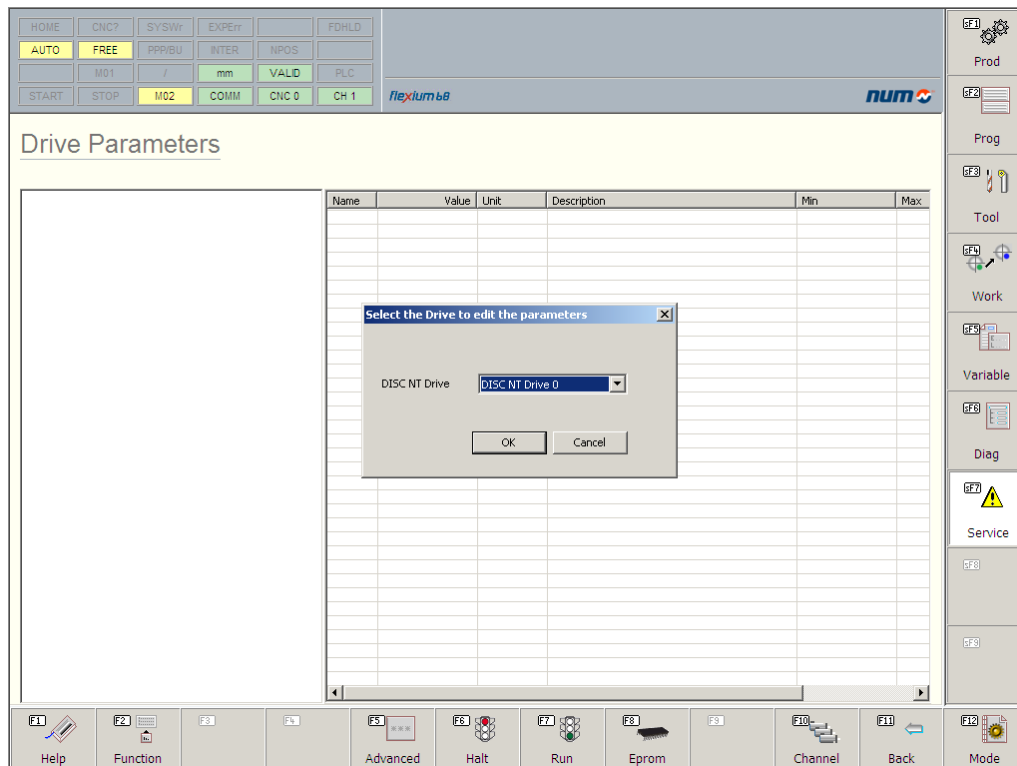
- Enter the new value according to the format indicated.
- Send will store the value in the NCK temporarily (until next reboot).
- Apply will store the value in the NCK permanently.



11.10.2 Drive parameters

When accessing this page the system will ask you on which drive you want to work.

To access this page, press **F4** in the 'Machine Data' page of the 'Service' context.



Click on a parameter to get access to its value. If the parameter is with advanced access you will not be able to continue until the advanced password is given.

Possible actions

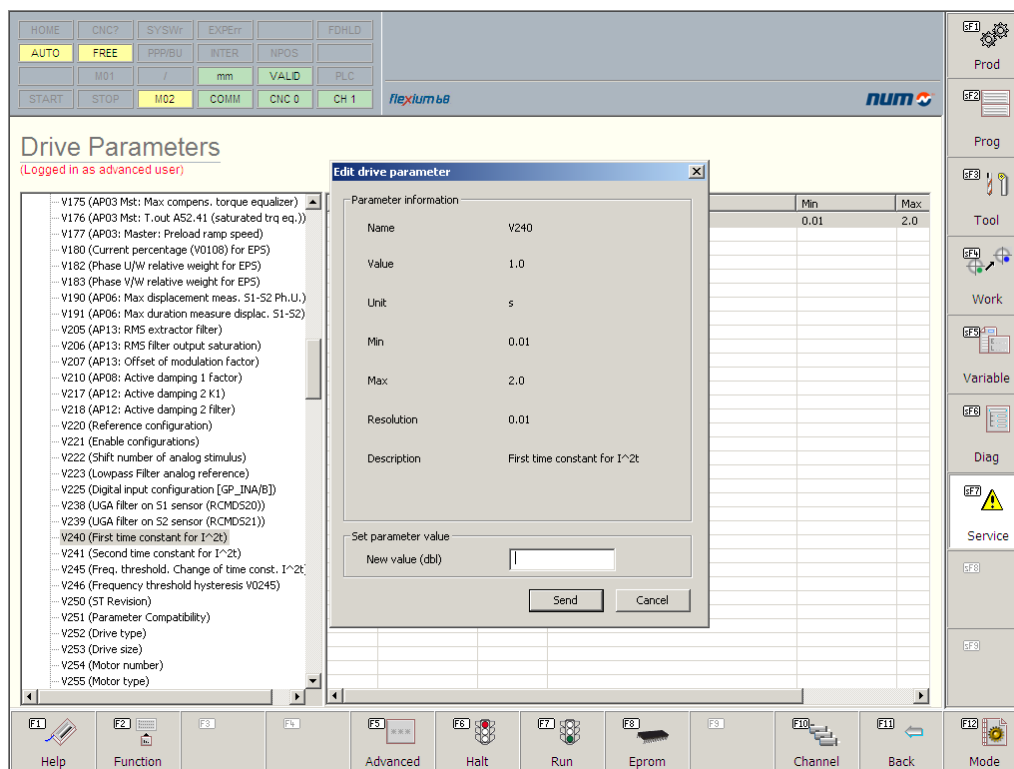
Press **F8** store the modified parameter in Eprom

Press **F7** to set the drive in Run

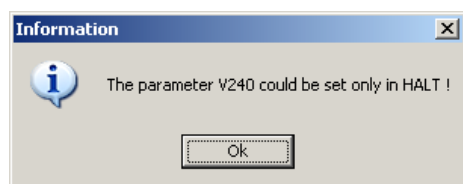
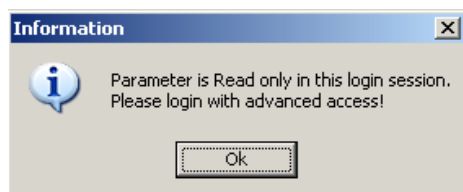
Press **F6** to set the drive in Halt

Press **F5** to get advanced access

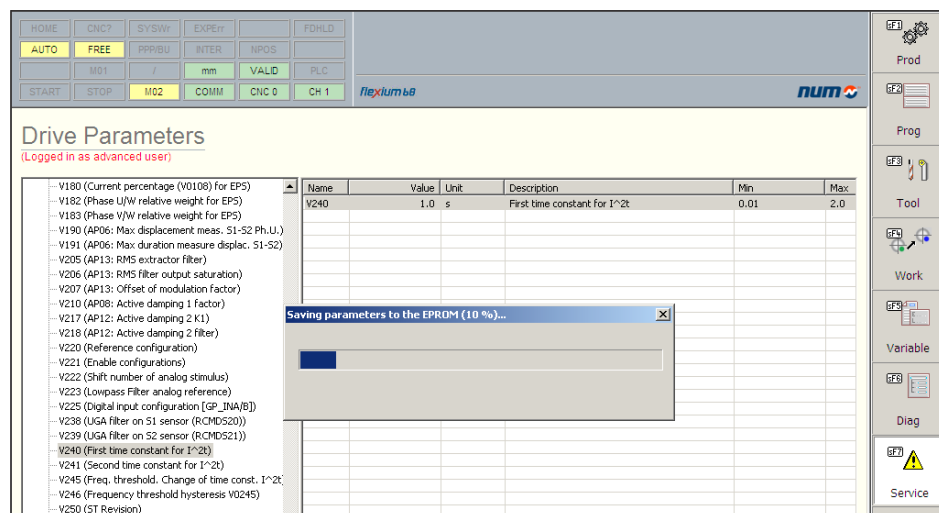
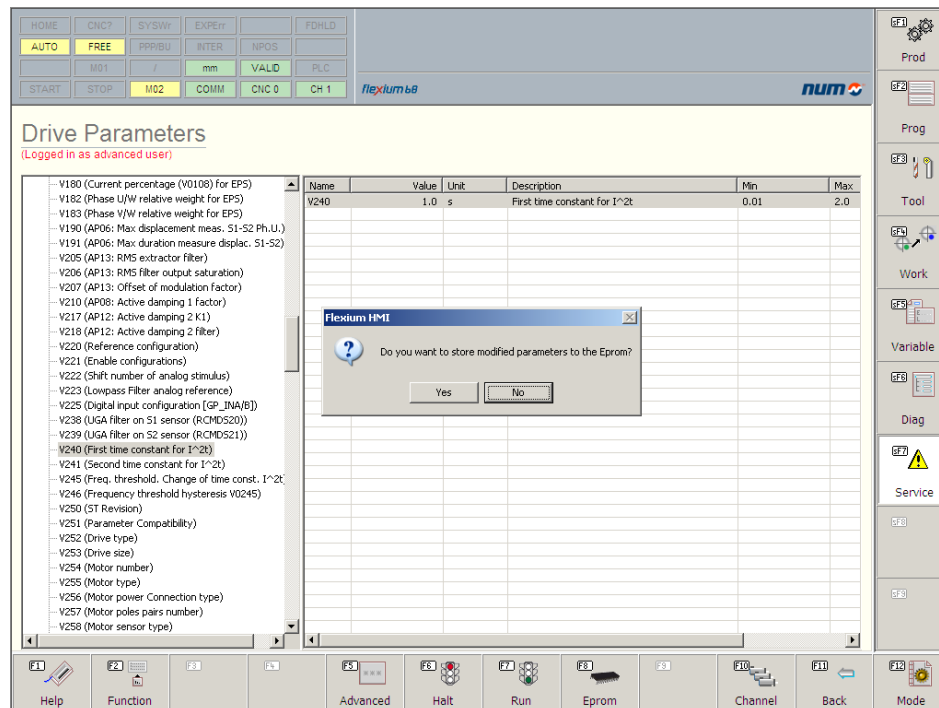
Once the drive selected the system will read and display all drive parameters. Click on one particular parameter to have edit access.



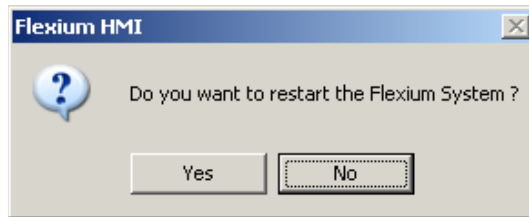
According to the parameter, it may be necessary to have advanced access or to set the drive in Halt.



Once the edit session is finished do not forget to store the parameters in EPROM to make the change permanent after setting the drive in Halt.



11.10.3 Restart



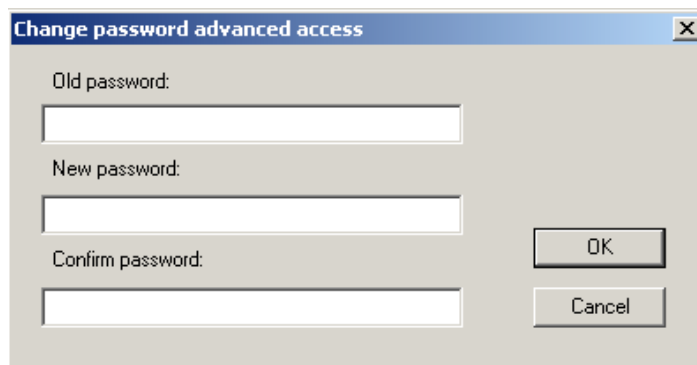
This command is intended to reboot the system in case of necessity after a parameter change to take into account the new value.

A restart is only possible with *Advanced* access.

To access this command, press **[F9]** in the '*Machine Data*' page of the '*Service*' context. The system will prompt for confirmation and optionnaly for the advanced access password.

11.10.4 Change Password

This prompt allows for changing the default password required for the advanced access. To access it, press **[F10]** in the '*Machine Data*' page of the '*Service*' context.



Page deliberately empty.

Summary

12	CopyExplorer.....	165
12.1	Access to CopyExplorer	165
12.2	Save	166
12.2.1	Save one file only.....	166
12.2.2	Save several successive files	166
12.2.3	Save several non successive files	166
12.2.4	Save all files	166
12.3	Restore.....	166

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

Page deliberately empty.

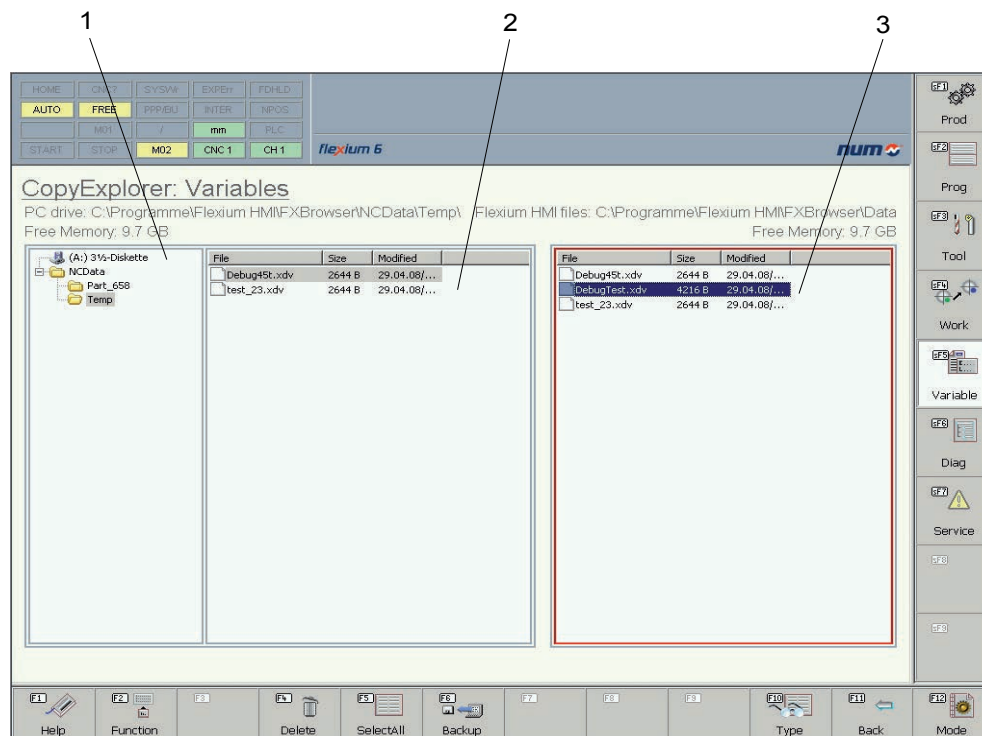
12 CopyExplorer

12.1 Access to CopyExplorer

CopyExplorer can be accessed from the Context “Variables” as well as from the “Exception” and “History” pages in the Context “Diagnostic”. It is used to save and restore files on disk or any other media which can be managed by the HMI.

The following file types can be transferred:

- Debug files for variables of the CNC and the PLC *.xdv.
- History files for messages and alarms *.log.
- Exception files *.exc.
- Device list files.



- List of drives and folders for save and restore.
For the selection of drives and folders, refer to chapter ‘11.4 Data and File Location Settings’ in this manual.
- List of files present in the selected folder.
- HMI files which can be saved.

Possible actions

Press [F10] to select the file type: Variables, Exception, History or Device List
Press [F6] to save or restore
Press [F5] to select all files of the list
Press [F4] to delete files
Press [F3] to create a new folder (the focus must be on the list of drives and folders)
Press [F2] to access all existing functions. Select the function with [↑] or [↓] and [Enter]

12.2 Save

12.2.1 Save one file only

Select the list of drives and folders	 Tab ...
Select drive and folder	 ↑ or ↓
Select the list of the HMI files as source	 Tab ...
Select the file type	 F10
Select the file	 ↑ or ↓
Save the file	 F6

12.2.2 Save several successive files

Select the first file	 ↑ or ↓
Mark the files to save	 ⇧ Shift + ↑ or ↓
Save the files	 F6

12.2.3 Save several non successive files

Select the first file	 ↑ or ↓
A) Enter select mode	 Ctrl + ↑ or ↓
B) Mark the next file to save	 Space
Return to A) to select further files	
Save the files	 F6

12.2.4 Save all files

Select all files	 F5
Save the files	 F6

12.3 Restore

The operations are the same as for “Save”.
The list of drives and folders must be selected as source.

Page deliberately empty.

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

Page deliberately empty.

Summary

13 The Virtual Keyboard 171

13.1 Use of the Keyboard 171

13.1.1 Activating the keyboard..... 171

13.1.2 Closing the keyboard display 171

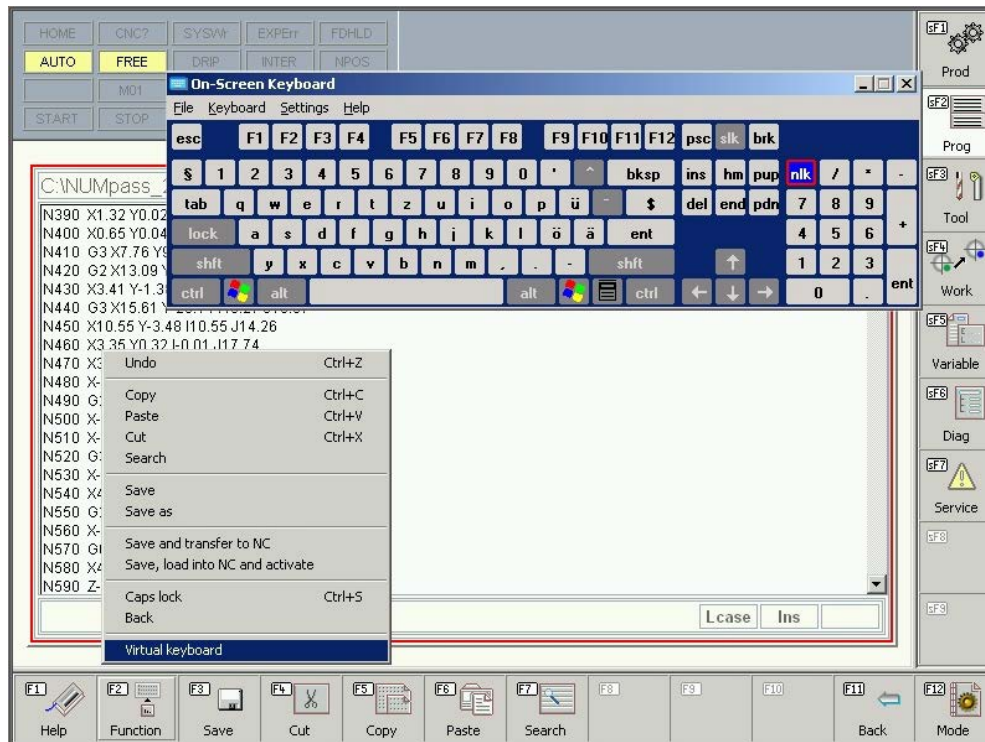


Page deliberately empty.

13 The Virtual Keyboard

13.1 Use of the Keyboard

The virtual keyboard offers the same possibilities for data input as a normal keyboard. It can be activated in every context or page allowing an input of data.



13.1.1 Activating the keyboard

Press **F2** "Function"

Select "Virtual keyboard"

13.1.2 Closing the keyboard display



Page deliberately empty.

Summary

14	Review	175
14.1	System Overview	175
14.1.1	Operating Mode	175
14.1.2	Defining a Program	175
14.1.3	Creating a Program	176
14.2	Review of Program Coordinate Systems	177
14.2.1	Review of Axis Definition and Direction	177
14.3	Machine Overview	178
14.3.1	General	178
14.3.2	Turning and Milling Machines	178
14.4	Turning Machine - Specific Features of Front and Rear Turrets	179
14.4.1	View of a Turning Machine with Rear Turret	179
14.4.2	View of a Turning Machine with Front Turret	179
14.5	Definition of Travels and Origins	180
14.5.1	Homing with Incremental or Combined Encoders Using Absolute Dimensions	180
14.5.2	Schematic Representation of Travels (Turning and Milling machine)	181
14.6	Offset Definitions	182
14.7	Turning Machine - Offsets Represented on the Z and X Axes	183
14.7.1	Offsets on the Z Axis (Solution 1 with DAT2)	183
14.7.2	Offsets on the Z Axis (Solution 2 with DAT2)	183
14.7.3	Offsets on the X Axis (Solution 1 with DAT2)	184
14.7.4	Offsets on the X Axis (Solution 2 without DAT2)	184
14.7.5	Schematic Representation of the Offsets on X and Z	185
14.8	Milling machine - Offsets Represented on the Z, X and Y Axes	186
14.8.1	Offsets on the Z axis	186
14.8.2	Offsets on the X axis	186
14.8.3	Offsets on the Y axis	187
14.8.4	Schematic Representation of All Offsets	188
14.8.5	Special Case of Milling Machines Equipped with Rotary Tables	189
14.9	Tool Axis Orientation	191
14.9.1	Using the G16 P+ function on turning or milling machine	192
14.10	Definition of Tool Dimensions and Offsets	193
14.10.1	Turning Machine - Dimensions, Tool Tip Radius and Orientation	193
14.10.2	Tool Tip Radius and Orientation	194
14.10.3	Milling Machine - Definition of Tool Dimensions	195
14.11	Definition of Tool Wear	196
14.11.1	Turning Machine	196
14.11.2	Turning Machine	197

Page deliberately empty.

14 Review

14.1 System Overview

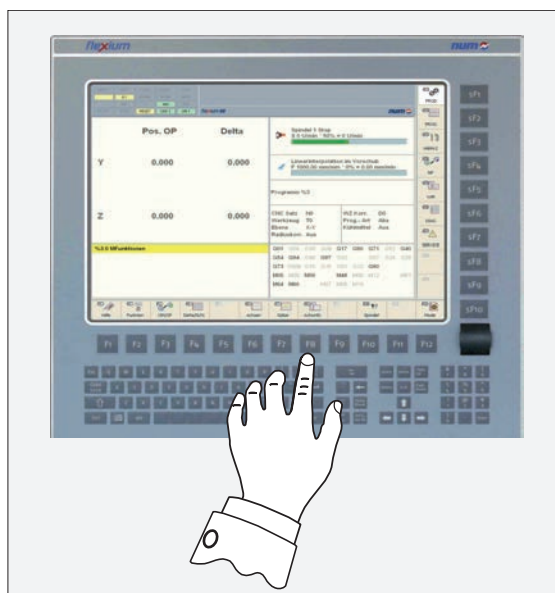
The aim of this chapter is to detail concepts to which the operator can refer, rather than to reflect the way an operator works on the machine.

For instance, in Section '14.6 Offset Definitions' the aim is to define the offsets and corresponding origins or zero points rather than give a method for measuring the offsets.

14.1.1 Operating Mode

The operator uses the computerised numerical control (CNC) in various operating modes accessible from the HMI.

Each key corresponds to a particular use of the CNC (program load, tool setting, automatic machining, etc.).



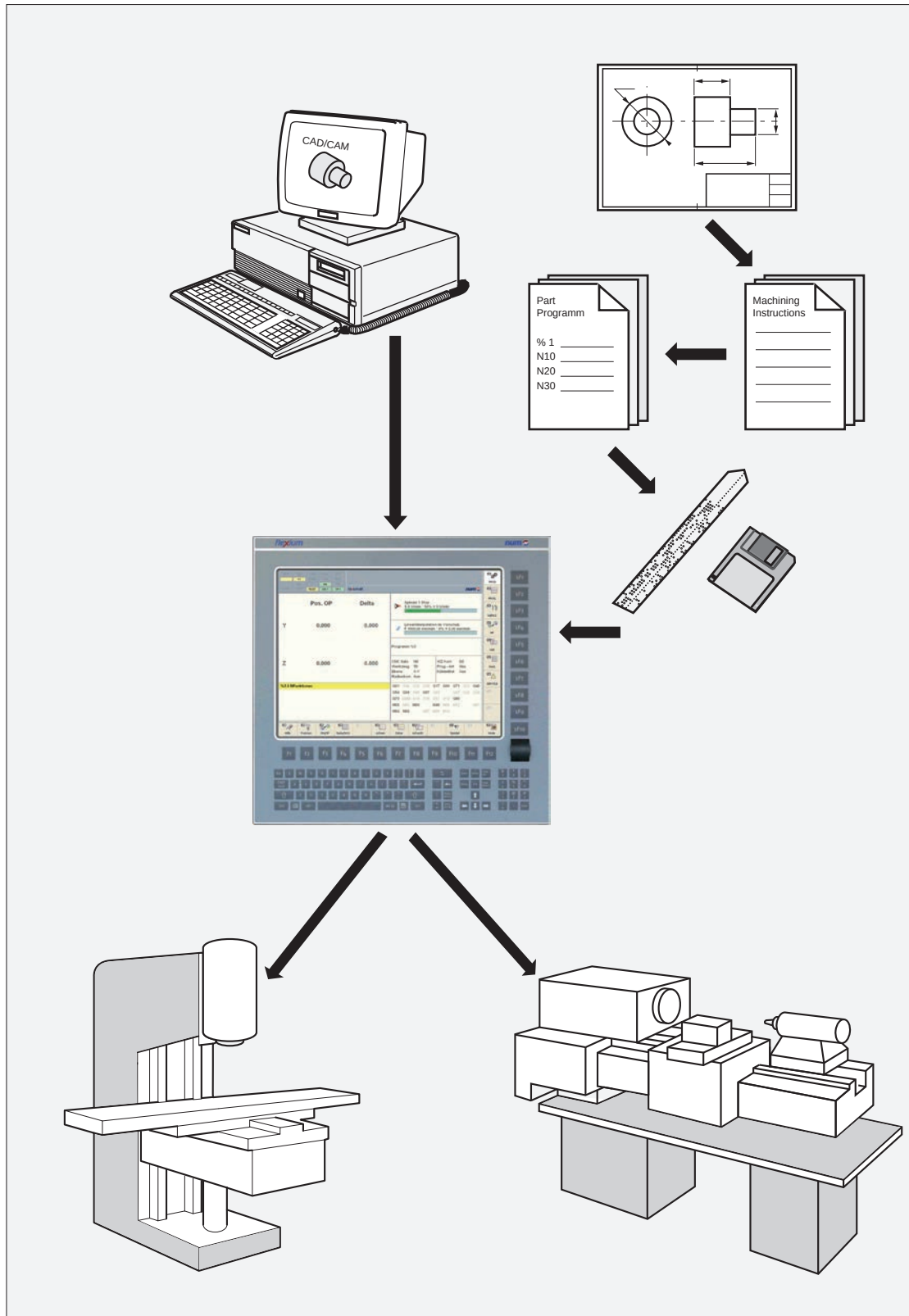
14.1.2 Defining a Program

A program is a sequence of instructions written in a programming language specific to the CNC (the most widely used is ISO code: International Organization for Standardization).

The CNC interprets the program to control actions on a machine tool.

14.1.3 Creating a Program

A part program can be created by traditional programming or using a CAD/CAM system.

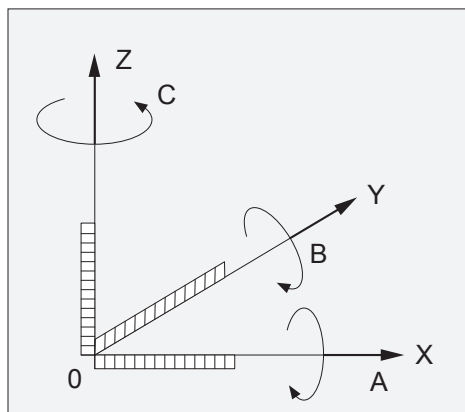


14.2 Review of Program Coordinate Systems

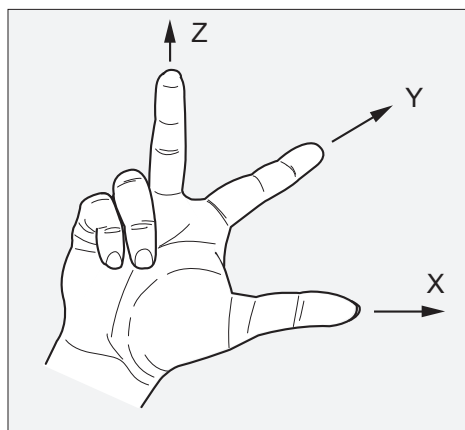
14.2.1 Review of Axis Definition and Direction

A coordinate system is used to identify the positions and movements of an object with respect to an origin or zero point.

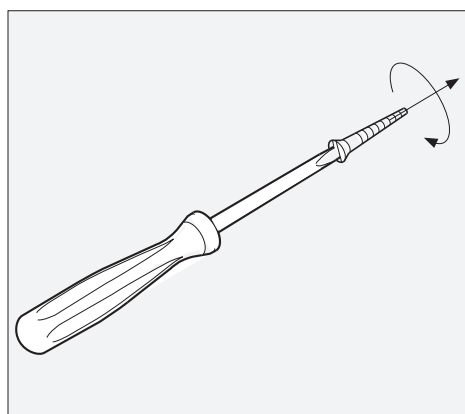
A rectangular Cartesian coordinate system is a direct three-axis system of three linear axes, X, Y and Z, with which are associated three rotary axes, A, B and C.



The direction of axes X, (Y) and Z is easily remembered by the right-hand rule.



The positive direction of rotation of a rotary axis corresponds to the direction of screwing of a right-hand screw on the associated axis.



14.3 Machine Overview

14.3.1 General

The manufacturer defines the coordinate system associated with the machine in accordance with standard ISO 841 (or NF Z68-020). The X, Y and Z axes, parallel to the machine slide ways, form a right-handed rectangular Cartesian coordinate system.

The coordinate system measures tool movements with respect to the part to be machined, assumed fixed.



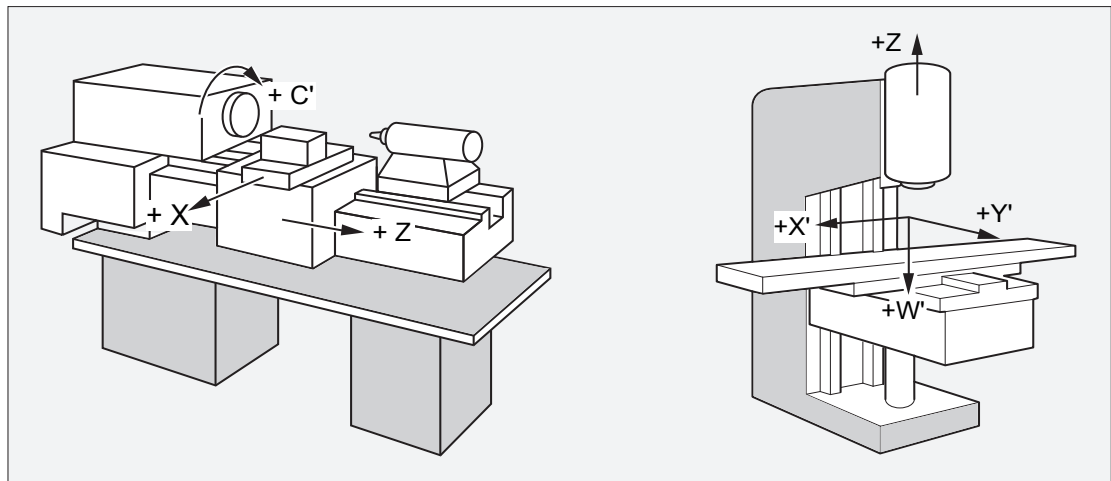
When it is the part that moves, it may be more convenient to identify its movements. In this case, axes X', Y' and Z', pointing in opposite directions from axes X, Y and Z, are used.

The direction of the axes of a machine depends on the type of machine and the layout of its components.

Rotary axes A, B and C define rotations around axes parallel to X, Y and Z.

Secondary linear axes U, V and W may or may not be parallel to primary axes X, Y and Z.

14.3.2 Turning and Milling Machines



Turning Machine

For a lathe:

- the Z axis is the spindle axis,
- the X axis is perpendicular to the Z axis and corresponds to radial movement of the tool head turret, the Y axis (generally imaginary for turning) forms an orthogonal coordinate system with the X and Z axes.

Positive movement on Z or X increases the distance between workpiece and tool.

For more details, refer to the above-mentioned standard.

Milling machine

For a milling machine:

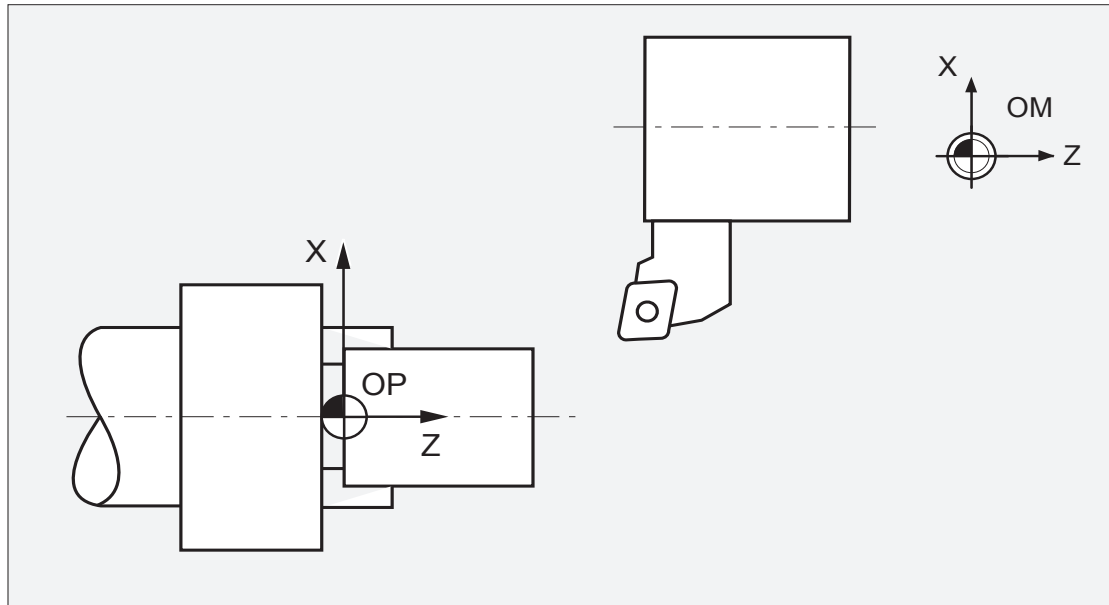
- The Z axis is the axis of the main spindle when this axis is parallel to one of the slide ways,
- Positive movement on the Z axis increases the distance between the part and tool,
- The X axis is perpendicular to the Z axis and corresponds to the largest excursion,
- The Y axis is perpendicular to the X and Z axes.

For more details, refer to the above-mentioned standard.

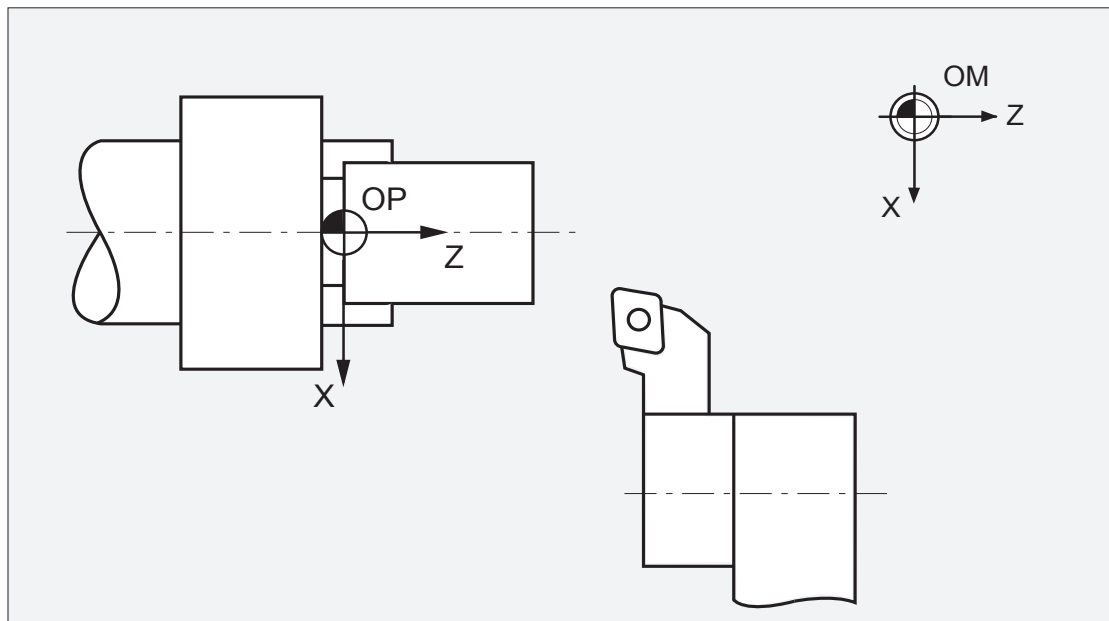
14.4 Turning Machine - Specific Features of Front and Rear Turrets

The front or rear position of the main turret on the turning machine defines the positive direction on the X axis, which affects definition of the tool tip orientation (see figure below and the section '14.10.1 Turning Machine - Dimensions, Tool Tip Radius and Orientation').

14.4.1 View of a Turning Machine with Rear Turret



14.4.2 View of a Turning Machine with Front Turret



14.5 Definition of Travels and Origins

All the CNC movements are defined with respect to the measurement origin or zero point of the machine.

When the system is turned on, it does not know the measurement origin. The mechanical travel on each machine axis is limited by maximum and minimum limit switches.

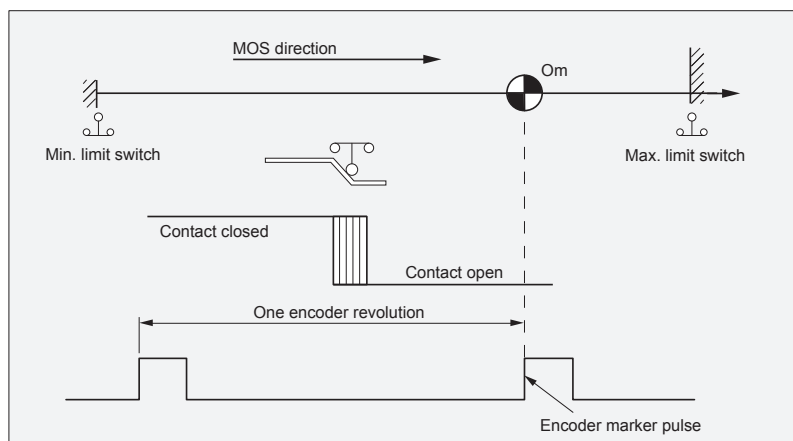
Om : 	The system establishes the measurement origin (OM) via a homing procedure (MOS).
OM : 	The origin switch is set in a specific physical location: the machine zero point (Om) may or may not be the same as the measurement origin (OM).

14.5.1 Homing with Incremental or Combined Encoders Using Absolute Dimensions

Homing with Incremental Encoders

In this case, the homing procedure is completed for each of the axes when:

- the origin switch is actuated in the direction of movement specified by the m/c manufacturer (MOS direction),
- the encoder which measures axis movement outputs its marker pulse.



Homing with Combined Encoders Using Absolute Dimensions

In this case, the axis travel is less than the encoder measurement travel. Homing is performed on any point of the axis travel when the CNC is turned on or reset.



The encoder zero point must be located outside the axis travel limits. The machine travels must be included with accurate real values in the manufacturer's machine manual.

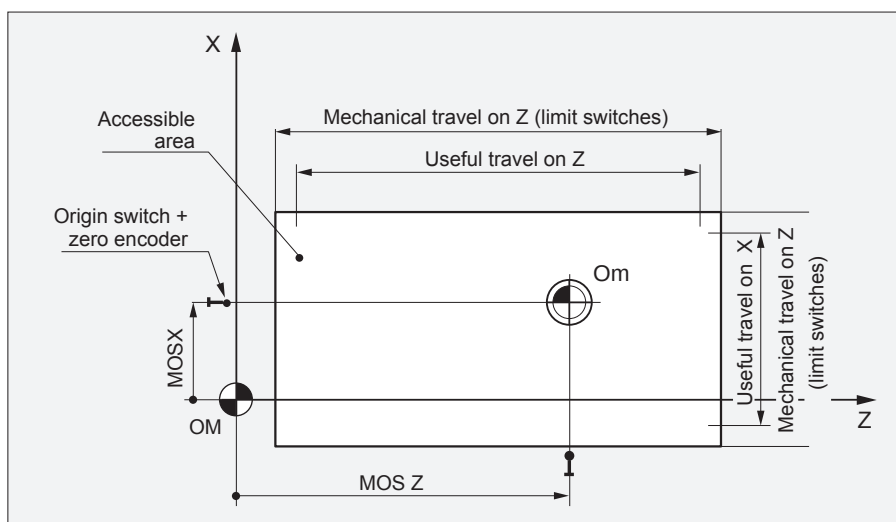
14.5.2 Schematic Representation of Travels (Turning and Milling machine)

When homing (MOS) is completed, the system applies the offsets defined by the manufacturer to each of the axes to establish the measurement origin (OM).

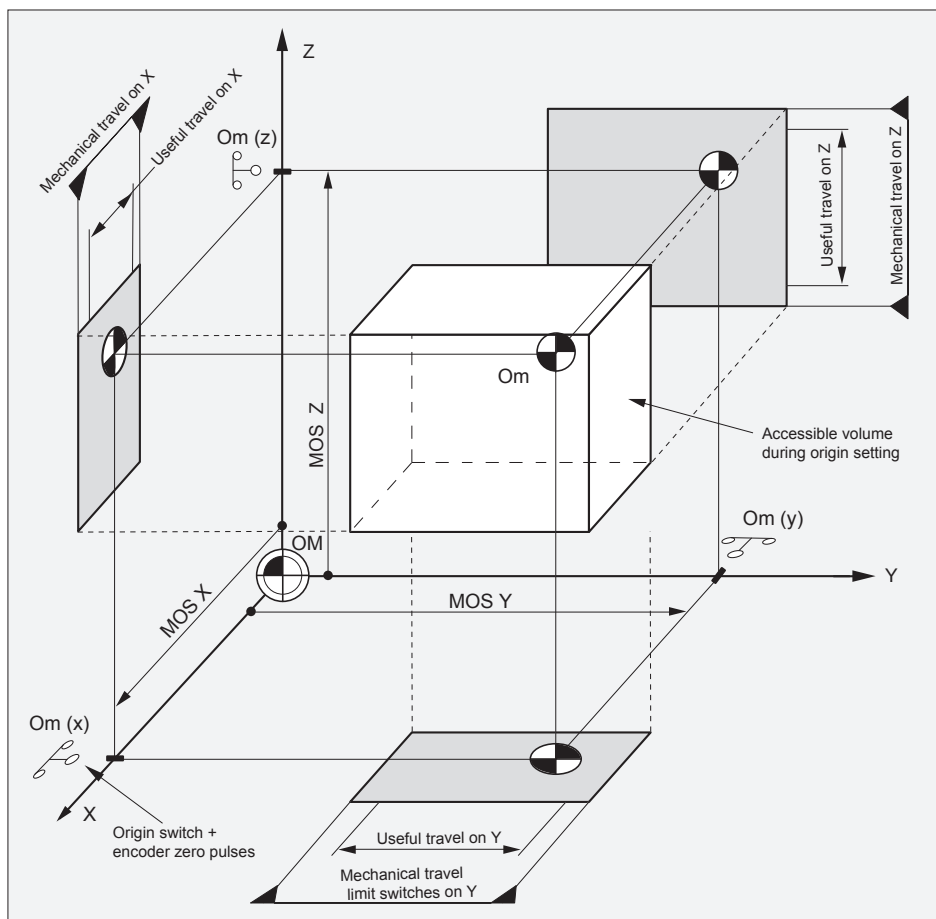
$$\text{Measurement origin offset (OM/OM)} = \text{MOS}$$

The useful travel on each axis is limited by software limit switches whose settings are defined by machine parameters.

Turning machine



Milling machine



14.6 Offset Definitions

To write a part program, the programmer chooses a program origin. The program origin is generally a starting point for dimensional measurements on the part drawing.

OP : 	The operator specifies the position of the program origin (OP) with respect to the part origin (Op).
Op : 	He sets (for each axis) a known, accessible point on the part, called the part origin (Op). This may be the same point as the program origin.

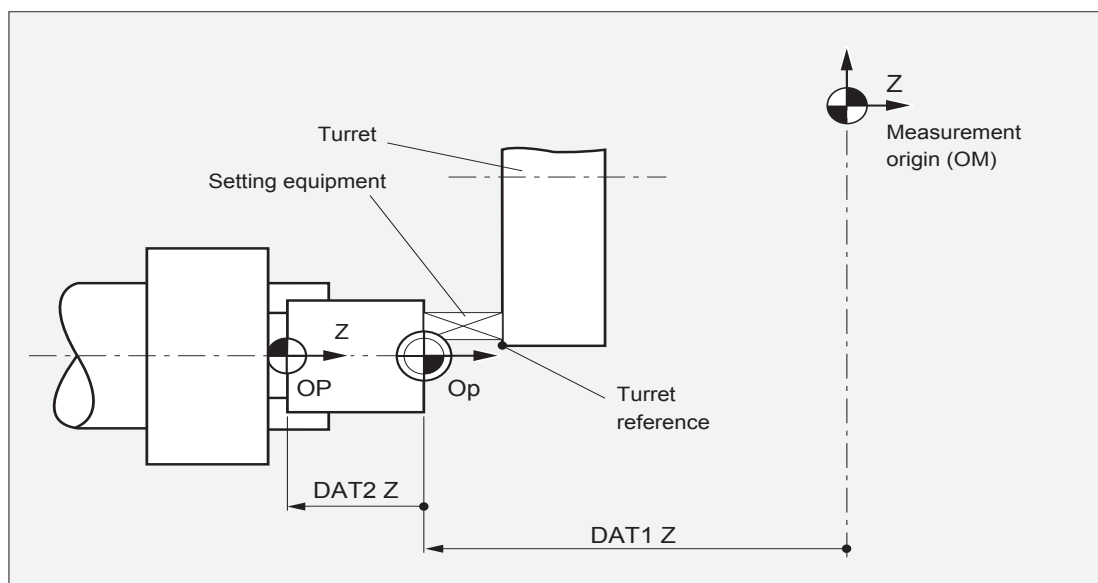
Part origin offset (Op/OM) = DAT1

The program origin offset is set with respect to the part origin.
(this can be done by programming).

Program origin offset (OP/Op) = DAT2

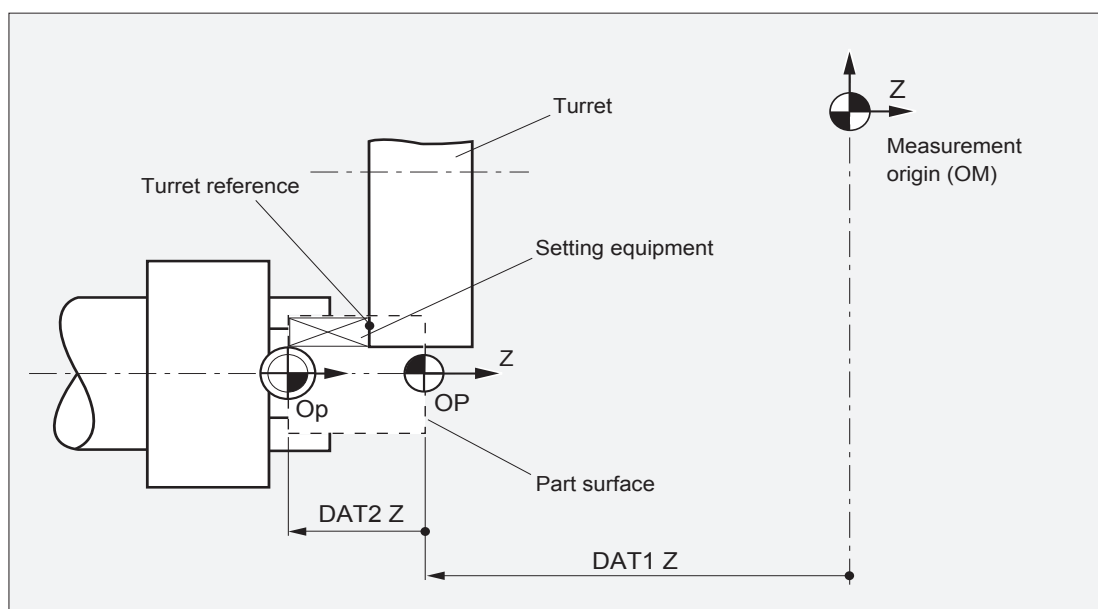
14.7 Turning Machine - Offsets Represented on the Z and X Axes

14.7.1 Offsets on the Z Axis (Solution 1 with DAT2)

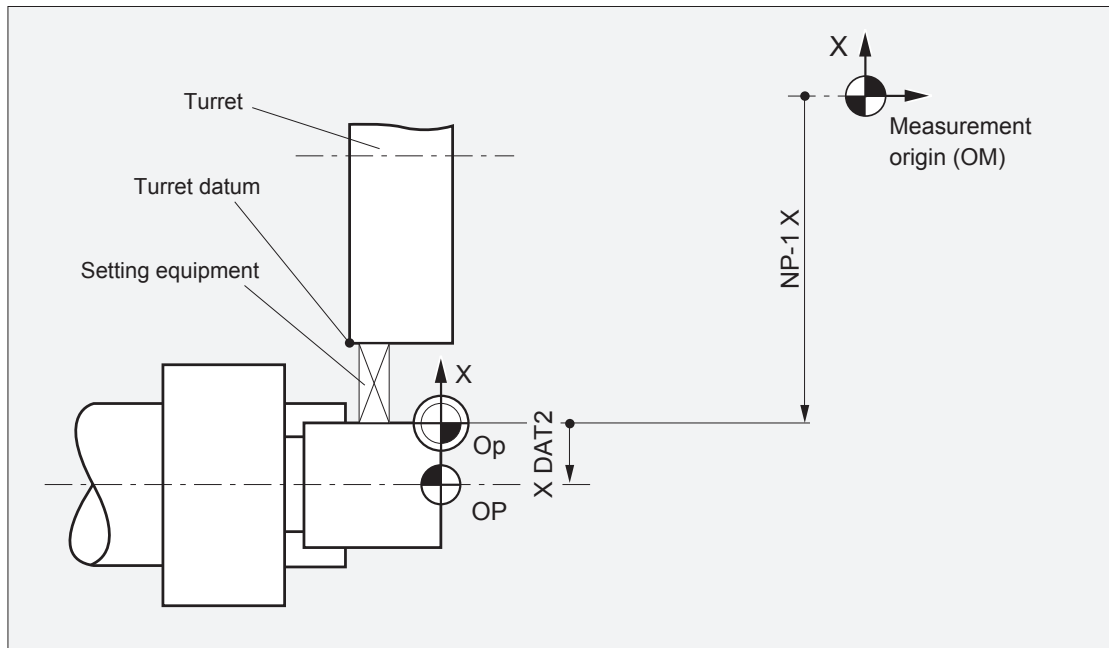


14.7.2 Offsets on the Z Axis (Solution 2 with DAT2)

DAT1 Z: Value measured between OM and the back of the jaws (workpiece not installed)

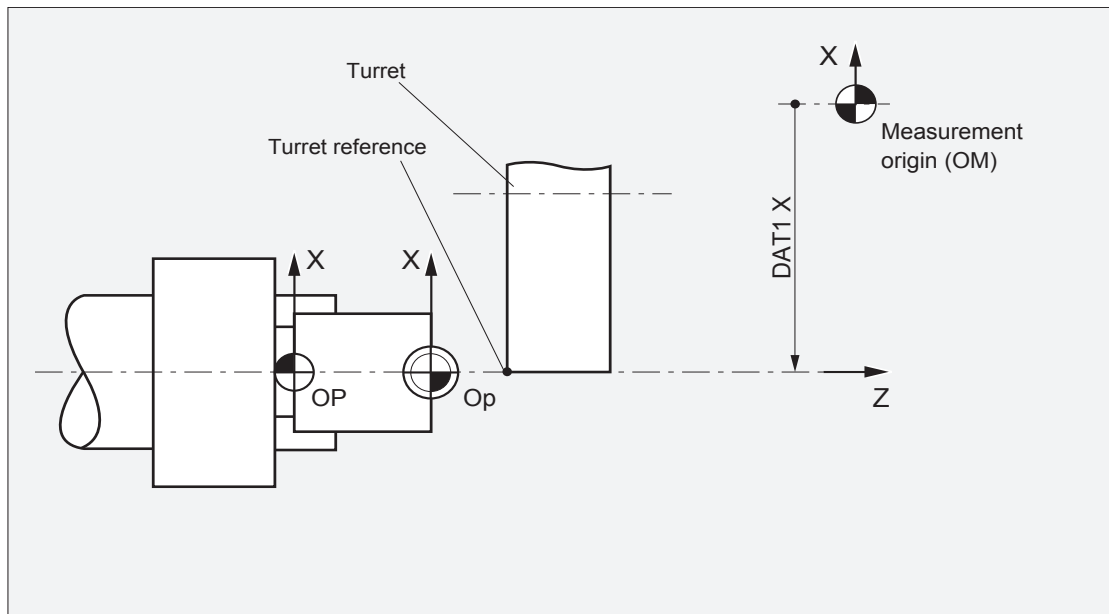


14.7.3 Offsets on the X Axis (Solution 1 with DAT2)



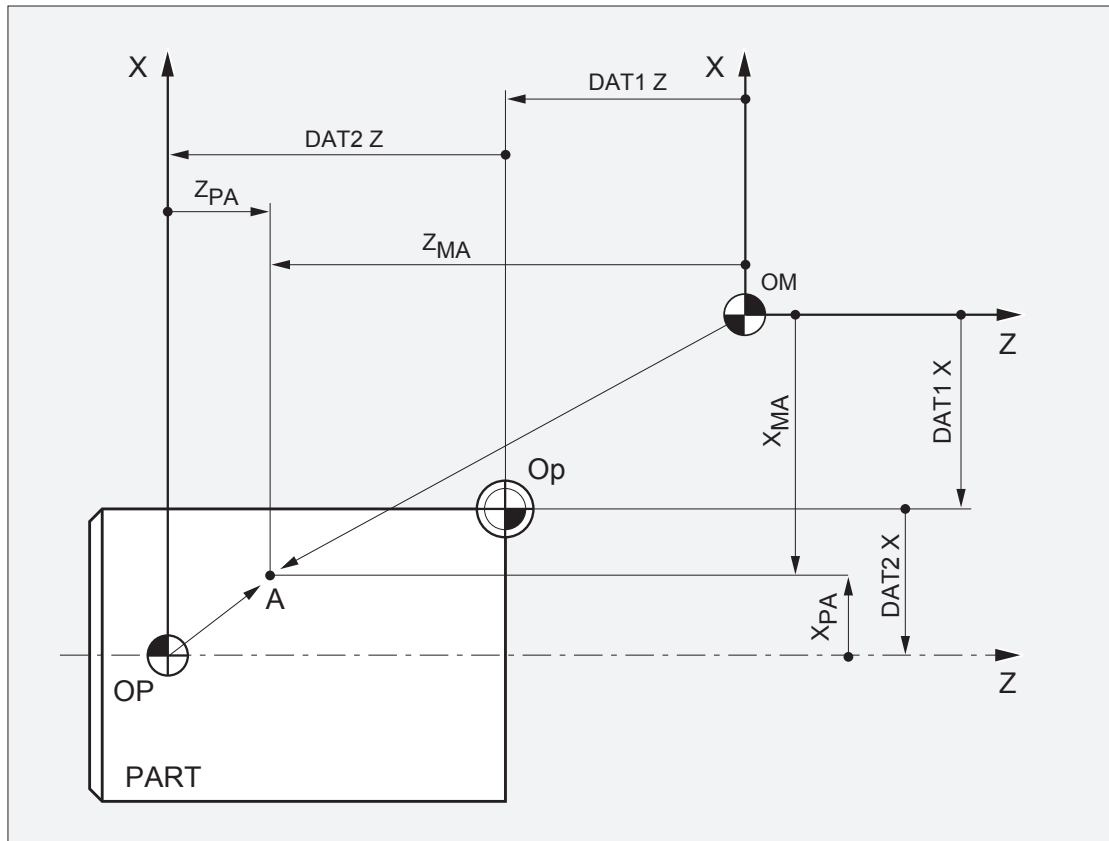
14.7.4 Offsets on the X Axis (Solution 2 without DAT2)

DAT1 X: Fixed value measured between OM and the spindle axis.



14.7.5 Schematic Representation of the Offsets on X and Z

The coordinates of a point (A) defined with respect to the program origin (OP) are converted by the CNC to coordinates with respect to the measurement origin (OM):

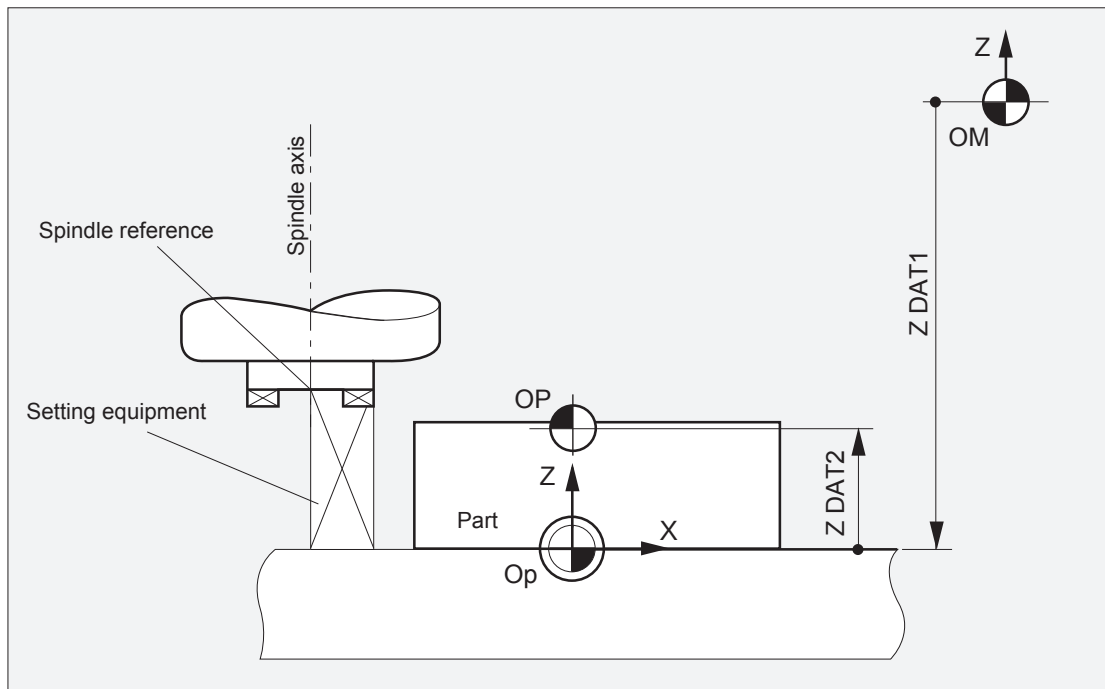


Program dimensions (with respect to OP)	Measurement dimensions (with respect to OM)
X_{PA}	$X_{MA} = X_{PA} + DAT1 X + DAT2 X$
Z_{PA}	$Z_{MA} = Z_{PA} + DAT1 Z + DAT2 Z$

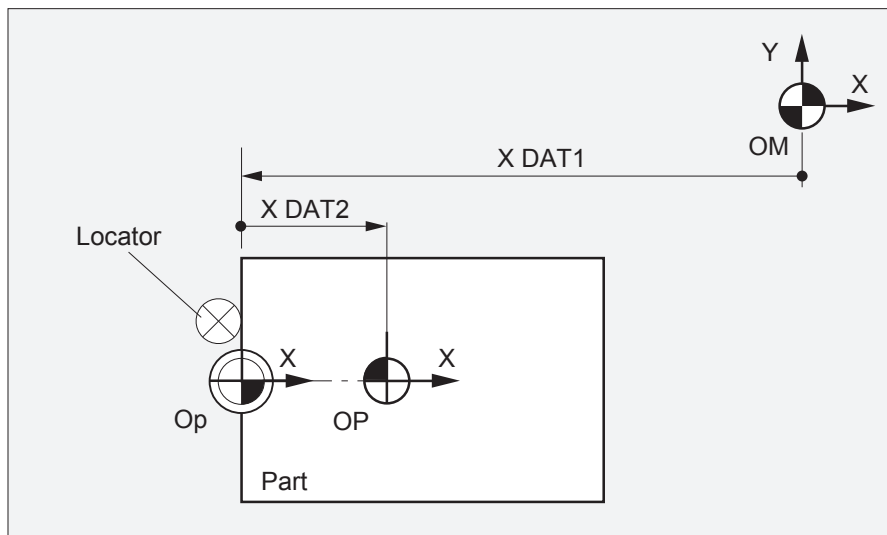
The dimensions are algebraic values.
Programmed shifts can be added to the measurement dimensions.

14.8 Milling machine - Offsets Represented on the Z, X and Y Axes

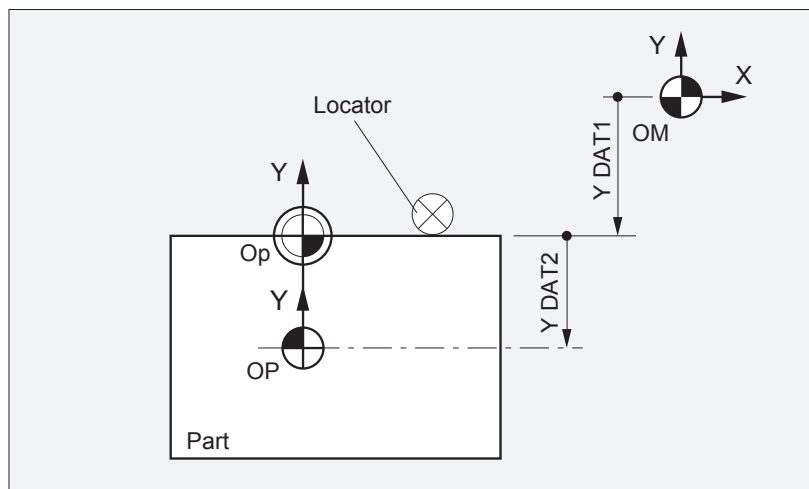
14.8.1 Offsets on the Z axis



14.8.2 Offsets on the X axis

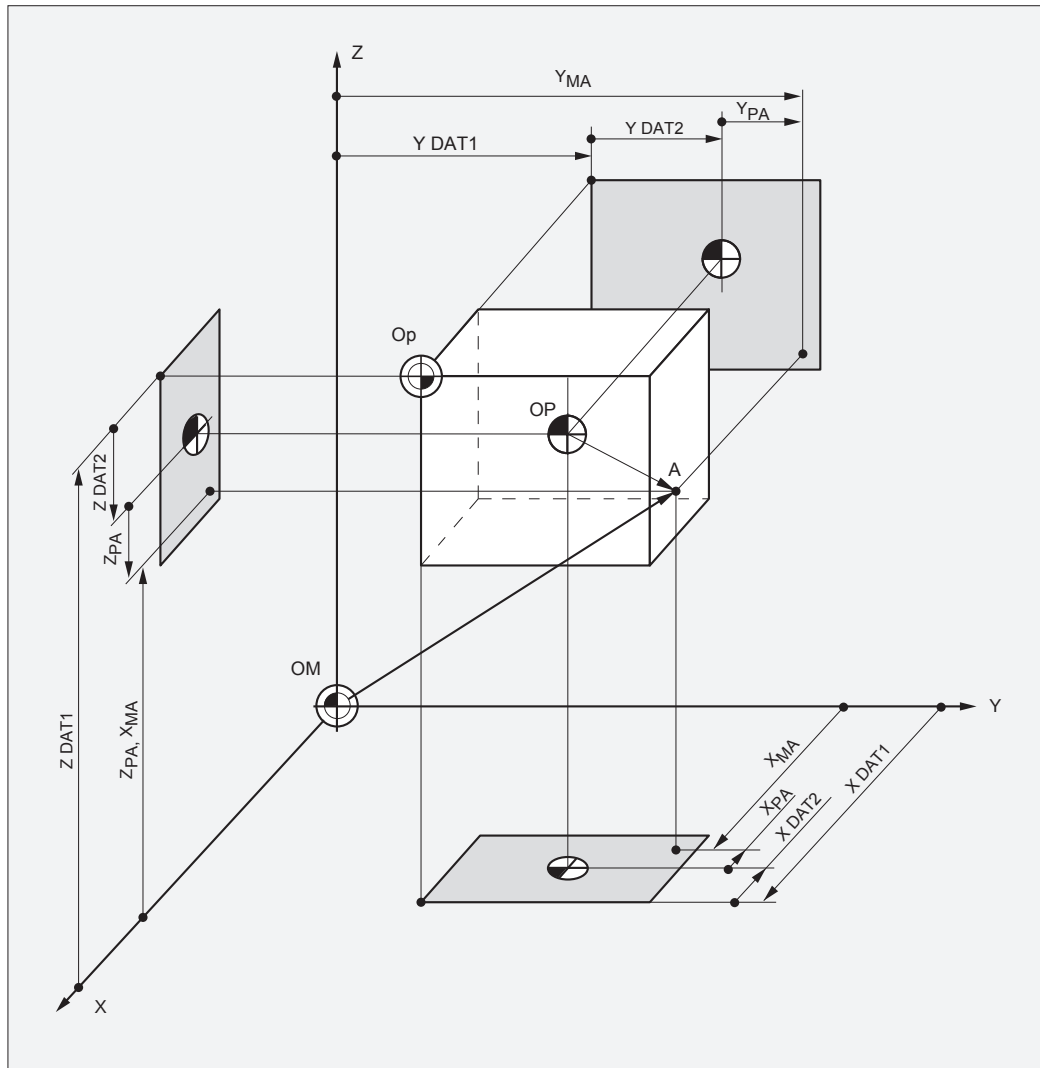


14.8.3 Offsets on the Y axis



14.8.4 Schematic Representation of All Offsets

The coordinates of a point (A) defined with respect to the program origin (OP) are converted by the CNC to coordinates with respect to the measurement origin (OM):



Program dimensions (with respect to OP)	Measurement dimensions (with respect to OM)
X_{PA}	$X_{MA} = X_{PA} + X\text{ DAT1} + X\text{ DAT2}$
Y_{PA}	$Y_{MA} = Y_{PA} + Y\text{ DAT1} + Y\text{ DAT2}$
Z_{PA}	$Z_{MA} = Z_{PA} + Z\text{ DAT1} + Z\text{ DAT2}$

The dimensions are algebraic values.
Programmed shifts can be added to the measurement dimensions.

14.8.5 Special Case of Milling Machines Equipped with Rotary Tables

The concept of part origin does not apply to the two axes affected by the rotation.
The centre of rotation of the table (OC) has a special role.

Offset of the centre of rotation (OC/OM) = DAT1
(axes affected by rotation)

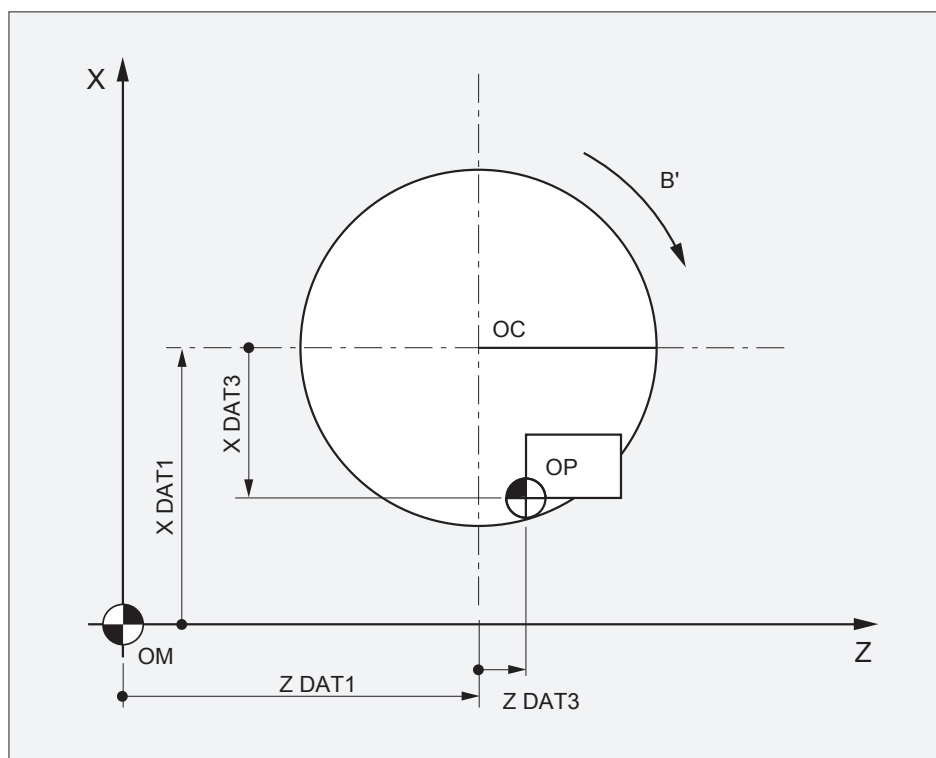
Offset of the part (OP/OC) = DAT3
(axes affected by rotation)



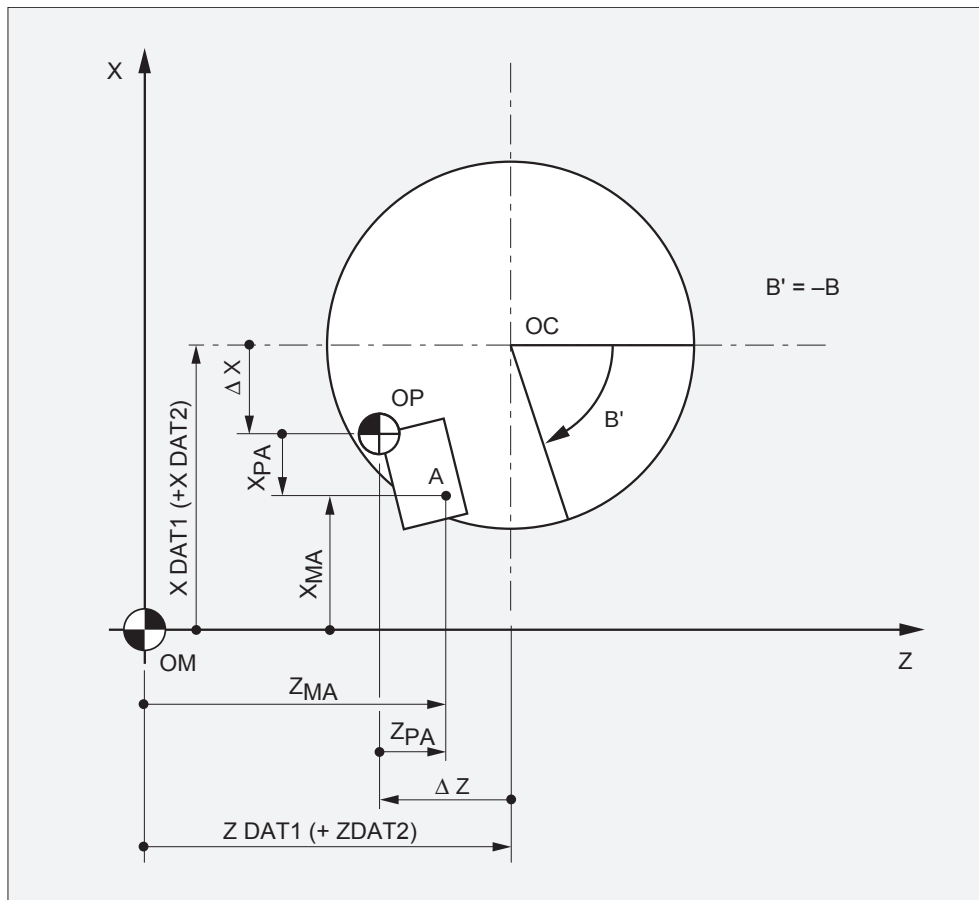
For axes other than those affected by rotation, the original definitions of DAT1 and DAT2 still apply.

Example: rotary axis B

Rotation is around an axis parallel to the Y axis. Therefore, the axes affected by the rotation are Z and X.



The coordinates of a point (A) defined with respect to the program origin (OP) are converted by the CNC to coordinates with respect to the measurement origin (OM):

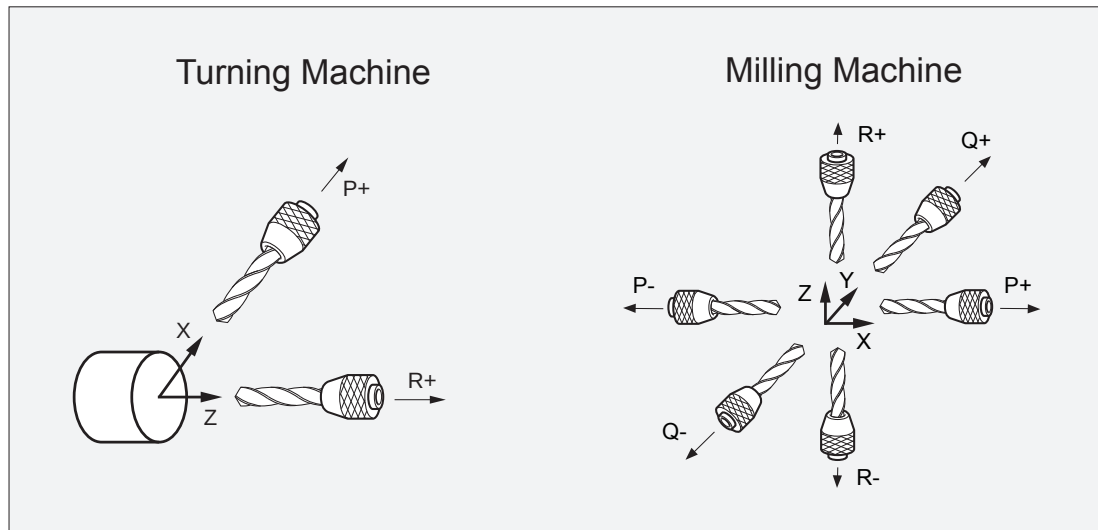


Program dimensions (with respect to OP)	Measurement dimensions (with respect to OM)
X_{PA}	$X_{MA} = X_{PA} + X \text{ DAT1 (+ X DAT2)} + \Delta X$ with $\Delta X = X \text{ DAT3} \times \cos B - Z \text{ DAT3} \times \sin B$
Y_{PA}	$Y_{MA} = Y_{PA} + Y \text{ DAT1 + Y DAT2}$
Z_{PA}	$Z_{MA} = Z_{PA} + Z \text{ DAT1 (+ Z DAT2)} + \Delta Z$ with $\Delta Z = Z \text{ DAT3} \times \cos B + X \text{ DAT3} \times \sin B$

14.9 Tool Axis Orientation

The default orientation of the tool axis on a standard turning or milling machine is set by the G16 R+ function at machine initialisation.

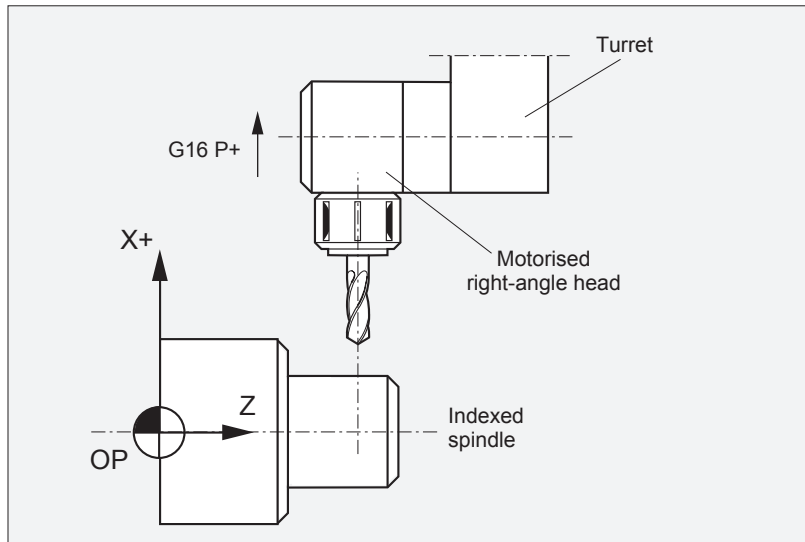
To perform radial machining when turning or milling (on X+), it is necessary to set a new tool axis orientation by programming function G16 P+. (Refer to the figures below and the complete description of the function G16 ... given in the programming manual).



14.9.1 Using the G16 P+ function on turning or milling machine

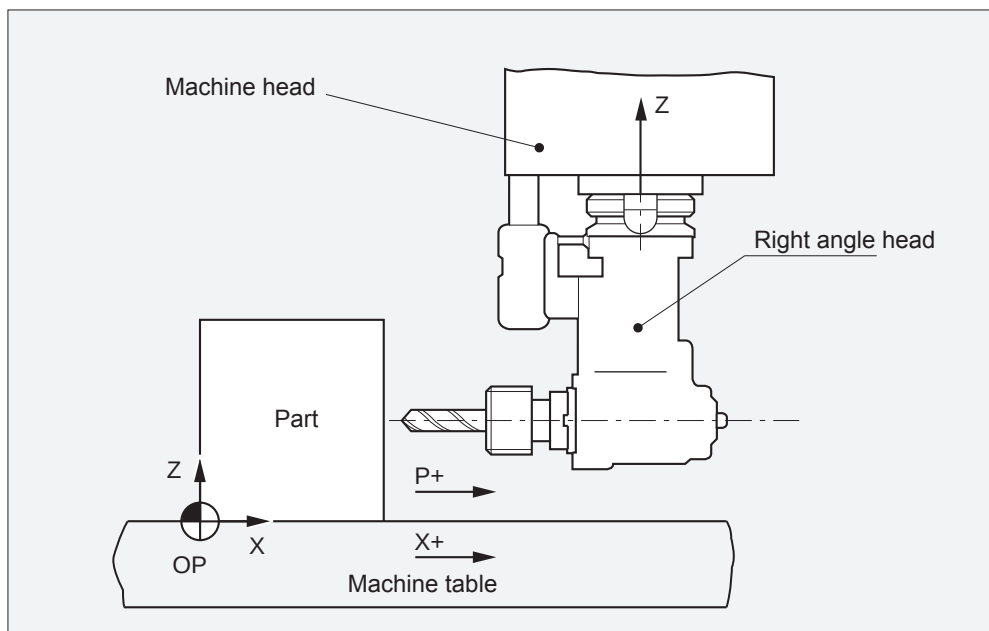
Use on Turning Machine

Machine equipped with a right-angle head (X+ orientation).



Use on Milling Machine

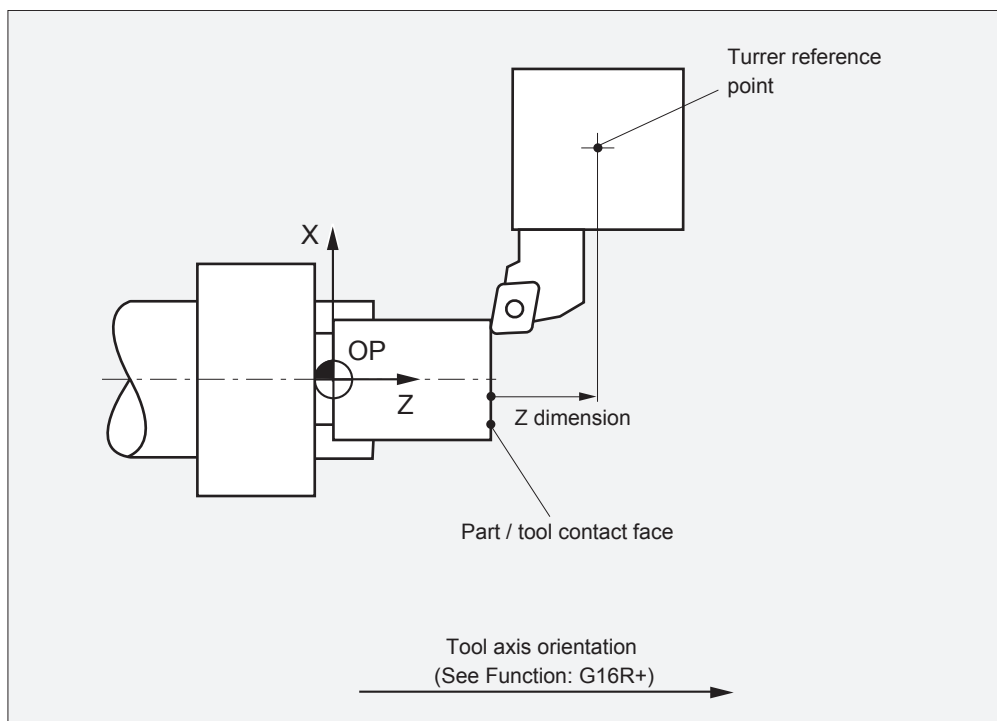
Machine equipped with a right-angle head (X+ orientation).



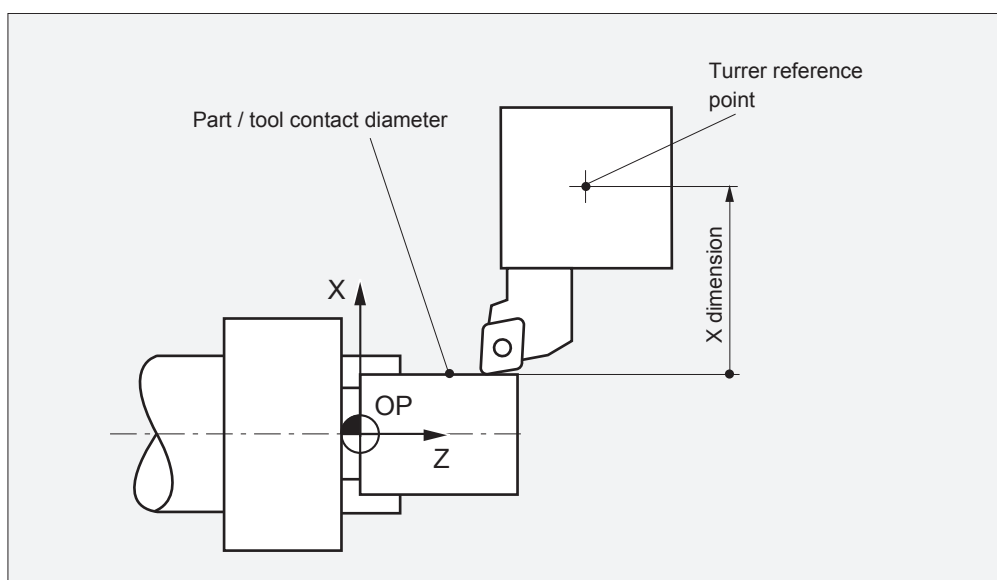
14.10 Definition of Tool Dimensions and Offsets

14.10.1 Turning Machine - Dimensions, Tool Tip Radius and Orientation

Tool dimension = Distance between tool cutting edge and turret reference point



Tool X dimension = X
Tool Z dimension = Z

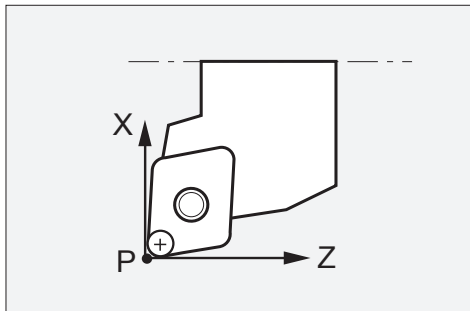


14.10.2 Tool Tip Radius and Orientation

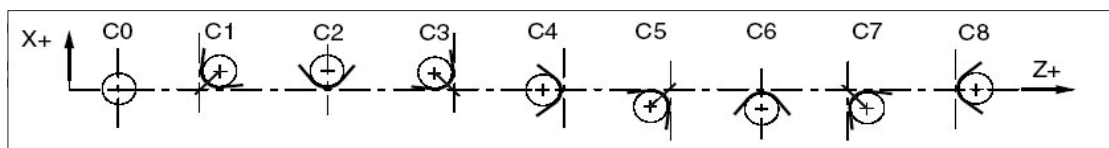
Tool tip orientation = code C0 to C8

The tool tip orientation code C0 to C8 allows the system to identify the centre (C) of the tool cutting edge from the theoretical cutting point (P).

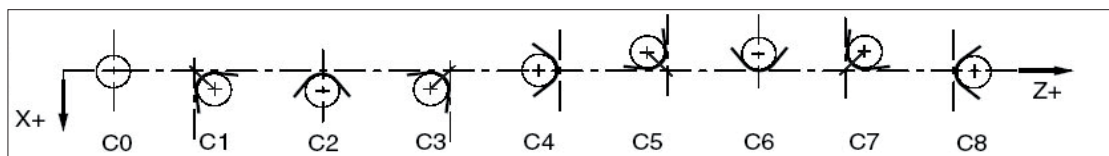
Example:



Tool Tip Orientation for a Rear Turret

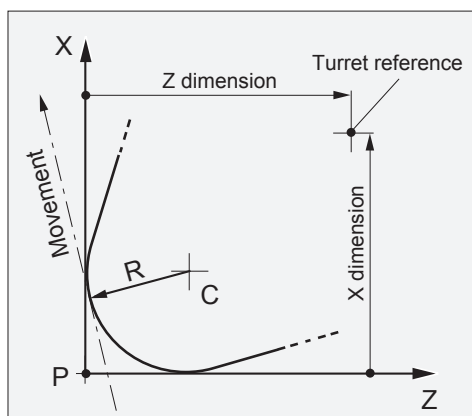


Tool Tip Orientation for a Front Turret



Tool cutting edge radius = R

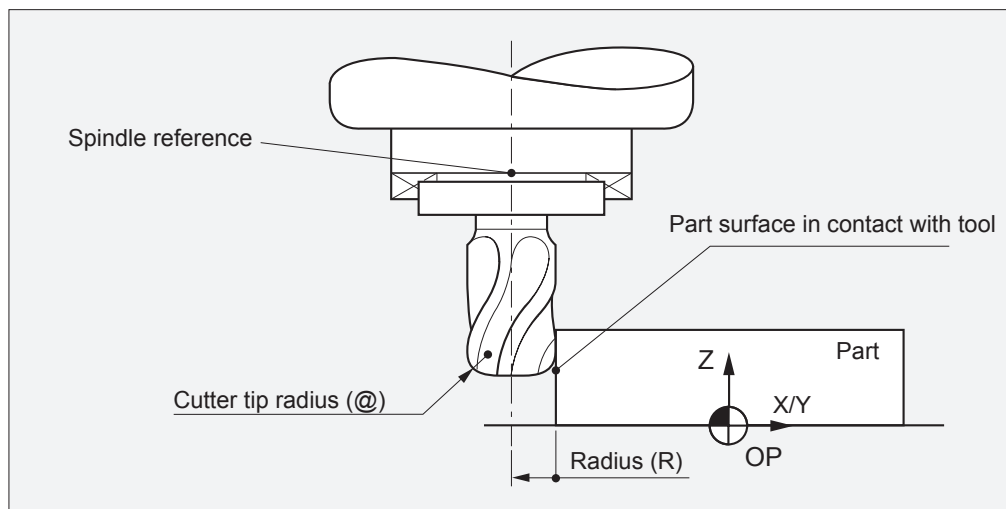
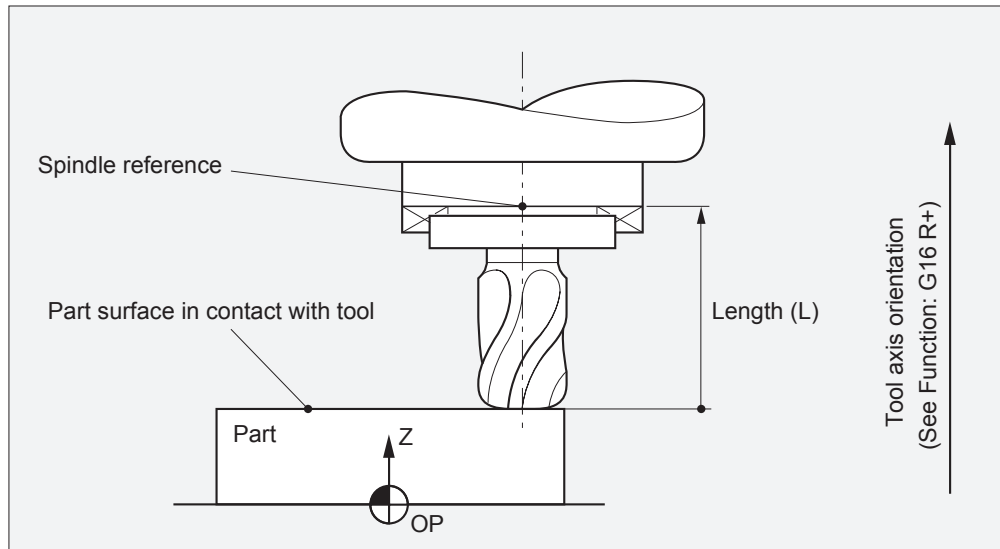
The real cutting point of the tool is obtained by applying a vector from centre "C" with length R perpendicular to the direction of movement.



14.10.3 Milling Machine - Definition of Tool Dimensions

Tool dimensions are defined as follows:

Tool dimension = distance from tool cutting edge to spindle reference point



Tool radius = R
Tool length = L

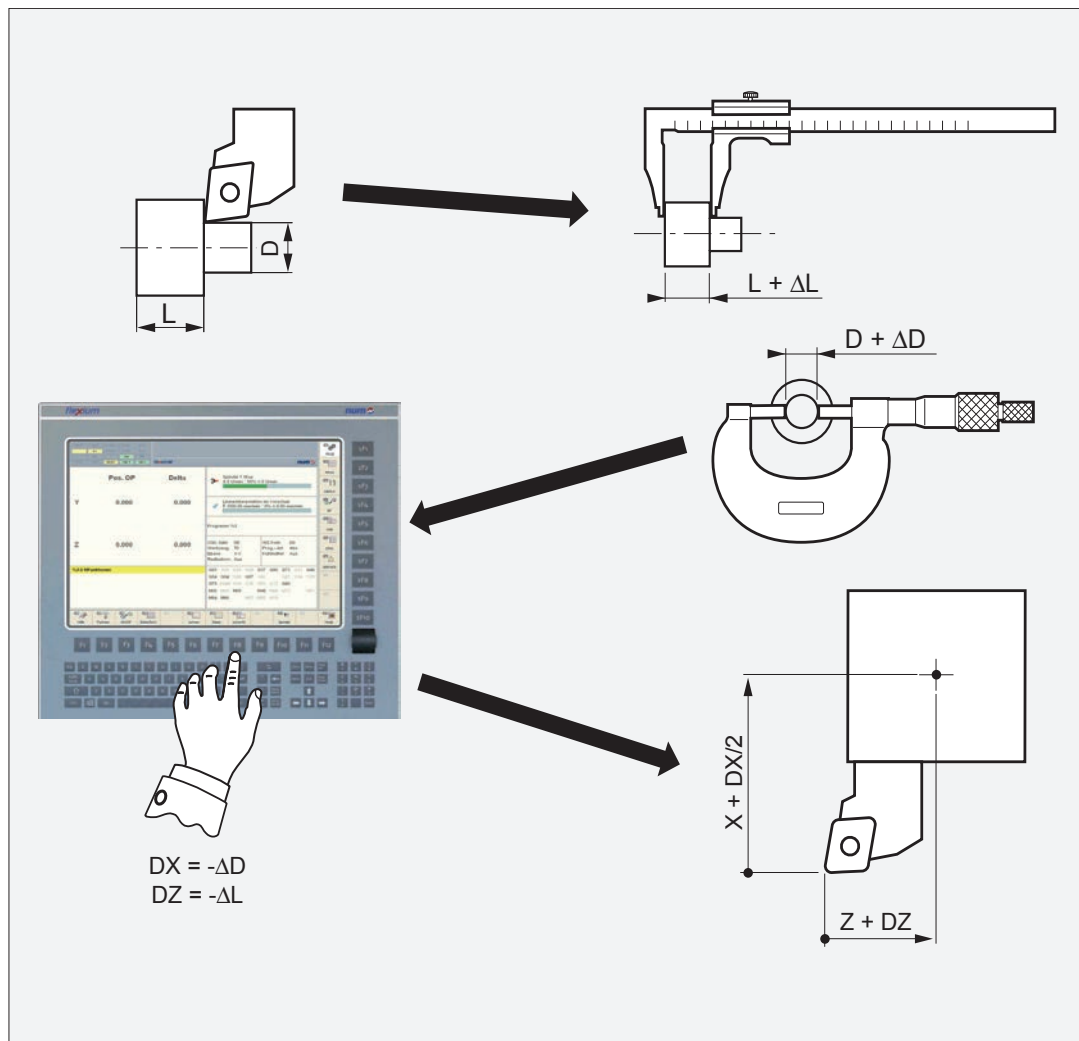
Cutter tip radius = @

14.11 Definition of Tool Wear

At any time (even during machining), the operator can enter tool wear offsets when he observes a difference between the expected and the actual dimensions on a part. These offsets (positive or negative) compensate for slight variations of the tool or part (wear, expansion).

14.11.1 Turning Machine

Tool wear offset on X = DX (by diameter)
Tool wear offset on Z = DZ

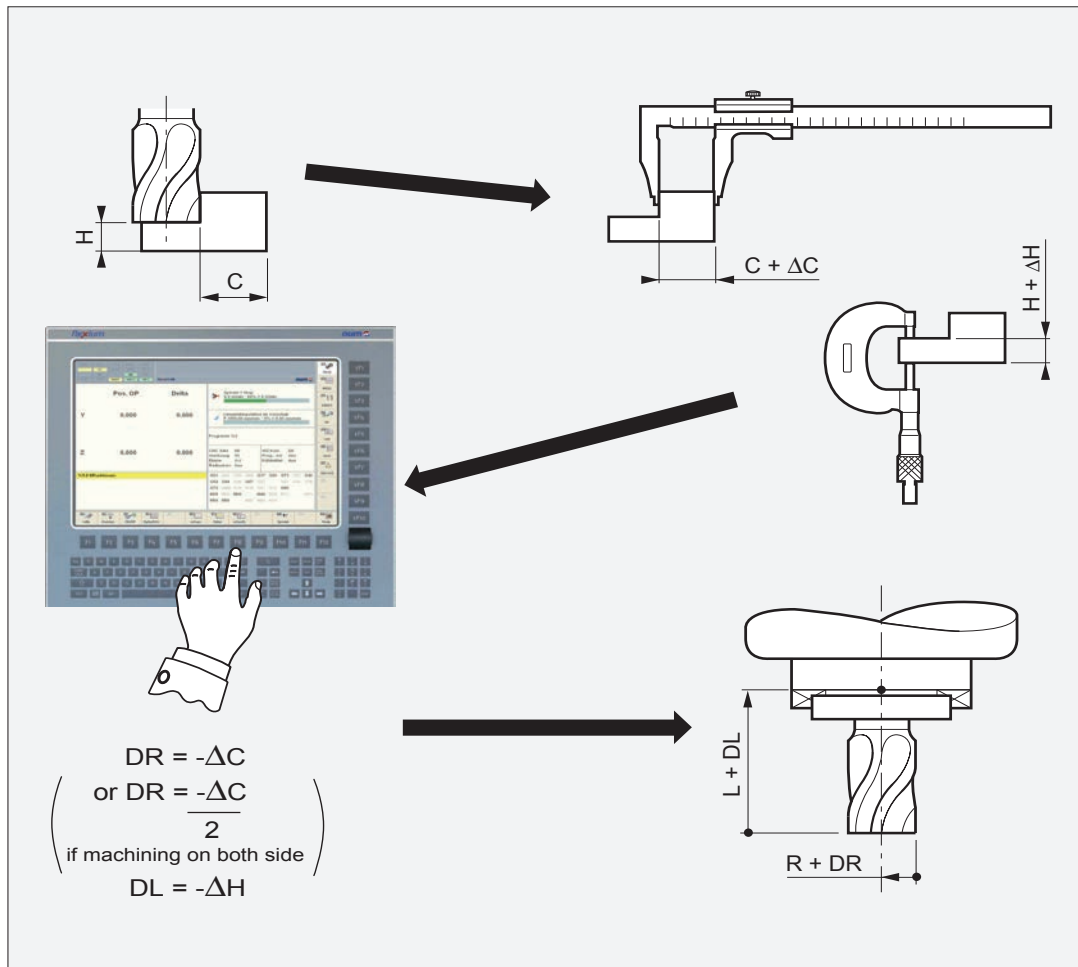


The system uses these offsets to correct the tool dimensions:

Corrected length on X = X dimension + DX/2
Corrected length on Z = Z dimension + DZ

14.11.2 Turning Machine

Tool radius offset = DR
Tool length offset = DL



The system uses these offsets to correct the tool dimensions:

Corrected radius = $R + DR$
Corrected length = $L + DL$

Page deliberately empty.

Summary

15	CNC Operation	201
15.1	Declaration of Axis Assignment	201
15.2	Execution of a Program	202
15.2.1	Program Test.....	202
15.3	Execution on the machine	203
15.3.1	Optional Stop and Block Skip Enable	203
15.3.2	Program Execution - General	203
15.3.3	Execution of a Program Step by Step.....	205
15.3.4	High Speed Execution of a Program	206
15.3.5	Execution of a Program in Automatic Mode	206
15.4	Operator Interventions	207
15.4.1	Unplanned Interventions.....	207
15.4.2	Interruption of Machining	207
15.4.3	Retracting the Axes	208
15.4.4	Manual Axis Recall	208
15.4.5	Automatic Axis Recall.....	209
15.4.6	Sequence Number Search	211
15.4.7	Extended possibilities in Search mode.....	213
15.4.8	Backtrack along Path.....	214
15.4.9	Return along Path	217
15.4.10	Emergency Retraction	218
15.4.11	Program Resumption after a Following Error	218
15.5	Programmed Interventions	219
15.5.1	Machining Stop Function M00 (or M01 if enabled)	219
15.5.2	Forcing an intervention by M12.....	219

Page deliberately empty.

15 CNC Operation

15.1 Declaration of Axis Assignment

Axis assignment is used to modify the $X_m Y_m Z_m$ physical reference system of the machine with respect to the $X_p Y_p Z_p$ part program reference system.

The part program reference system is then adapted to the workpiece orientation on the milling machine. Axis assignment can be positive or negative.

Axis assignment can be set by entering the values directly or in a program by external parameter E69003.

After each program axis (X, Y, Z), the function P, Q, R identifies the corresponding machine axis (X, Y, Z). The plus or minus sign indicates whether assignment is positive or negative.

The default values are $XP+YQ+ZR+$.

Example:

Axis assignment: $XR-YQ+ZP-$

$XR-$: program axis X assigned to machine axis Z with the opposite sign

$YQ+$: program axis Y unchanged

$ZP-$: program axis Z assigned to machine axis X with the opposite sign.

Assignment is refused if the command format is incorrect or the channel to which it applies is not in reset state. If the command is accepted, the tool direction (G16 ...) takes the assignment of program axis Z.

It should be noted that axis assignment:

- reorients the programmed dimensions, the interpolation plane (possibly with reversal of the direction of rotation of circles), the PQR material vectors, the coefficients of polynomial functions, the 2D tool offsets (possibly with reversal of G41/G42) and the tool direction (G16...)
- is applied downstream of programmed geometric transformations such as angular offset (ED...), programmed offset (G59), mirroring (G51...) and scaling factor (G74),
- is not applied to the dimensions programmed with respect to the measurement origin (G52...)
- does not affect offsets DAT1, DAT2 and DAT3.

15.2 Execution of a Program

15.2.1 Program Test

The mode “Test” is used to check a part program in the operating conditions of the machine and system. The system analyses and executes the program without moving the axes, which is far quicker than real machining.

The test checks:

- the syntax,
- the compatibility of the programmed instructions with the machine data accessible to the CNC (e.g. movements within limit travels including application of tool corrections, axes to be moved declared in the system, etc.)..

Functions M00 and M01 are not taken into account during the test.

Requirements
Measurement origin setting performed on the axes included in the program. No mode being executed. If a mode is being executed, selection of the test mode is only taken into account after an interruptible block. Program selected as active program.
Start program execution by pressing the “START” button on the machine panel
Display of “START” in the status window.

15.3 Execution on the machine

15.3.1 Optional Stop and Block Skip Enable

A part program may contain functions which, when enabled, modify its execution (functions for debugging for example). These include:

- optional stops (M01),
- block skips ("/" at start of block).

When optional stops are enabled by the operator, all "M01" functions encountered in the program are treated as "M00" functions (See Programming manual). Otherwise, the "M01" functions are ignored.

When block skips are enabled by the operator, all blocks preceded by "/" are ignored. Otherwise, the blocks are executed normally.

The optional stop (M01) and the block skip (/) in the status window indicate that the functions are enabled.

15.3.2 Program Execution - General

A part program can be executed in three modes:

- single step,
- dry run or rapid,
- automatic.

This section describes the functions common to all three modes.

Display of program execution

During program execution, all the display pages are accessible.

Temporary stops during program execution

Program execution can be restarted (in the mode active before the stop or in another mode) from where it was interrupted.

Feed interruption

Turn down the feed rate potentiometer on the machine panel to zero.

The "START" indicator remains displayed in the status window.
Feed stop performed, current mode not cancelled.

Perform the desired operations or checks.
Restart movement by turning the feed rate potentiometer back up.

Mode change
The most frequent cases when the operator wishes to change the current mode are to: - continue to execute machining in single step mode "Single", - execute instructions outside the program in manual data input mode "MDI".
Select another mode.
At the end of an interruptible block, the system cancels the current execution mode. The "START" indicator disappears from the status window, the newly selected mode replaces the previous mode.
Perform the desired operations.
If needed, return to the initial mode.
Resume execution by pressing the "START" button on the machine panel.

Feed stop using the "STOP" button
Movements can be interrupted immediately using the "STOP" button on the machine panel.

Programmed stop
Reading function M00 (or M01 if enabled) interrupts program execution and requires operator intervention.

Forcing of recall mode
The recall mode is forced when function M12 is read during program execution and requires operator intervention.

15.3.3 Execution of a Program Step by Step

A program can be executed step by step (blocks or parts of blocks) in single step mode (Single) to check that the machining procedure is being performed correctly.

Requirements	
Measurement origin setting performed on the axes included in the program. No mode being executed. If a mode is being executed, selection of the single step mode is only taken into account after an interruptible block. Program selected as active program.	
Select the mode "Single".	
Start program execution by pressing the "START" button on the machine panel.	
Display of "START" in the status window.	
Execution of the block or part of block.	
	If the function M997 is programmed, the system operates as though it were in automatic mode and sequences blocks until it encounters an M998, M999 or M02 function.
	At the end of the block, the "START" indicator disappears.
Start execution of the next block by pressing the "START" button.	

15.3.4 High Speed Execution of a Program

A program can be executed at high speed (maximum speed authorised by the system) in order to check movements, tool changes, etc.

High speed execution can be performed with the part in place, in which case a modification of the work offsets (DAT1 or DAT2) is required to complete the paths outside the part.

Requirements
Measurement origin setting performed on the axes included in the program. No mode being executed. If a mode is being executed, selection of the rapid mode is only taken into account after an interruptible block. Program selected as active program.
Select the mode "Rapid".
Start program execution by pressing the "START" button on the machine panel.
Display of "START" in the status window.
Execution of the program at high speed.
At the end of program, the "START" indicator disappears.
Return to the old values of work offsets DAT1 or DAT 2.

15.3.5 Execution of a Program in Automatic Mode

When machining parts in automatic mode:

- the program is executed from the beginning (%...) up to function M02,
- programmed speeds are applied.

Requirements
Measurement origin setting performed on the axes included in the program. No mode being executed. If a mode is being executed, selection of the automatic mode is only taken into account after an interruptible block. Program selected as active program.
Select the mode "Auto".
Start program execution by pressing the "START" button on the machine panel.
Display of "START" in the status window.
Execution of the machining procedure.
At the end of program, the "START" indicator disappears.

15.4 Operator Interventions

During execution of a program, two types of interventions may be required of the operator:

- unplanned intervention when a machining incident has occurred, or
- programmed intervention when a manual operation is required for the program to continue.

15.4.1 Unplanned Interventions

Unplanned interventions are initiated by the operator. Certain of these interventions for which the procedures are described herein require having corresponding features present in the system and a machine panel with pushbuttons to activate them.

By convention, the pushbuttons are considered as existing on the machine panel, but their actual locations (and names) on the machine are left up to the machine manufacturer (see manufacturers data).

15.4.2 Interruption of Machining

When machining is interrupted, all the movements on all the axes are stopped.

Press the "STOP" button on the machine panel.
Display of "STOP" in the status window. The "START" indicator remains displayed.
Machining is interrupted immediately, but the spindle continues to rotate.
The axes can be retracted (see 15.4.3).
Restart machining by pressing the "START" button on the machine panel.

15.4.3 Retracting the Axes

Axis retraction is related to interruption of machining (see 15.4.2).
The axes are retracted manually by moving them with the jogs or handwheels.

Requirements
Machining interrupted ("STOP" indicator lit).
Press the "AXRCL" button on the machine panel.
"INTER" indicator flashing in the status window. The axis jogs or the handwheels on the machine panel are enabled.
Retract the axes with the jogs or handwheels.
In the current position pages, the "DELTA" column displays the distance from the stop point.
Perform the operations required.
There are two ways of returning the axes to their starting position: - Manually (see 15.4.4), - Automatically (see 15.4.5).
Restart machining by pressing the "START" button on the machine panel.

15.4.4 Manual Axis Recall

The axis recall function supplements interruption of machining.
It is used to return the axes to their positions prior to retraction using the axis jogs or handwheels.

Requirements
Machining interrupted on the system; axes moved ("INTER" indicator flashing).
Press the "AXRCL" button on the machine panel.
"AXRCL" indicator flashing in the status window.
The only movements authorised are those which return the axes to their starting position. This is displayed in the "DELTA" column of the current position pages.
Bring the axes back to their initial position using jogs or handwheels.
Move the axes to the position they occupied when machining was interrupted ("DELTA" = 0).
Select the desired mode and continue machining by pressing "START".

Exit from the procedure
Once the axis recall function has been selected, it is possible to return to jog operations.
Press the "AXRCL" button on the machine panel again.
"INTER" indicator flashing in the status window.
The axes can be moved in both directions again.

15.4.5 Automatic Axis Recall

Automatic axis recall is related to interruption of machining.

When machining is interrupted, this function allows the axes to be returned automatically to their starting position along the same path as was used to retract them by the jogs, since the path was stored (see figure under Notes).



Automatic axis recall can be combined with the functions "Backtrack along path" (see 15.4.7) and "Return along path" (see 15.4.8).

Requirements
Machining interrupted on the system; axes retracted ("INTER" indicator flashing).
Press the "Automatic Axis Recall" button on the machine panel (hold depressed).
The axes are automatically returned to the position they occupied when machining was interrupted (see Speed of movement and Approach distance "d" under Notes).
In the current position pages, the "DELTA" column shows the distance from the stop point decreasing until it reaches zero.

Notes on Automatic Axis Recall

Axis recall after retraction by handwheel
Caution: After retraction of the axes by the handwheels, automatic axis recall is enabled, but in this case, the axes may be recalled in any order. It is therefore not recommended to use this mode if the number of points stored in word N2 of machine parameter P114 is different from zero (see Parameter Manual).
If the number of points is equal to zero, the axes are recalled by linear interpolation.

Number of retraction points stored

The number of retraction points that can be stored (maximum 10) is declared in word N2 of machine parameter P114 (see Parameter Manual).

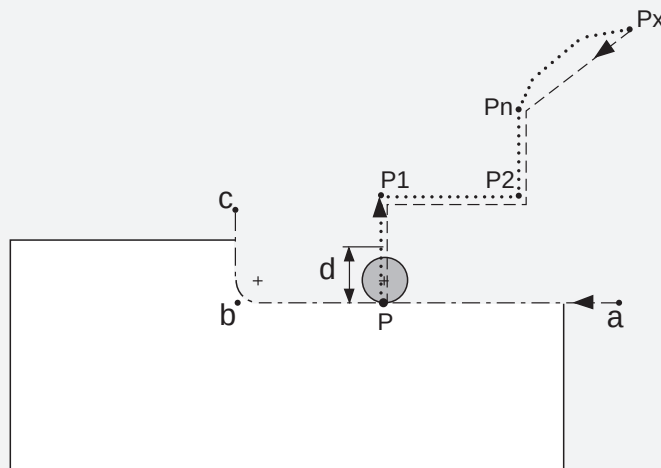


When the number of retraction steps exceeds the number of points that can be stored, the axes are recalled by linear interpolation up to the last point stored, then paraxially on each axis up to the starting point.

Speed of movement and approach distance “d”

Automatic axis recall takes place at the specified jog rate. However, approach of the recall point at the work rate can be specified at an approach distance “d” declared in word N3 of machine parameter P114 (see Parameter Manual).

When the axes reach the point at distance “d” from the point where work is to be resumed, the feed rate applied is that programmed in the current block.



a, b, c : Programmed tool path
P: Tool stop point (machining interrupted)
P1 to Pn... : Retraction path (maximum 10 points)
Pn to Px : Retraction path not stored
Pn : Last point stored

Automatic axis recall

Px to Pn ... : Recall linear up to last stored point Pn,
Pn to P : Return to restart point P (see d)
d : Approach distance.

15.4.6 Sequence Number Search

The sequence number search mode "Search" is used to start execution of a program from a given block.

This mode can be used to restart machining which has been interrupted for emergency retraction once the incident has been sorted out.

Requirements	
Measurement origin setting performed on the axes included in the program. No mode being executed. If a mode is being executed, selection of the search mode is only taken into account after an interruptible block. Program selected as active program.	
Select the mode "Search"	
Display of "SEARCH" in the status window.	
Enter the coordinates of the block to be searched (see syntax)	
Syntax (repeat as many times as there are axes channels)	
N[block No.].[channel No.] H[S/R No.]; [No. of times]	
N [block No.]	Number of the block to be selected (belonging to the main program or the specified subroutine, this item is mandatory)
.[channel No.]	Axes channel number; this item must be specified for each of the axes channels when using a multichannel system
H[S/R No.]	Subroutine number; this term is not used unless the block searched for belongs to a subroutine
; [No. of times]	Number of times the block number is to be read before stopping the search.
Start the search by pressing the "START" button on the machine panel.	
Display of "START" indicator in the status window.	
Analysis of blocks from the start of the program: - store values, - coded M functions (list supplied by the OEM) sent to the PLC, - decoded M functions and T functions (tools) sent to the PLC if allowed by the configuration (function of parameter P7 defined by the OEM).	
Stop on the block preceding the block to be found.	
The "START" indicator disappears.	
Press the "START" button on the machine panel.	
Display of "START" indicator in the status window.	
Execution by the system of the auxiliary functions (decoded M functions, S and T functions) stored during the search.	
The "START" indicator disappears.	
Display of the "STOP" and "AXRCL" indicators in the status window if the axes are not in position.	

Move the axes into position (see 15.4.4).
Press the "START" button on the machine panel.
The "STOP" indicator is cleared from the status window.
Select one of the modes automatic, single, rapid or test and continue program execution.

Notes
Searches for unnumbered blocks cannot be performed. In such cases, search for the preceding numbered block.

Exit from the procedure
The search can be interrupted by pressing "RESET".

15.4.7 Extended possibilities in Search mode

From the Flexium HMI version 3.3.0.0 on, two new types of search have been added.

This means that three types of search are accepted:

- **Standard search by the sequence number**
Syntax: *N[block No].[Channel No] H[S/R No];[No of times]*
- **String search**
Syntax: *"string to search".[Channel No] H[S/R No];[No of times]*
- **Line number search**
Syntax: *:[Line No].[Channel No] H[S/R No];[No of times]*

With:

N[block No] : The block number is preceded by the character N

"String to search" : The string must be between two quotes ("). Tabs and spaces are ignored. The search is case insensitive. As the size of the search command is limited to 114 characters, the length of the "string to search" depends on the number of CNC channels, the presence of spaces in the command and the presence of the fields [Channel No], H[S/R No], ;[No. of Times]

[Line No] : The line number is preceded by the character (:).



On a multi-channel system, it is possible to define the search type for each channel. For example, it is possible to perform a search by sequence number for the first channel, a search by line number for the second channel and a string search for a third channel.

The search can be done in the main program or in a subroutine, in an embedded part program or in drip feed mode. The ;[No. of times] is linked to main program or subroutine.

Examples

N100 H200;4

Search block number N100 in subroutine %200 of a mono channel system.
Search completed at the 4th occurrence in %200.

"T07 D125 M06" H345;10

Search string T07 D125 M06 in subroutine %345 of a mono channel system.
Search completed at the 10th occurrence in %345.

:456778

Search line number 456778 in the main part program of a mono channel system.

:145000.1 H125;3 "S3000".2 H10;5 N300.3;4

Search by line number in CH1; by string in CH2; and by block number in CH3 with a 3 channel CNC System (in common mode).

15.4.8 Backtrack along Path

After an interruption in machining (see 15.4.2), the Backtrack along path function is used to repeat the programmed tool path blocks in the reverse direction.

When the backtrack point is reached, the user can return to the initial position by the Return along path function (see 15.4.8).

Requirements
Machining interrupted
If necessary, set a tool/workpiece offset (see Tool/Workpiece offset under Notes).
Press the "Backtrack along path" button on the machine panel (hold depressed).
Backtrack takes place in the mode selected (see Movement mode under Notes).
In the current position page, the "DELTA" column displays the distance moved by the axes.
Release the "Backtrack along path" button when the backtrack point is reached.
Backtrack is immediately stopped.
Perform the actions which made backtrack necessary.
The system allows a return to a selected position on the tool path to be made by the Return along path function (see 15.4.8) or programming to be resumed by cancelling the machining interruption (see Program resumption under Notes).

Exit from the procedure
If no Tool/Workpiece offset was programmed or this offset has been cancelled (see Tool/Workpiece offset under Notes), machining can be resumed at any time.
Release the "Backtrack" button.
The current backtrack is immediately stopped.
Press the "START" button to resume machining.
The "START" indicator comes on in the status window and program execution is resumed (see Movement mode under Notes for the speed of movement). During resumption, a new machining interruption can be requested by pressing "STOP".

Notes on Backtrack along Path and Return along Path

Backtrack (and return) along path are possible in automatic (AUTO), single step (SINGLE) or dryrun (Rapid) mode:

- In automatic mode, the blocks are sequenced automatically as long as the operator holds the "Backtrack" or "Return" button depressed, at the feed rate declared in the block being executed.
- In single step mode, movement is continuous until the beginning of the current block (movement is stopped if the operator releases the "Backtrack" or "Return" button). To backtrack (or return) to the beginning of the next block, it is necessary to press "Backtrack" or "Return" again. Movement is at the speed declared in the current block.
- Execution in dryrun or rapid mode is the same as in automatic mode except that movement is at high speed.



A change of mode (automatic, single step or rapid) is always possible at the end of a block. In single step mode, pressing "Backtrack" or "Return" at the end of a block forces a cycle start. When resuming the block interrupted by "STOP", the initial mode is restored.

Tool/Workpiece offset

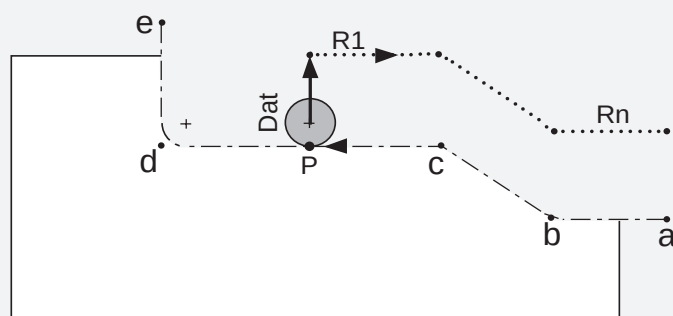
An offset of the tool with respect to the workpiece can be applied to backtrack or return at any time (with machining interrupted and "STOP" indicator lit).

To shift the toolhead (for further information, see Sec. 15.4.3, Retracting the Axes), press the "AXRCL" button on the machine panel (the "AXRCL" indicator is lit in the status window) and retract the toolhead using the axis jogs.

This procedure is also used to change or cancel the offset.



An offset cannot be applied, edited or cancelled unless the "Backtrack" or "Return" button is released.



a, b, c, d, e : Programmed tool path
 P: Tool stop point (machining interrupted)
 Dat : Tool/workpiece datum shift (optional)
 R1 to Rn... : Backtrack along path.

Resumption of program execution

If no tool/workpiece offset is applied (or if the offset has been cancelled), machining can be resumed upstream ("Backtrack" and "Return" buttons released), i.e. the machining interruption can be cancelled (the "STOP" indicator in the status window goes out) and program execution can be resumed in the current mode by pressing "START".

During resumption, a new machining interruption can be requested by pressing "STOP".

Number of blocks stored during backtrack along path

The number of blocks (maximum 100) on which backtrack can be executed is declared in word N1 of machine parameter P114 (see Parameter Manual).

Restriction on use of backtrack along path

It should be noted that:

- Only the functions related to axis movements are stored (G0, G1, G2, etc.) for backtrack, but not the other functions (M, T, S, etc.).
- The user cannot (even if the storage capacity allows) backtrack along the path beyond the point where the tool in use was called.
- The rigid tapping, rigid tap removal, inclined plane, RTCP and block skip functions cancel the remainder of the blocks stored and therefore stop execution of backtrack along path (the same is true for smooth polynomial interpolation)..
- Sequenced thread cutting (G38) is performed as a linear interpolation (G1) during backtrack and return along path or program resumption (see above).

Tool offsets

Dynamic tool offsets can be applied at any time to the current tool offset of the axes channel, provided they are less than 0.1 mm. This offset is applied immediately in the block being executed and over the complete stored backtrack path.

Backtrack/Return with multiple axes channels

With multichannel, backtrack and return are possible only for the first axes channel.

15.4.9 Return along Path

The return along path function is indissociable from the backtrack along path function (see 15.4.7). After a backtrack along path, the return along path returns the toolhead to the position it occupied when machining was interrupted.

Requirements
Machining interrupted, backtrack along path completed ("Backtrack" button released).
Cancel (immediately or later) the tool/workpiece offset (see Tool/Workpiece offset under Notes)..
Press the "Return" button on the machine panel (hold depressed).
The return along path takes place in the movement mode selected (see Movement mode under Notes).
In the current position page, the "DELTA" column displays the distance from the end point.
Release the "Return" button when the selected return position is reached.
Resume machining by pressing the "START" button on the machine panel.
The program continues.

Exit from the procedure
If no Tool/Workpiece offset was programmed or this offset has been cancelled (see Tool/Workpiece offset under Notes), machining can be resumed at any time.
Release the "Return" button.
The return movement is stopped immediately.
Press the "START" button to cancel the machining interruption.
The "START" indicator comes on in the status window and program execution is resumed (see Movement mode under Notes for the speed of movement). During resumption, a new machining interruption can be requested by pressing "STOP".

Notes on Return along Path
The notes for return along path are the same as for backtrack along path (see 15.4.7).

15.4.10 Emergency Retraction

The emergency retraction function allows movement to be interrupted so that the tool can be retracted from the part following an incident (e.g. broken tool). The emergency retraction function executes a pre-planned retraction procedure provided in the program being executed (see Programming manual).
Press the "Emergency Retract" button on the machine panel.
Immediate interruption of machining and retraction in accordance with the procedure defined in the part program.
The "STOP" indicator is displayed in the status window, the "START" indicator remains displayed.
The axis recall "AXRCL" indicator flashes in the status window.
In the current position pages, the "DELTA" column displays the distance from the stop point.
Perform the operations required.
Return the axes to their interruption position (see 15.4.3).
Restart the machining procedure by pressing the "START" button.

15.4.11 Program Resumption after a Following Error

When the system detects a following error (excessive lag) during axis movement, it generates one of the error messages 40 to 71 and interrupts execution of the program.

The function "Program resumption after an error" allows the following error detected to be cancelled and the program to be resumed. However, this function is only available for certain types of automatic controls.

Requirements
Program interrupted, fault indicator displayed in the status window.
Press the "Resume on error" button on the machine panel.
If the current mode is automatic (AUTO), single step (SINGLE), manual data input (MDI) or dryrun (RAPID), the system automatically goes into manual axis recall mode ("AXRCL" indicator flashing). If this is not the case, see "Restrictions" below.
Return (if necessary) the CNC and PLC axes along the programmed path using the jogs.
In the current position page, the "DELTA" column displays the distance from the end point.
Press the "START" button to cancel the program interruption.

Restrictions
If the system is not in automatic (AUTO), single step (SINGLE), manual data input (MDI) or dryrun (RAPID) mode, axis recall is impossible. In this case, the only way to resume the cycle is by a reset.
 The following error cannot be cancelled with certain applications which calculate references using dynamic operators.

15.5 Programmed Interventions

Programmed interventions are planned within a program to require operator intervention.

15.5.1 Machining Stop Function M00 (or M01 if enabled)

After program execution has been stopped by the function M00 (or M01 if enabled), the operator must perform the planned operation and then restart program execution.

Display of the "M00" indicator in place of the "START" indicator in the status window.

Perform the operation.
Restart program execution by pressing the "START" button.

15.5.2 Forcing an intervention by M12

After program execution has been stopped by the function M12, the operator must perform the planned operation and then restart program execution.

The "STOP" indicator is displayed in the status window, the "START" indicator remains displayed.
The axis jog controls on the machine panel are active, the spindle continues to rotate.
The operator must perform the required operation.
Perform the operation (including any jog operations).
Restart program execution by pressing the "START" button.

Page deliberately empty.

Index

Symbol

*.dat 55
 *.jau 55
 *.usi 55
 *.xdv 111
 *.xpc 55
 *.xpi 55
 .xsk 132

A

Absolute Dimensions 180
 Access switching 139
 Active Part Program 82
 Addresses 67
 alarm history 121
 approach distance 210
 area 140
 Customer area 140
 NUM area 140
 OEM area 140
 Automatic Mode 206
 Axes Positions Page 47, 48
 Axis 34
 axes information 42
 Axes information 45
 axes positions 42
 Axis Definition and Direction 177
 not in position 34
 Axis assignment 201
 Axis recall 34, 209
 AXRCL 34

B

Backtrack 214
 block 32
 display 32
 error 32
 Block skip 34
 Boring tool 95
 Box PC 141
 BOX PC Access 138
 BrkPnt 34

C

CAD/CAM 176
 Cartesian coordinate 177
 Change Tool Offsets 140
 Channel number 34, 35

Channel selection 139
 CNC address 34, 35
 CNC blocks 42
 CNC Blocks Page 46
 CNC cyles 140
 Edit 140
 CNC editor 153
 CNC error 34
 CNC memory 55, 153
 CNC memory information 56
 CNC memory zone 56
 CNC message 147
 CNC number 34
 CNC Operation 201
 CNC system 19
 CNC variables 113
 Compact part program 62
 Context 31, 91
 Diagnostic 121
 Production 41
 Programming 55
 Service 137
 Variables 111
 work area 31
 CopyExplorer 132
 Access 165
 Save 166
 Customer area 140
 Customer Tools Access 138
 Custom tools 143
 Cycle 34

D

DAT1 106
 data locations 55
 Date 144
 DbStep 34
 Debug step 34
 Delete 57
 Deleting Files
 NC Memory 86
 PC Disk 86
 Delta 42
 Device List 132
 Diagnostic 121
 alarm history 121
 configuration data 121
 exception record 121
 Diagnostic Settings 146
 Dialog field 32, 50

- Data input 50
- Directory 57
- Display 139
- Display unit 34
- DPLC Access 141
 - DPLC setting 141
- Drip feed mode 34
- Drip Feed Mode 75
 - Principle 75
- drip file display 56
- Drive parameters 158
- Drives parameters 155
- Drive warning 147
- duplicated axes 42

E

- Edit CNC cycles 140
- Editing 74
 - NUM CNC Cycles 74
 - Program directly in NC Memory 84
- Editor protection 139
- Encoders 180
- Exception error 34
- Exception file 139
- EXPErr 34

F

- FDHLD 34
- Feed interruption 34
- feed rate 36
 - Interpolation mode 36, 37
 - options 149
 - override graph bar 36, 37
 - Settings 148
 - state 36, 37
- file 153
 - NC file manager 153
 - PC file 153
 - USI file manager 153
- files existing 55
- Filters 125
 - selection window 125
- Folder 57
- Font 153
 - Editor 153
 - size 153
- Format 63
 - Compact part program 63
- FREE 34
- Front Turret 179

G

- G16 192

H

- HANDWH 34
- Help 122
 - Extended Help 122
 - help page 123
- Help Window 25
 - contents window 26
 - existing functions 26
 - navigation list 25
 - sub-context bar 25
- Hidden programmes 61
- Hide Shutdown 140
- High Speed Execution 206
- History 151
- History setting 146
- HMI 32
 - Axis position information 32
 - CNC status window 32
 - Dialog field 32
 - Main contex 32
 - Messages 32
 - start-up page 32
- HOME flashing 34

I

- include function 65
- INTER 34
- Interpolation mode 36, 37
- ISO 61
 - ISO 841 178

J

- JOG increment 34

K

- Keyboard 171
 - Virtual Keyboard 171
- Korrektur 149

L

- Lag 42
- LAN Access 141
- Language 139
- Loop and Include 65

M

- M00 34
- M01 34
- M02 34
- Machine data 155
- Machine Data 155
 - Drives parameters 155
 - NCK parameters 155
- Machines 178
 - Milling 178
 - Turning 178
- Machining stop 34
- Management Settings 152
- manual override 34
- measuring points 128
- Memory Areas 140
- message 147
 - Drive message 147
- Milling tool 94
- mode bar 139
- Mode waiting 34

N

- NC file 153
- NCK parameters 155
- NC Memory 84
 - Editing a Program 84
- NC Memory Zones 80
- new tool offset 66
- N_POS 34
- NUM area 140
- Number Search 211
- NUM CNC Cycles 74

O

- OEM area 140
- Offset Definitions 182
 - Z Axis 183
- OM 42
- OP 42
- Origins 180
- Override information 32

P

- part program 34
 - ISO 61
- password 137
- Password management 154

- Password Settings 146
- PC disk 57, 70
 - Directory 57
 - Folder 57
 - Keyboard functions 71
 - new tool offset 66
 - part program 60
 - Usinage 72

- PC file 153

- PLC 34
 - PLC state 34
 - run 34
 - stop 34

- PLC-HMI 140
 - Communication 140

- PLC project 111
- PLC state 35
- PLC variable 113
- PLC variables 111

- PQR 201

- Production 41
 - axes channel 42
 - duplicated axes 42
 - OP, OM 42
 - spindle monitor 42
 - Spindle monitor 43

- Program Coordinate Systems 177

- Programmed feed rate 36, 37

- Programming 56
 - active program 56
 - CNC memory information 56
 - Directory and folder 56
 - drip file display 56
 - File list 56
 - Free disk size 56
 - Hidden programs 61
 - subroutines 61

- programs 55
 - Compact part 55
 - Part programs 55
 - Usinage 55, 65

- Program Test 202

R

- Rear Turrets 179
- Refreshing 140
- Relative 42
- Relative origin 139
- Rename 57
- RESET 34

Restart , 161
retraction points 210
R or @ field 96

S

safety module 129
SAM 121
 diagnostic page 121
Security 139
Service 137
 Diagnostic Settings 146
 password 137
 setting option 139
 Settings Page 138
 Start window 137
 time date 144
Service Mode 140
Spindle monitor 43
Spindle Settings 150
Spindle state 36, 37
START 34
stop 34
STOP 34
subroutines 61
Symbolic CNC 111
System messages 147
System Time 144

T

Teach-in 94
test point 121
 measuring points 128
 monitor page 121
 Test Point Monitor 128
TOO 66
tool dimensions 92
Tool offset 67
Tool offset files 55
Tool Offset table 93
Tools 91
Toolstd 66
Tool Tip Radius 193
Tool Type 92
Tool Wear 196
travel limits 48
Travels 181

U

UNITE message 147
USI file 153
Usinage 65

V

variables
 Display 113
 PLC variables 114
Variables 111
 comments 113
 Entering variables 113
 list 116

W

warning 147
 Drive warning 147
Windows protected 139
Windows Protection 139
work area 31
Work Offset 106
Work offset files 55
 files created 69
Workpiece 96
Write variables 139

Published by NUM Spa

Via F. Somma 62
20012 Cuggiono (MI) (Italy)
Tel. : +39 02 979 691
website: www.num.com

Printed May 2016