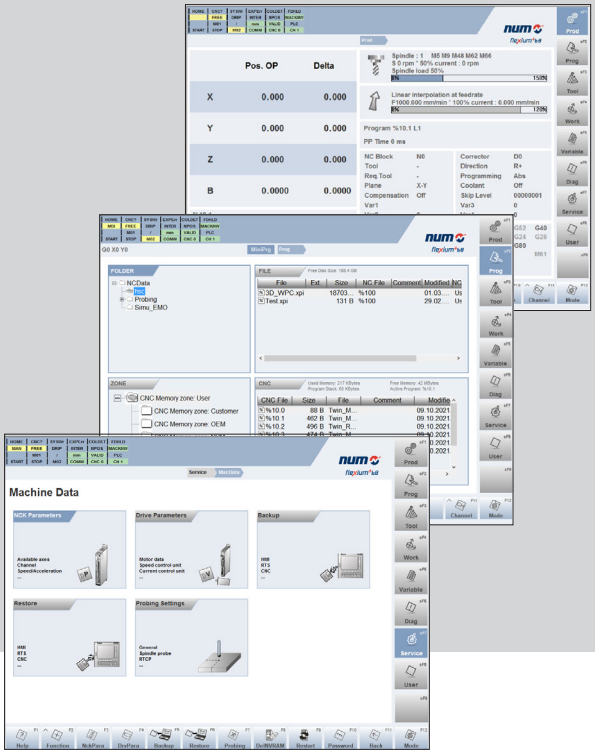
Flexium+ HMI

Operator's manual

(system release 4.2.0.0)

M00039EN-07

Edition 3.2023



flexium+

www.num.com

num
CNC HighEnd Applications

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.

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.

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 13

1.1 Important notice 13
1.1.1 Use of this manual 13
1.1.2 CNC mode selection by the PLC 13
1.2 Use of the Flexium+ HMI manual 14
1.2.1 Operations 14
1.2.2 Possible action during an operation 14
1.3 Document revisions 15
1.4 Use of symbology in the manual 17
1.5 NUM Agencies 17

2 General 21

2.1 Product presentation 21
2.2 Turning ON and OFF the system 22
2.2.1 Turning on the System 22
2.2.2 Turning off the System 22
2.2.3 HMI Startup 22
2.2.4 Start-up sequence 23
2.3 Context 25
2.4 General information on the HMI start-up page 26
2.5 Flexium+ HMI Design Philosophy 28
2.5.1 Main targets 28
2.5.2 Basic structure 28
2.6 Status Windows 30
2.7 Spindle and feed rate information 31
2.7.1 Display with spindle speed and override 31
2.7.2 Display with spindle speed and spindle load 32
2.7.3 Display with spindle speed and spindle current 33
2.8 Help Window 34
2.9 Access to the Help Window 35
2.10 Context menu 36
2.11 Setup the communication 37
2.11.1 FXServer Status in the Taskbar 39

3 Context “Production” 43

3.1 Access to the Context “Production” 43
3.1.1 Context “Production” display when starting the HMI 43
3.2 Channel selection 46
3.3 Spindle monitor 46
3.4 Duplicated axes 47
3.5 Axes information 48
3.5.1 Measurement corrections 49
3.5.2 Tool orientation 50
3.6 CNC Blocks Page 51
3.6.1 Split view 52

3.7 Axes Position Page 53
3.7.1 Axes travel limits 54
3.8 CNC functions 55
3.9 Dialog field 57
3.9.1 Data input into the dialog field 57
3.9.2 Deleting the dialog list 58
3.10 MDI mini program 59
3.10.1 Manage mini programs 60
3.10.2 Functions menu 60
3.10.3 Usage 61

4 Context “Programming” 65

4.1 Access to the Context “Programming” 65
4.1.1 Representation of data locations in your system 65
4.1.2 Definition of the files existing on PC disk and in CNC memory 65
4.1.3 Start-up window of the context “Programming” 66
4.2 Moving between Windows 67
4.3 Selecting Directory and Folder on the PC disk 67
4.4 Selecting a File on PC disk 68
4.5 Creating a new part program on PC disk 71
4.5.1 Hidden programs and subroutines 72
4.5.2 Main program 72
4.5.3 Part programs created with another editor or on an external system 72
4.5.4 Format of an ISO part program 72
4.6 Creating a new Compact part program on PC disk 73
4.6.1 Compact programs created with another editor or on an external system 73
4.6.2 Format of a Compact part program 74
4.7 Creating a new “Usinage” Program on PC disk 75
4.7.1 “Usinage” programs created with another editor or on an external system 75
4.7.2 Format of a “Usinage” program 76
4.7.3 The include function 76
4.7.4 The Loop and Include functions 76
4.8 Creating a new tool offset file on PC disk 77
4.8.1 Tool offset files created with another editor or on an external system 78
4.9 Creating a new work offset file on PC 79
4.9.1 Work offset files created with another editor or on an external system 80
4.10 Editing a file on PC disk 81
4.10.1 Keyboard functions 82
4.11 Editing a “Usinage” file on PC disk 83
4.11.1 Keyboard functions 84
4.12 Including a Part Program in an “Usinage” File 84
4.13 Creating a Loop in an “Usinage” File 84
4.14 Editing NUM CNC Cycles 85
4.15 Drip Feed Mode 86
4.15.1 Principle of drip feed mode 86
4.15.2 Drip feed on machines with several axes channels 87
4.15.3 Block search with drip feed mode 88
4.16 Loading a file into CNC memory 89
4.16.1 Load one file only 89
4.16.2 Load several successive files 89
4.16.3 Load several non successive files 89
4.16.4 Load a full folder 90
4.17 Protected Memory Area Management 91
4.17.1 Selecting CNC Memory Zones 92
4.17.2 Closing the memory zones 92

4.18 NC Memory 93
4.19 Selecting the Active Part Program 94
4.20 Saving Files to PC Disk 95
4.20.1 Save one file only 95
4.20.2 Save several successive files 95
4.20.3 Save several non successive files 95
4.21 Editing a Program directly in NC Memory 96
4.22 Deleting Files on PC Disk or NC Memory 98
4.22.1 Delete one file only 98
4.22.2 Delete several successive files 98
4.22.3 Delete several non successive files 98

5 Context “Tools” 103
5.1 Access to the Context “Tools” 103
5.2 Entering tool dimensions and miscellaneous values 104
5.3 Selecting the Tool Type 105
5.4 Resetting the whole Tool Offset table 106
5.5 Teach-in of the Tool Dimensions 107
5.5.1 Milling tool setting page 107
5.5.2 Turning tool setting page 108
5.5.3 Boring tool setting page 108
5.5.4 Procedure 109

6 Context “Work Offset” 113
6.1 Access to the Context “Work Offset” 113
6.1.1 Milling display 113
6.1.2 Turning display 114
6.2 Measuring cycles 115
6.3 Entering DAT2 values 119
6.4 Entering DAT1 values 120
6.5 Setting the Work Offset (DAT1) by Teach-in for Milling 121
6.6 Setting the Work Offset (DAT1) by Teach-in for Turning 123
6.7 RTCP compensation 125
6.8 Selecting DAT3 axes and other parameters 126
6.8.1 Entering Scaling Factor 127
6.8.2 Entering Axes Assignment 127

7 Context “Variables” 131
7.1 Access to the Context “Variables” 131
7.1.1 Variables context window 132
7.2 Entering variables and comments 133
7.3 CNC variables and comments 134
7.3.1 Display of Symbolic variables 134
7.4 PLC variables and comments 135
7.5 Modification of variables values 136
7.6 Creating a list of consecutive variables 137

8 Context “Diagnostic” 141
8.1 Access to the Context “Diagnostic” 141
8.1.1 Diagnostic context window 141
8.2 Extended Help 142
8.2.1 Example of help page 143
8.3 Alarm History Page 144

8.3.1 History Page and Filters 145
8.4 Exception Page 147
8.5 Test Point Monitor 148
8.6 CNC and DISC NT+ Configuration 149
8.6.1 Device List 150
8.6.2 System Functions Page 151
8.6.3 Drive History 152
8.6.4 Project information page 154
8.6.5 CNC task information 155
8.6.5.1 Dynamic Operators monitor 155
8.6.5.2 Available Dynamic Operators programs in C 156
8.7 SAMX Diagnostic 157
8.7.1 Safety module diagnostic 159
8.7.2 STOX Safety diagnostic 159
8.8 Operation time 160
8.9 Export log files 161

9 Context “Service” 165

9.1 Access to the Context “Service” 165
9.1.1 Start window of the context “Service” 165
9.2 General Settings Page 167
9.2.1 Setting selection 167
9.2.2 System Time and Date Settings 170
9.2.2.1 Setting the system time and date 170
9.2.3 Security Settings 171
9.2.4 Location Settings 174
9.2.4.1 Keyboard functions 174
9.2.5 Custom tools 175
9.2.6 Password Settings 176
9.2.7 Box PC Access 177
9.3 Diagnostic Settings 178
9.3.1 Custom message colors 180
9.3.1.1 System message colors 1 180
9.3.1.2 System message colors 2 181
9.3.1.3 PLC message colors 182
9.3.2 History 183
9.3.2.1 History setting 183
9.3.3 PLC Visu 184
9.3.4 Axes view 185
9.3.5 Spindle Settings 188
9.3.6 Production Settings 191
9.4 Management Settings 193
9.4.1 FXCAM 195
9.4.2 Password management 196
9.4.3 NUMCoss 197
9.5 Machine Data 198
9.5.1 Change Password 199
9.5.2 Restart 199
9.5.3 Clear the NVRAM 200
9.5.4 Probing 201
9.5.5 Restore 202
9.5.6 Backup 204
9.5.7 Drive parameters 207
9.5.8 NCK parameters 210
9.6 Diagnostic 212
9.6.1 Oscilloscope 213
9.6.1.1 Oscilloscope functions 214
9.6.1.2 Chart 216

9.6.1.3 Tabs 218
9.6.1.4 Configure page 221

10 CopyExplorer and Virtual keyboard 229

10.1 Access to CopyExplorer 229
10.2 Save 230
10.2.1 Save one file only 230
10.2.2 Save several successive files 230
10.2.3 Save several non successive files 230
10.2.4 Save all files 230
10.3 Restore 230
10.4 Virtual Keyboard 231
10.4.1 Activating the keyboard 231
10.4.2 Closing the keyboard display 231

11 CNC Operation 235

11.1 Declaration of Axes Assignment 235
11.2 Execution of a Program 236
11.2.1 Program Test 236
11.3 Execution on the machine 237
11.3.1 Optional Stop and Block Skip Enable 237
11.3.2 Program Execution - General 237
11.3.3 Execution of a Program Step by Step 239
11.3.4 High Speed Execution of a Program 240
11.3.5 Execution of a Program in Automatic Mode 240
11.4 Operator Interventions 241
11.4.1 Unplanned Interventions 241
11.4.2 Interruption of Machining 241
11.4.3 Retracting the Axes 242
11.4.4 Manual Axis Recall 242
11.4.5 Automatic Axes Recall 243
11.4.6 Sequence Number Search 245
11.4.7 Search Mode Dialog wizard 247
11.4.7.1 Search Block number 247
11.4.7.2 Search Line number 248
11.4.7.3 Search String 249
11.4.7.4 Restart at Interrupted position 250
11.4.8 Extended possibilities in Search mode 251
11.4.9 Backtrack along Path 252
11.4.10 Return along Path 255
11.4.11 Emergency Retraction 256
11.4.12 Program Resumption after a Following Error 256
11.5 Programmed Interventions 257
11.5.1 Machining Stop Function M00 (or M01 if enabled) 257
11.5.2 Forcing an intervention by M12 257

12 Additional functions 261

12.1 Introduction 261
12.2 Teach-in 262
12.2.1 Access to the Teach-in function 262
12.2.2 Teach-in editor window for part programmes 264
12.2.3 Explanation of the fields and functions 265
12.2.4 Teach-in procedure 266
12.2.5 Teach-in directly in NC Memory 267
12.3 Extended tool table 271
12.3.1 Displays 271
12.3.2 Channel display 272
12.3.3 Channel assignment and comments 273

12.3.4 Assigning a tool to a channel....273
12.3.5 Assigning a name and a comment....273
12.4 Symbolic names....274
12.4.1 Set up of the symbolic names....274
12.4.2 Axes configuration....275
12.4.3 Entering axes names....275
12.4.4 Giving a name to an axis address assignment....276
12.4.5 Deleting the name given to an axis address assignment....276
12.4.6 Display of axis name of address....276
12.4.7 Axes channel configuration....277
12.4.8 Display of symbolic names....280
12.4.9 Display in the status window....281
12.4.10 Channel selection....282
12.4.11 Particularities of the channel selection....283
12.4.12 Particularities with the function Multi-CNC....284
12.4.13 Specific features of the extended tool table....284
12.5 Extensions for NUMROTOplus®....285
12.5.1 Particularities in the context bar....285
12.5.2 Setting in the context “Service”....286
12.5.3 Wheel compensation....288
12.6 PLC visualization....289
12.6.1 Integrated PLC visualization....289
12.6.2 Configuration – Softkey sF8 / sF9....293
12.6.3 Sample of PLC Visu Pages....294
12.7 FXKeyboard....295
12.7.1 Settings....297
12.8 Flexium CAM....301
12.8.1 Settings....302
12.8.2 Install a configuration....303
12.8.3 Operation....304
12.8.3.1 Block number....306
12.9 NUMcoss....308
12.9.1 Access to the NUMcoss Configuration page....309
12.9.2 NUMcoss Configuration....310
12.9.2.1 Fine Finishing....310
12.9.2.2 Theoretical Background and Smoothing....311
12.9.2.3 Explanation of all NUMcoss Settings....312
12.9.3 NUMcoss in the Context Programming....315
12.9.4 Programmable NUMcoss parameter....317
12.10 Tool management....318
12.10.1 Tool view....319
12.10.1.1 Search a tool....320
12.10.1.2 Load a tool....321
12.10.1.3 Unload a tool....321
12.10.1.4 Create a new tool....322
12.10.1.5 Function list menu....323
12.10.2 Magazine View....324
12.10.2.1 Function list menu....326
12.10.2.2 Buffer View....327
12.10.2.3 Spindle View....327
12.10.3 Tool data View....328
12.10.3.1 Function list menu....330

13 Review....335

13.1 System Overview....335
13.1.1 Operating Mode....335
13.1.2 Defining a Program....335
13.1.3 Creating a Program....336
13.2 Review of Program Coordinate Systems....337

13.2.1 Review of Axes Definition and Direction 337
13.3 Machine Overview 338
13.3.1 General 338
13.3.2 Turning and Milling Machines 338
13.4 Turning Machine - Specific Features of Front and Rear Turrets 339
13.4.1 View of a Turning Machine with Rear Turret 339
13.4.2 View of a Turning Machine with Front Turret 339
13.5 Definition of Travels and Origins 340
13.5.1 Homing with Incremental or Combined Encoders Using Absolute Dimensions 340
13.5.2 Schematic Representation of Travels (Turning and Milling machine) 341
13.6 Offset Definitions 342
13.7 Turning Machine - Offsets Represented on the Z and X Axes 343
13.7.1 Offsets on the Z Axis (Solution 1 with DAT2) 343
13.7.2 Offsets on the Z Axis (Solution 2 with DAT2) 343
13.7.3 Offsets on the X Axis (Solution 1 with DAT2) 344
13.7.4 Offsets on the X Axis (Solution 2 without DAT2) 344
13.7.5 Schematic Representation of the Offsets on X and Z 345
13.8 Milling machine - Offsets Represented on the Z, X and Y Axes 346
13.8.1 Offsets on the Z axis 346
13.8.2 Offsets on the X axis 346
13.8.3 Offsets on the Y axis 347
13.8.4 Schematic Representation of All Offsets 348
13.8.5 Special Case of Milling Machines Equipped with Rotary Tables 349
13.9 Tool Axis Orientation 351
13.9.1 Using the G16 P+ function on turning or milling machine 352
13.10 Definition of Tool Dimensions and Offsets 353
13.10.1 Turning Machine - Dimensions, Tool Tip Radius and Orientation 353
13.10.2 Tool Tip Radius and Orientation 354
13.10.3 Milling Machine - Definition of Tool Dimensions 355
13.11 Definition of Tool Wear 356
13.11.1 Turning Machine 356
13.11.2 Milling Machine 357

ANNEX 1 - HMI Structure....361

ANNEX 2 - HMI functions 381

Page deliberately empty

Summary

1 Foreword....13
1.1 Important notice....13
1.1.1 Use of this manual....13
1.1.2 CNC mode selection by the PLC....13
1.2 Use of the Flexium+ HMI manual....14
1.2.1 Operations....14
1.2.2 Possible action during an operation....14
1.3 Document revisions....15
1.4 Use of symbology in the manual....17
1.5 NUM Agencies....17

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 described in the chapter *“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 do not have to 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 Use of the Flexium+ HMI 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.

This manual should be considered as a supplement to the Flexium+ Programming manual. (Reference : M00040)

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

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 have to 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
Press F6 to scroll the contents window one page up

1.3 Document revisions

Date	Index	Reason for revision	See section
09 - 2014	00	First manual version is conform to software version 4.0.0.0.	See section
12 - 2015	01	Manual version is conform to software version 4.0.20.0. - New graphic presentation for >70% of the main HMI pages. - New Main program. - New Load a full folder. - New Entering Current Offset values. - New Entering DAT1 values. - New Entering DAT2 values. - New Operation time . - New Setting options. - New Box PC Access. - New Custom tools. - New Systems RTS Messages. - New History. - New History setting. - New Password management. - New Machine Data section. - New FXKeyboard Multitouch handling, Shut down. - New Flexium CAM section.	 .... 4.5.2 4.16.4 6.2 6.3 6.4 8.6.3 9.2 9.2.2 9.2.4 9.3.1 9.3.4 9.3.4.1 9.4.2 9.5 12.7 12.8
06 - 2017	02	Manual version is conform to software version 4.1.0.0. - New graphic presentation for >60% of the main HMI pages. - New Status Windows. - New Display with spindle speed and spindle current. - New Access to the Help Window. - New Spindle monitor. - New Selecting a File on PC disk functions. - New NC Memory functions. - New Extended Help. - New Drive History. - New SAMX Diagnose. - New Safety module diagnostic. - New SAMX Safety diagnostic. - New STOX Safety diagnostic. - New Security Settings. - New Diagnostic Settings messages. - New Spindle Settings options. - New Management Settings options. - New Machine Data commands. - New Clear the NVRAM section. - New Backup section. - New Restore section. - New Extended tool table pages. - New PLC visualization actions and setting options. - New Configuration - Softkey sF8 / sF9. - New Sample of PLC Visu Pages. - New NUMcoss section. - New Tool management section.	 .... 2.6 2.7.3 2.9 3.3 4.4 4.18 8.2 8.6.4 8.6.5 8.6.6 8.6.6.1 8.6.6.2 9.2.7 9.3.1 9.3.2 9.4 9.5 9.5.5 9.5.6 9.5.7 12.3 12.6 12.6.2 12.6.3 12.9 12.10

12 - 2018	03	Manual version is conform to software version 4.1.10.10. - New graphic presentation for >60% of the main HMI pages. - Setup the communication. - FXServer Status in the Taskbar. - New Restart at Interrupted position. - Measurement corrections. - Context “Production” display when starting the HMI. - New CNC task information. - New Dynamic Operators monitor. - New Available Dynamic Operators programs in C. - New Time Settings. - New Spindle color settings. - Explanation of the setting options. - New Search Mode Dialog wizard. - New Search Block number. - New Search Line number. - New Search String. - New Restart at Interrupted position. - New Production Settings. - New Settings in the context “Service”. - New functions on PLC visualization “Communication buffer size and Paint buffer size. - Block number.	 2.4 2.10 2.10.1 3.1.1 3.5.1 8.6.7 8.6.7.1 8.6.7.2 9.3.2 9.3.3.2 9.3.4.1 9.3.6 11.4.7 11.4.7.1 11.4.7.2 11.4.7.3 11.4.7.4 12.5.2 12.6 12.8.3.1
2 - 2019	04	- Custom message colors. - System message colors 1. - System message colors 2. - PLC message colors.	9.3.3 9.3.3.1 9.3.3.2 9.3.3.3
2 - 2020	05	- Status Windows page. - Setup the communication. - CNC Blocks Page. - Split view. - Dialog field page. - MDI mini program. - Editing a file on PC disk page. - Editing a “Usinage” file on PC disk page. - Editing a Program directly in NC Memory page. - Teach-in of the Tool Dimensions page. - Context “Service” page. - Location Settings page. - Security Settings page. - Diagnostic page. - Oscilloscope. - Teach-in page.	2.6 2.10 3.6 3.6.1 3.9 3.10 4.10 4.11 4.21 5.5 9.1 9.2.5 9.2.7 9.6 9.6.1 12.2
11 - 2021	06	- Upgrade to system version 4.1.30.0 - Image update - HMI start-up page - Information update - Context menu - Information update - Axes information - Tool orientation - Reordering of paragraphs - Reordering of paragraphs - New paragraph - Probing - Correction and updating of texts and tables.	 2.4 2.10 3.5 3.5.2 8 9 9.5.4
03 - 2023	07	- Upgrade to system version 4.2.0.0 - Image update - HMI - Information update - HMI Startup sequence - Axes information - Tool orientation - Reorder chapter - Measuring cycles - Selecting DAT3 axes and other parameters - Export log files - Reorder chapter - PLC Visu - Axes view - FXCAM - NUMCoss - Annexes	 2.2.3 2.2.4 3.5 3.5.2 6 6.2 6.8 8.9 9 9.3.3 9.3.4 9.4.1 9.4.3 A

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

Page deliberately empty

Summary

2 General 21

2.1 Product presentation 21

2.2 Turning ON and OFF the system 22
2.2.1 Turning on the System 22
2.2.2 Turning off the System 22
2.2.3 HMI Startup 22
2.2.4 Start-up sequence 23

2.3 Context 25

2.4 General information on the HMI start-up page 26

2.5 Flexium+ HMI Design Philosophy 28
2.5.1 Main targets 28
2.5.2 Basic structure 28

2.6 Status Windows 30

2.7 Spindle and feed rate information 31
2.7.1 Display with spindle speed and override 31
2.7.2 Display with spindle speed and spindle load 32
2.7.3 Display with spindle speed and spindle current 33

2.8 Help Window 34

2.9 Access to the Help Window 35

2.10 Context menu 36

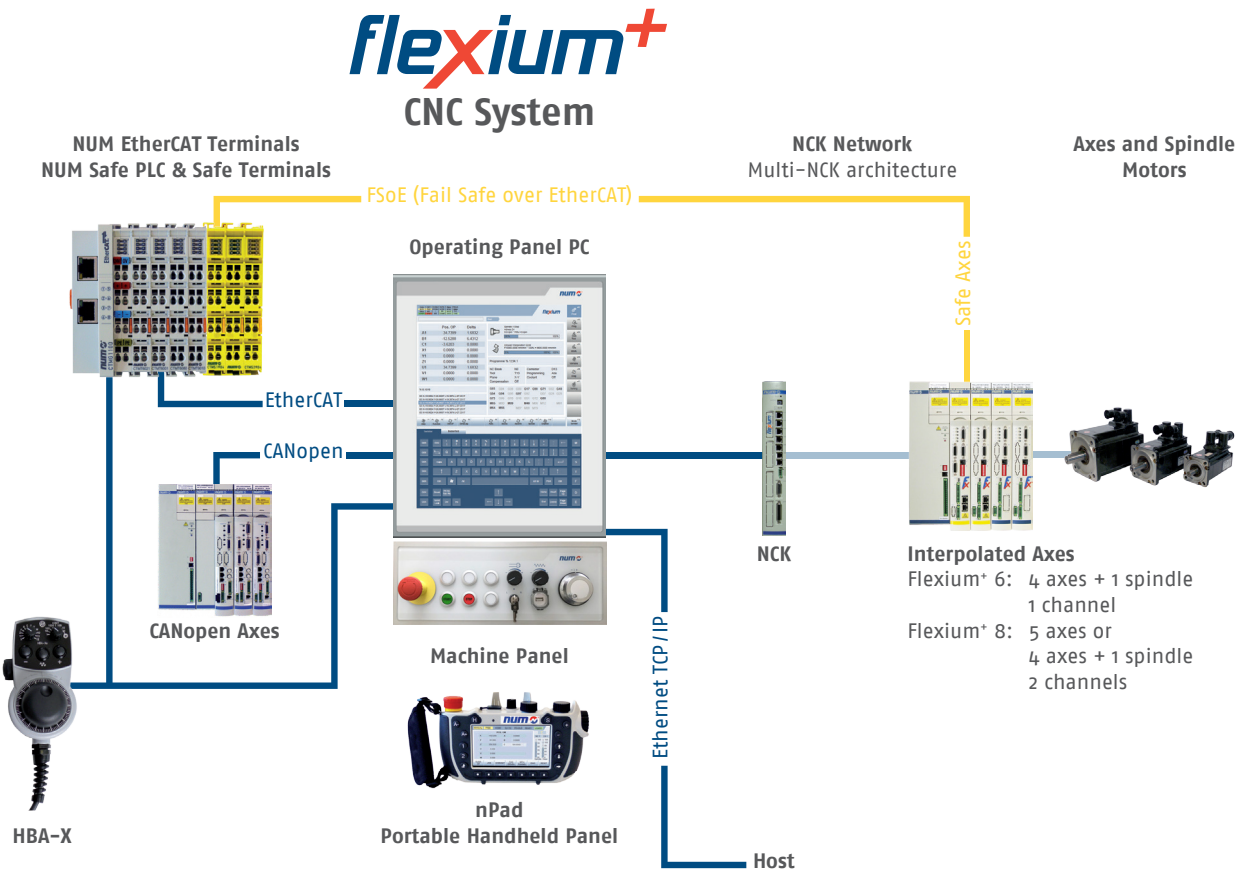
2.11 Setup the communication 37
2.11.1 FXServer Status in the Taskbar 39

Page deliberately empty

2 General

2.1 Product presentation

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



2.2 Turning ON and OFF the system

2.2.1 Turning on the System

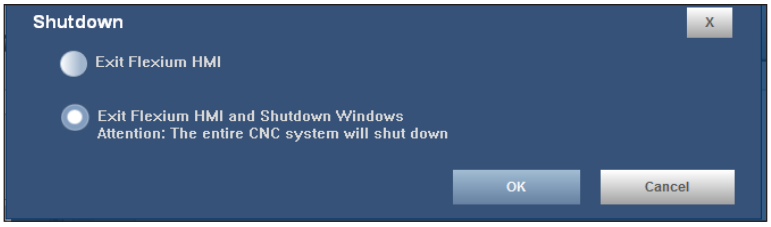
Power up the NCK and the PC.

2.2.2 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	☞ ⇧ Shift + F12 or ☞ Alt + F4

The shutdown dialog shows two options:



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 fully stopped before selecting this option, as the machining could be stopped in an uncontrolled way.

It is possible to hide this window via the service menu (see chapter 9).
If this dialog is hidden pressing Alt + F4 will exit Flexium HMI only.

2.2.3 HMI Startup

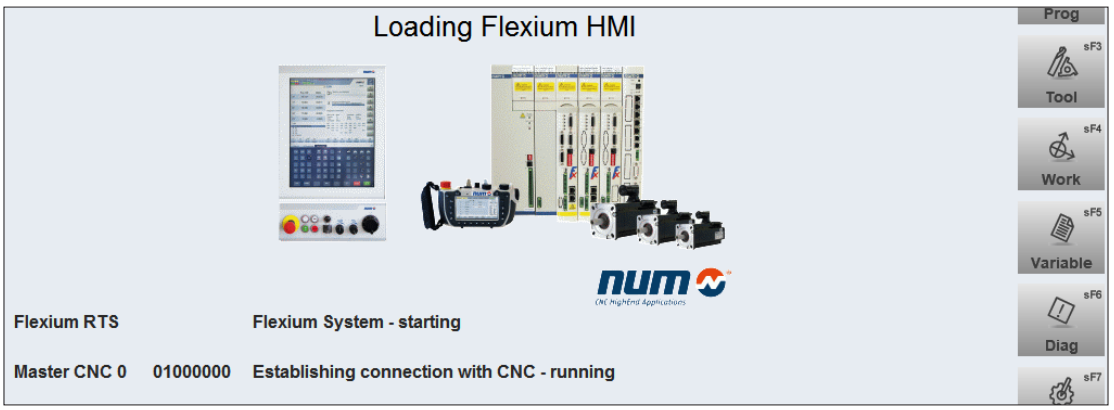
By default the HMI is launched automatically at power up. An icon is available on the desktop to relaunch it if necessary.
It is possible to launch a second instance of this HMI by the command : “C:\Program Files (x86)\NUM\Flexium HMI\FXBrowser\Bin\FXBrowser.exe” -SecondInstance.

When running on a larger display, the HMI size can be adapted, refer to chapter 9.2

2.2.4 Start-up sequence

During the system start up, several messages will be displayed to show the status of the initialisation. They are fugitive and the production page should be shown at the end of the process.

Should a message ne permanent the following chapters will help to detect the issue.
The messages are structured like below



The bottom line indicates on what NCK the HMI is connected followed by a code and the start-up phase in clear text.

The code structure is of the form: aa bb cc dd

- a = Start-up phase
- bb = Status message
- cc = Index (e.g. 00 for Memory zone 0 or 01 for drive 01 etc.)
- dd = Specific NCK message.

The different start-up phases (aa) are:

- 01 Establishing connection with NCK
  - Communication between the PC Panel and the NCK
- 02 Loading: Licence
  - Loading and checking the licence file.
- 03 Loading: Parameters
  - Loading the parameters of the NCK and the drives.
- 04 Loading: Memory area cc
  - cc corresponds to memory zone 0 (00), memory zone 1 (01) etc.
- 05 Loading: Interaxis calibration
- 06 Loading: Axis calibration
- 08 Loading: Dynamic Operators in C
- 09 Initializing
- 0A Loading: NCK software
- 0B Starting: NCK software
- 0C Reading: Checking file integrity
  - The NCK checks if the system was shut down correctly the last time and compares the data in the zone “User”.
- 0F Cyclic communication
  - Establishing continuous communication between NCK and PLC

Additional information (bb) about the start-up phase (aa)

- 00 Running.
- 01 Completed.
- 02 File cannot be opened.
- 03 Transmission cannot be started.
- 04 Segment error.
- 05 Transmission cannot be stopped.
- 06 Communication error.
- 07 Error.
- 09 Cancelled by user.
- 0F Reading RTS licences.
- 20 Boot loader and NCK software not compatible.
- 21 NCK Device and NCK software not compatible.

During a normal start-up phase, the different messages will be replaced by the next one until the PROD page of the HMI is displayed.

In the case, the start-up is locked on a message, different actions can be taken:

- Establishing connection with NCK - running
  - If this message is displayed for a longer time:
    - check the address of the NCK;
    - compare address switch on the NCK with the address in your project;
    - check the connection of the Ethernet cable between the panel and the NCK (Eth1 <-> X5);
    - check the NCK power supply (Power LED should be on);
    - If everything above is ok, press the Reset switch on the NCK.

- Loading: Licence - running
  - If this message is displayed for a longer time:
    - Ethernet not initialized correctly - Stop RTS;
    - Switch off power of the NCK;
    - Switch on power of the NCK;
    - Start RTS.

- Loading: Licence - File cannot be opened
  - The license file has not been loaded or is corrupted. Refer to the commissioning manual to load the license file again.

- Loading: NCK software - NCK Device and NCK software not compatible
  - The Firmware version of the NCK in your project must match with the version of the NCK Software. This could occur after a replacement of the NCK unit of a software upgrade not realised correctly.

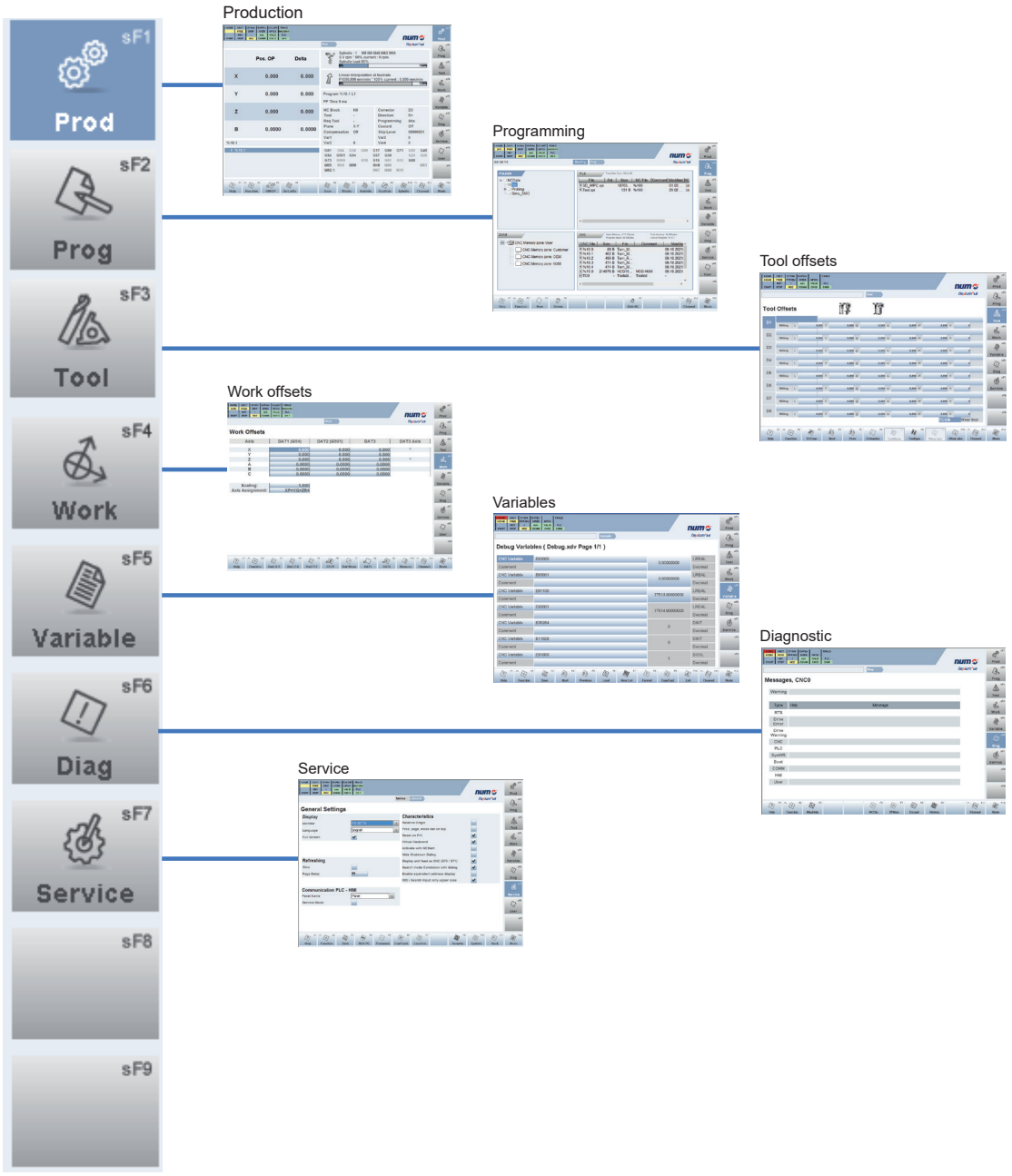
- Loading: NCK software - File cannot be opened
  - The CNC firmware file is missing. Again, this could occur after a software update not executed correctly. Contact your maintenance service or NUM if this message remains after a second tentative.

- Starting: NCK software – running
  - Restart NCK and PC Panel. If the error remains present this can be a failure of the NCK unit or a peripheral.

- Cyclic communication - communication error
  - Check the connection between NCK (X5) and PC Panel.

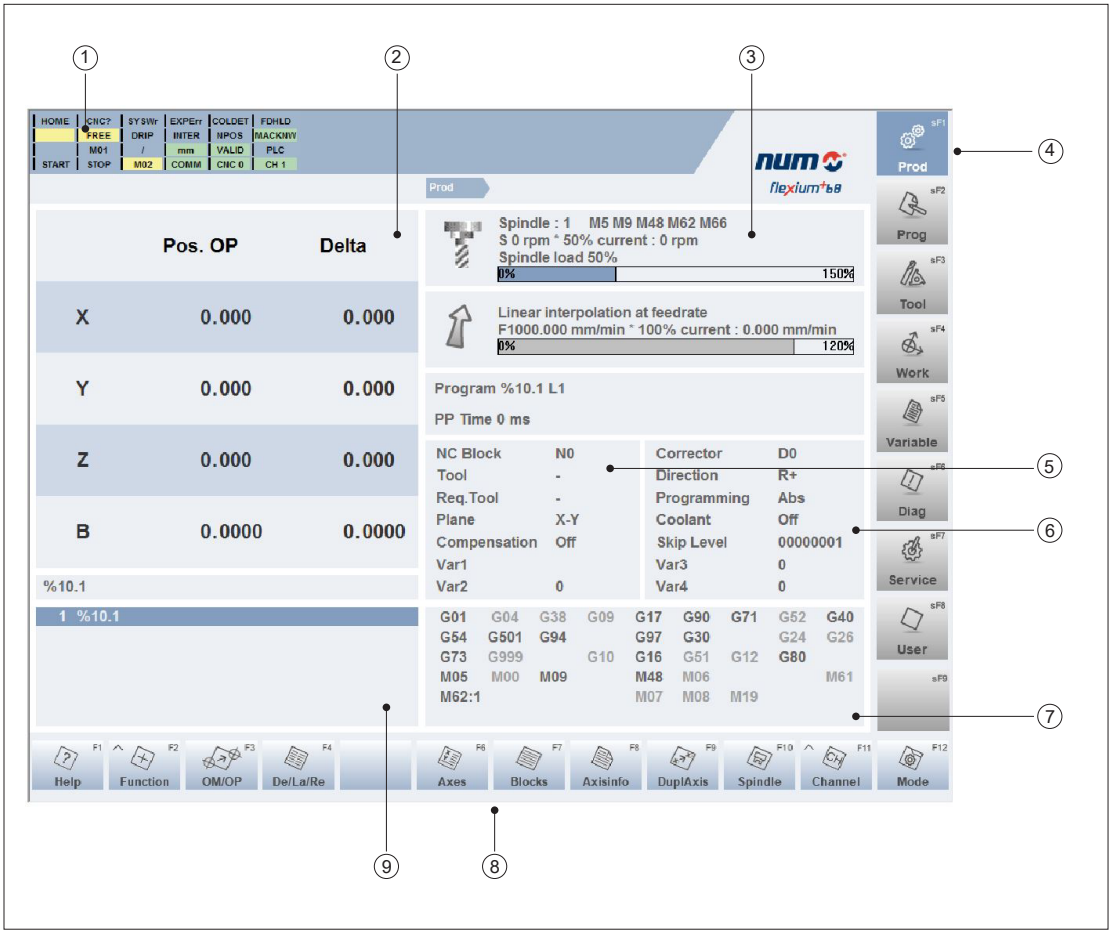
2.3 Context

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



2.4 General information on the HMI start-up page

Display of the HMI start-up page.

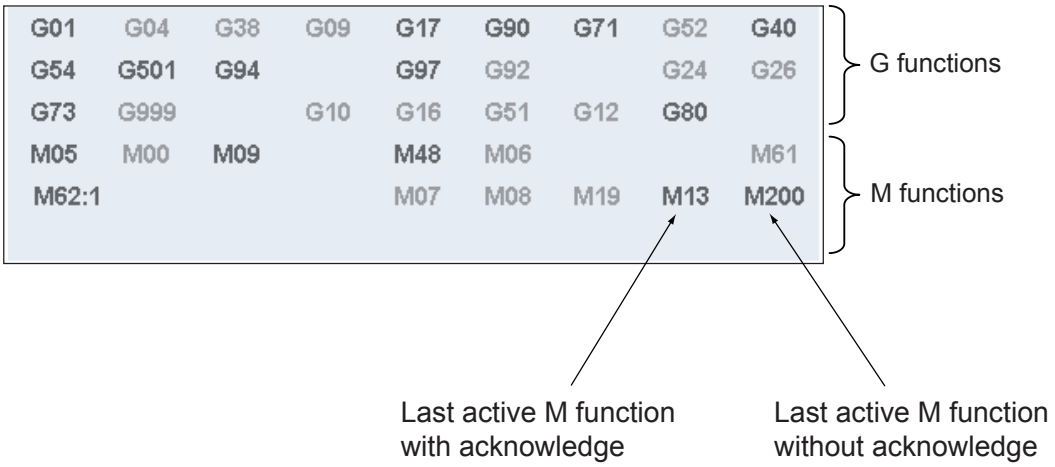


1	CNC status window.
2	Axes position information (axes which are not homed are shown in red).

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

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

Detailed function information



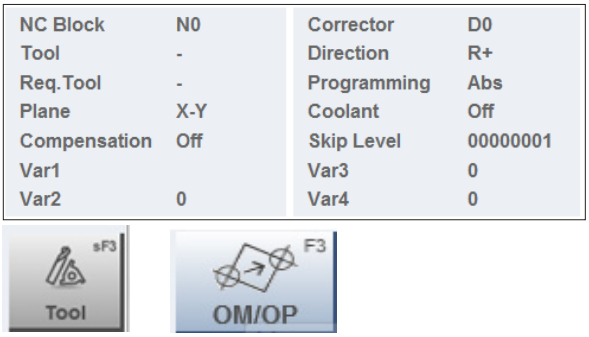
2.5 Flexium+ HMI Design Philosophy

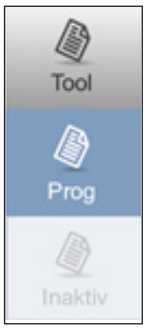
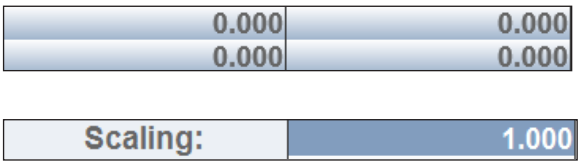
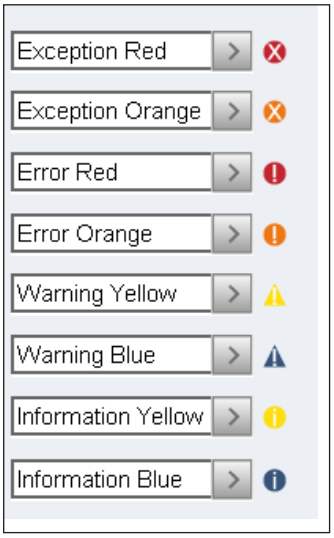
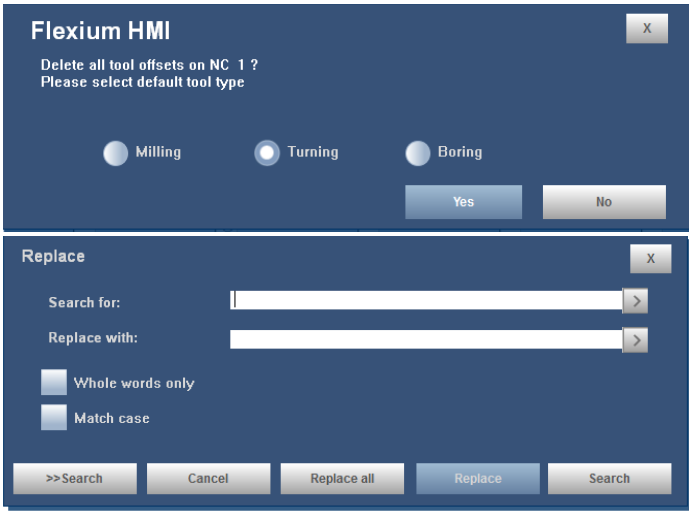
2.5.1 Main targets

The new Flexium+ HMI is designed to reflect the corporate identity of the NUM. Target of its appearance is trustful and long lasting. The main targets and differences to the previous design are listed below:

Orientation	The eye is orientated through simplification, clear structures and direct references, by areas of color and monochrome backgrounds. It can be operated intuitively, steps are facilitated.
Color	Systematic color use can increase the usability of complex systems, but the associated orientation effect could be reduced with an increasing number of different colors. The new Flexium+ HMI is mainly limited to blues and grays. This color combination is perceived as pleasant and eye-friendly. Only important references such as errors or warnings are displayed in color. This ensures that labels stand out and could be registered.
Navigation	The visual separation of the context menu (vertical bar) and the sub navigation (horizontal bar) created the same navigation hierarchy as it was before. But the two menus were now separated from the content not only by their position, but also by their color.
Icon language	Used icon language is simplified but more meaningful.

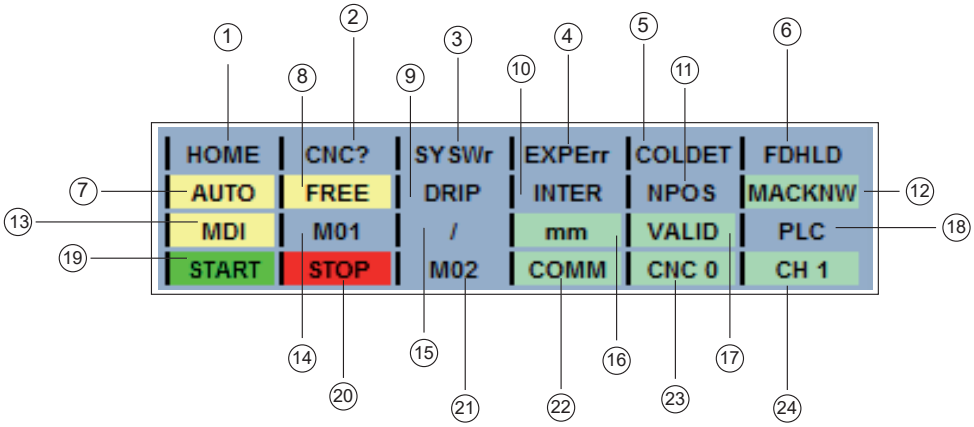
2.5.2 Basic structure

Buttons and edit area	The structure of the screen helps the user to get a quick overview of the relevant content. The separation of two-and three-dimensional elements with shadows and gradients, makes sure, that the user can quickly grasp what information is a purely display field (unique color background) and which components are operated respectively oppressive and adjustable: 
Grid	All surfaces, headings and buttons are aligned to the Grid. It is important that both vertical and horizontal references and other operational input fields are easy and direct to find.

Status of buttons	Meaning	Status of Edit areas
	**Operational** **Pressed or active** **Inactive**	
Messages	Message icons with different meaning: 	
HMI Dialogs	All dialogs are customized in NUM design: 	

2.6 Status Windows

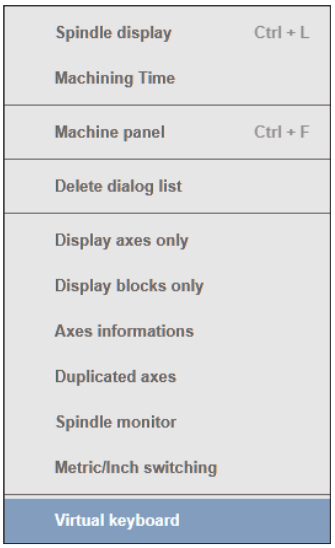
The Status Window shows the status of the CNC at any given time via indicators displayed in 24 fields. **When no indicators are required in a field, it is not displayed.**



1	Homing not performed (HOME flashing).
2	CNC error (CNC ? flashing).
3	System warnings (see Flexium⁺ Programming manual Reference M00040).
4	CNC Exception error has occurred.
5	Status of the Online Collision Detection of NCK/Flexium 3D.
6	Feed interruption (FDHLD). Axes' movement not allowed by PLC.
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	M Function not completed. The NCK is waiting for the function finish (PLC acknowledge).
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 (ready), STOP: PLC in stop, EXPErr: Exception error, DbStep:Debug step BrkPnt: Stopped on Breakpoint, 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.

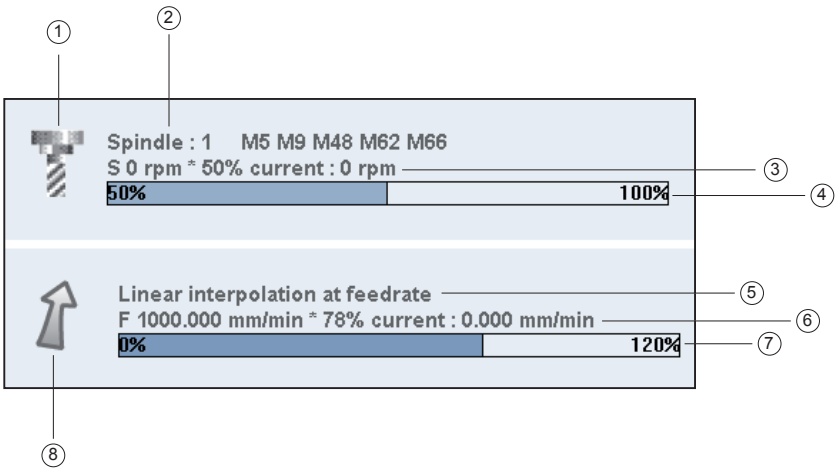
2.7 Spindle and feed rate information

This window may show various information. To switch between displays, use the function *"Spindle Display"* in the *"Production"* context. (Whenever "F2 *Function"* does not appear press "F12 *Mode"*)



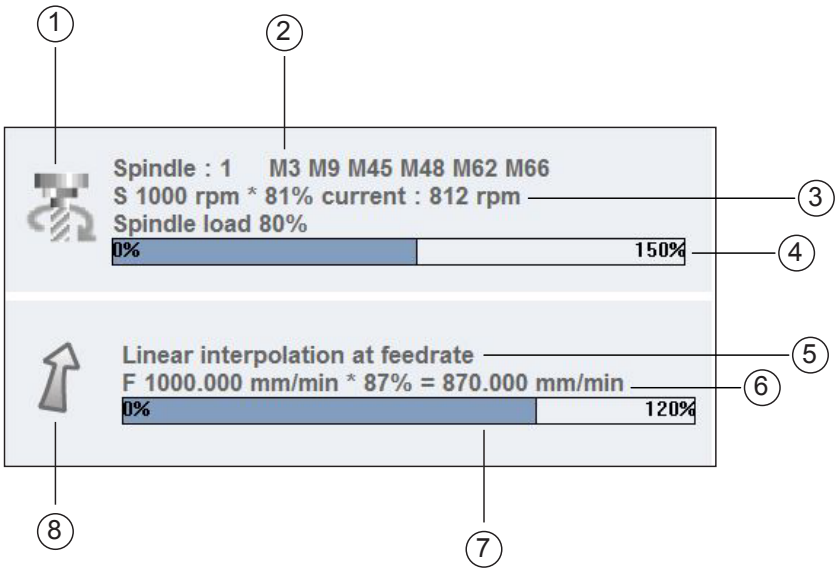
- Spindle display : Switch between spindle speed and spindle load
- Machining time : Switch between machining time and part program time
- Machine panel : Replace the right half of the display by a custom made PLC graphic page
- Delete dialog list : Clear the previous dialog memory

2.7.1 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.

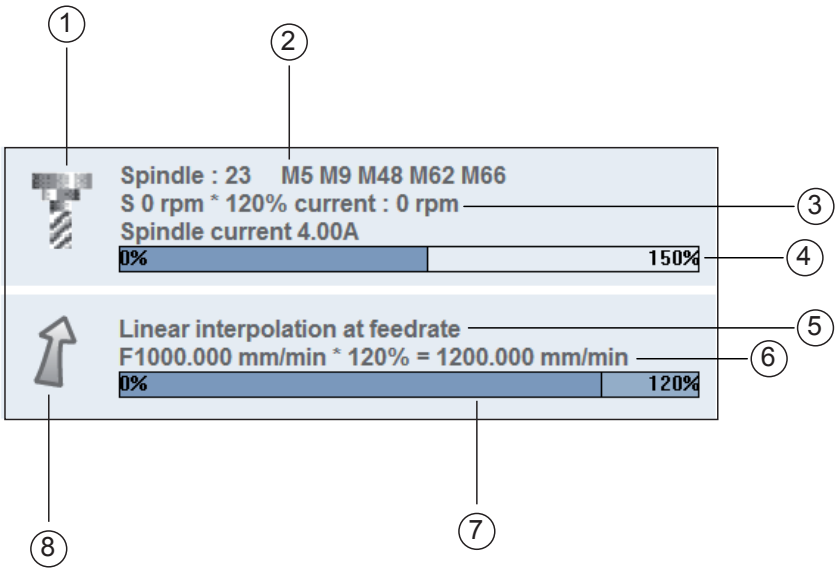
2.7.2 Display with spindle speed and spindle load



1	Spindle state icon.
2	Spindle state description and spindle logical number.
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.

The ability to display the spindle load is validated by the PLC

2.7.3 Display with spindle speed and spindle current



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

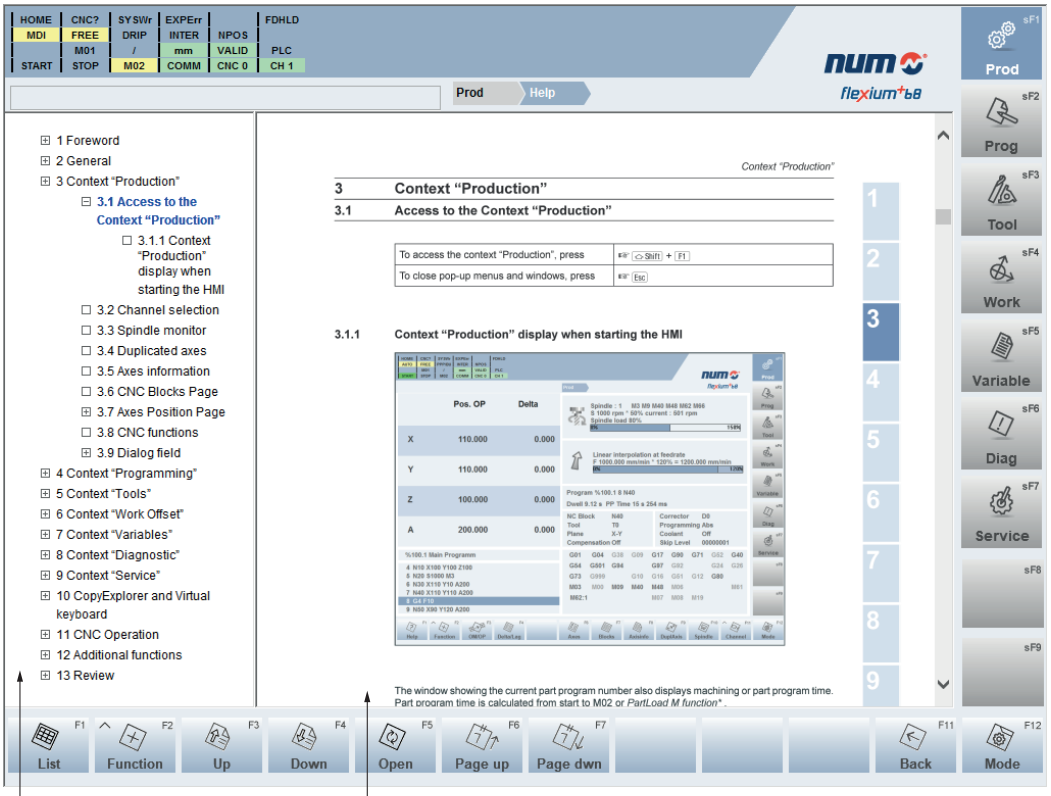
2.8 Help Window

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 related to current displayed Flexium+ HMI context.

2.9 Access to the Help Window

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



To exit the Help window	☞ F11

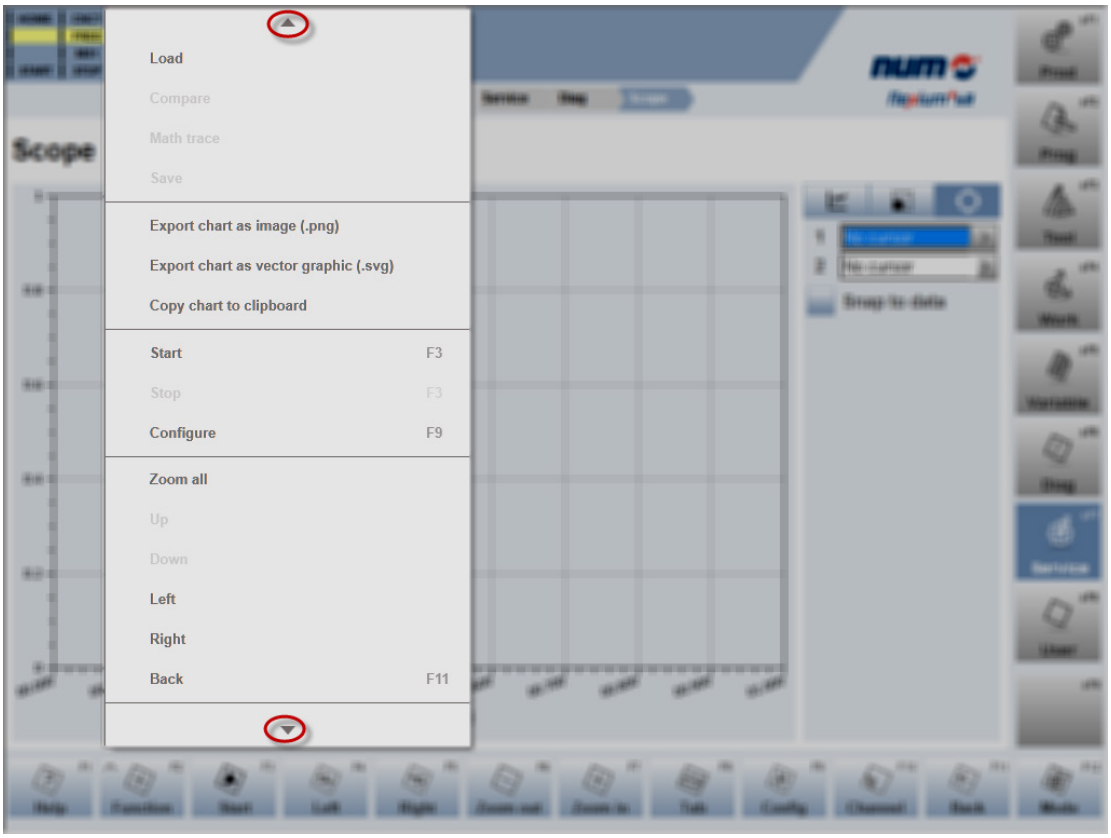
Possible actions

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.

Help is context sensitive.

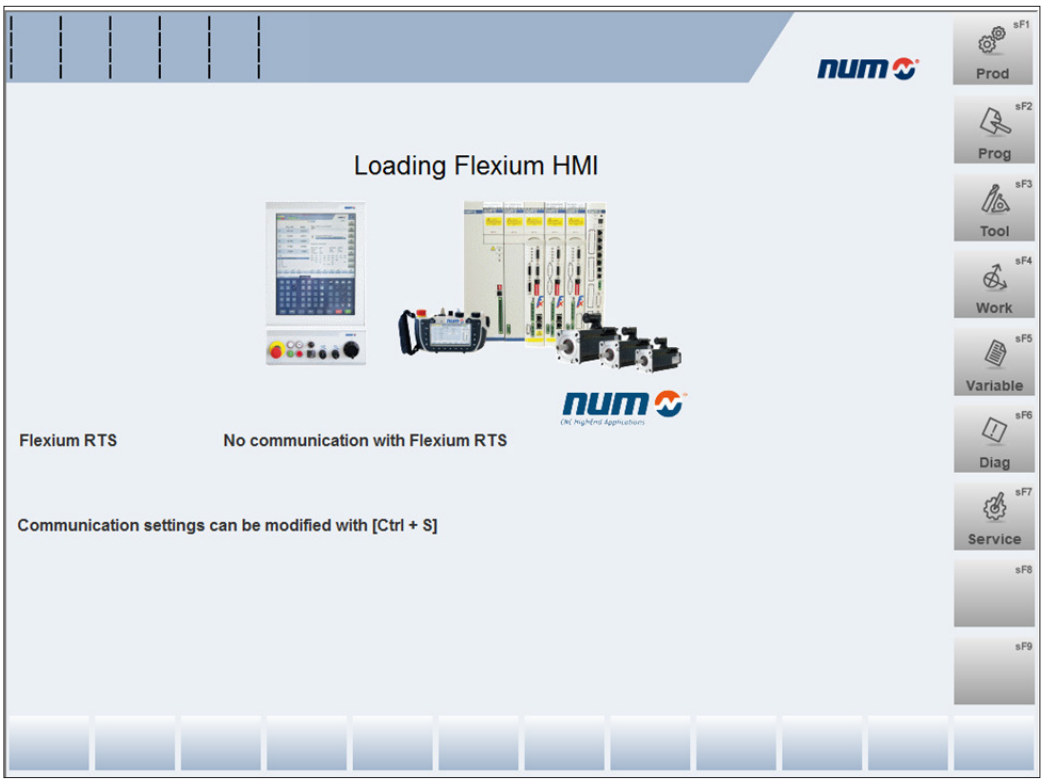
2.10 Context menu

In most pages a context menu is available by pressing function key F2. In some cases the context menu can expand further vertically as interline has been increased to comply with a touchscreen panel. Notice the sign on top and/or bottom of this menu to access all items.



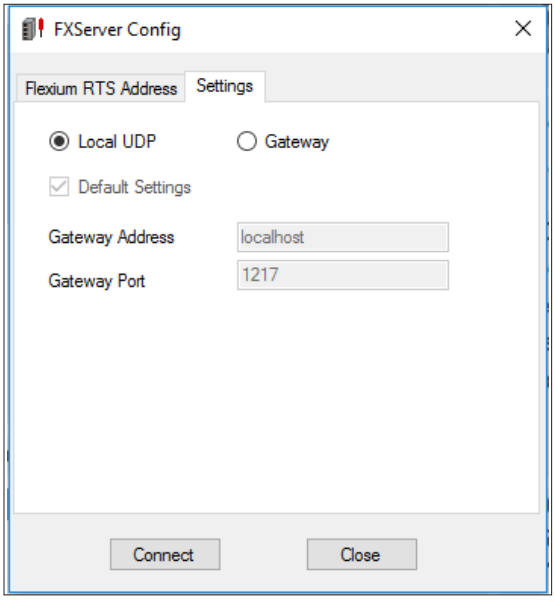
2.11 Setup the communication

If there is no communication at HMI startup this page is displayed:

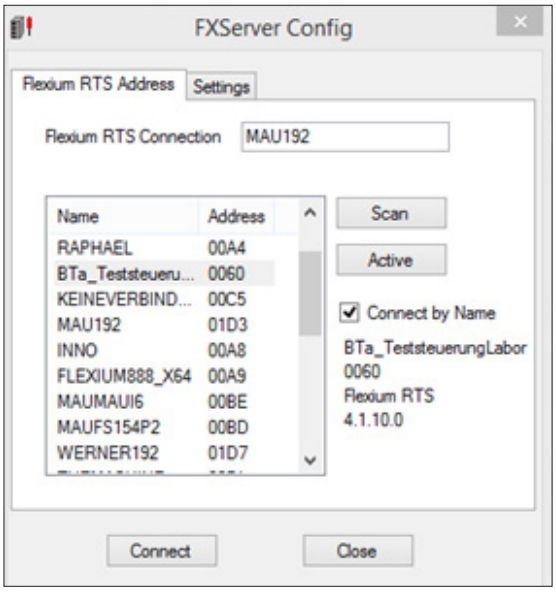


Open the FXServer communications settings with Ctrl+Shift or with right click on the *FXServer Icon* and select the *Settings menu* in the Windows taskbar.

After this, the following settings Dialog will be open:



Gateway	Select the Gateway to communicate over the Flexium Gateway. In this way all the Flexium System in the Network can connected to the HMI.
Local UDP	With Local UDP is only possible to connect to a Flexium System on the same System. This communication is independent from the Network configuration on the local System.
Gateway Address	Used only if the Gateway is used. This is the IP Address of the Gateway.
Gateway Port	Used only if the Gateway is used. This is the Port number of the Gateway.



Scan	To scan the Network with all available Flexium Systems.
Active	Active the selected Flexium System.
Connect by Name	Connect the Flexium System over the Name.

All changes must be activated with the *Connect* button.

2.11.1 FXServer Status in the Taskbar

	No communication with the Flexium System.
	Some Responds are in timeout and repeated.
	Successful connected to the Flexium System.

Page deliberately empty

Summary

3 Context “Production” 43

3.1 Access to the Context “Production” 43
3.1.1 Context “Production” display when starting the HMI 43
3.2 Channel selection 46
3.3 Spindle monitor 46
3.4 Duplicated axes 47
3.5 Axes information 48
3.5.1 Measurement corrections 49
3.5.2 Tool orientation 50
3.6 CNC Blocks Page 51
3.6.1 Split view 52
3.7 Axes Position Page 53
3.7.1 Axes travel limits 54
3.8 CNC functions 55
3.9 Dialog field 57
3.9.1 Data input into the dialog field 57
3.9.2 Deleting the dialog list 58
3.10 MDI mini program 59
3.10.1 Manage mini programs 60
3.10.2 Functions menu 60
3.10.3 Usage 61

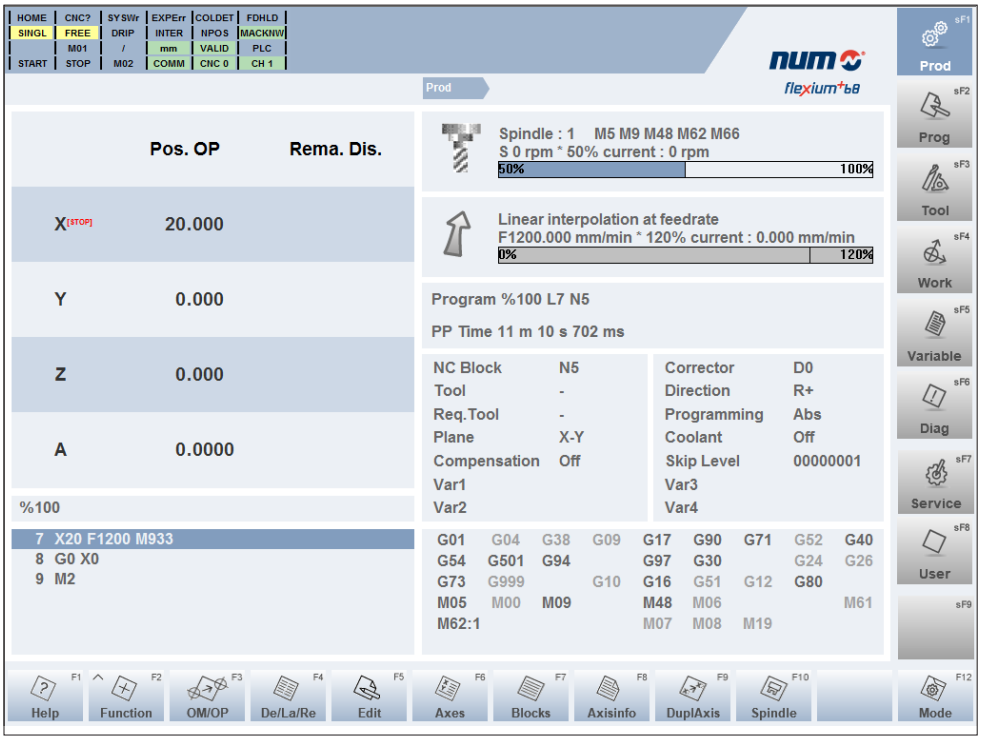
Page deliberately empty

3 Context “Production”

3.1 Access to the Context “Production”

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

3.1.1 Context “Production” display when starting the HMI



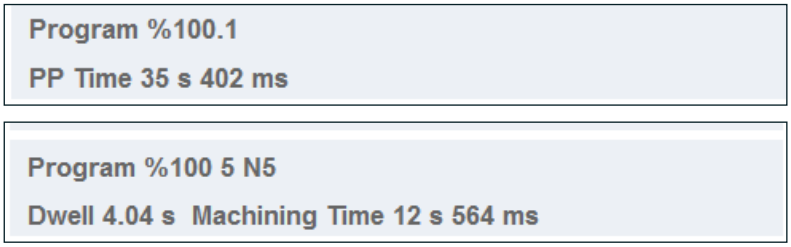
The window showing the current part program number also displays machining or part program time. Part program time is calculated from start to *M02* or *Part Load M function** .

Machining time is only incremented when the axes are moving in G1, G2 or G3 and with spindle rotation active.
Please note the red suffix [STOP] at the X label in the axis position field of this page. It indicates that the said axis movement is inhibited by the PLC Flag WCNC.Axis[..].Stop. This situation has been done to simplify the diagnostic and is the only case where a feed hold is not indicated in the status field

(*) Part Load M function is an M code defined by machine parameter P37 N1 used to indicate that the current part is finished and another one can be loaded. It is useful to count the time to execute one part when the program is continuously executed in a loop.

On lathe channels a symbol next to the radial axis position indicates when the display is shown at the diameter (see below).

Additionally when a dwell is active (G4 F..), this windows shows the remaining time.



Red block is reserved to display 4 individual values configured on service context sF7.
See Chapter 9.3.7.

NC Block	N0	Corrector	D0
Tool	-	Direction	R+
Req.Tool	-	Programming	Abs
Plane	X-Y	Coolant	Off
Compensation	Off	Skip Level	00000001
Piece Counter	549		
		Piece ID	46874

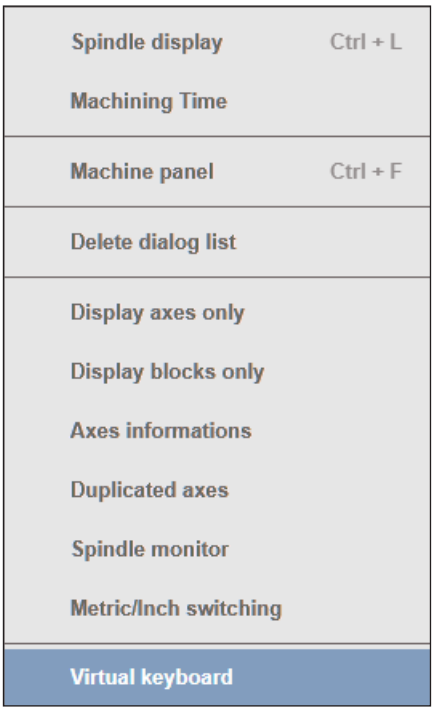
Example of configuration

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 F5 to select Edit mode (current program in NC memory)
Press F4 to toggle between Delta and Lag display.
Press F3 to toggle between OP (Program origin), OM (Machine origin) and Relative display.

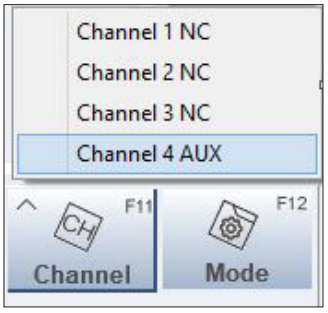
The values of the Relative display are reset as soon as this mode is selected. The description of the relative measurement of distances can be found in chapter ‘3.7 Axes Positions Page’. The Relative display must be activated in the context “Service”. For detailed information, refer to chapter ‘9.2 General Settings Page’ in this manual.

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



3.2 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.



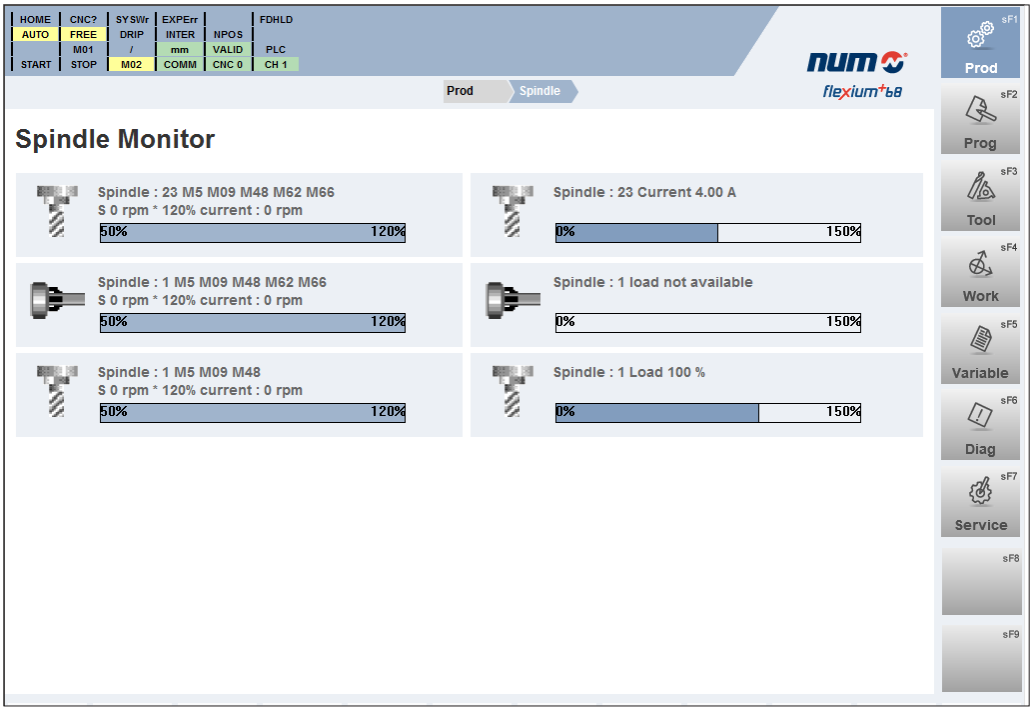
Open the channel selection	☞ F11
Select the desired channel	☞ ↑ or ↓
Validate	☞ Enter ↵

3.3 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 or spindle current (according to the machine settings).

To access the spindle monitor page	☞ F10



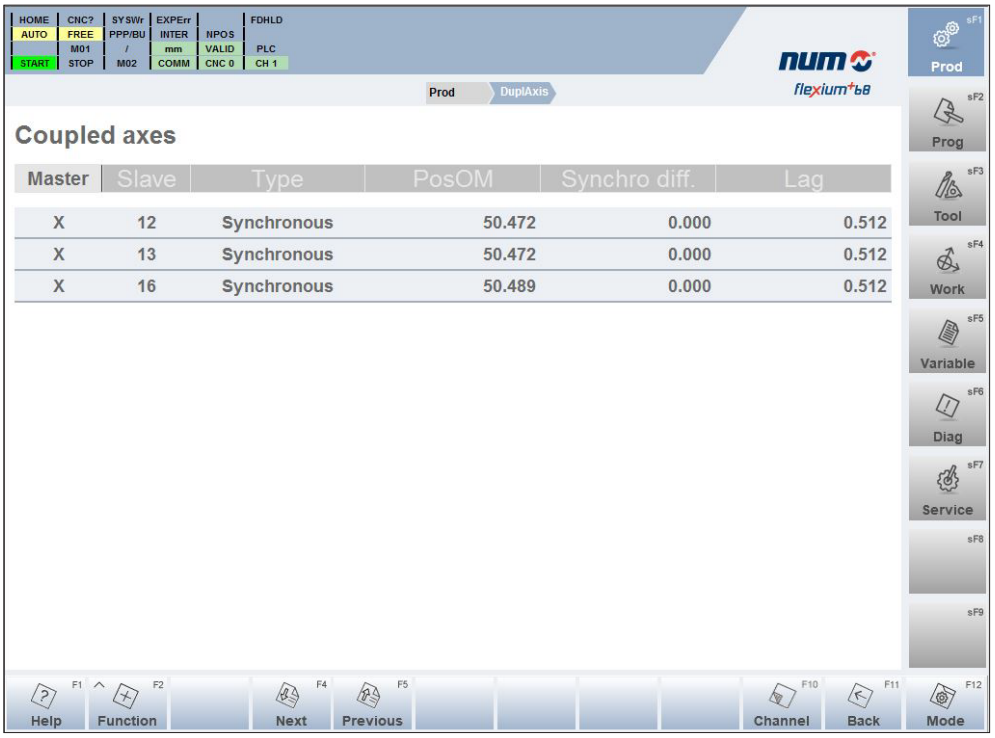
To exit the spindle monitor page	☞ F11

3.4 Duplicated axes

This page gives information on the coupling, duplication and synchronization of axes.
For detailed information about axes duplication and synchronization, please refer to the Flexium+ Programming manual M00040.

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

To access the Duplicated Axes page	☞ F9



To exit the Duplicated Axes page	☞ F11

3.5 Axes information

This page gives supplementary information on axes and program states.

To access the axes information page	☞ F8

The field ‘Programmed Circle’ gives the coordinates of the center and the radius value (if programmed) in case of circular interpolation.

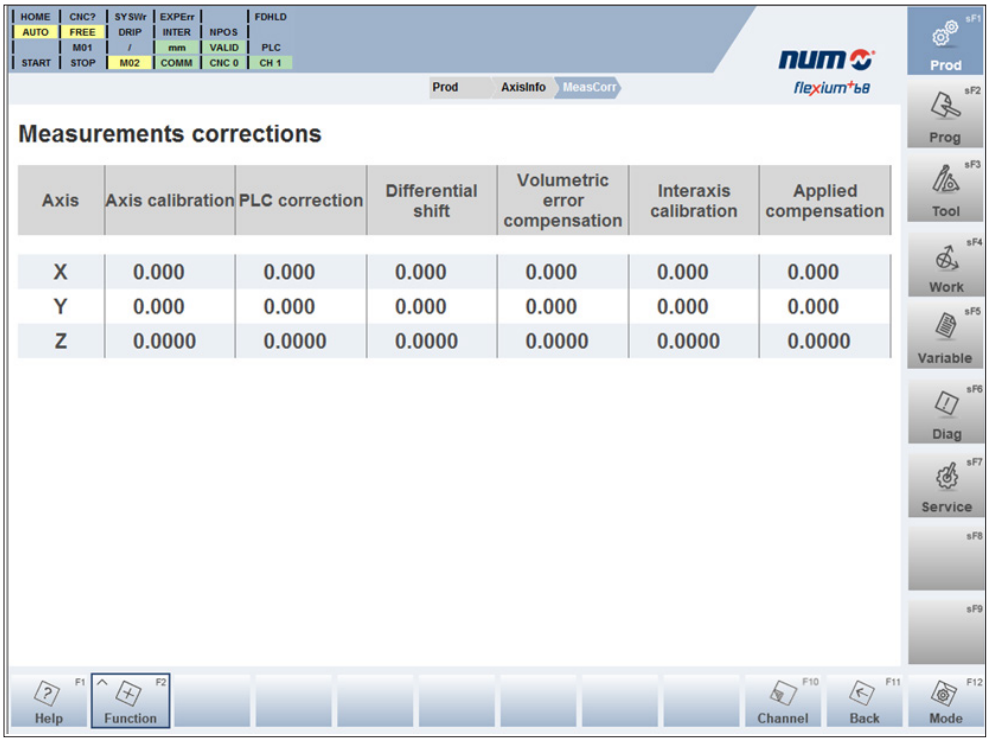
To exit the axes information page	☞ F11
To enter the Measurement corrections	☞ F8
To select Tool Orientation	☞ F7

Please note the line @X2=Y in the Axis Address Table.
This is related to the function Indexed Axis Name described in the Extended Programming Manuel.

3.5.1 Measurement corrections

This page gives the overview over all active measurement corrections.

Access this page in the Axis information’s	☞ F8



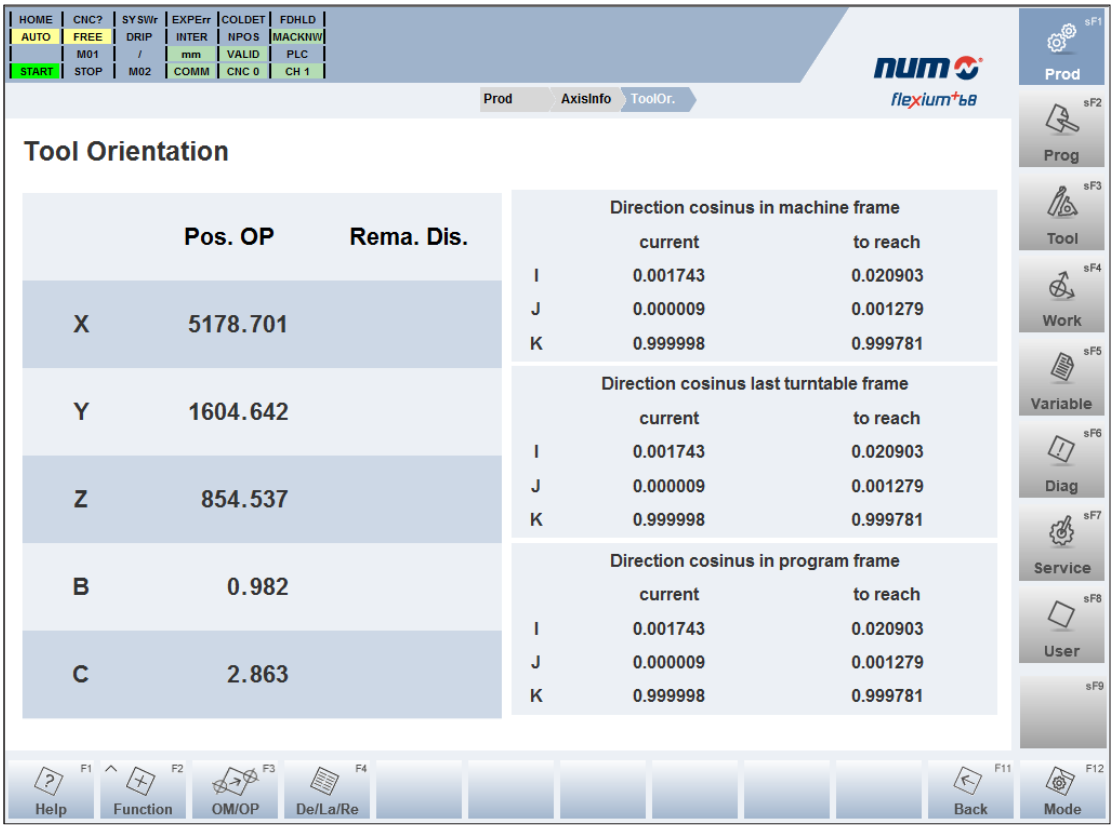
To exit the Measurement corrections	☞ F11

Axis calibrations	Active correction from the axis calibrations.
PLC corrections	Corrections from the PLC.
Differential shift	Differential offset value.
Volumetric error Compensations	Measure correction for “Volumetric error compensations” (Vecomp).
Interaxis calibrations	Active correction from the interaxis calibrations.
Applied compensation	Sum of all active Axis (or Spindle) measurement corrections.

3.5.2 Tool orientation

This page displays the tool orientation in case of RTCP or Inclined plane.

Access this page in the axis information	☞ F7



The left window defines the direction cosinus of the tool direction in relation to the machine, table and program referentials. This page is only shown when RTCP is active

To exit the tool orientation page	☞ F11

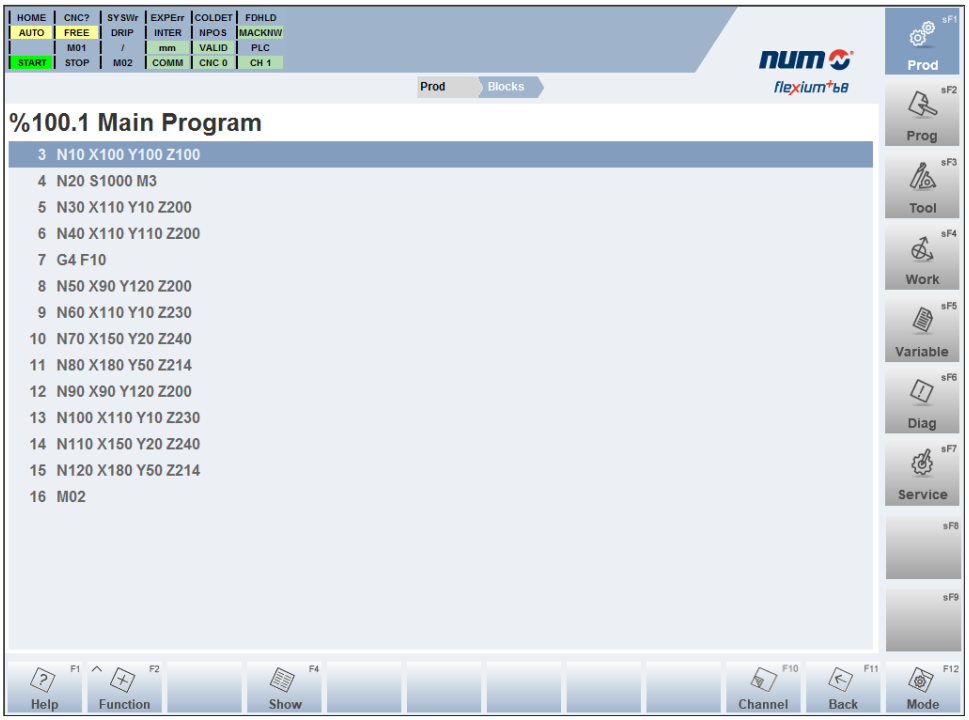
3.6 CNC Blocks Page

This page displays the CNC blocks of an already started part program 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 blue bar and the lower part of the screen shows the blocks still to be executed.

If the part program is executed completely or NC-Reset is pressed only the first line of the part program is displayed.

To access the CNC Blocks page in the main page *“Production“* context.	☞ F7

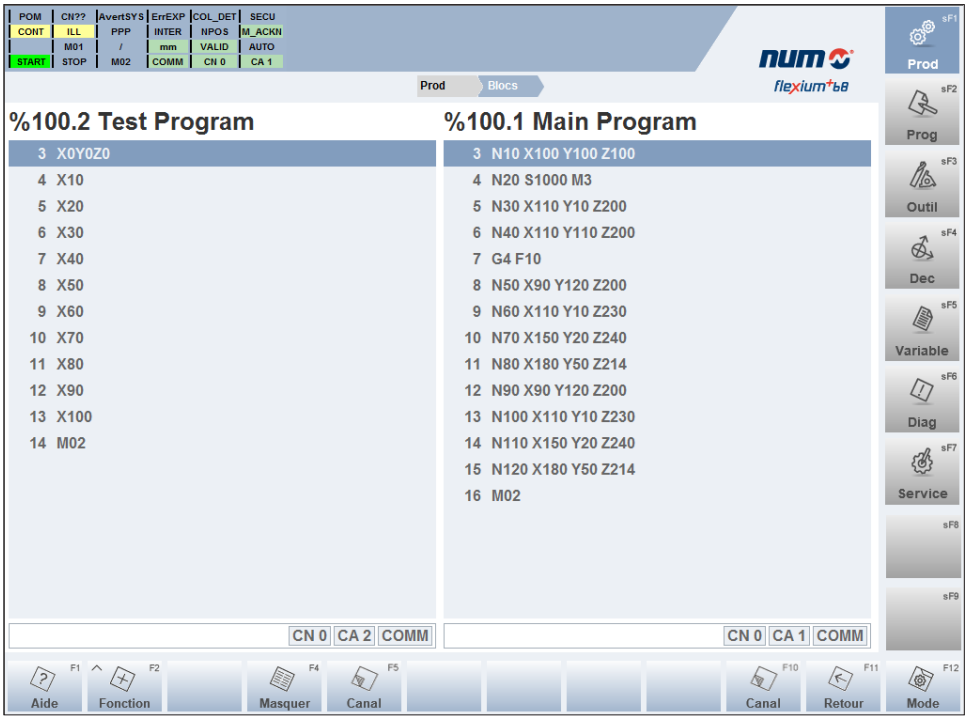


Possible actions

Press F4 to show the second blocks view.

3.6.1 Split view

This page shows two views side by side. Press F4 in the CNC blocks page to access this view.



By means of this page, the user has the possibility to show the CNC blocks of two channels at the same time. On the right side, the blocks of the currently selected channel are visible.

The second view on the left shows the blocks of another channel.

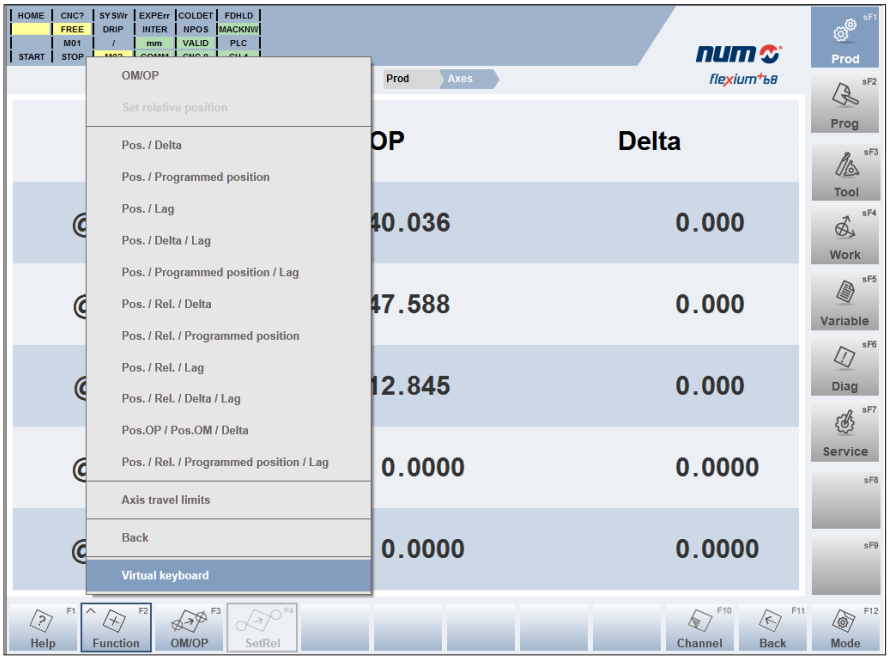
Possible actions

Press F4 to hide the second blocks view.
Press F5 to change the channel of the second blocks view.

3.7 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*”.

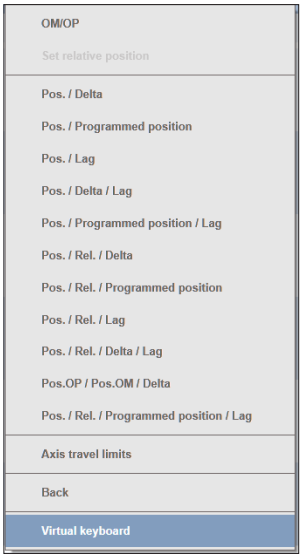
To access the axes position page	☞ F6



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↵

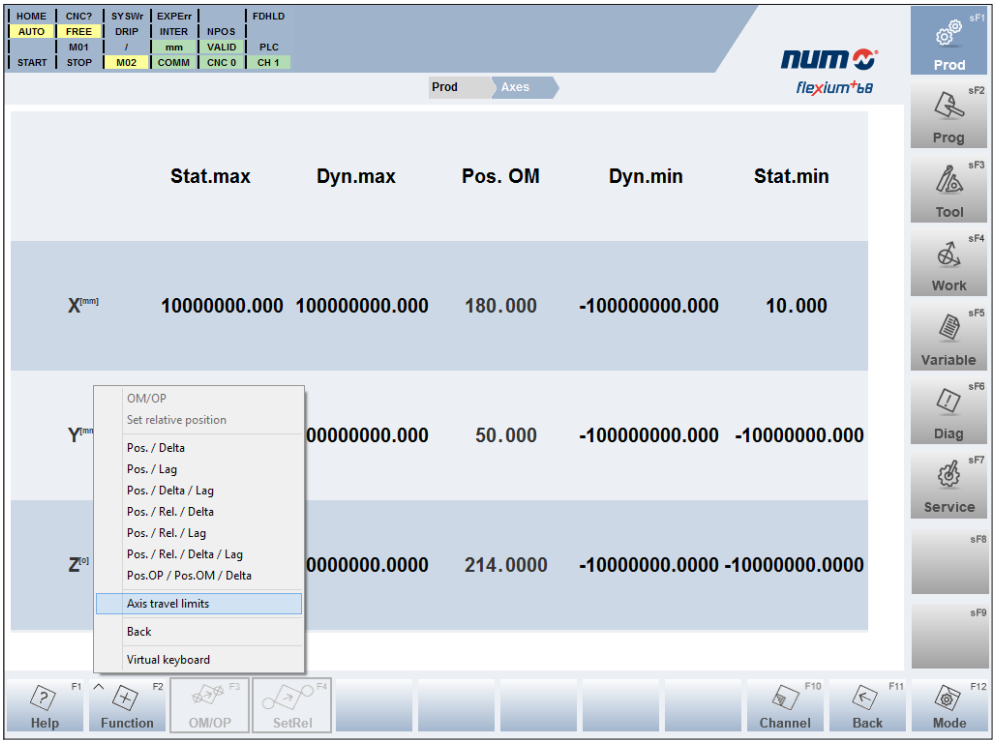
Starting with version 4.2 display of the programmed position is available as an additional option.
The following display modes may be selected: this text and the following picture must remain together.



3.7.1 Axes travel limits

For machine setup, knowing the static as well as current dynamic travel limits can be useful.

To access the axes travel limits page, select *“Axis travel limits”* in the function menu F2.



3.8 CNC functions

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

Navigation between windows and fields in a given context (mostly with T, t, b, l, r) can only be accomplished when the CNC function bar is not activated!

For further information about CNC functions, refer to chapter 13 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 (Manual Data Input).
Press F2 to select the single mode.
Press F1 to select the automatic mode.

In addition to activate M01 and Block skip, the button *NCFunc* (F10) also permits to set the block skip level.

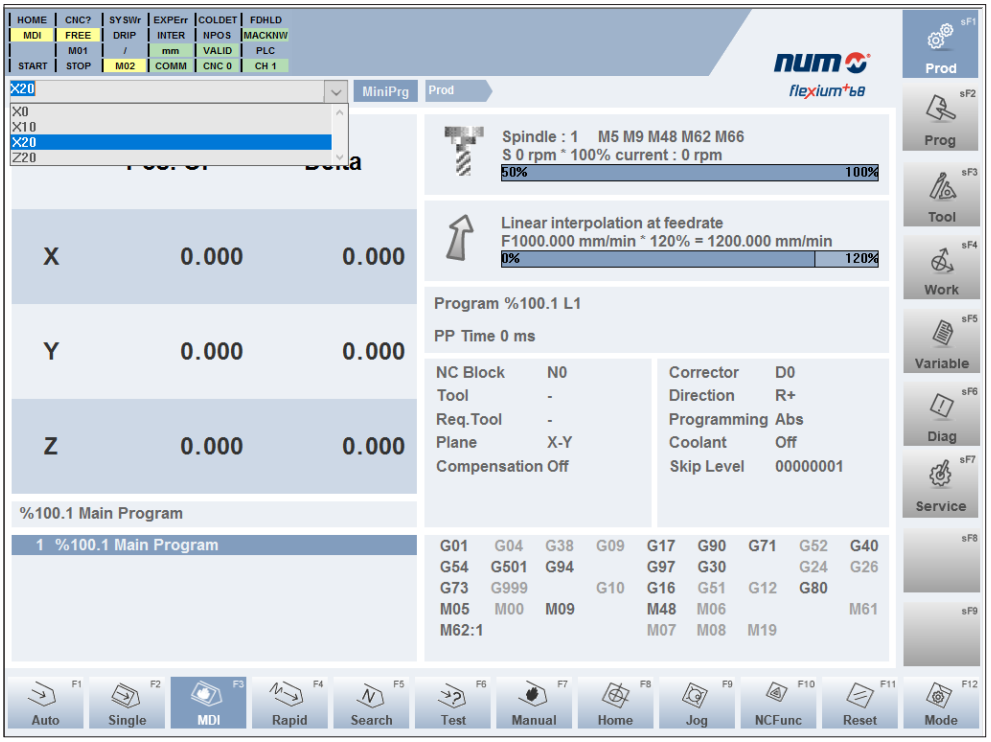
Block skip level is used to select among 'skippable' blocks the ones that should be skipped or executed (Refer to block skip in Flexium+ Programming Manual M00040.

When block skip level is selected with the F10 button, the following dialog will be shown:

3.9 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 and it is also used in the case of a message display with wait for an operator response.

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



3.9.1 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

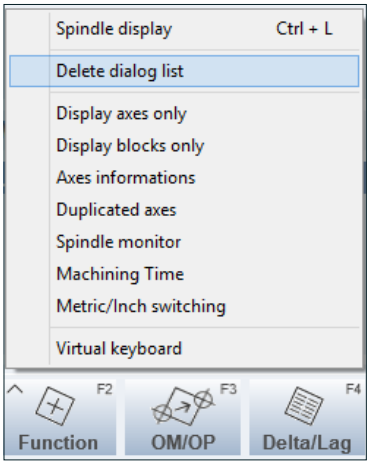
Contrary to previous version, the text in the dialog can be entered in lower or or uppercase (in order to offer string handling to end user). It is possible to force dialog in uppercase only by a setting in the service context. If the dialog list is already open, entering the first character of data sets the focus to the first similar value contained in the list.
See General settings in Service context.

3.9.2 Deleting the dialog list

The entire dialog list can be deleted if needed.

This function is only available in the context “Production”.

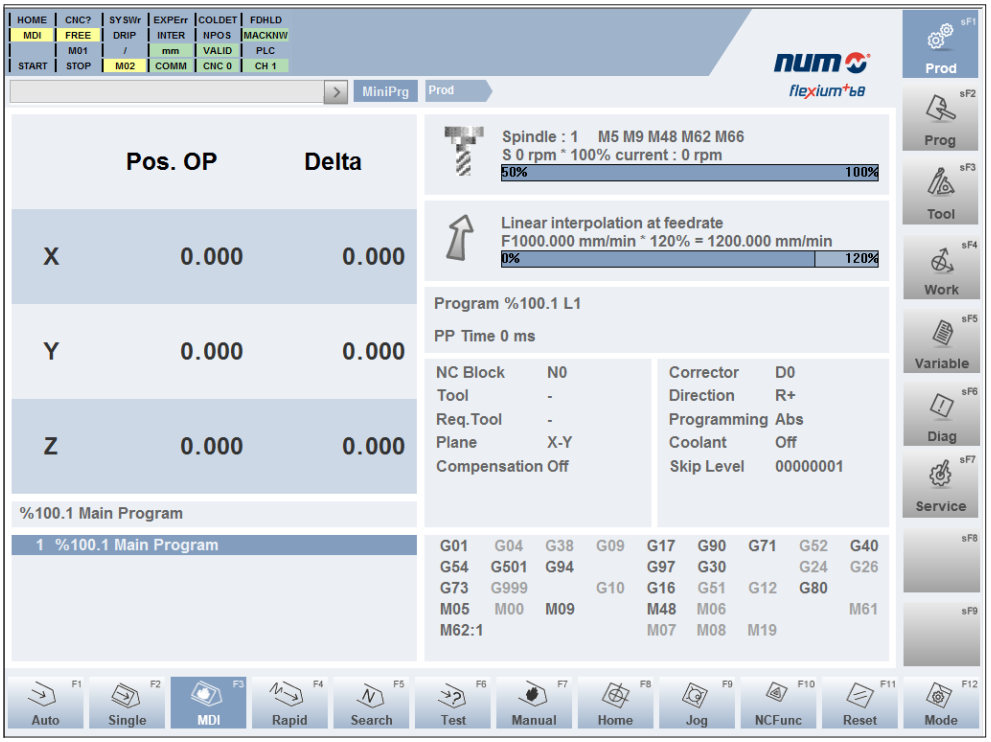
Activate the sub-context bar	☞ F12
Select function	☞ F2



Select “Delete dialog list”	☞ ↑ or ↓
Validate	☞ Enter ↵

3.10 MDI mini program

When the system is in MDI mode, the button *“MiniPrg”* is available at the top, near the dialog field.



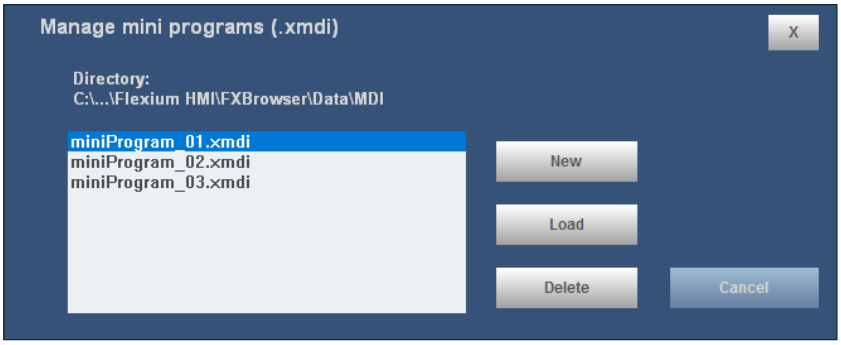
Press the button *“MiniPrg”* or use the shortcut Ctrl + [M] to select the mini program mode.

The dialog *“Manage mini programs”* shows up. If already a mini program is active, the user gets access to the functions menu instead of the mentioned dialog.

3.10.1 Manage mini programs

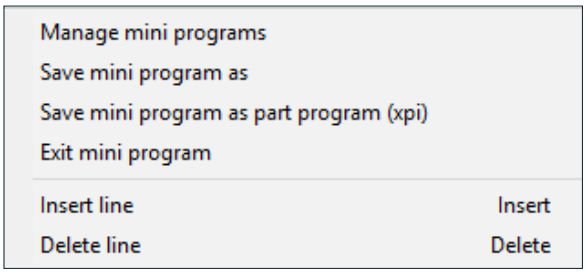
To start a MDI mini programs, choose *"New"* to create a new one or *"Load"* to load an existing one. An existing mini program can be deleted with *"Delete"*. The used extension is always *"xmdi"*.

Note that when a mini program is active, it is saved automatically after every change.



3.10.2 Functions menu

Press the button *"MiniPrg"* or use the shortcut Ctrl + [M] while a mini program is active to access this menu.

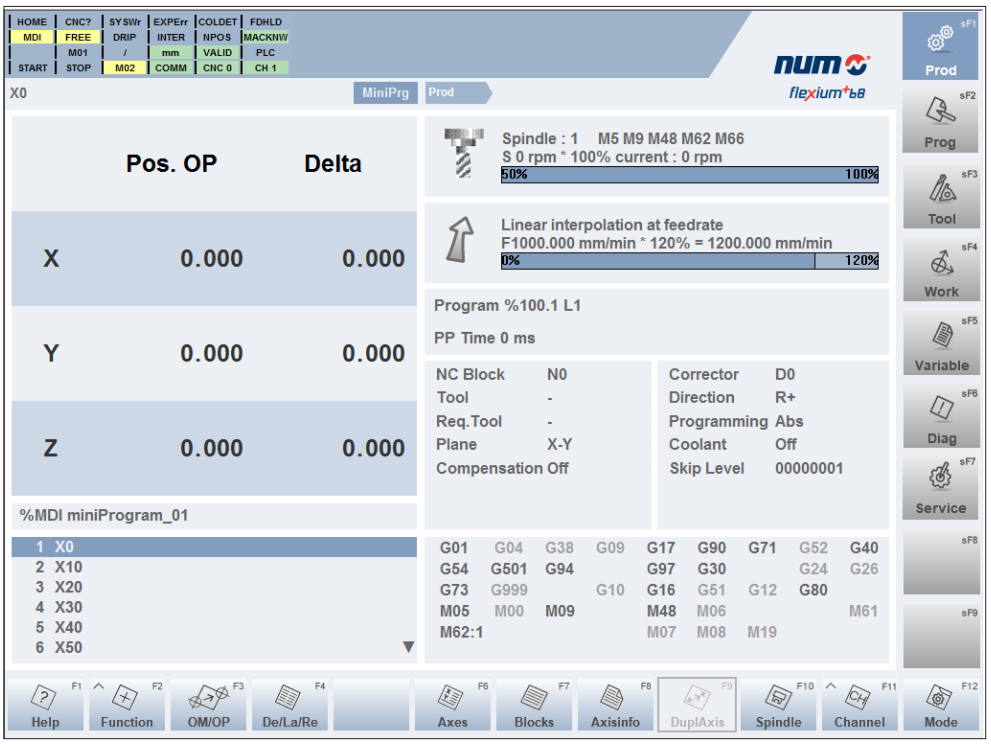


Possible actions

Manage mini programs	Open the dialog *"Manage mini programs"* to create, load or delete mini program files.
Save mini program as	Open a dialog to save the mini program with a different name.
Save mini program as part program (xpi)	Open a dialog to save the mini program as a part program.
Exit mini program	Exit the mini program and go to standard MDI mode.
Insert line	Insert a new line at the current position.
Delete line	Delete the current line.

3.10.3 Usage

If a mini program is active, the block display in the sF1 production context is used for user input.



Possible actions

Press ↑ or ↓ to change the active line.
Press Enter to edit the current line.
Press Enter while editing a line to apply changes.
Press Esc while editing a line to abort changes.
Press PgUp or PgDn to show the next six lines.
Press Home to go to the first line.
Press End to go to the last line.

When scrolling over the last line, a new blank line is inserted.

The active line gets incremented when a block has been executed.

Page deliberately empty

Summary

4 Context “Programming” 65

4.1 Access to the Context “Programming” 65
4.1.1 Representation of data locations in your system 65
4.1.2 Definition of the files existing on PC disk and in CNC memory 65
4.1.3 Start-up window of the context “Programming” 66
4.2 Moving between Windows 67
4.3 Selecting Directory and Folder on the PC disk 67
4.4 Selecting a File on PC disk 68
4.5 Creating a new part program on PC disk 71
4.5.1 Hidden programs and subroutines 72
4.5.2 Main program 72
4.5.3 Part programs created with another editor or on an external system 72
4.5.4 Format of an ISO part program 72
4.6 Creating a new Compact part program on PC disk 73
4.6.1 Compact programs created with another editor or on an external system 73
4.6.2 Format of a Compact part program 74
4.7 Creating a new “Usinage” Program on PC disk 75
4.7.1 “Usinage” programs created with another editor or on an external system 75
4.7.2 Format of a “Usinage” program 76
4.7.3 The include function 76
4.7.4 The Loop and Include functions 76
4.8 Creating a new tool offset file on PC disk 77
4.8.1 Tool offset files created with another editor or on an external system 78
4.9 Creating a new work offset file on PC 79
4.9.1 Work offset files created with another editor or on an external system 80
4.10 Editing a file on PC disk 81
4.10.1 Keyboard functions 82
4.11 Editing a “Usinage” file on PC disk 83
4.11.1 Keyboard functions 84
4.12 Including a Part Program in an “Usinage” File 84
4.13 Creating a Loop in an “Usinage” File 84
4.14 Editing NUM CNC Cycles 85
4.15 Drip Feed Mode 86
4.15.1 Principle of drip feed mode 86
4.15.2 Drip feed on machines with several axes channels 87
4.15.3 Block search with drip feed mode 88
4.16 Loading a file into CNC memory 89
4.16.1 Load one file only 89
4.16.2 Load several successive files 89
4.16.3 Load several non successive files 89
4.16.4 Load a full folder 90
4.17 Protected Memory Area Management 91
4.17.1 Selecting CNC Memory Zones 92
4.17.2 Closing the memory zones 92
4.18 NC Memory 93
4.19 Selecting the Active Part Program 94

4.20 Saving Files to PC Disk 95
4.20.1 Save one file only 95
4.20.2 Save several successive files 95
4.20.3 Save several non successive files 95
4.21 Editing a Program directly in NC Memory 96
4.22 Deleting Files on PC Disk or NC Memory 98
4.22.1 Delete one file only 98
4.22.2 Delete several successive files 98
4.22.3 Delete several non successive files 98

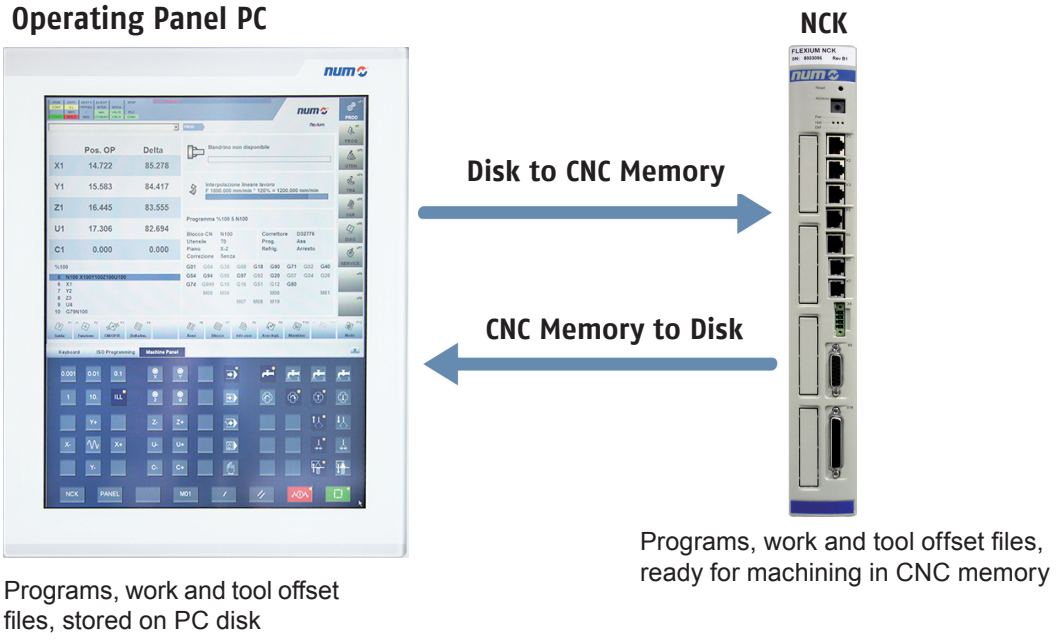
4 Context “Programming”

4.1 Access to the Context “Programming”

To access the context “*Programming*”	☞ Shift + F2
To close pop-up menus and windows, press	☞ Esc

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

4.1.1 Representation of data locations in your system



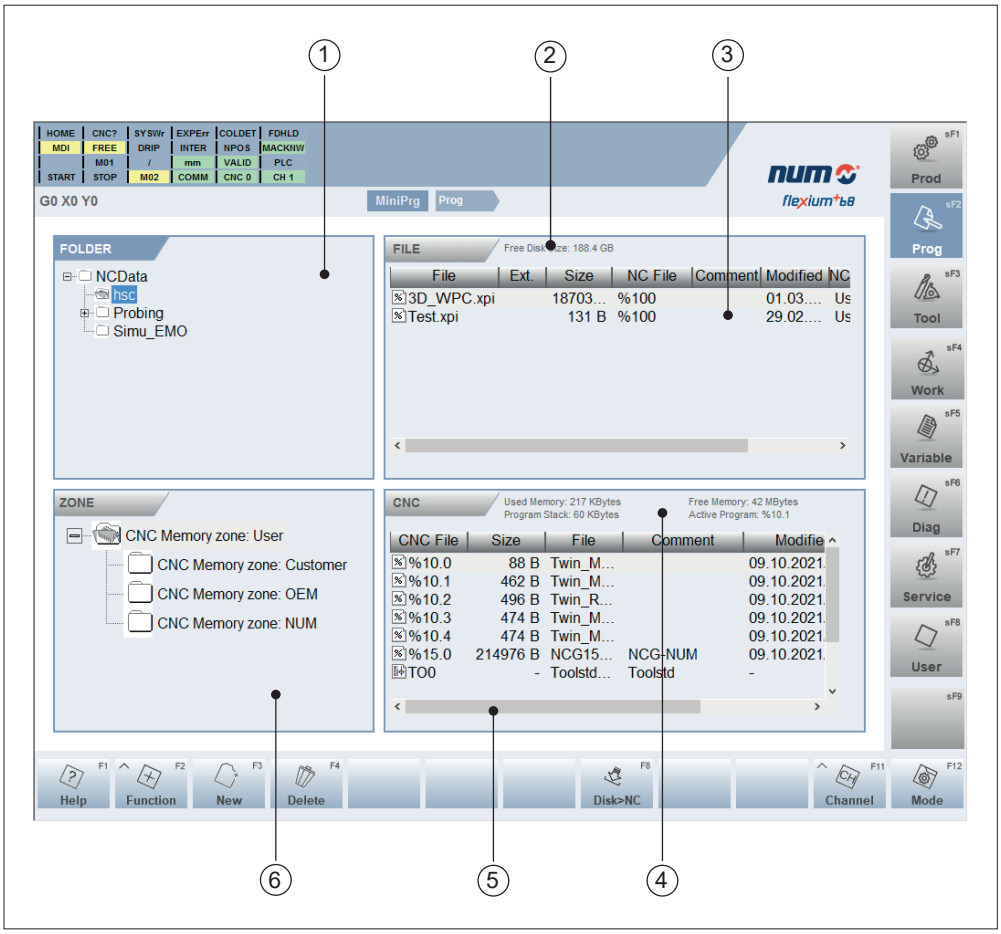
Programs, work and tool offset files, stored on PC disk

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

4.1.2 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 M00040.
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.
Usinage programs *.usi	A usinage 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 have to be loaded according to the tools being used.
Work offset files *.dat	Contain all work offset data. They are resident in CNC memory and have to be loaded according to the part being machined.

4.1.3 Start-up window of the context “Programming”



It is important to switch off the system properly (see 2.2.2)
If the system is switched off abruptly, the programs may be not up to date and error 2007 will be issued.
In this case, once the programs have been checked it is possible to acknowledge this error via the F2 button (see next page)

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.

The active window (Folder, File, Zone, CNC) is identified by a darker upper-left label (Folder in this case).

4.2 Moving between Windows

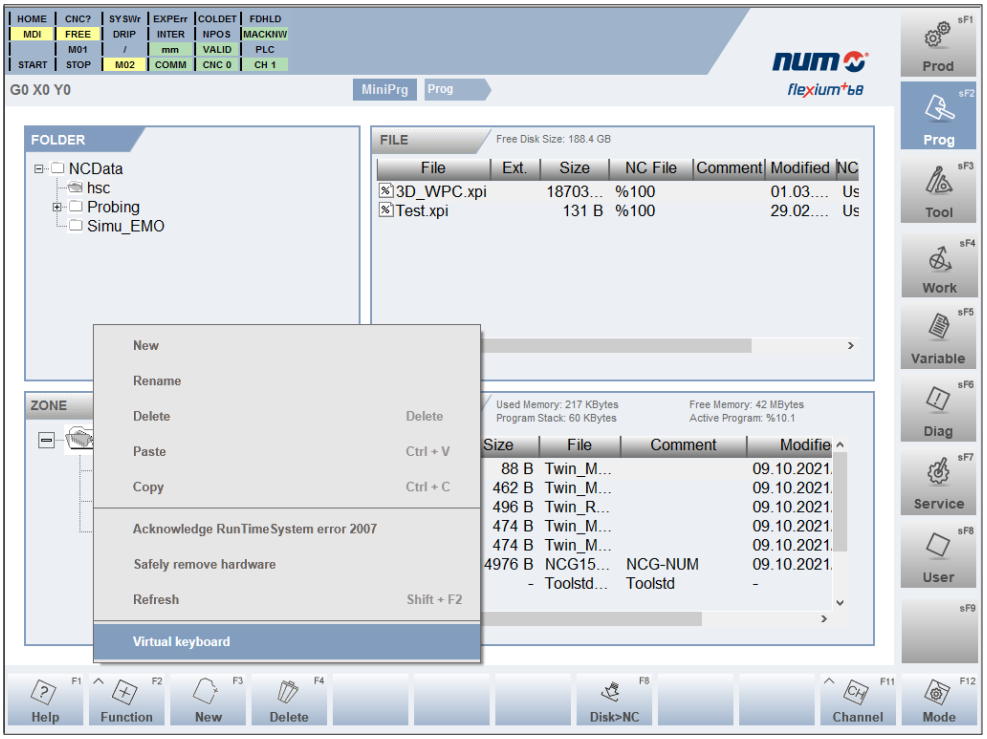
Select the window	☞ Tab⇆

4.3 Selecting Directory and Folder on the PC disk

Select the window	☞ Tab⇆...
Select the desired directory or folder	☞ ↑ ↓, ← or →
Validate	☞ Enter↵

STOP

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

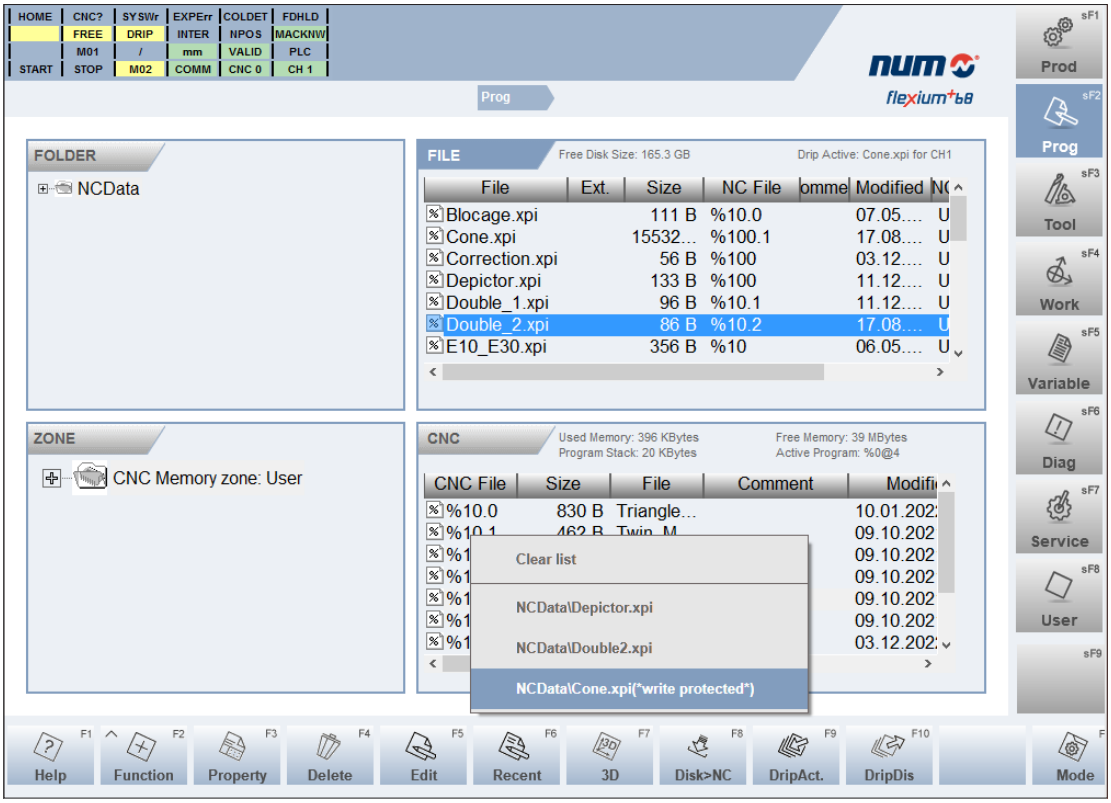
Press F4 or Delete to delete a folder
Press F3 to create a folder
Press F2 to copy, paste, delete, rename, create a folder

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.

4.4 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 of a line	☞ ← or →

It is possible to display a list of the recently opened program by pressing F6. A double click in this list will open the selected file for edition.

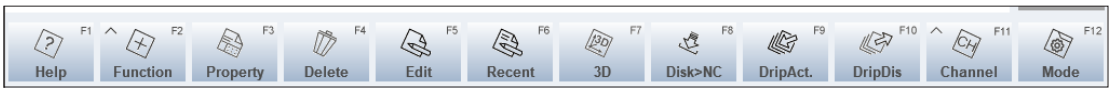


Flag description

The PC disk view has an extended property column called “Attribute”.
This attribute returns the following properties of the respective file.

NC File	Comment	Modified	NC Memd	Attribute
30.1	commande DELETE	05.05.10/09:04	User	---a
30.9	nd channel	06.06.17/14:35	User	h-ma
32.0	BSET BCLR	01.06.17/10:40	User	----

Flag	Description
h	Hidden programs and subroutines (see Chapter 4.5.1).
m	Main program (see Chapter 4.5.2).
a	Archive flag, the file has changed since the last download.

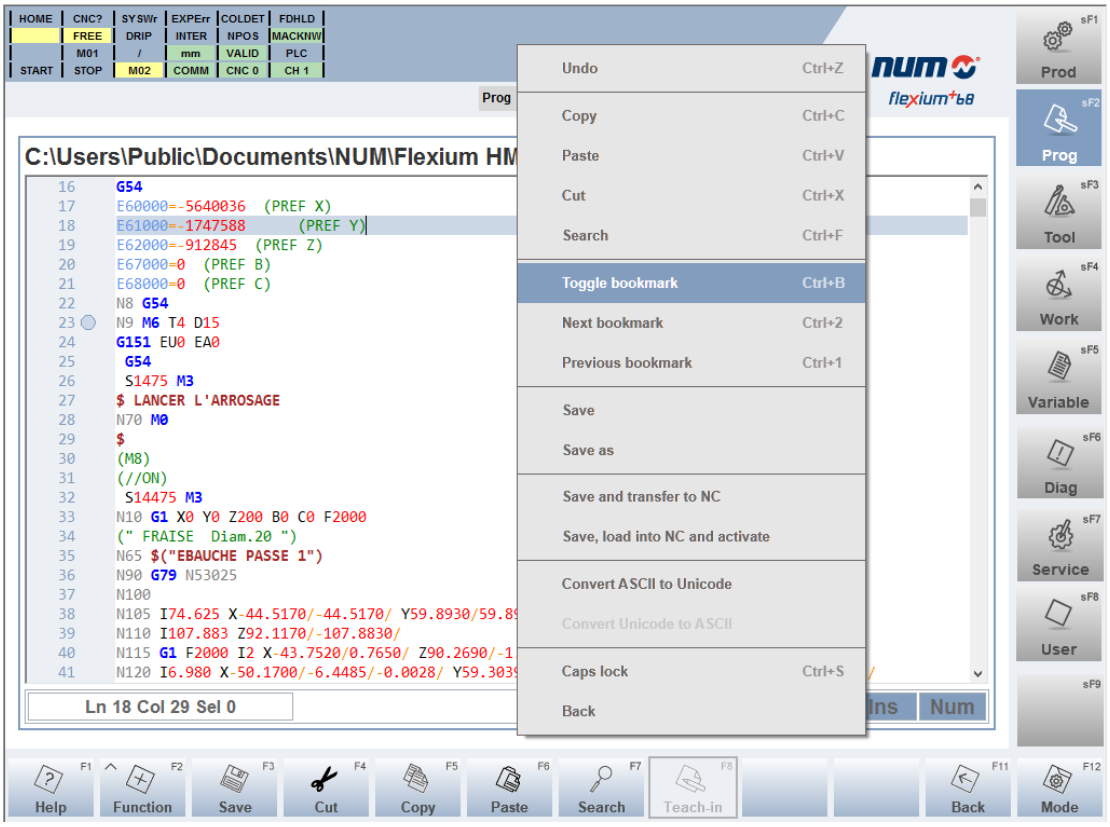


Possible actions

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 F6 to display a list of the 10 most recently edited files.
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↵

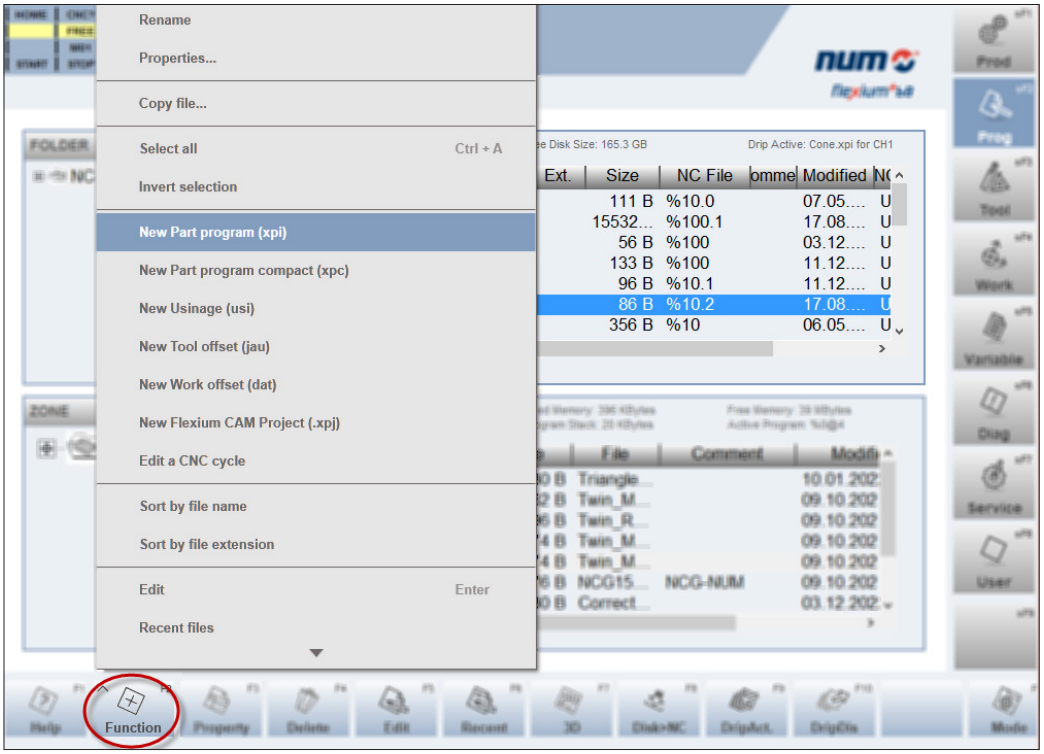
If the editor protection has been activated in the context “Service” or by the PLC, the following will not be available:
F8 Disk>NC
F5 Edit
F4 Delete
F3 Property
In the F2 Function menu: Delete, Rename and Property.

From 4.2 a new file editor is available with syntax colorisation and additional features (Undo, Bookmark, Caps, conversion etc.). These functions can be invoked by a right click



4.5 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).

Flexium CAM project (.xpj)
A Flexium CAM project is an interactive programming method specific to a certain and optional type of operation (Milling, Grinding, OEM specific etc.) Refer to the particular manual for each case.

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 ↵

4.5.1 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.

4.5.2 Main program

The check box *‘Main Program’* is used in case of transfer of a full folder to the CNC memory.
The part program whose *‘Main Program’* box is checked will be automatically activated.

4.5.3 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 ‘4.4 Selecting a File on the PC disk’)	
Select *“Properties”* Enter the properties as described above	☞ F3

4.5.4 Format of an ISO part program

Refer to the Flexium+ Programming manual (Reference: M00040).

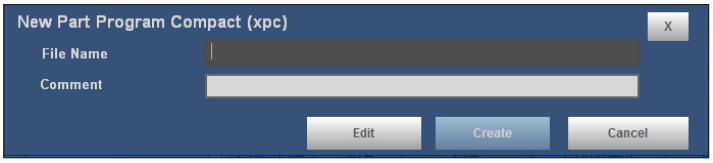
4.6 Creating a new Compact part program on PC disk

A compact part program is a succession of several ISO program in a single file. it is generally made via the function “Unload, All programs” of a prior generation NUM CNC.
To be able to manage such files with the HMI, they need to have the extension *.xpc.

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.

4.6.1 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 ‘4.4 Selecting a File on the PC disk’)	
Select Properties Enter the properties as described above	☞ F3

4.6.2 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
```

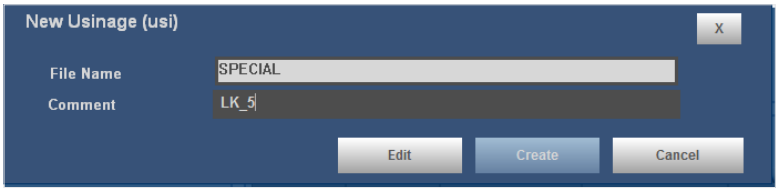
4.7 Creating a new “Usinage” Program on PC disk

An “Usinage” part program is an ISO program with some meta commands like loops and inclusions.

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 Usinage (.usi)	☞ ↑ 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 Usinage file appears in the right hand window of PC disk and may now be edited.

4.7.1 “Usinage” 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 ‘4.4 Selecting a File on the PC disk’)	
Select Properties Enter the properties as described above	☞ F3

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

4.7.2 Format of a “Usinage” program

The programming format is the same as an ISO part program, but the usinage program offers some new possibilities
Normal ISO code:

```
%100 (Usinage 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
```

4.7.3 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 (Usinage 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 (Usinage with Include command only)
#include c:\ncdata\cn100k.xpi
N99 M2
```

4.7.4 The Loop and Include functions

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

```
%100 (usinage 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
```

4.8 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.

4.8.1 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 '4.4 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.

The Format of an extended tool table is described in chapter 12.4.15.

4.9 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, comment, channel and dimension.	

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.

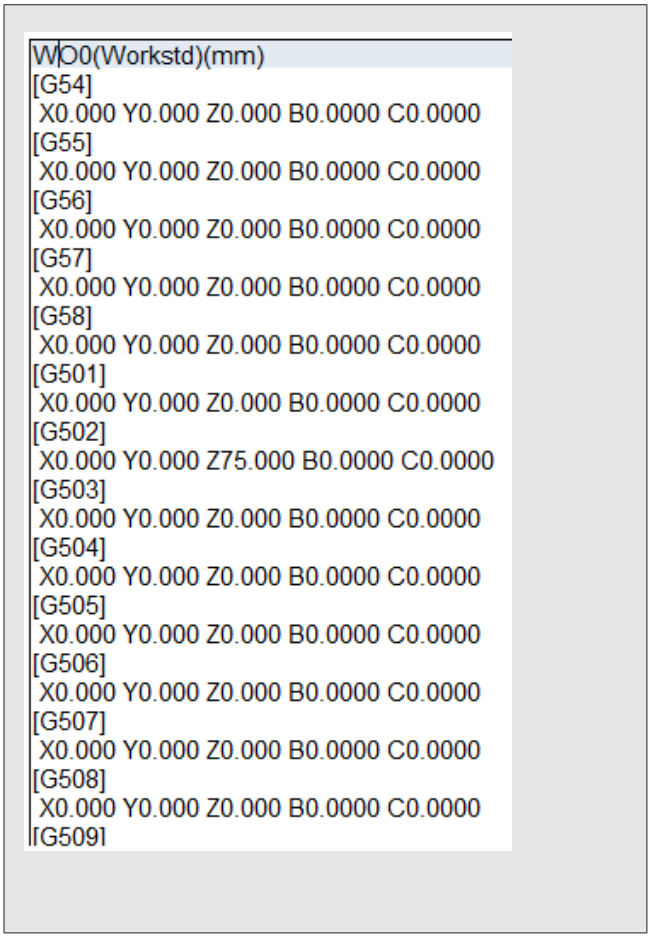
Select a dimension button	☞ Tab ⇆ ...
Toggle between “mm” and “inch”	☞ ← or →
In multi-channel configuration, the channel number corresponding to the work offset file must be entered. After having entered all necessary properties of the file, press Enter ↵ .	
The new Work offset file appears in the right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

4.9.1 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 '4.4 Selecting a File on the PC disk')	
Select Properties	☞ F3
Enter the properties as described above	

Format of a work offset file (partial)



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.

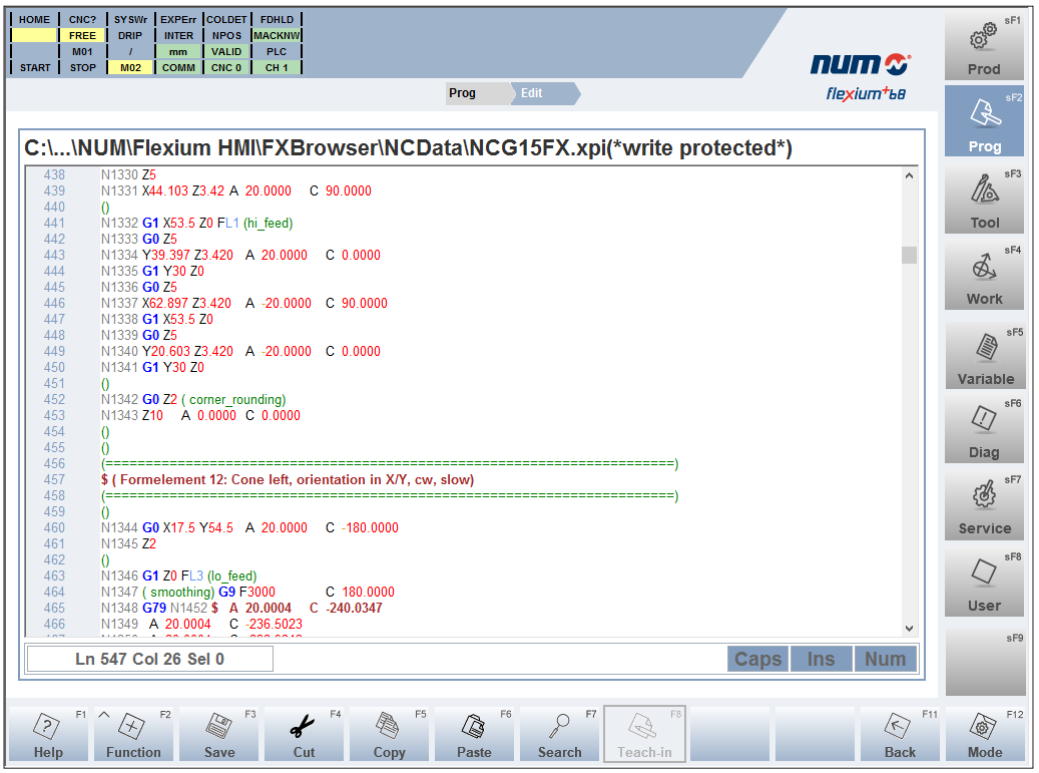
4.10 Editing a file on PC disk

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 “*Usinage*” file, the special “*Usinage 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 F8 for Teach-in (optional)
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↵

4.10.1 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

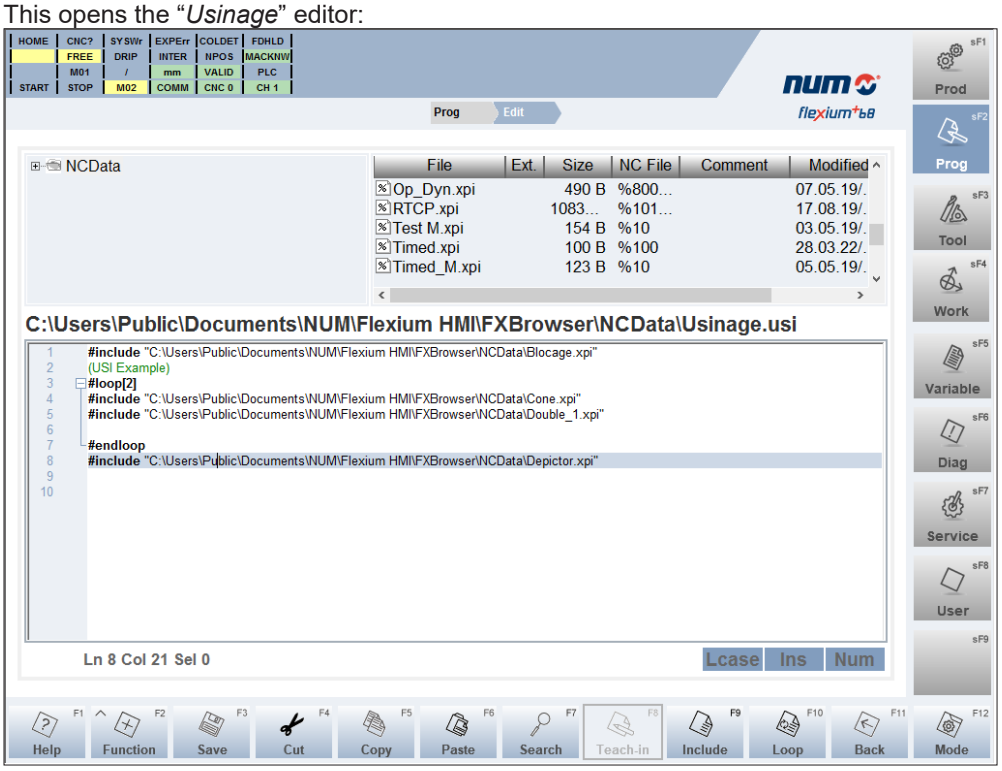
4.11 Editing a “Usinage” file on PC disk

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

Usinage files are a type of file to describe a full machining process. It contains CNC commands and part programs to include with possibility of loops (see example below).

Select the right hand window on the PC disk	☞ Tab
Select the desired *“Usinage 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 “*Usinage*” editor:



Possible actions

Press F10 to insert a loop command.
Press F9 to include a selected program.
Press F8 for Teach-in
Press F7 to search for a string.
Press F6 to paste.
Press F5 to copy.
Press F4 to cut.
Press F3 to save the *"Usinage"* file.
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter

4.11.1 Keyboard functions

They are the same as for the file editor.

To enter a tabulator: ☞ Ctrl + Tab⇆

4.12 Including a Part Program in an “Usinage” 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

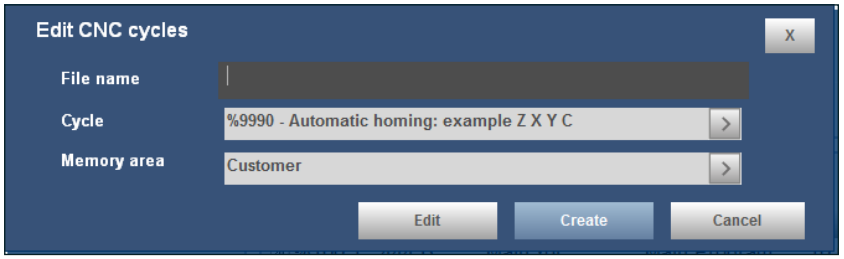
4.13 Creating a Loop in an “Usinage” File

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

4.14 Editing NUM CNC Cycles

Using this function 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

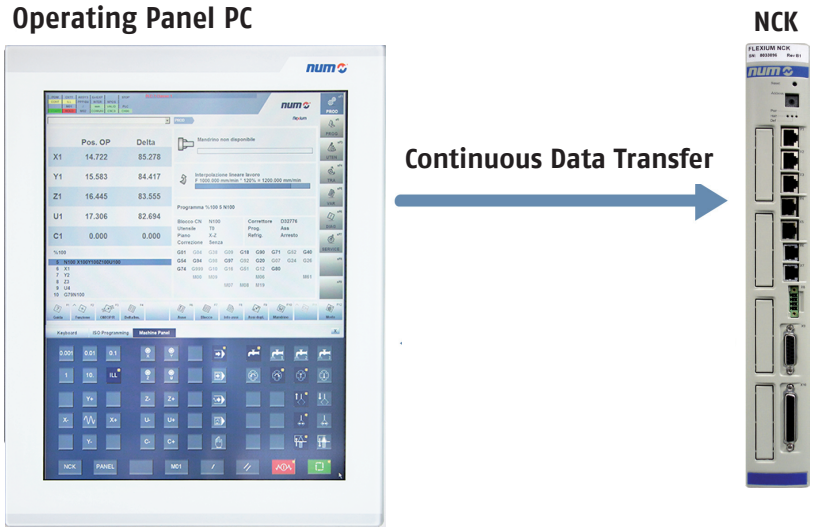
These cycles are available if the license “Turning cycles” is present:
%10033 / %10063 / %10063.2 / %10063.4 / %10064 / %10064.1 / %10065 / %10066
%10081 / %10082 / %10083 / %10084 / %10085 / %10087 / %10089

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

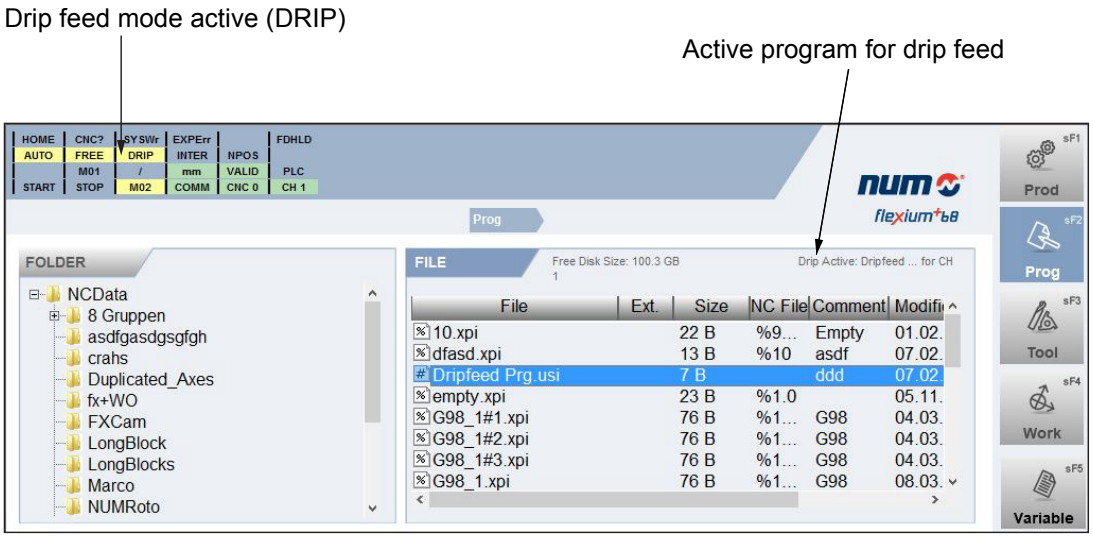
4.15 Drip Feed Mode

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.

4.15.1 Principle of drip feed mode



Select the right hand window on the PC disk	☞ Tab⇆...
Select the desired part or *"Usinage"* 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



Press the *"NC-Start" or "Cycle-Start"* button 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.

4.15.2 Drip feed on machines with several axes channels

It is possible to use the drip feed mode on machines with several axes channels, but only one channel can receive its program by drip feed.

Drip feed is only available on common channels.

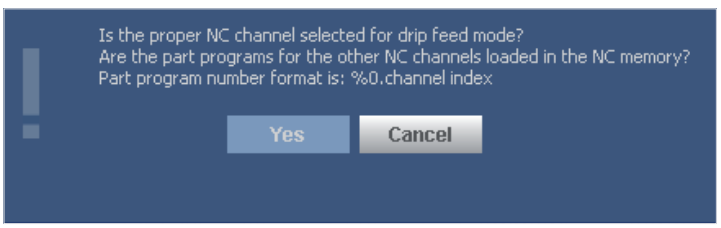
Operation:

- 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 an validate with Yes
Press the *“CYCLE”* key on the machine panel to execute the program.

To exit drip feed mode: select an active part program.
(See chapter 4.19 Selecting the active part program).

4.15.3 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 11 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 11.4.6 "*Sequence Number Search*" in this manual.

4.16 Loading a file into CNC memory

4.16.1 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 ‘*9.2 General Settings Page*’.

4.16.2 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

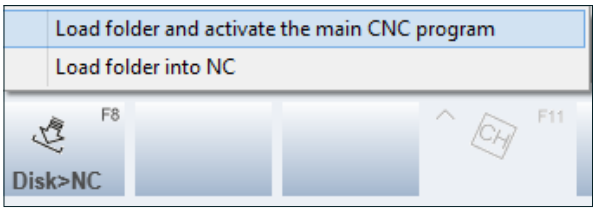
4.16.3 Load several non successive files

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 has to be in *Reset / M02* state.

4.16.4 Load a full folder

Select the left hand window on the PC disk	☞ Tab⇆...
Select the desired folder	☞ ↑ 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 folder into CNC memory. If a part program in this folder has got the Main Program property it can be activated if you select *'Load Folder and Activate'*.	☞ F8



4.17 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).

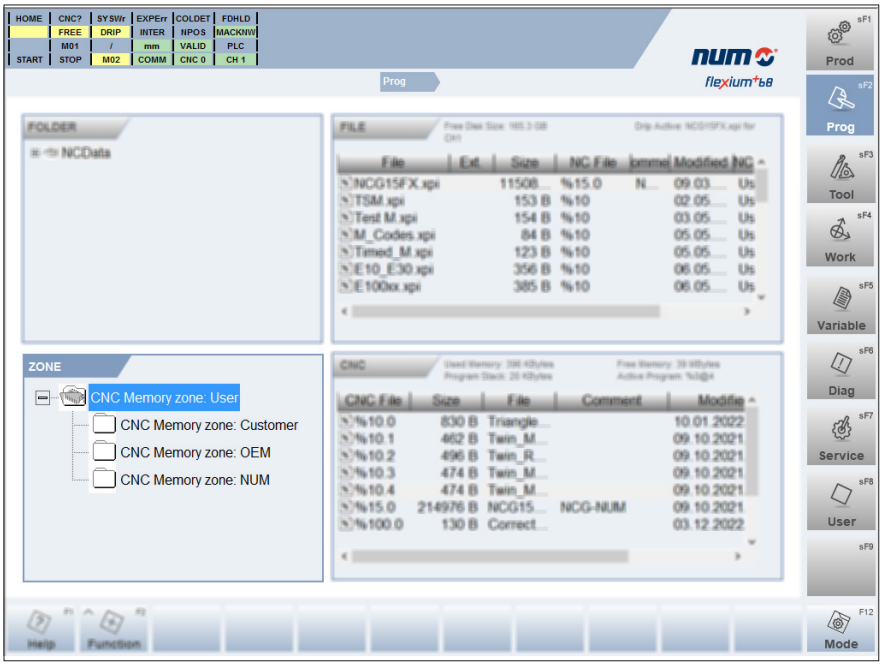
Protected areas are divided into two categories:
- No password protection: free access to all programs in this specific area
- Password protection: all programs in this area can be executed, but they cannot be copied to PC Disk or deleted without entering the correct password.

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

4.17.1 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.

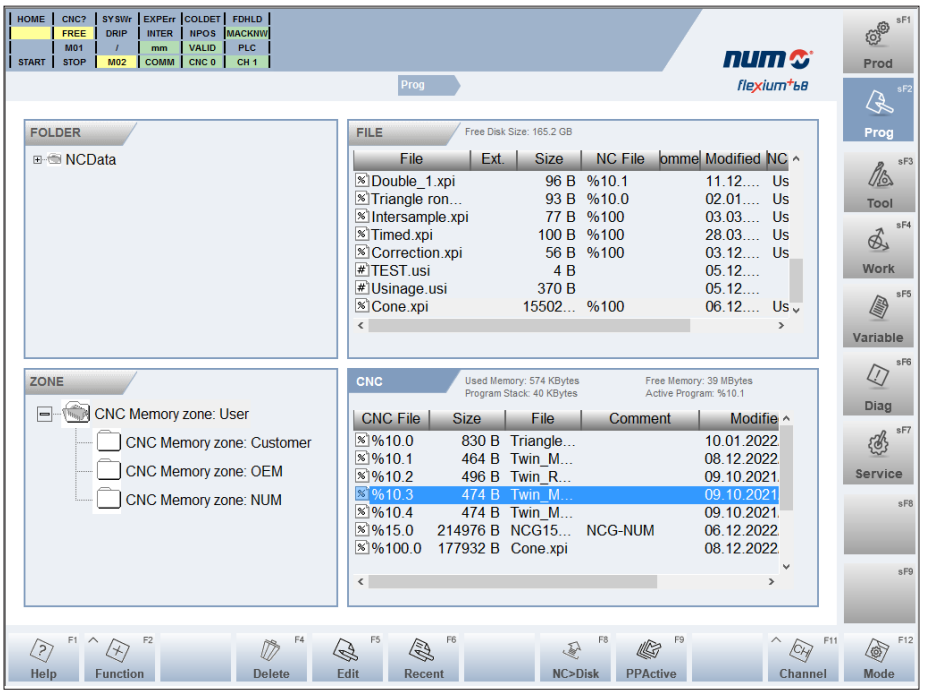


4.17.2 Closing the memory zones

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

4.18 NC Memory

Select the right hand window in the NC memory area	☞ Tab⇆...



Flag description

Flag	Description.
H	Hidden programs and subroutines (see Chapter 4.5.1).
A	Archive: the file has been changed since the last download. This flag will be reset at the next upload.

Possible actions

Press F9 to activate a part program.
Press F8 to save a file to PC disk.
Press F6 to display a list of the 10 most recently edited files.
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.

4.19 Selecting the Active Part Program

Select the right hand window in NC memory	☞ Tab...
Select the desired part 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 part program	☞ F9

4.20 Saving Files to PC Disk

4.20.1 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

4.20.2 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

4.20.3 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

4.21 Editing a Program directly in NC Memory

It is possible to edit a part program directly in NC memory, even during execution.

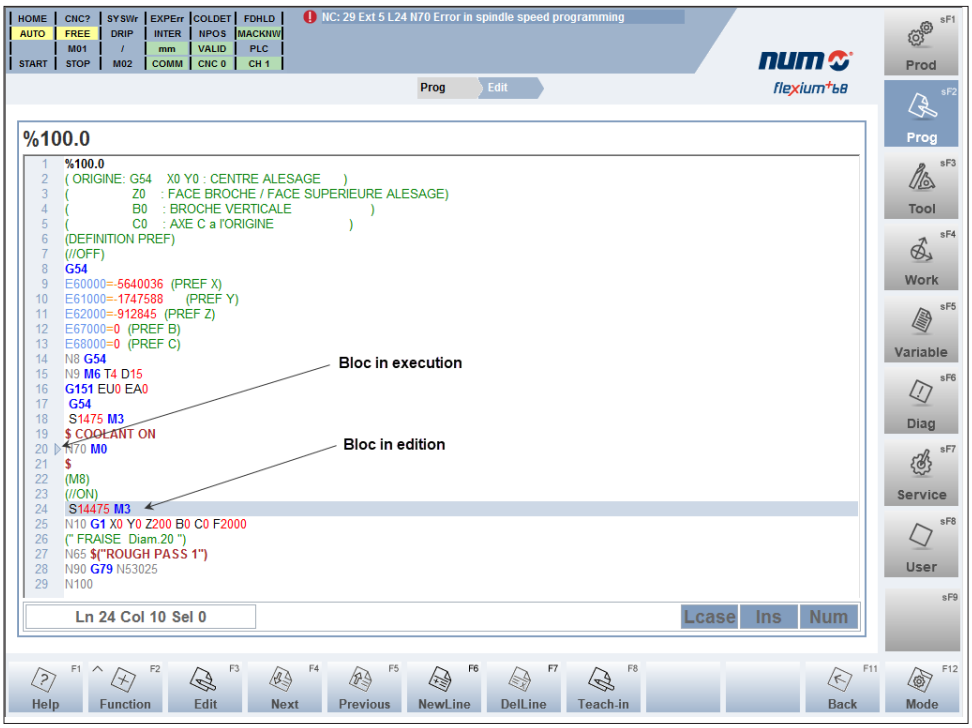
The access to the editor is only enabled if:
- The memory area *"User"* (area 0) is selected
- The selected file is a part program
- The editor is not protected by settings in the context *"Service"* or by the PLC.

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):

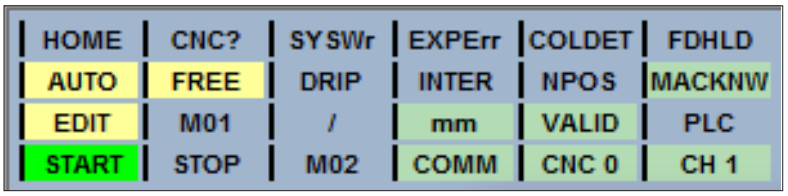


Possible actions

Press F11 to leave the editor.
Press F8 for Teach-in
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: auto scroll”* in the context *“sF7 Service”*, Management” must be activated.
To enable an automatic upload, on leaving the editor after a change, the checkbox *“CNC editor: Auto Upload”* in the context *“sF7 Service, Management”* has to 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.

4.22 Deleting Files on 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.

4.22.1 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

4.22.2 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

4.22.3 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

Page deliberately empty

Summary

5 Context “Tools”.....103
5.1 Access to the Context “Tools”.....103
5.2 Entering tool dimensions and miscellaneous values.....104
5.3 Selecting the Tool Type 105
5.4 Resetting the whole Tool Offset table.....106
5.5 Teach-in of the Tool Dimensions 107
5.5.1 Milling tool setting page 107
5.5.2 Turning tool setting page 108
5.5.3 Boring tool setting page.....108
5.5.4 Procedure 109

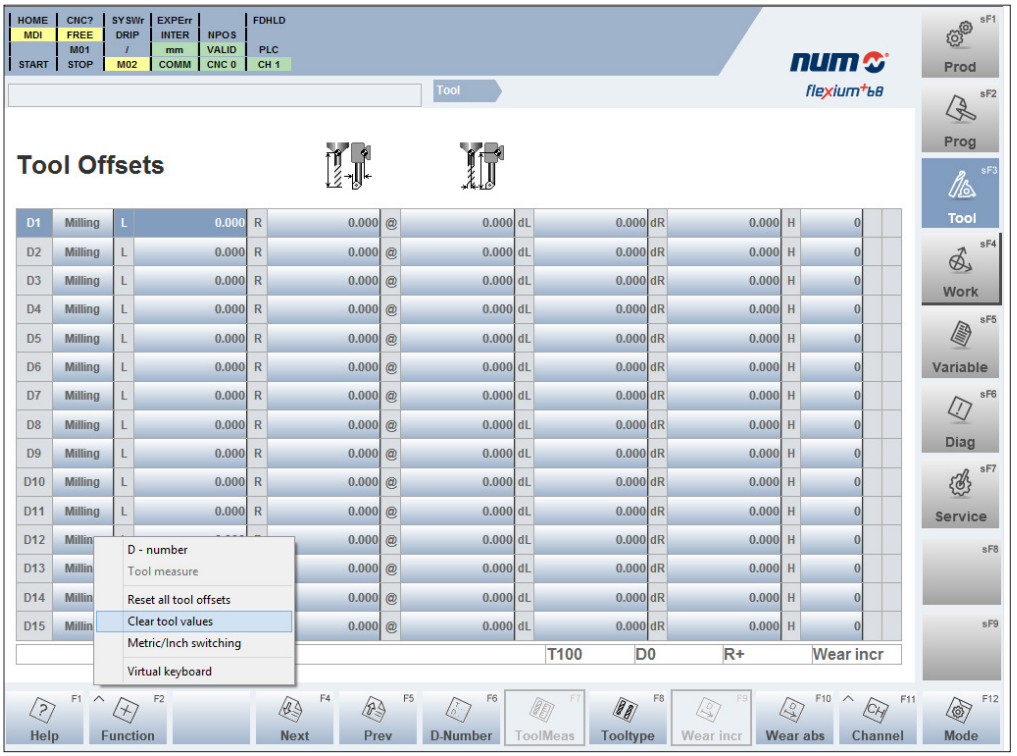
Page deliberately empty

5 Context “Tools”

5.1 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 *‘13 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 ↵

5.2 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 tool type cursor can only be moved between fields with gradient. When entering values in the field for turning or boring tools, it may be necessary to back off the cursor to the left before moving it up or down.

For security purposes it is only possible to modify a tool offset when the system is in RESET status (after an M02 or an operator Reset)

It is however possible to bypass this security by a proper setting in the Service context. Any tool (including the current) can then be changed; operator's attention is required.

5.3 Selecting the Tool Type

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

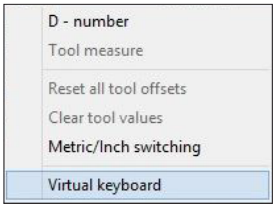
The following tool types may be chosen: milling, turning and boring.

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

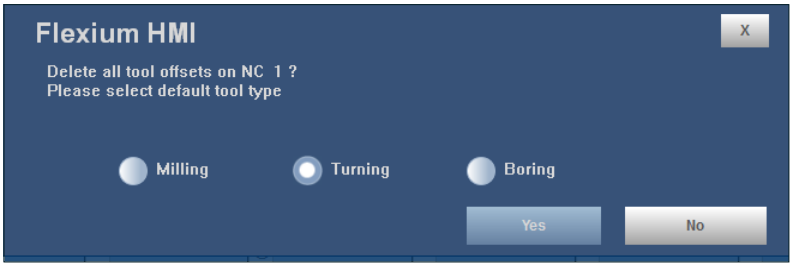
5.4 Resetting the whole Tool Offset table

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 ↵

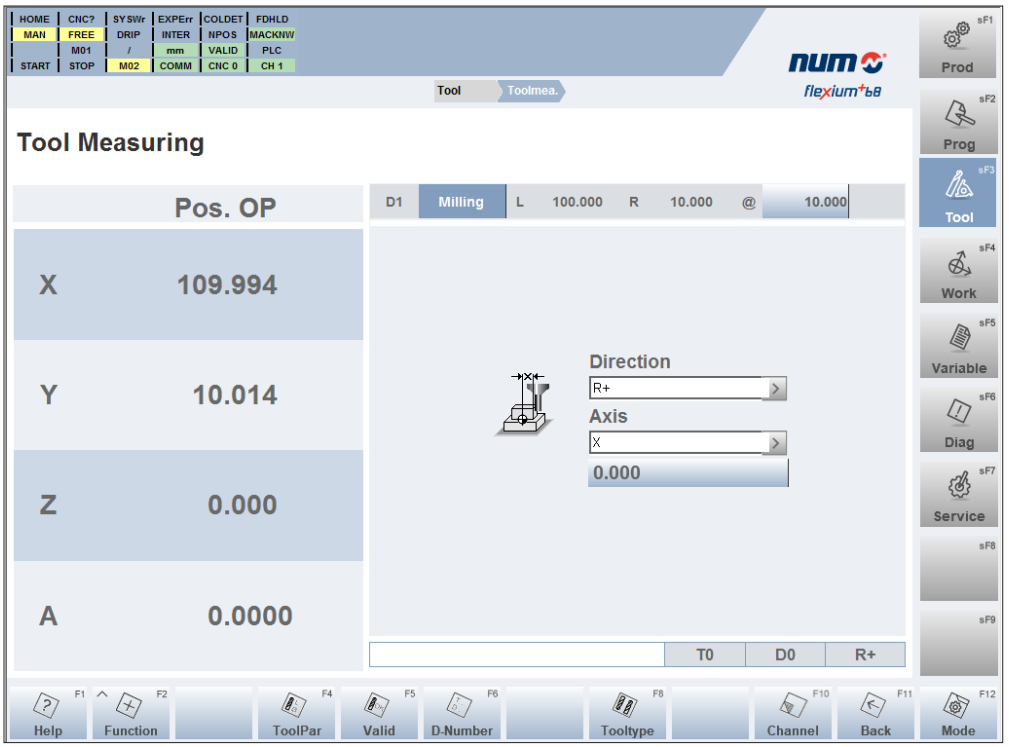
The function "Clear Tool values" allows for clearing the data of the selected offset. The tool type remains unchanged

5.5 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

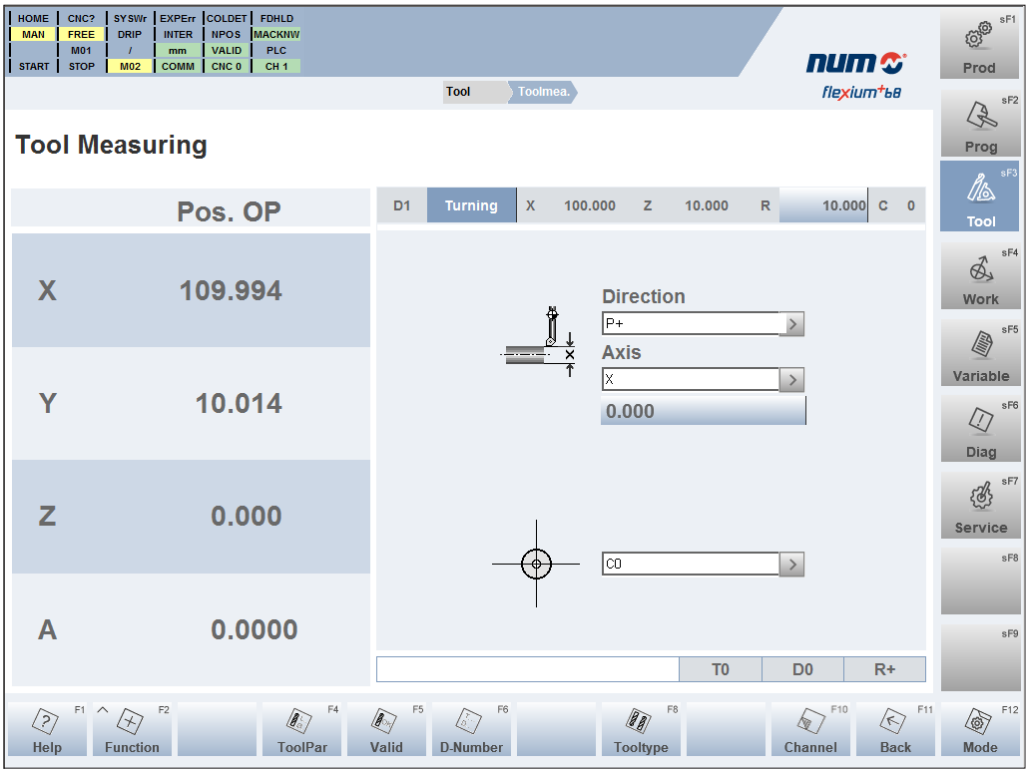
5.5.1 Milling tool setting page



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↵

5.5.2 Turning tool setting page



5.5.3 Boring tool setting page

This page is similar to the turning tool setting page.

5.5.4 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

Summary

6 Context “Work Offset” 113

6.1 Access to the Context “Work Offset” 113
6.1.1 Milling display 113
6.1.2 Turning display 114
6.2 Measuring cycles 115
6.3 Entering DAT2 values 119
6.4 Entering DAT1 values 120
6.5 Setting the Work Offset (DAT1) by Teach-in for Milling 121
6.6 Setting the Work Offset (DAT1) by Teach-in for Turning 123
6.7 RTCP compensation 125
6.8 Selecting DAT3 axes and other parameters 126
6.8.1 Entering Scaling Factor 127
6.8.2 Entering Axes Assignment 127

Page deliberately empty

6 Context “Work Offset”

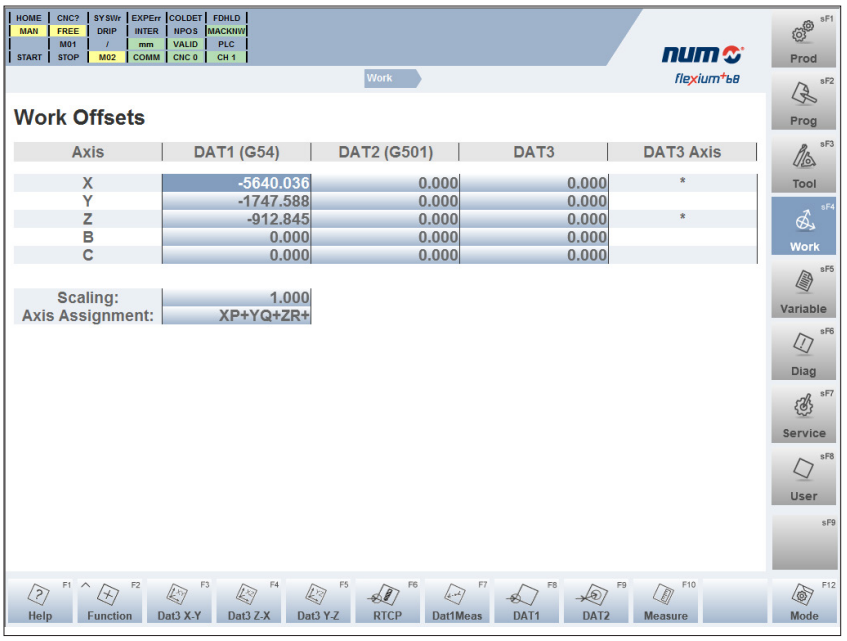
6.1 Access to the Context “Work Offset”

To access the context “Tools”	☞ ⇧ Shift + F4
To close pop-up menus and windows	☞ Esc

This context is used to enter and modify work offsets, scaling and axes assignment values manually. It also offers the possibility to measure the work offset automatically.
Flexium provides 5 different DAT1 (G54 to G58) and up to 99 DAT2 (G501 to G599). The page Work offset displays the values of the currently selected DAT1 and DAT2.

For detailed information about work offsets, refer to chapter ‘13 Review’ in this manual.
The display differs according to the type of machine (milling or turning).

6.1.1 Milling display

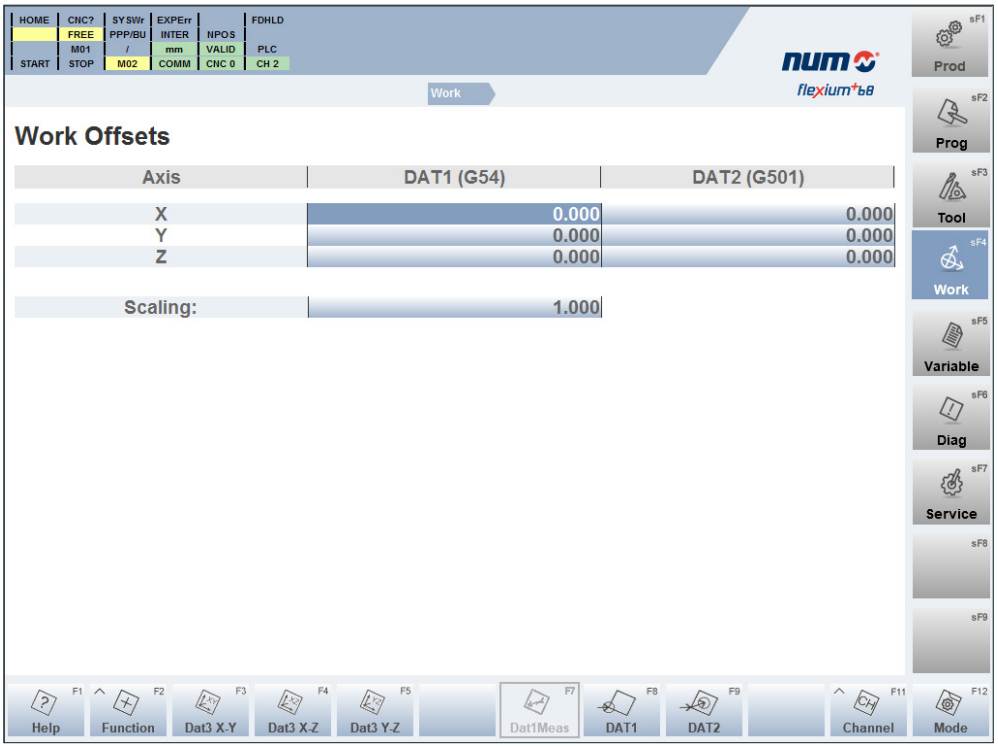


Possible actions

Press F10 to access the measuring cycles (optional)
Press F9 to access the DAT2 offsets
Press F8 to access the DAT1 offsets
Press F7 to access DAT1 measuring
Press F6 to access RTCP and 3D workpiece compensation (optional)
Press F5 to set the DAT3 for Y and Z
Press F4 to set the DAT3 for X and Z
Press F3 to set the DAT3 for X and Y
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter↵

The DAT3 setting functions F5 F4 F3 are only displayed if the CNC is in the reset state.

6.1.2 Turning display



Possible actions

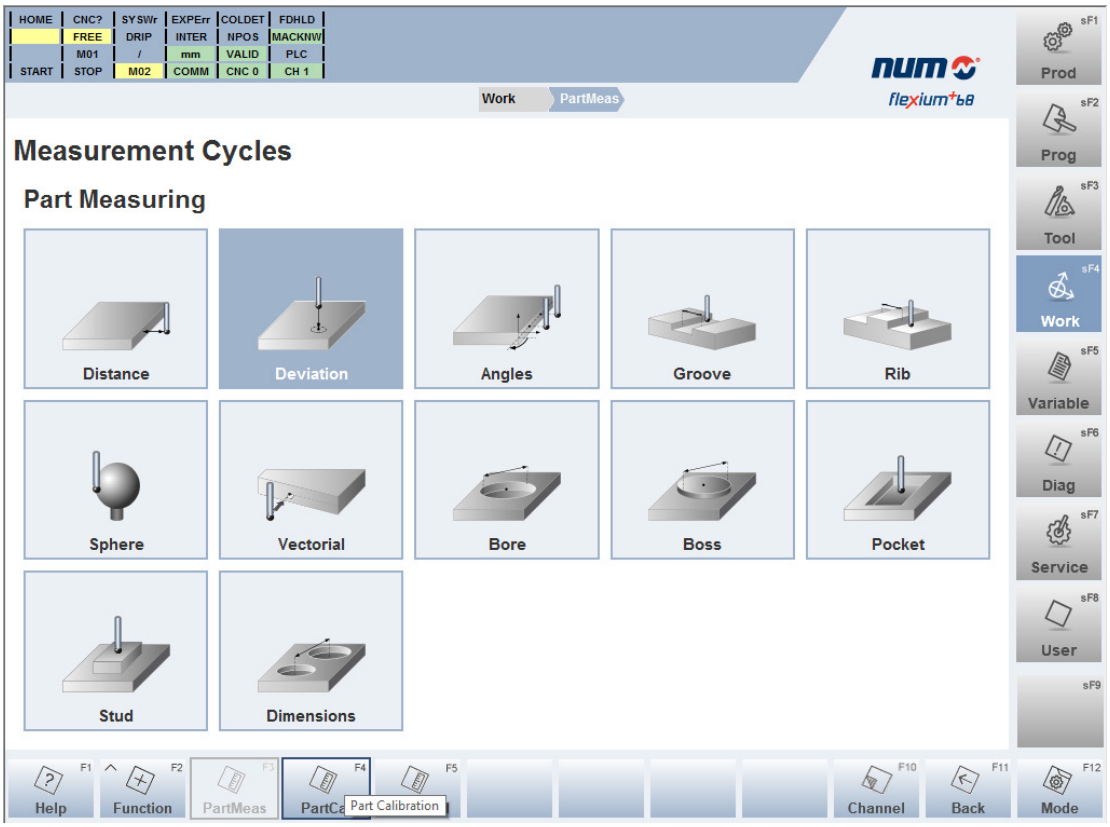
Only DAT1, DAT2 and Measuring cycles are available for lathe configuration.

6.2 Measuring cycles

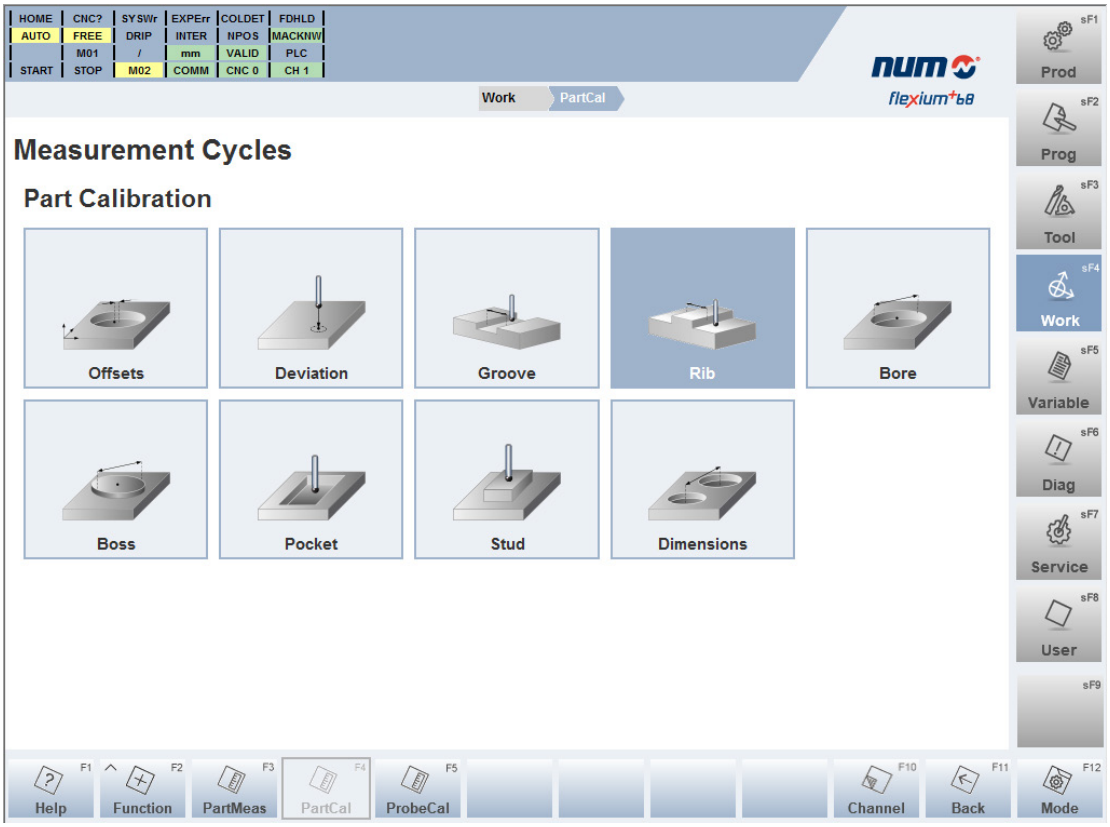
This page introduces the measuring cycles (probing).

The picture below are only given for reference. A dedicated manual M00057 Flexium+Workpiece Measuring Cycles - Operator`s Manual is available for in depth description and use of those cycles.

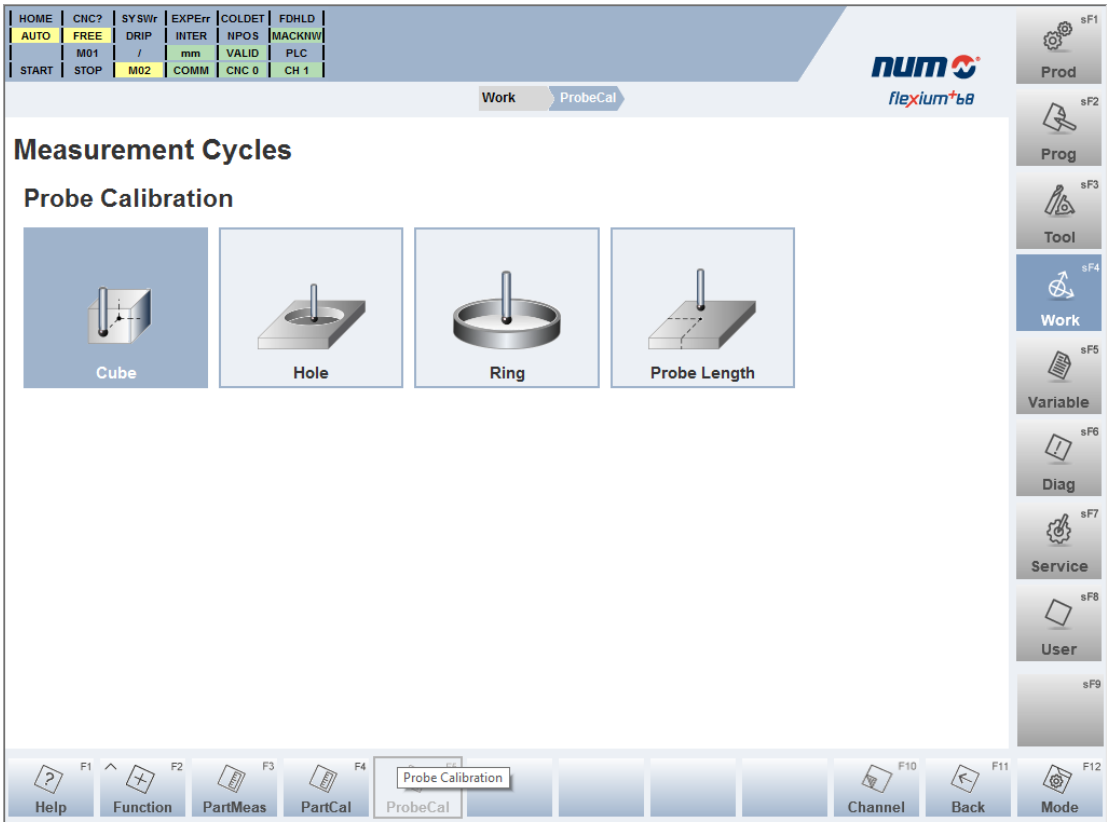
Select the Measuring Page	☞ F10
Select Part Measure, Part Calibration or Probe Calibration	☞ F3, ☞ F4 or ☞ F5



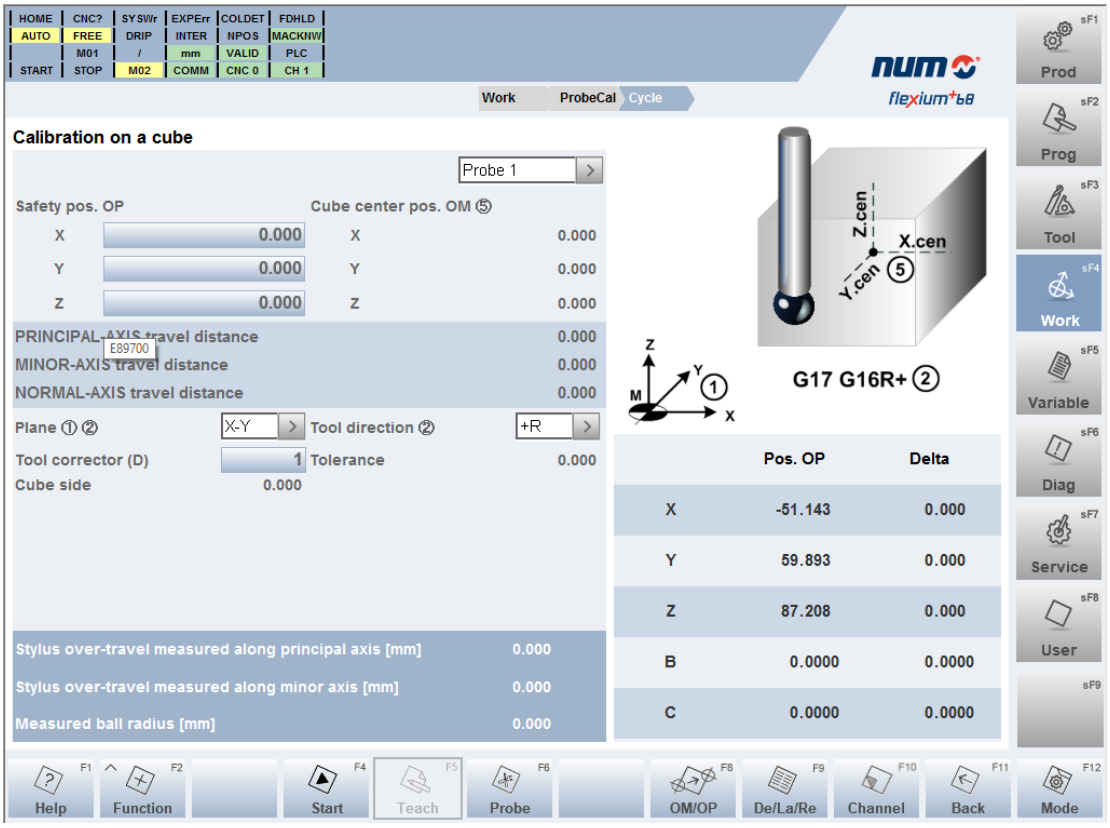
Cycles of part measuring



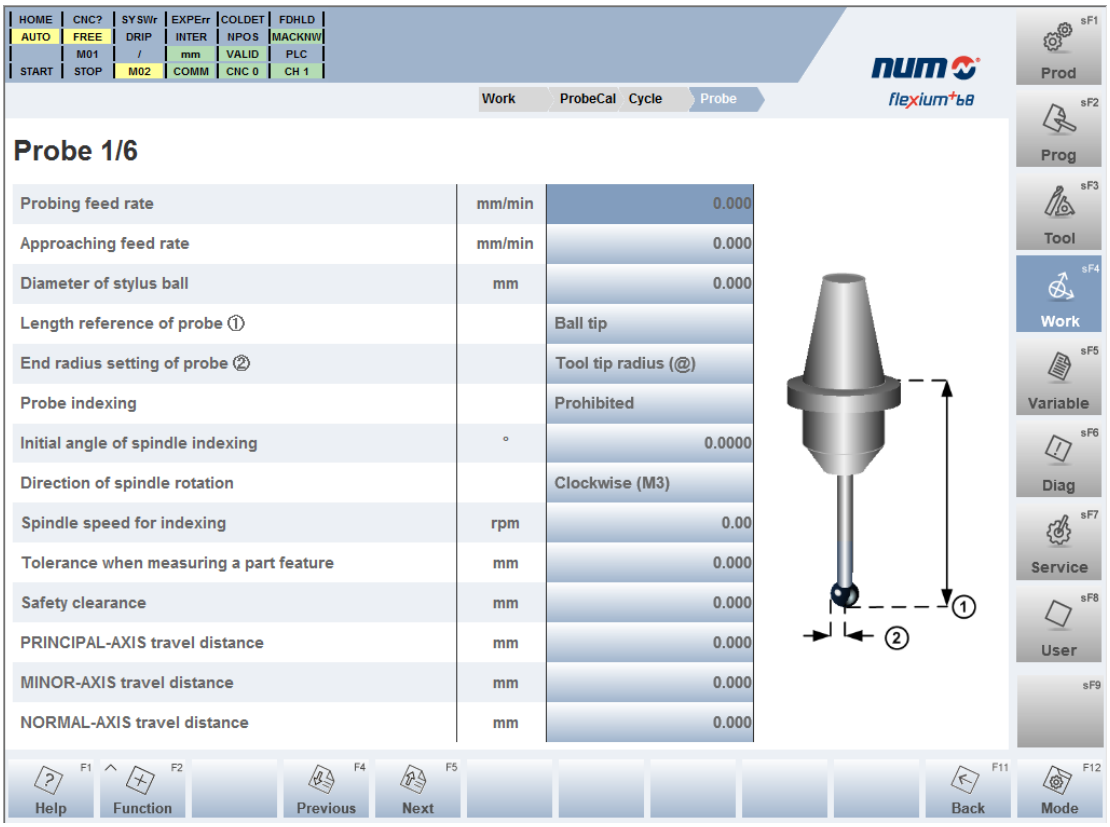
Cycles of part calibration.



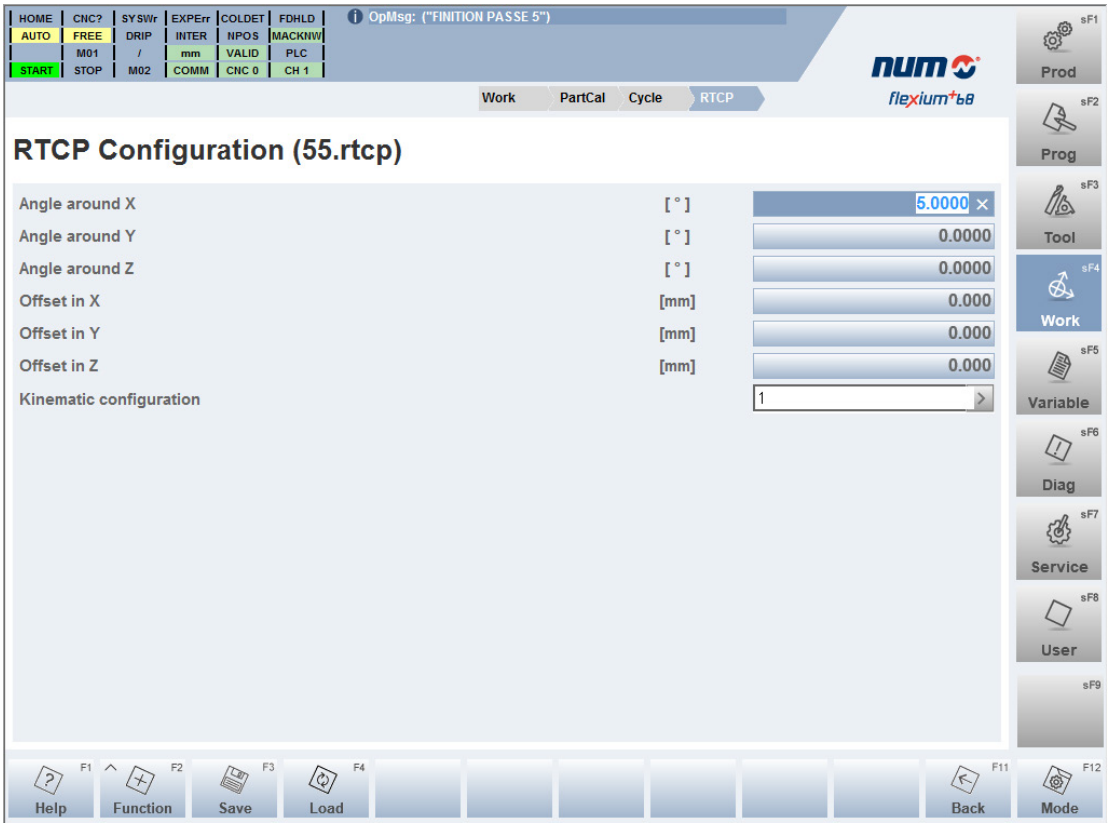
Cycles of probe calibration



Example of probe calibration cycle on a cube



Data page for one of the six possible probes

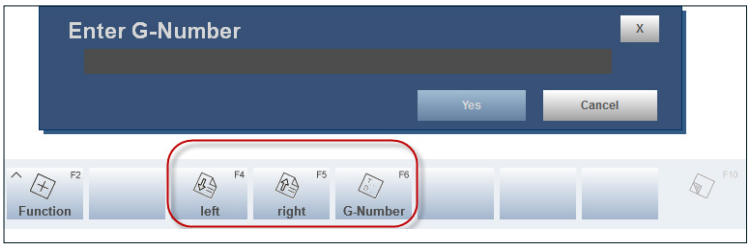


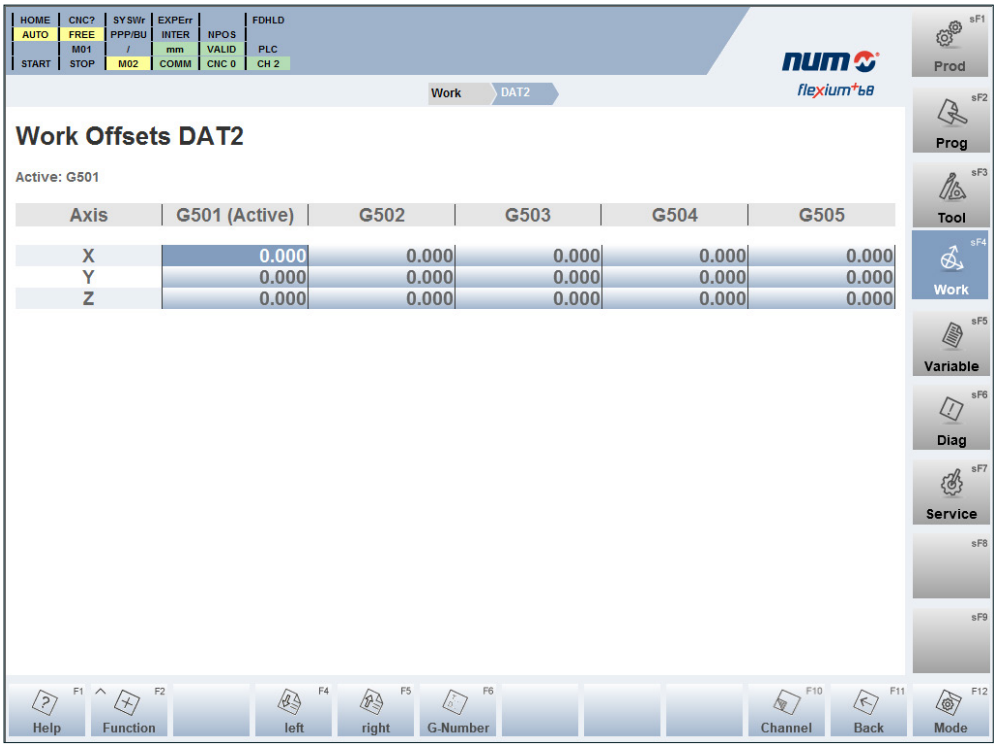
In case of probing with RTCP or iPlane active a dedicated page is added to define the configuration

6.3 Entering DAT2 values

Select the DAT2 table	☞ F9
Select the appropriate column by cursor or mouse or G-number (F6)	

The following page will be displayed :



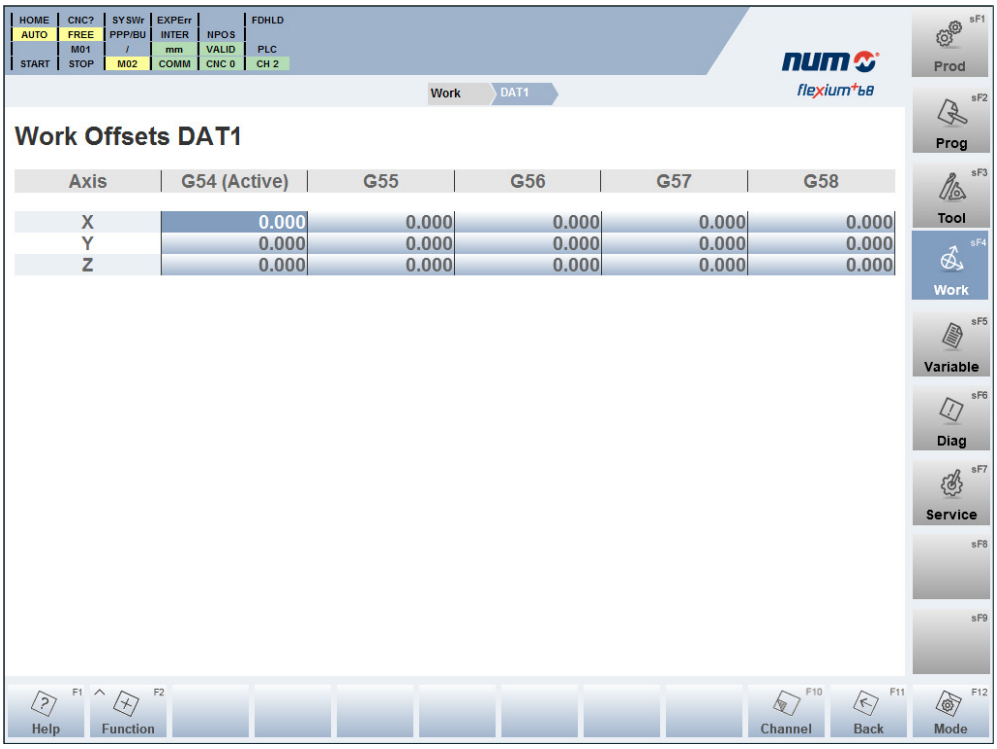


Select the appropriate field	☞ ← or →; ↑ or ↓
Open the field	☞ Enter ↵
Enter the value and validate	☞ Enter ↵

6.4 Entering DAT1 values

Select the DAT1 table	☞ F8
Select the appropriate column by cursor or mouse	

The following page will be displayed :



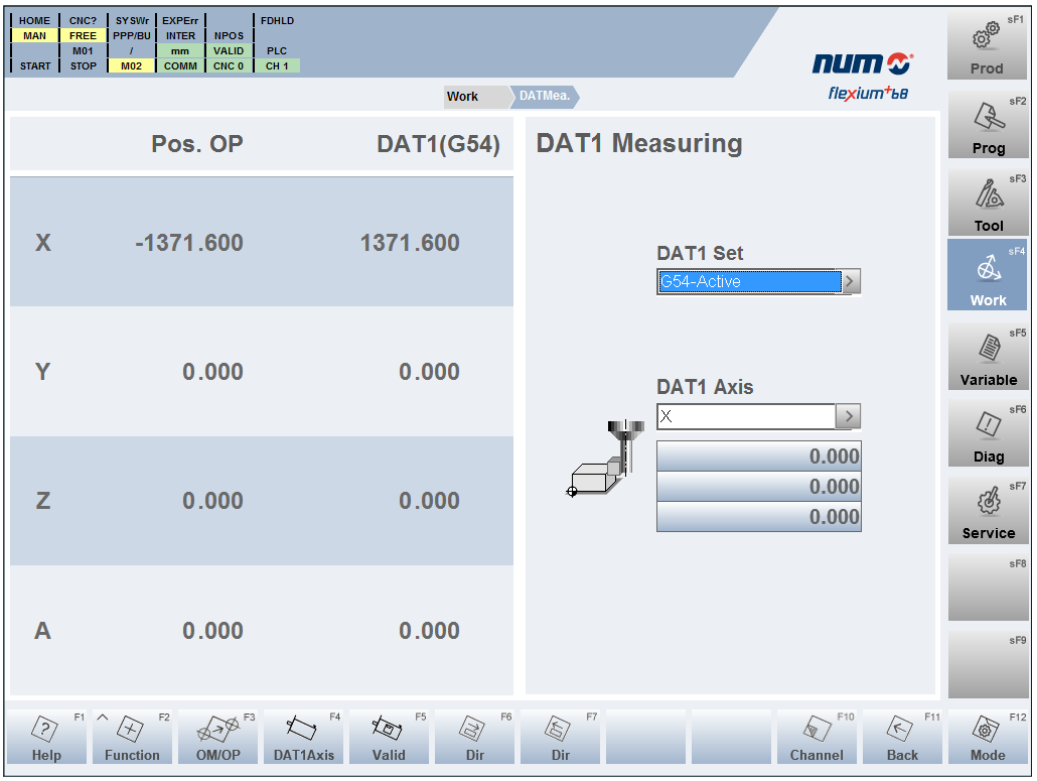
Select the appropriate field	☞ ← or →; ↑ or ↓
Open the field	☞ Enter ↵
Enter the value and validate	☞ Enter ↵

6.5 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.

The result of measurement will be entered in the currently selected DAT1 parameter.

Select DAT1 measuring	☞ F7



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

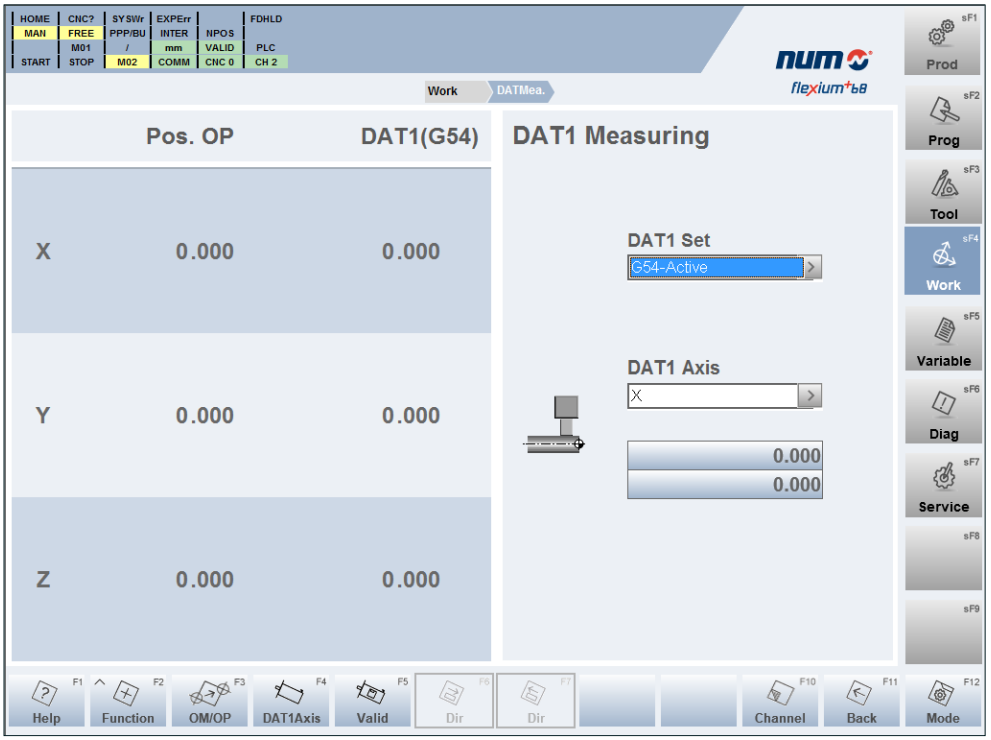
Workpiece and setting piece dimensions are stored and can be used again for a further measurement.

6.6 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.

The result of measurement will be entered in the currently selected DAT1 parameter.

Select DAT1 measuring	☞ F7



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↵

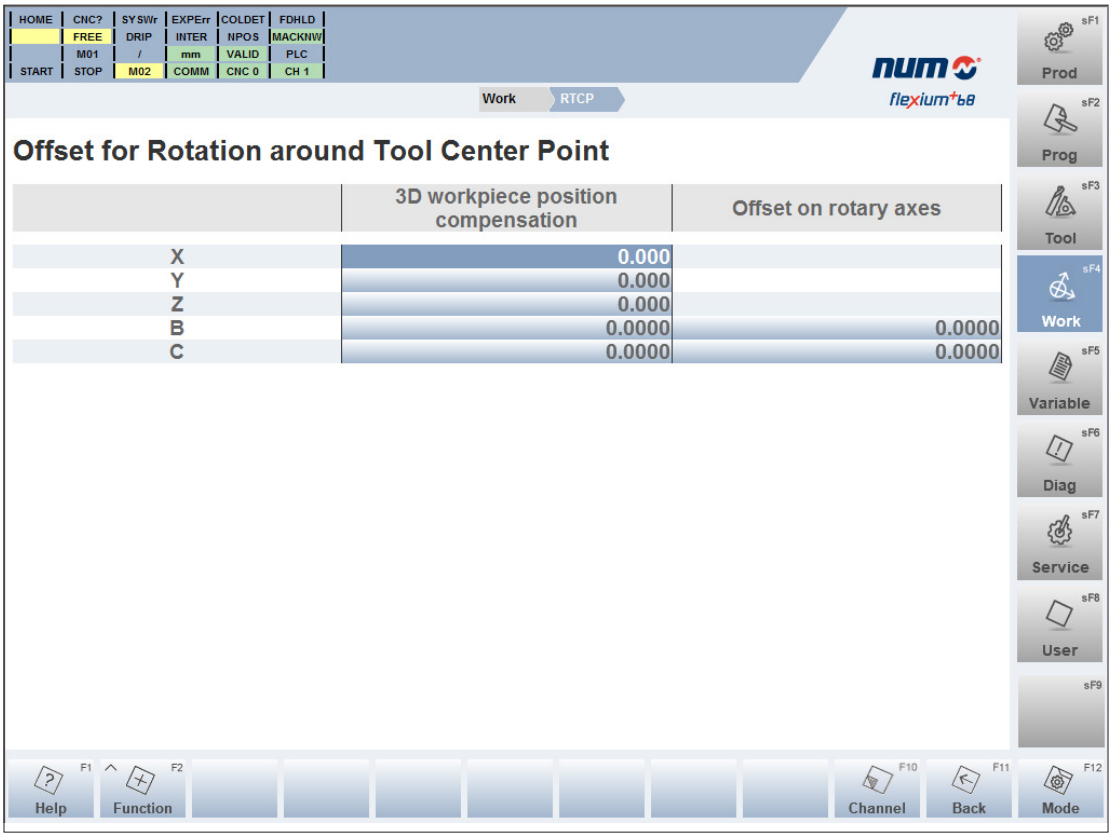
Move the axis to position the tool head in contact with the setting tool	
Validate the setting	☞ F5
The measured value is now displayed in the DAT1 column in the left hand window.	
To exit DAT1 measuring mode	☞ F11

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

6.7 RTCP compensation

This page permits to enter the data about 3D workpiece position compensation as well as the rotary axes offets
For details about the workpice position compensation refer to the extended programming manual

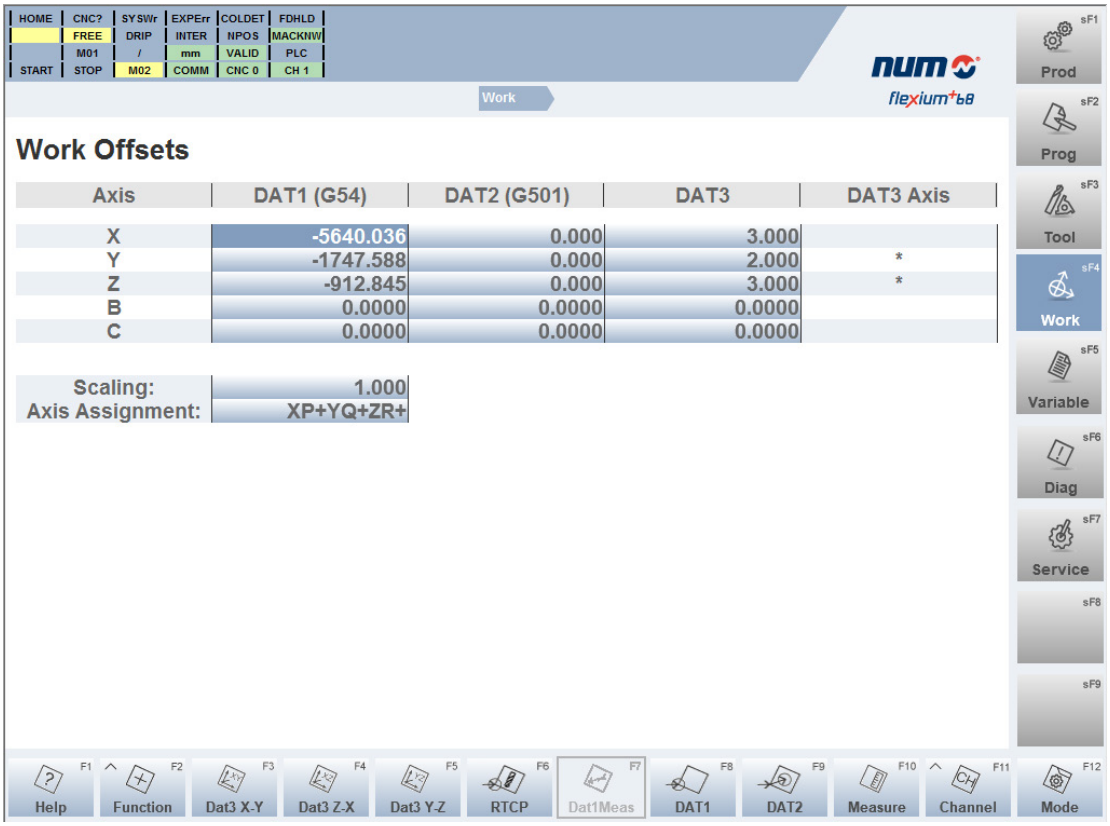
Select the RTCP Page	☞ F6
Select the appropriate column by cursor or mouse	



6.8 Selecting DAT3 axes and other parameters

Select the appropriate axes group	☞ F3 to F5
Select the appropriate line and column by cursor or mouse	

This table also permits to enter the other DAT data.



Select the scaling factor field	☞ ← → ☞ ↑ ↓
Open the field	☞ Enter
Enter the value and validate	☞ Enter

6.8.1 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.

6.8.2 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 ‘11 CNC Operation’ of this manual.

Page deliberately empty

Summary

7 Context “Variables” 131
7.1 Access to the Context “Variables” 131
7.1.1 Variables context window 132
7.2 Entering variables and comments 133
7.3 CNC variables and comments 134
7.3.1 Display of Symbolic variables 134
7.4 PLC variables and comments 135
7.5 Modification of variables values 136
7.6 Creating a list of consecutive variables 137

Page deliberately empty

7 Context “Variables”

7.1 Access to the Context “Variables”

To access the context *“Variables”*	☞ Shift + F5
To close pop-up menus and windows	☞ Esc

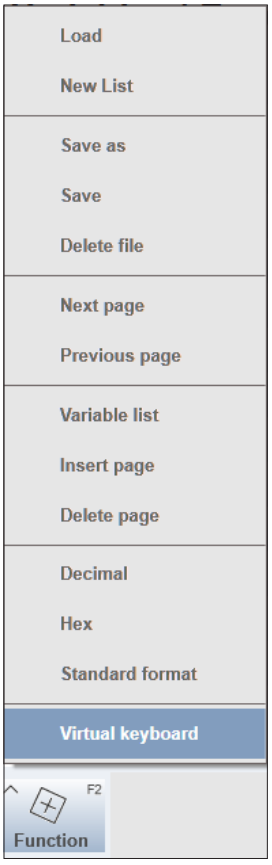
The Variable context is intended as a control and debugging tool.
It is used to display, save and load variable lists described in the debug files *.xdv.

The variables that may be displayed are:
- External parameters E
- L variables
- Symbolic CNC program variables
- PLC variables: according to the PLC project.

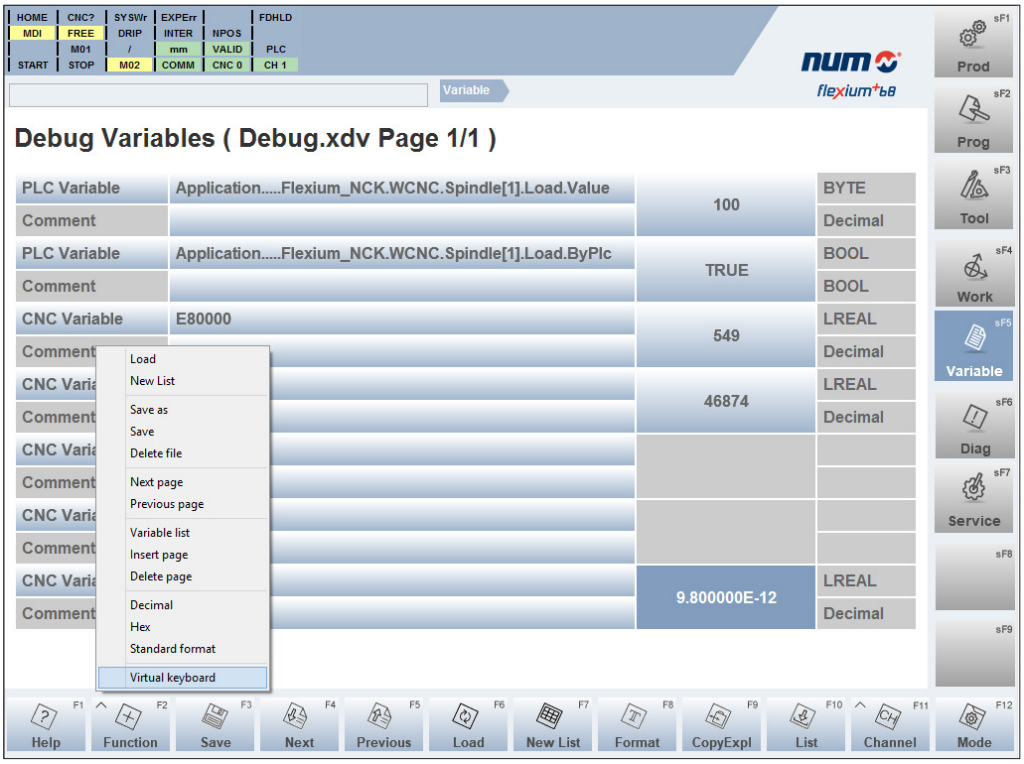
In PLC programming, the whole set of variable data types according to IEC 61131-3 is supported.

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:



7.1.1 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 *'10 CopyExplorer'* in this manual.
Press F8 to select the type of display (hexadecimal and decimal) To choose a format : ↑ or ↓ and Enter
Part program 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

7.2 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

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

7.3.1 Display of Symbolic variables

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

1 Symbolic variables are selected, but not yet declared.
The message “Invalid variable” is displayed:

CNC Symbolic	PosX	Invalid variable	LREAL
Comment			Decimal
CNC Symbolic	VALUE	Invalid variable	LREAL
Comment			Decimal

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

```
%10.1(Test SymVariable)()

VAR
         [COUNTER]
         [VALUE]
         [PosX]
ENDV

[COUNTER] = 0
[VALUE] = 125
[PosX] = 10

WHILE [COUNTER]  < 100 DO
         X [PosX]
         [PosX]  = [PosX]  + [COUNTER]
          [COUNTER] =  [COUNTER] + 1
ENDW

M02
```

3 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	COUNTER	17	LREAL
Comment			Decimal
CNC Symbolic	PosX	146	LREAL
Comment			Decimal
CNC Symbolic	VALUE	125	LREAL
Comment			Decimal

Please refer to Flexium Extended Programming manual M00055 for more detailed information.

7.4 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 M03 from the Master Spindle:
Application.IOCONFIG_GLOBALS.Flexium_NCK.RCNC.Channel[1].DecodeM.Spindle[0].M03

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

7.5 Modification of variables values

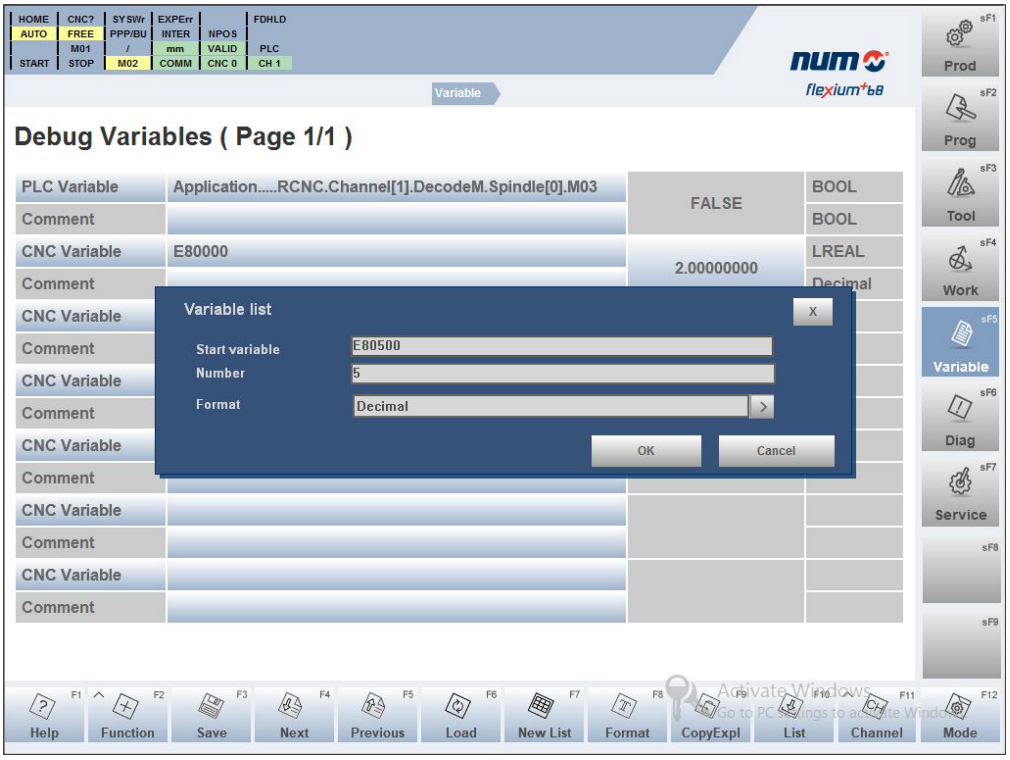
The values of variables can be modified. This is only possible if the box “*Write variables*” in the Context “*Service*” has been checked. and if writing is enabled by the PLC.

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

7.6 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

Summary

8 Context “Diagnostic” 141

8.1 Access to the Context “Diagnostic” 141
8.1.1 Diagnostic context window 141
8.2 Extended Help 142
8.2.1 Example of help page 143
8.3 Alarm History Page 144
8.3.1 History Page and Filters 145
8.4 Exception Page 147
8.5 Test Point Monitor 148
8.6 CNC and DISC NT+ Configuration 149
8.6.1 Device List 150
8.6.2 System Functions Page 151
8.6.3 Drive History 152
8.6.4 Project information page 154
8.6.5 CNC task information 155
8.6.5.1 Dynamic Operators monitor 155
8.6.5.2 Available Dynamic Operators programs in C 156
8.7 SAMX Diagnostic 157
8.7.1 Safety module diagnostic 159
8.7.2 STOX Safety diagnostic 159
8.8 Operation time 160
8.9 Export log files 161

Page deliberately empty

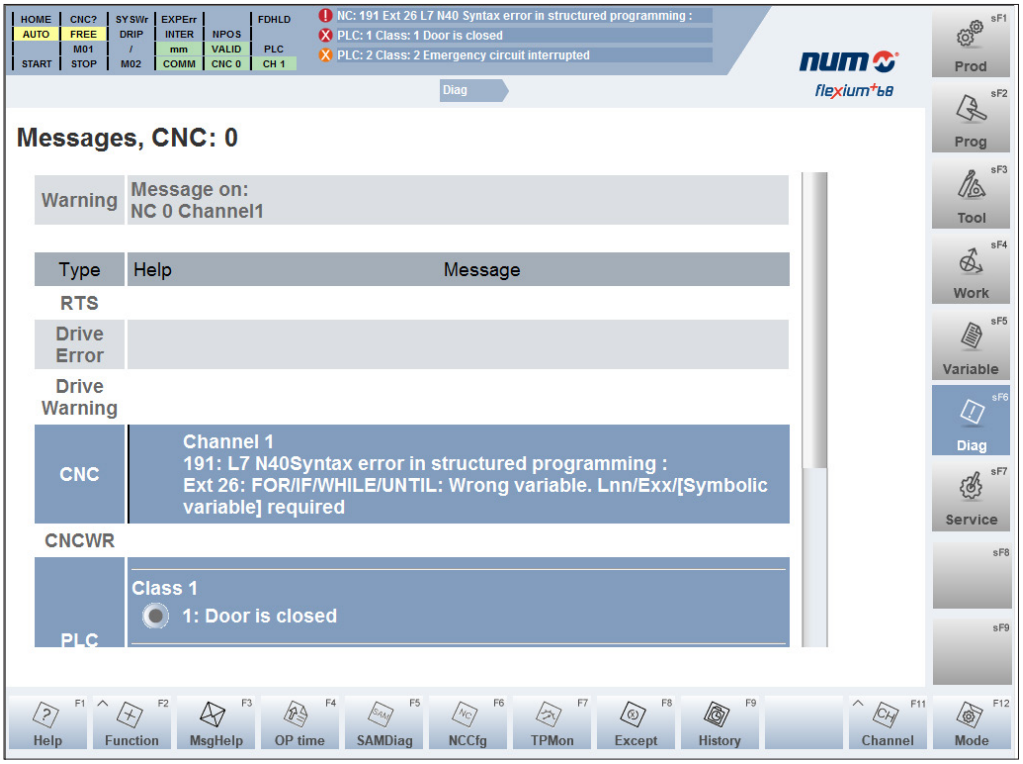
8 Context “Diagnostic”

8.1 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.

8.1.1 Diagnostic context window



Possible actions

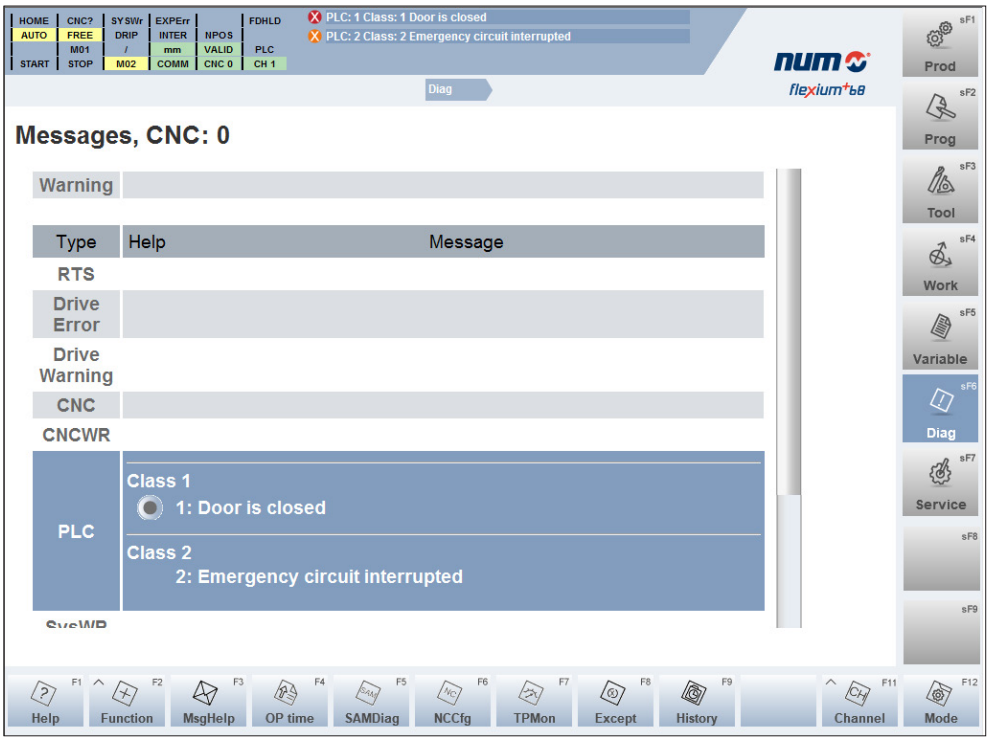
Press F9 to access the alarm history page.
Press F8 to access the exception records.
Press F7 to access the test point monitor page.
Press F6 to display the CNC and DISCNT configuration data.
Press F3 to open the extended help page.
Press F2 to access all existing functions. Select the function with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

The menu F2 also allows for exporting all log files in a single compressed (.7z) file

8.2 Extended Help

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

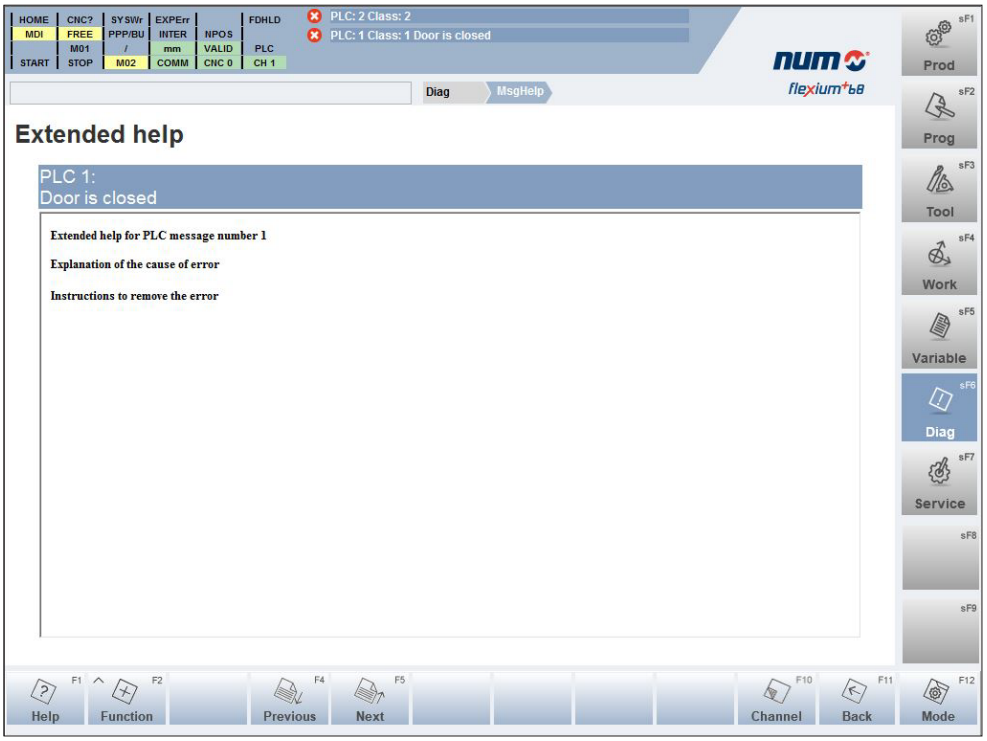
Once a message has been configured with an extended help, it will be displayed with a radio button and shown as follows:



Possible actions

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

8.2.1 Example of help page



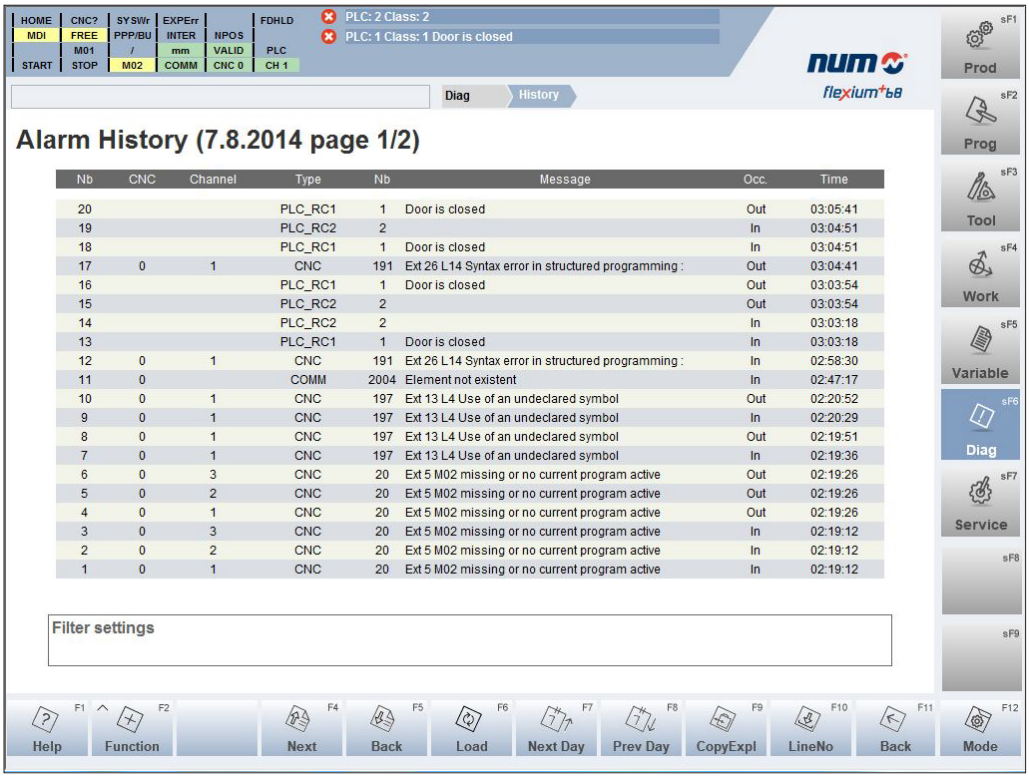
Possible actions

Press F5 to display the help page of the next message.
Press F4 to display the help page of the previous message.

8.3 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



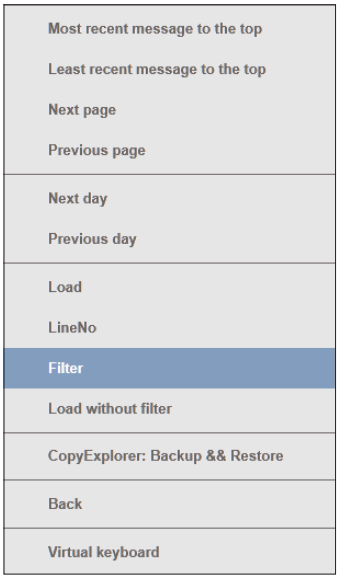
Possible actions

Press F10 to display messages starting at a specific line number.
Press F9 to access CopyExplorer. For the description of CopyExplorer, refer to chapter *'10 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↵

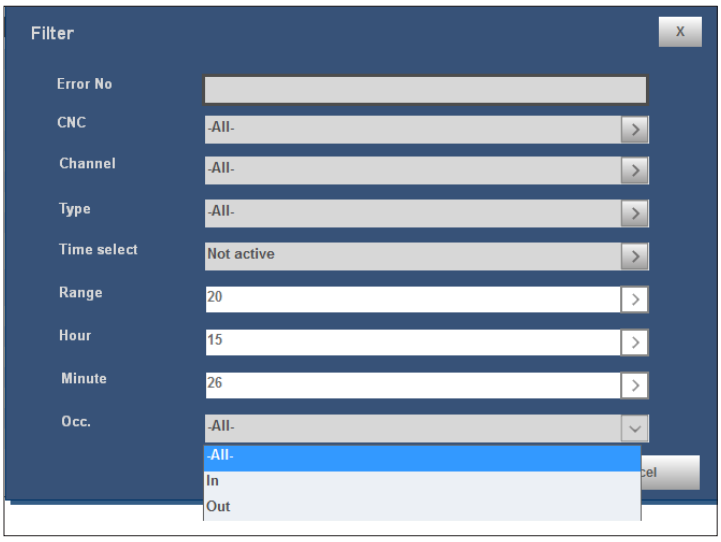
8.3.1 History Page and Filters

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”*	

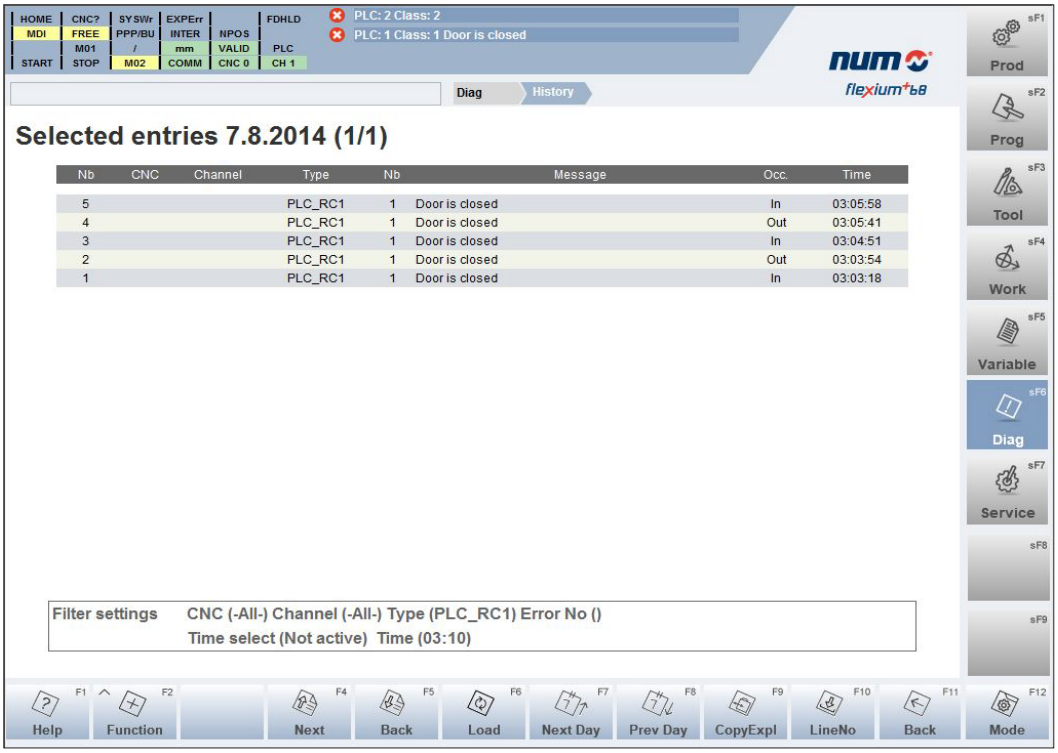


This opens the filter selection window:



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↵

Occ. gives you the possibility to select the occurence : In, Out or All.



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.

8.4 Exception Page

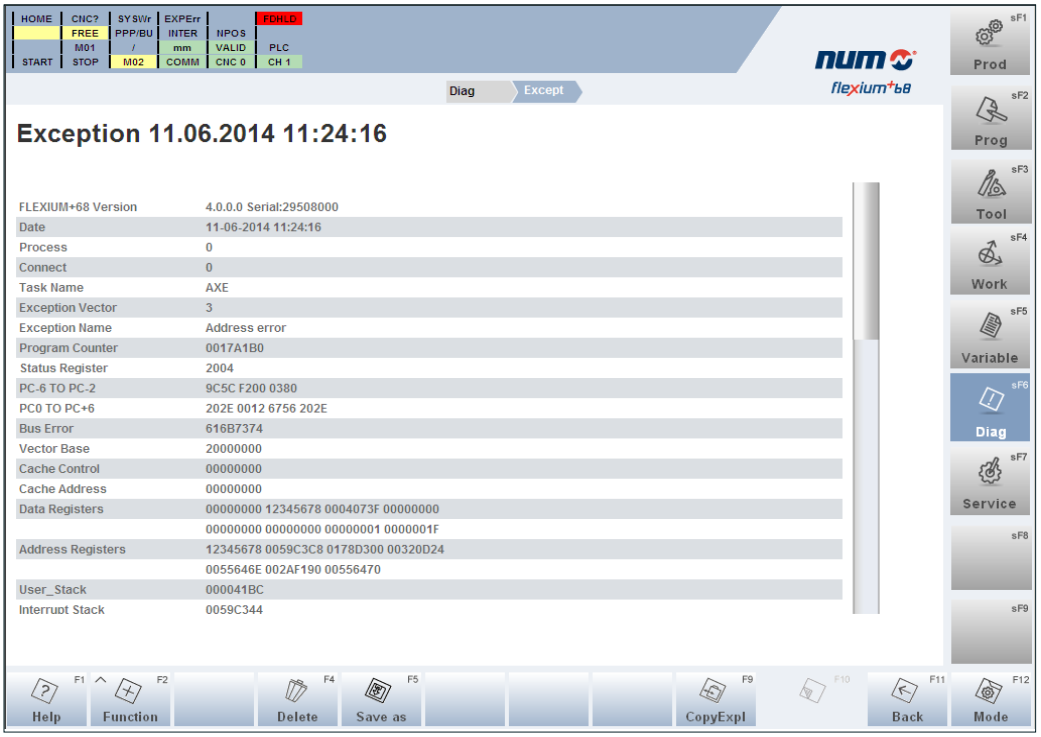
An exception file contains the data of a CNC system error caused by hardware or software failure or by a wrong CNC configuration. This data can only be interpreted by NUM.

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.

This page is only shown if authorized in the Service context.

Select the exception page	☞ F8

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



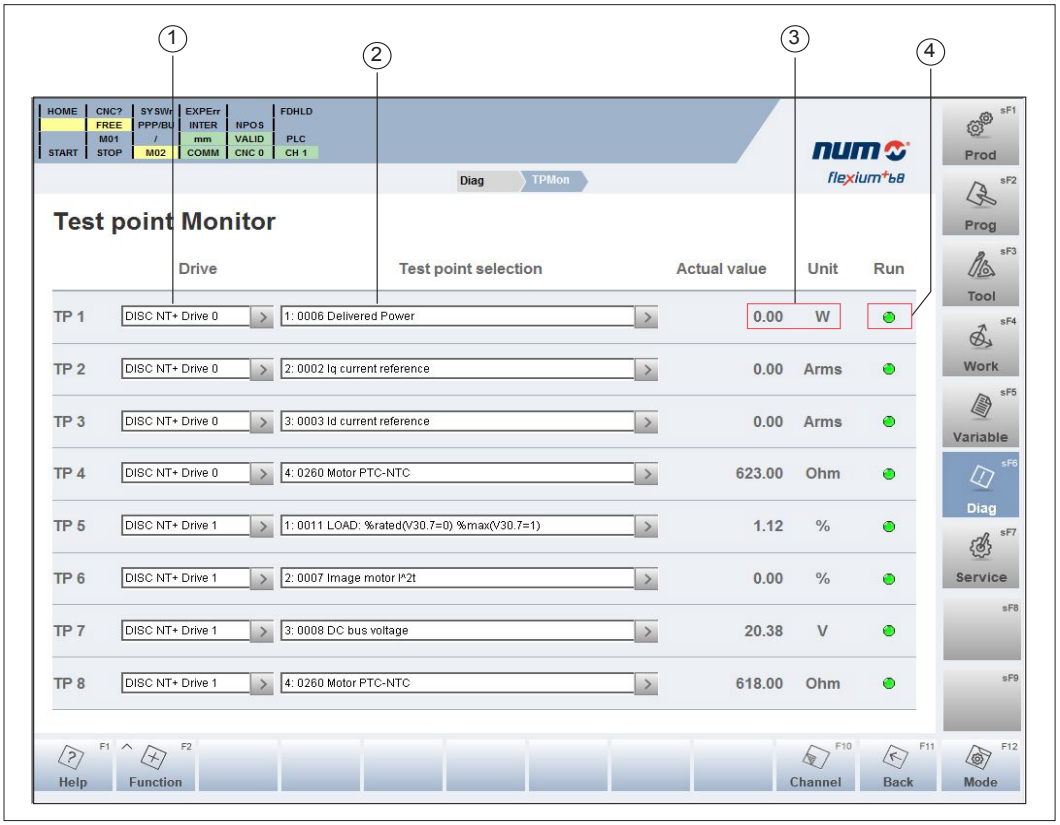
Possible actions

Press F9 to access CopyExplorer.	
For the description of CopyExplorer, refer to chapter *‘10 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

8.5 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 M00044).

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, grey = Halt.

Navigation and selection

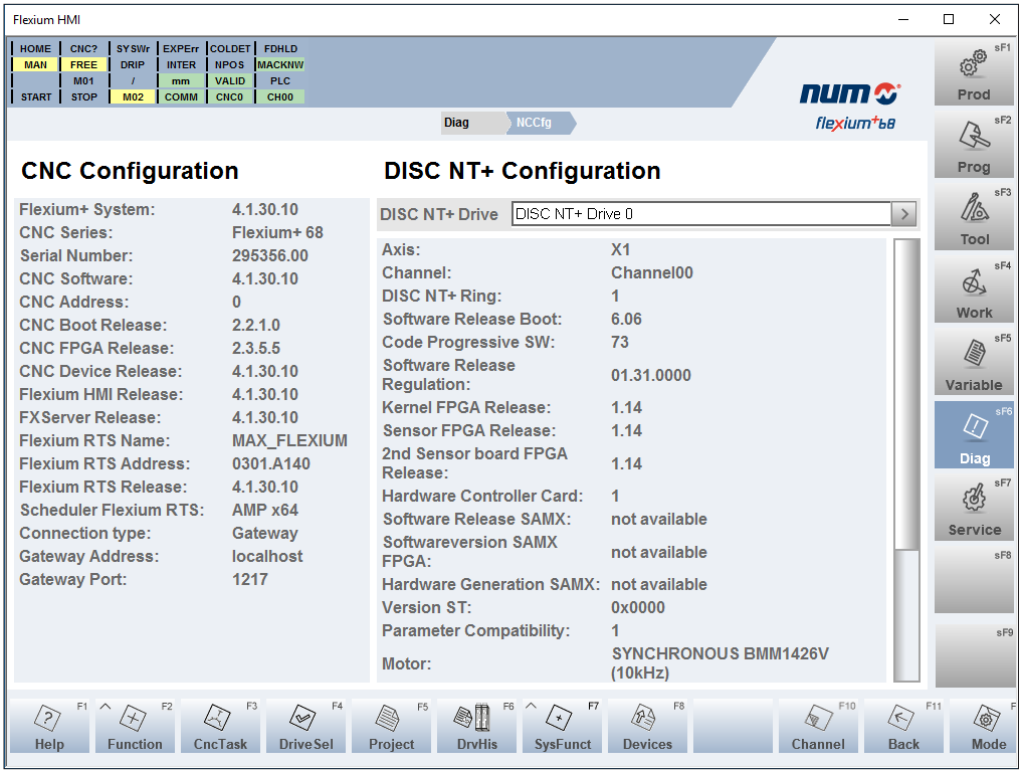
Select a field	☞ Tab
Open a list or an entry field	☞ Enter
Select an entry in an open list	☞ ↑ or ↓
Validate a selection or an entry	☞ Enter

8.6 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 F8 to access the Device List page.	
Press F7 to open the system function page.	
Press F6 to open the Drive history page	
Press F5 to open the Project information page	
Press F4 to select the DISC NT+ drive information.	
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter↵	
To open the drive select box	☞ F4
To select the drive	☞ ↑ or ↓ and Enter↵
To exit the configuration page	☞ F11

8.6.1 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 5 will be saved in *C:\Users\Public\Documents\NUM\Flexium HMI\FXBrowser\Data* *with the extension .xsk if you like to change the folder, go to sF7 -> General Settings -> F7*
If you wish to have a more flexible access to these files, it is recommended to use the functions of CopyExplorer.

8.6.2 System Functions Page

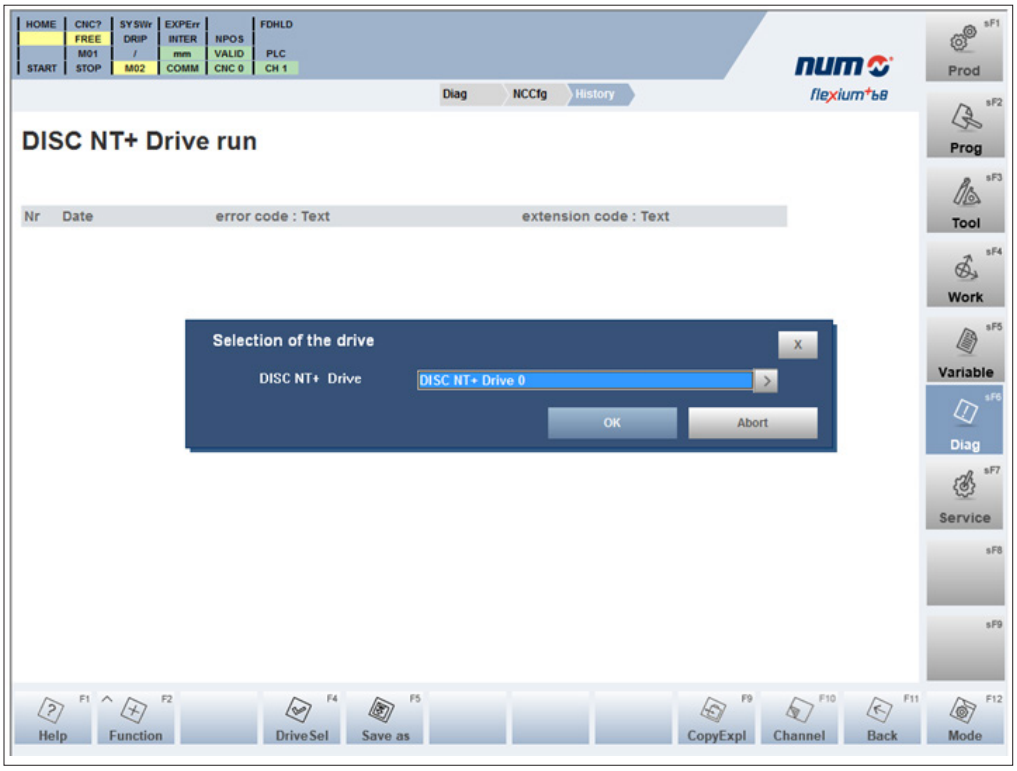
This page contains a list of all available functions enabled in the Flexium+ system.

System Functions Page	☞ F7

8.6.3 Drive History

This page contains all Drive alarms stored in the DISC NT+ Drive
When accessing this page the system will ask you on which drive you want to work.

To access this page, press F4 in the *'NC Config'* page of the *'Diagnostic'* context.



Possible actions

Press F4 to select an other Drive.
Press F5 to save the file on the PC Disk.
Press F9 to access to the Copy Explorer.

Files generated using the function 5 will be saved in
C:\Users\Public\Documents\NUM\Flexium HMI\FXBrowser\Data
with the extension .drvlog
if you like to change the folder, go to sF7 -> General Settings -> F7.

8.6.4 Project information page

This page shows the main information about the project (PLC program + settings).

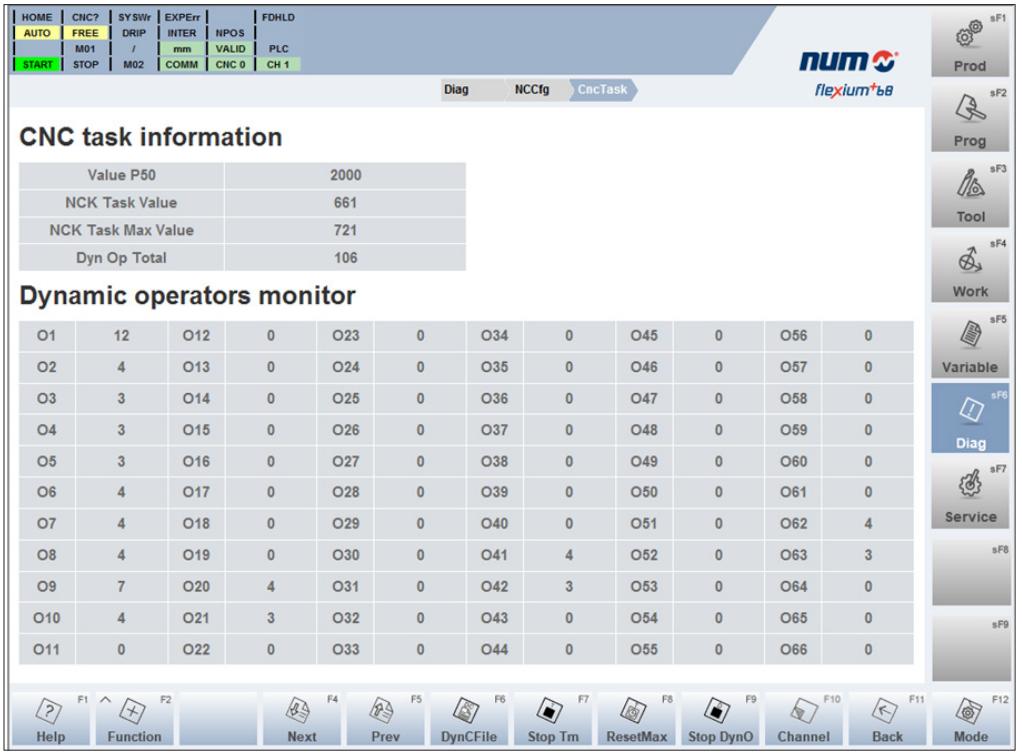
Select this page	☞ F5

8.6.5 CNC task information

This page displays the load of the CNC Task.
To access this page, press F3 in the *‘CNC Configuration’* page in the *‘Diagnostic’* context.

8.6.5.1 Dynamic Operators monitor

This page displays CNC Task information and Dynamic Operators monitoring.



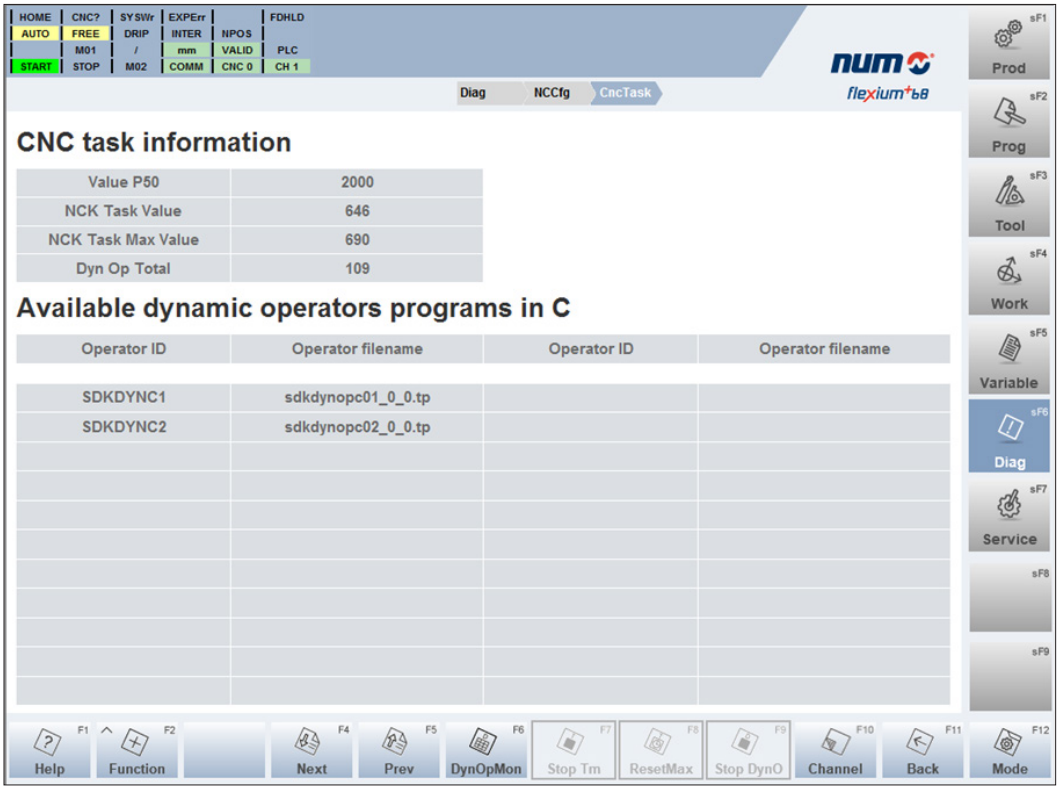
Value P50	Task time of the CNC in micro seconds.
NCK Task Value	Last task time of the CNC in micro seconds.
NCK Task Max Value	Max task time of the CNC in micro seconds.
Dyn Op Total	Total task time value of all Dynamic Operator in micro seconds.
O1 – O128	Task time value of the Dynamic Operator in micro seconds.

Possible actions

Press F4 to select the Next page.
Press F5 to select the Previous page.
Press F6 to switch the available Dynamic Operators programs in C.
Press F7 to Start / Stop Task Monitor.
Press F8 to Reset Max value.
Press F9 to Start / Stop Dynamic Operator Monitor.

8.6.5.2 Available Dynamic Operators programs in C

This page displays the CNC Task information and available Dyn Op programs. Toggle between Standard Dynamic Operators and Dynamic Operators in C monitoring via the <F6> key.



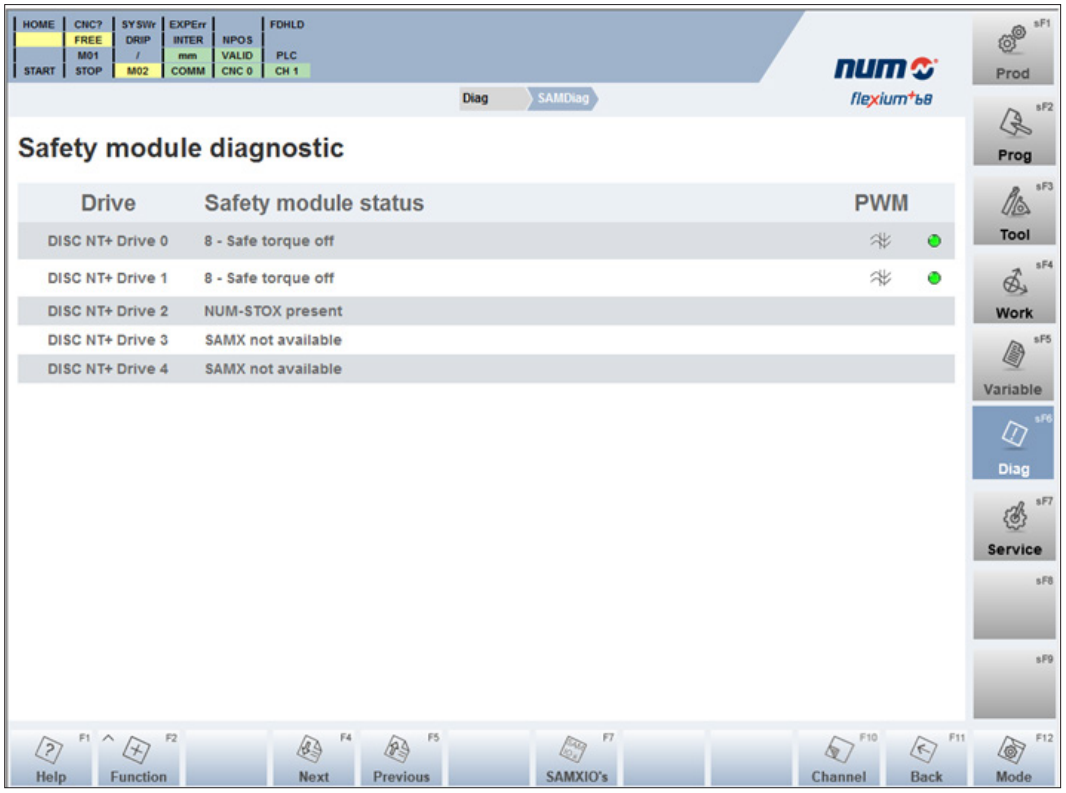
Possible actions

Press F4 to select Next page.
Press F5 to select Previous page.
Press F6 to select Dynamic Operators monitor.

8.7 SAMX Diagnostic

This page contains all DISC NT+ drive states of the Safety Modules.

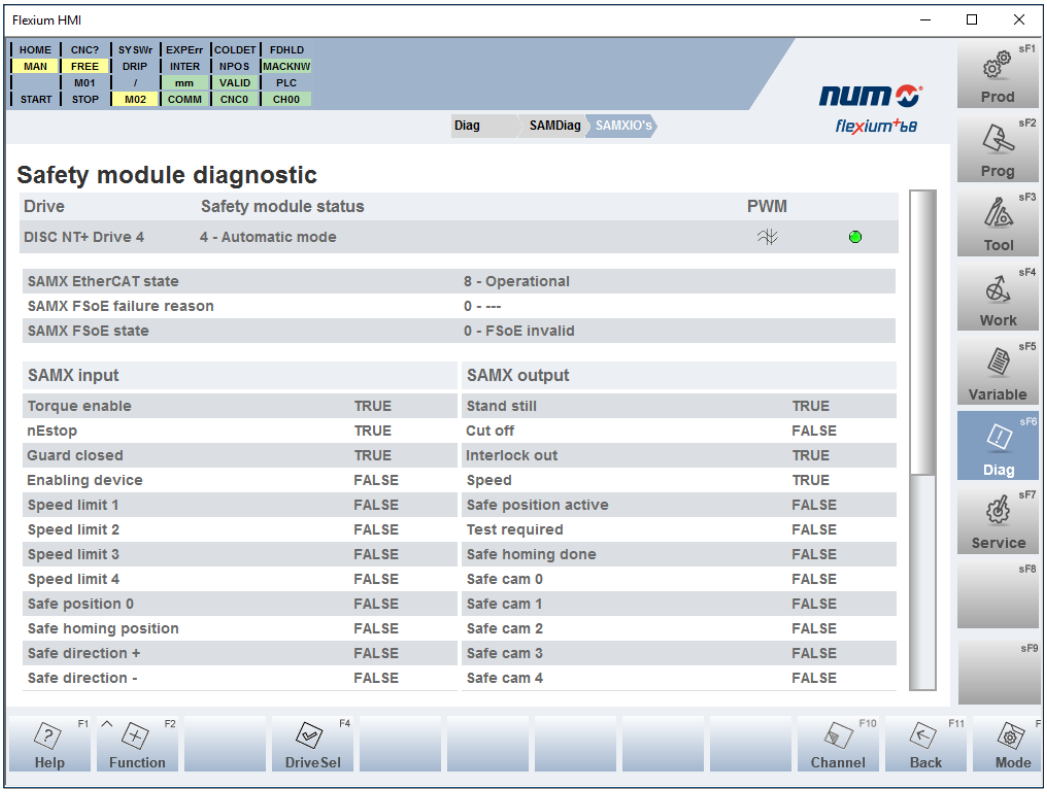
To access this page, press F5 in the ‘*Diagnostic*’ context.



PWM	Description of PWM signal (Pulse-with Modulation)
	PWM Signal is off (red circle).
	PWM Signal is on (green circle).

Possible actions

Press F4 for Next Page.
Press F5 for Previous Page.
Press F7 to access the SAMX IO’s page.



Possible actions

Press F4 to select an other Drive.

8.7.1 Safety module diagnostic

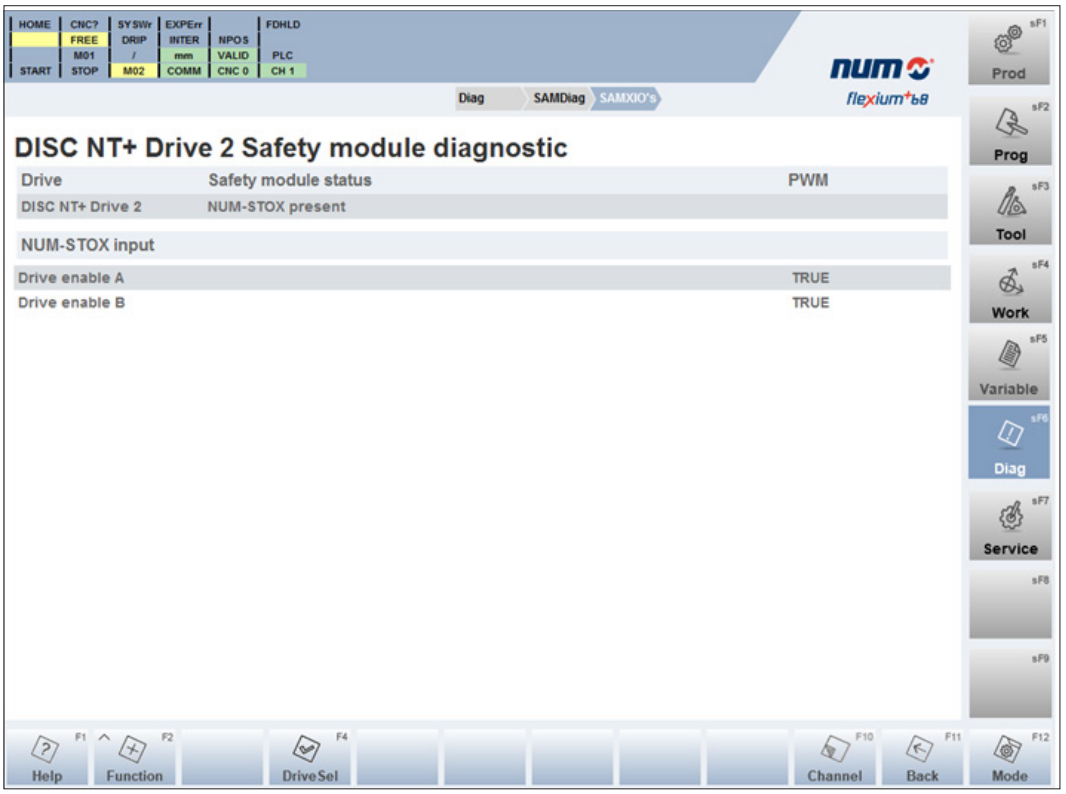
This page contains all IO’s of the DISC NT+ drive safety Modules
When accessing this page the system will ask you on which drive you want to work.

To access the page, select a drive F4 and, press F7 in the ‘*SAMX Diagnostic’* page in the *‘Diagnostic’* context.

8.7.2 STOX Safety diagnostic

This page contains all IO’s of the DISC NT+ drive Safe Torque Off module.
When accessing this page the system will ask you on which drive you want to work.

To access this page, press F7 in the ‘*SAMX Diagnostic’* page in the *‘Diagnostic’* context.



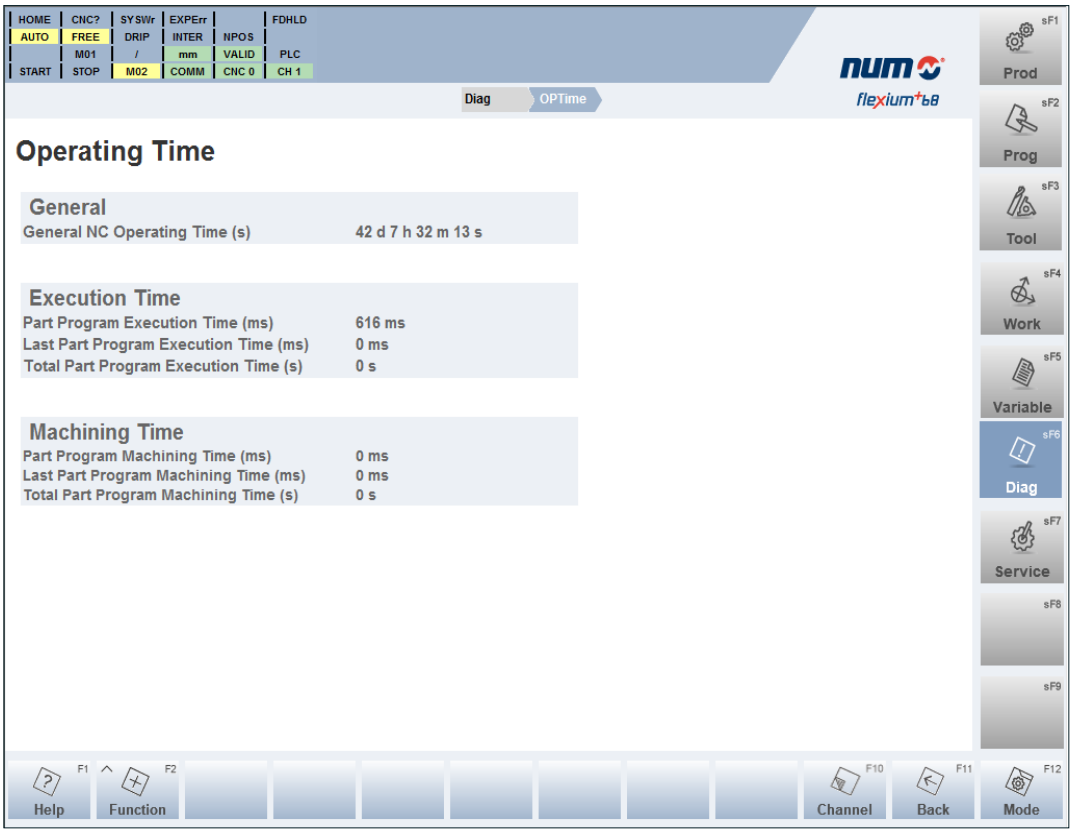
Possible actions

Press F4 to select an other Drive.

8.8 Operation time

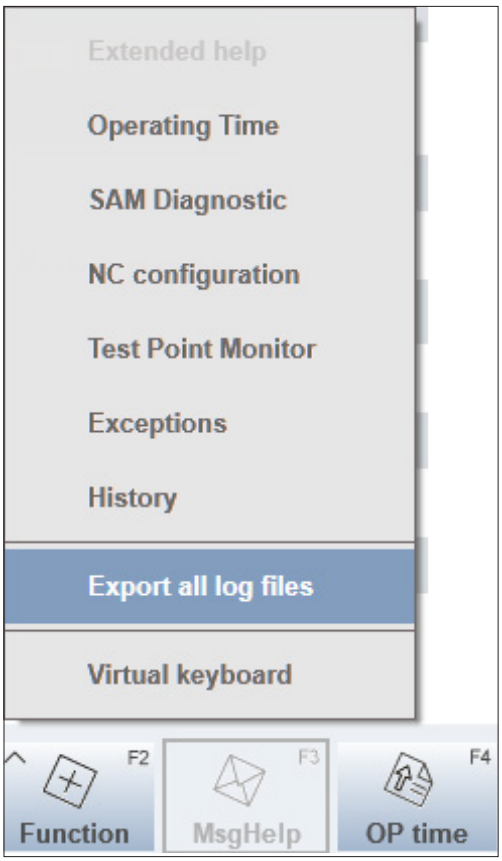
This page recaps the different operational times recorded: To access this page, press <F4> in the '*Diagnostic*' context.

General: • General NC Operating Time	Time under which the NCK has been powered.
Execution Time: • Part Program Execution Time • Last Part Program Execution Time • Total Part Program Execution Time	Time of part program duration (see Prod page).
Machining Time: • Part Program Machining Time • Last Part Program Machining Time • Total Part Program Machining Time	Time of machining with spindle running and axes moving in interpolation mode.



8.9 Export log files

For maintenance purposes, it is possible to export the different Log files. This function can be achieved via the F2 softkey.



The files will be compacted in a .7z document stored in a selectable directory.

The files are :
- HMI logs (FXLog)
- FXServer.DDriveHistory from all drives.
- PLC log files from the RTS Folder.
- FXServerLlog
- NUMROTO logs file
- Exception File
- Device list

Page deliberately empty

Summary

9 Context “Service” 165

9.1 Access to the Context “Service” 165
9.1.1 Start window of the context “Service” 165

9.2 General Settings Page 167
9.2.1 Setting selection 167
9.2.2 System Time and Date Settings 170
9.2.2.1 Setting the system time and date 170
9.2.3 Security Settings 171
9.2.4 Location Settings 174
9.2.4.1 Keyboard functions 174
9.2.5 Custom tools 175
9.2.6 Password Settings 176
9.2.7 Box PC Access 177

9.3 Diagnostic Settings 178
9.3.1 Custom message colors 180
9.3.1.1 System message colors 1 180
9.3.1.2 System message colors 2 181
9.3.1.3 PLC message colors 182
9.3.2 History 183
9.3.2.1 History setting 183
9.3.3 PLC Visu 184
9.3.4 Axes view 185
9.3.5 Spindle Settings 188
9.3.6 Production Settings 191

9.4 Management Settings 193
9.4.1 FXCAM 195
9.4.2 Password management 196
9.4.3 NUMCoss 197

9.5 Machine Data 198
9.5.1 Change Password 199
9.5.2 Restart 199
9.5.3 Clear the NVRAM 200
9.5.4 Probing 201
9.5.5 Restore 202
9.5.6 Backup 204
9.5.7 Drive parameters 207
9.5.8 NCK parameters 210

9.6 Diagnostic 212
9.6.1 Oscilloscope 213
9.6.1.1 Oscilloscope functions 214
9.6.1.2 Chart 216
9.6.1.3 Tabs 218
9.6.1.4 Configure page 221

Page deliberately empty

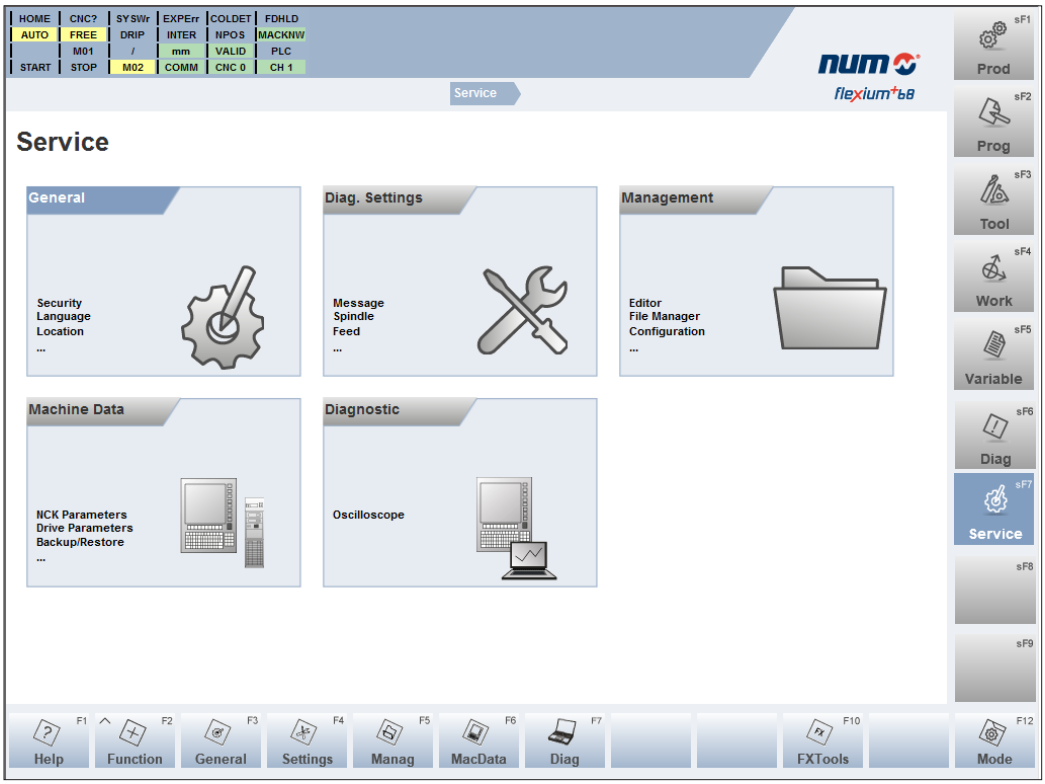
9 Context “Service”

9.1 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.

9.1.1 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 communicated by 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 F7 to access the *"Diagnostic"* page.

Press F6 to access the *"Machine setup Data"* page.

The page *'Machine Setup Data'* allows for modification of the machine setup parameters. Modification of parameters by non specialist personnel can have serious or dangerous consequences. If this page is accessed by mistake return immediately to the previous menu by F11.

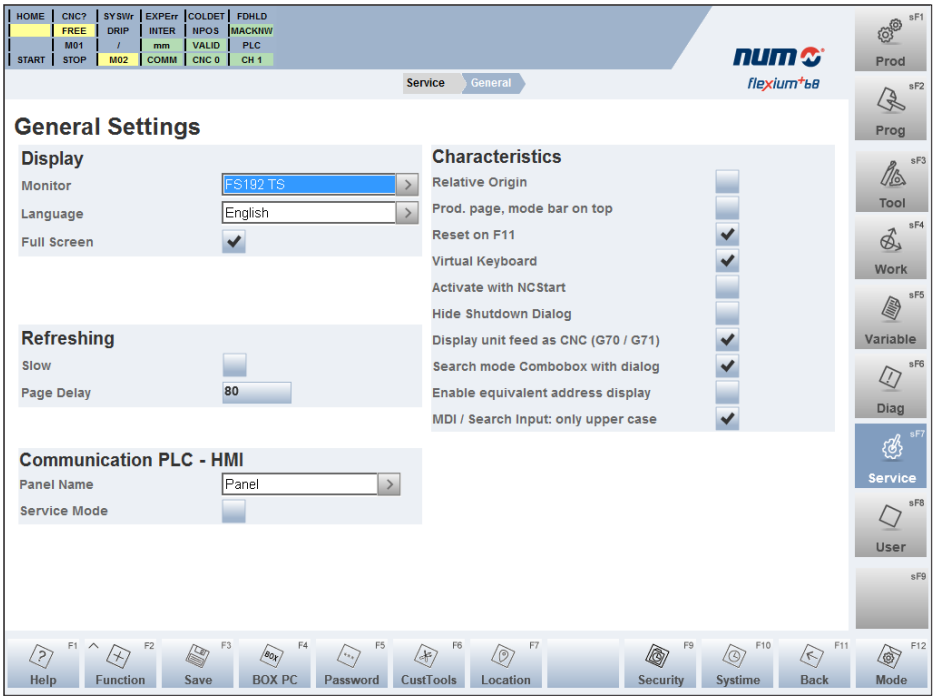
Press F5 to access the *"Management settings"* page.

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↵

9.2 General Settings Page



Press F3 in the Main page of the *‘Service’* context to access this page.

Possible actions

Press F10 to access the system time and date settings.
Press F9 to access the Security Settings.
Press F7 to access the data and file location settings.
Press F5 to access the password settings.
Press F4 to access the Box PC.
Press F3 to save the settings.
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter↵

9.2.1 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↵

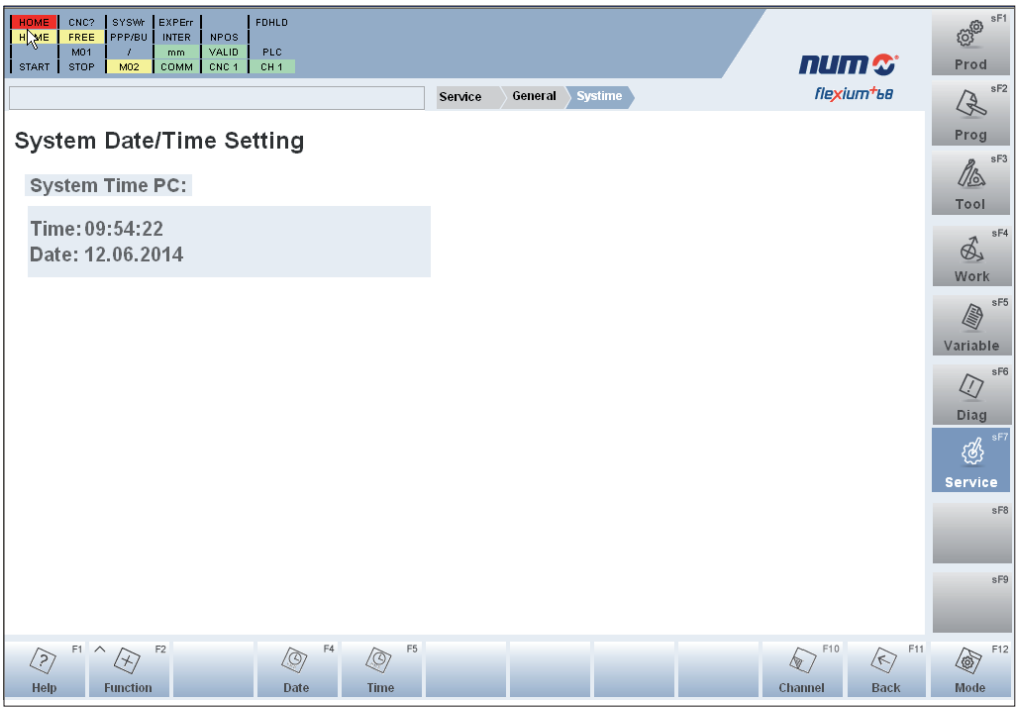
Explanation of the setting options

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.
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.
Characteristics	
Relative Origin	Allows the activation of the relative axes positions display in the context “Production”.
Prod. page, mode bar on top	Entering the production page (sF1), the CNC function bar will be selected by default.
Reset on F11	Activate this box to display the “Reset” key in the CNC function bar. .
Virtual Keyboard	Virtual keyboard allows to call up an “on screen” virtual keyboard with the key F2 “Function” in all pages providing a data input possibility.
Activate with NCStart	When several programs are running simultaneously, it is possible that the HMI is switched to the background and is no more visible on the screen. Activating this box causes the HMI to be displayed again in foreground as soon as the CNC Start button is pressed.
Hide Shutdown Dialog	Checking this box hides the shutdown dialog when pressing Alt + F4. The HMI will be closed immediately and the system remains under power.

Display unit feed as CNC (G70 / G71)	Display the Feed unit depending on the reference dimension of CNC (G70/G71). If it is deactivated, the, the Feed unit is displayed using the axes display unit.
Search mode	Search mode Combo box with Dialog in the Search mode is a Dialog with a wizard use. See Chapter 11.4.7 Search Dialog wizard.
Enable equivalent address display	Activate the display of axes address equivalence on the PRO context For information about address equivalence, refer to the programming manual. **Pos. OP** **Delta** **@X=X** **5640.036** **0.000** **@Y=Y** **1747.588** **0.000**
Mdi /Search Input in upper case	The dialog in MDI or Search mode will be in uppercase regardless of the keyboard Caps Lock Status.

9.2.2 System Time and Date Settings

This page allows to set the system date and time.
To access this page, press F10 in the *General Settings* page of the context *'Service'*.



Possible actions

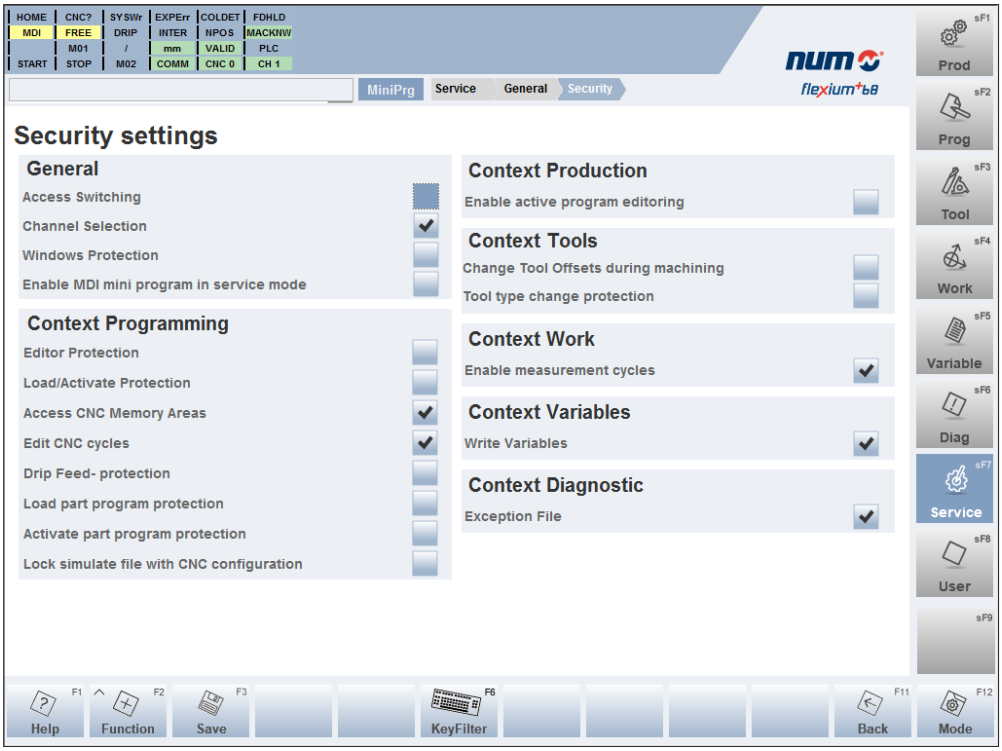
Press F10 to enter in the main page.
Press F9 for Logging and history of the messages
Press F8 for PLC visu settings
Press F7 for Axes display settings
Press F5 to set the system time.
Press F4 to set the system date.
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter ↵

9.2.2.1 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 ↵

9.2.3 Security Settings

Press F9 ‘Security’ in the General Settings Page of the *‘Service’* context to access this page.



Possible actions

Press F3 to save the settings.
Press F2 access all existing functions. Select the function with ↑ or ↓ and Enter ↵

F6 (Key filter) available with Windows 10 gives the possibility to lock some keys of the standard keyboard. It require the Keyboard filter device to be enabled in Windows. Refer to the Windows help for further information.

Setting options description

General	
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 + F11 .
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.
Windows protection	Activating this option prevents a direct access to Windows. Full functionality of the HMI remains available. STOP Use extreme caution with this field. Windows functions will not be accessible any-more without entering a password.
Enable MDI mini program in service mode	Activating this option allows to enter the MDI mini program mode when service mode is active.

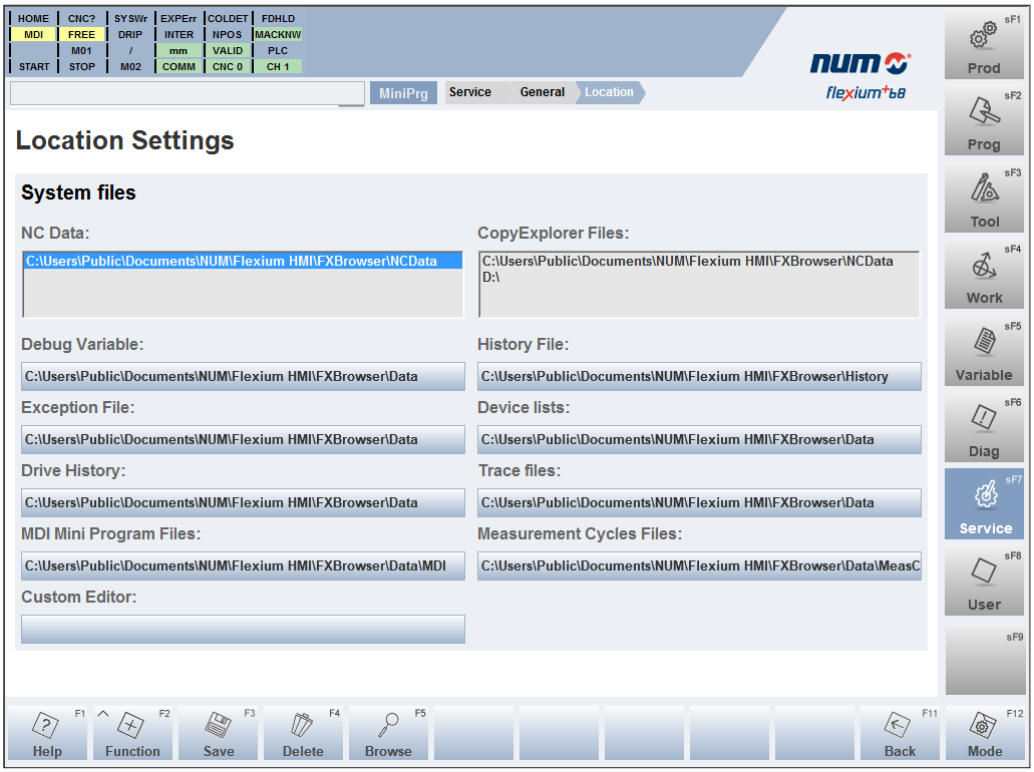
Context Programming	
Editor protection	If this box is activated, there is no possibility to access the editor in the Programming context. No file on PC disk can be modified or deleted by the HMI.
Load/activate protection	Usually, the functions "*Save, load and activate"* of the editor as well as *"Load and activate"* when loading a part program into the CNC are enabled. These functions can be disabled by checking the box.
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. Additional information can be found in the Commissioning manual M00044.
Drip Feed- protection	Prevents the Dripfeed Mode.
Load part program protection	Prevents the possibly to load Part Programs in the CNC.
Activate part program protection	Prevents the possibly to activate Part Programs in the CNC.
Lock Simulate file with CNC configuration	To prevent loading of CNC configuration from NCK (Tool list, Offset table and Macro files) before Simulation start activate this checkbox.
Lock Offline Teach In	To prevent Teach In in CNC Editor from files stored on the PC disk, activate this checkbox. Note: This checkbox is only available with the Options FXSW282114.
Lock Online Teach In	To prevent Teach In in CNC Editor from files stored on the CNC Memory, activate this checkbox. Note: This checkbox is only available with the Options FXSW282114.
Context Production	
Enable active program editing	Activate the possibility, to Edit the active programs with F5 in the context “Production”.
Context Tools	
Change Tool Offsets during machining	Change Tool Offsets during Machining. Checking this box allows for changing the tool length values during a stop.
Tool type change protection	Prevents the possibility to change the Tool type in the context “Tool”.
Context Work	
Enable measurement cycles	Enable F10 in the work context to give access to the measurement cycles.
Context variables	
Write variables	Enable writing the values of the R/W variables displayed.
Context diagnostic	
Exception file	Enable F8 in the diagnostic Context (Display of last Exception).

9.2.4 Location Settings

This page shows and allows changing the path for the different files in the system. It is generally not necessary to change such fields.
A common use is for adding an NCData or Copy Explorer folder. (on the network or an USB Stick).

To access this page, press F7 in the *General Settings* page of the *'Service'* context.



Uncontrolled changes can cause system dysfunction.

Possible actions

Press F7 to open the Location Settings page.
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↵

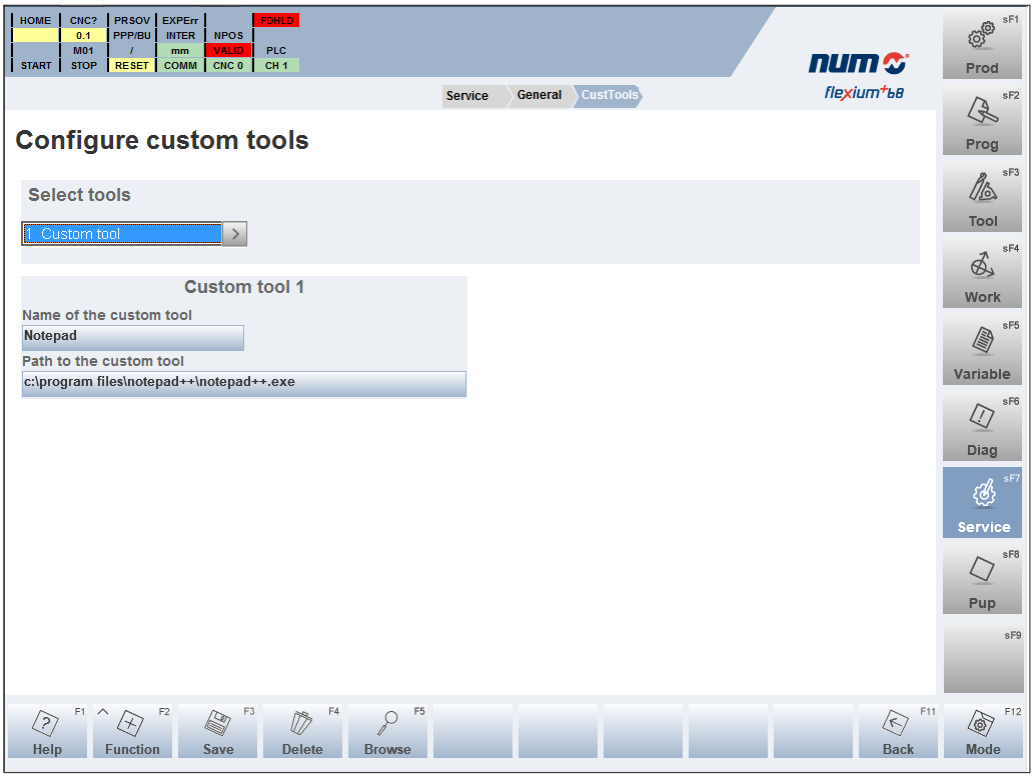
9.2.4.1 Keyboard functions

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 ←

9.2.5 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.

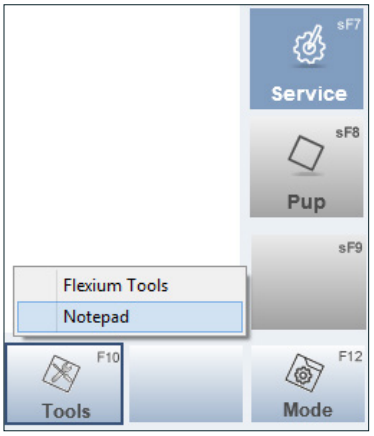
To access this page, press F6 in the *General Settings* page of the *‘Service’* context



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.



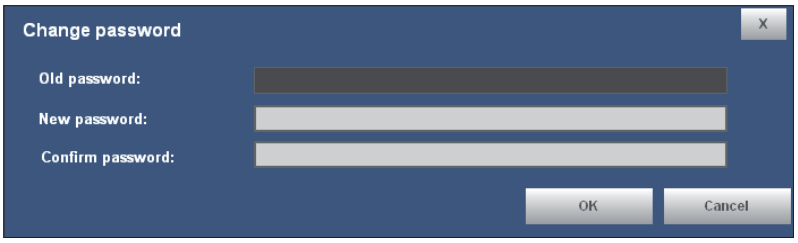
9.2.6 Password Settings

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

As said in the chapter 9.1 it is important to change this password after commissioning the machine to prevent unauthorized access.

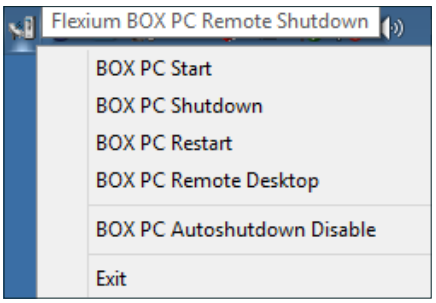
The prompt to change the password is obtained by pressing F5 in the *General Settings* page of the *'Service'* context.

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



9.2.7 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).

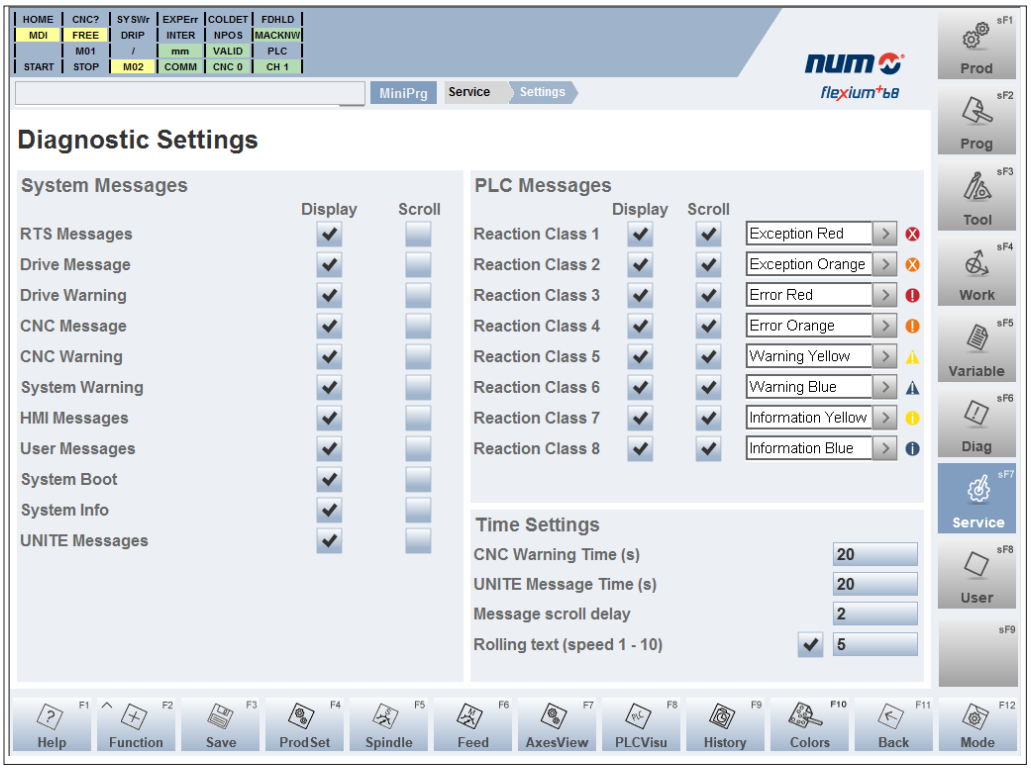


9.3 Diagnostic Settings

This page defines which messages are displayed in the Message banner and the Diagnostic Page.

In the message banner messages can be selectively defined as scrolling or not.

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



Possible actions

Press F10 to enter the Colors settings.
Press F9 to access the history setting.
Press F8 to access the PLC Visu setting (Optional). For this function, please refer to the chapter PLC visualisation in the additional functions chapter.
Press F7 for axes view settings.
Press F6 to access the feed rate display settings page.
Press F5 to access the spindle display setting page.
Press F4 to enter in the “Productions Setting” page.
Press F3 to save the settings.

System messages

The check box in front of each category defines if the type of message should be displayed or not. This type of message can scroll or not according to the setting in the dedicated column.

RTS Messages	Messages issues by the RTS system.
Drive message	Messages of the digital drives.
Drive warning	Warnings of the digital drives.
CNC Warning	Warnings issued by CNC.
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.
System Info	Operating system messages.
UNITE message	UNITE messages are displayed a short time in the message field every time a communication error with the CNC is detected.

PLC messages

Adjust the settings in columns Display and Scroll to get the desired behaviour.

Reaction class	Two settings are available for each reaction class: • Defines if this class of message should be displayed. • Icon : Represent the message importance.

Time Settings

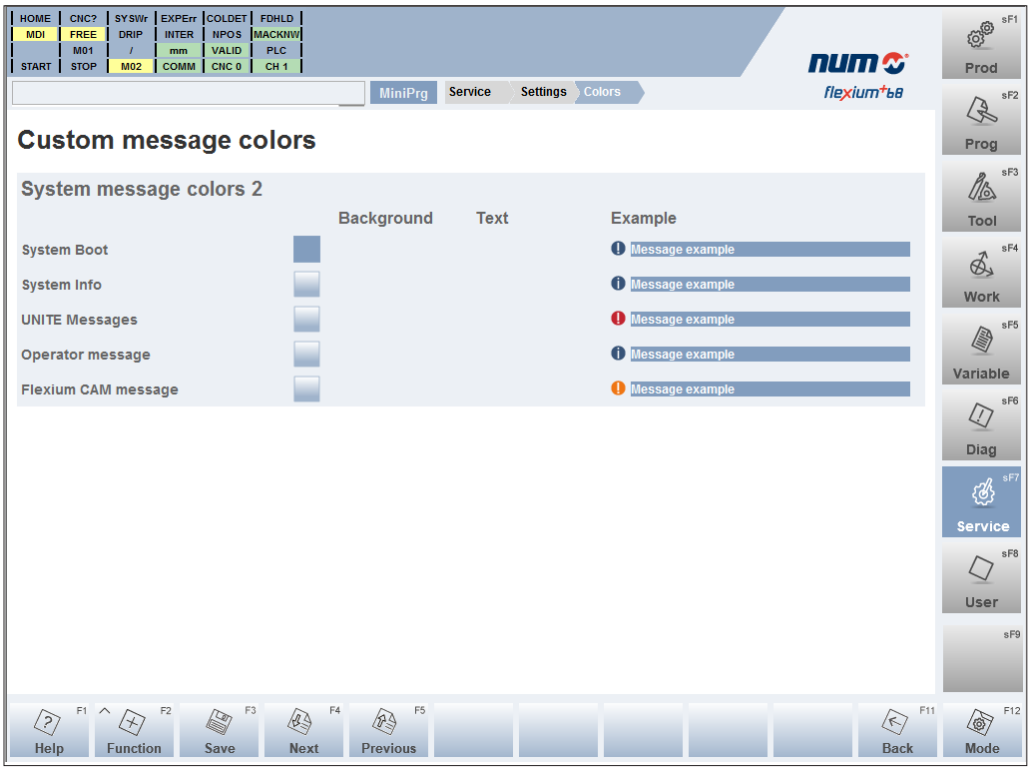
CNC Warning Time	The default display time is set at 20 seconds. You may modify it in a range from 1 to 60 seconds.
UNITE Message Time	The default display time is set at 20 seconds. You may modify it in a range from 1 to 60 seconds.
PLC Msg Scroll Delay	Display of the messages in the message field. If more than 3 messages are pending, the display will be scrolled in a predefined time to allow the next messages to be displayed.
Rolling text active	Activate the rolling in the message banner.
Rolling text speed factor	Change the speed of the Rolling text in the message banner.

9.3.1 Custom message colors

9.3.1.1 System message colors 1

Press “Service” then “ F4 Diagnostic Service” and F10 colors buttons.

Enable the checkbox to change the background and the text colors of the system message colors.



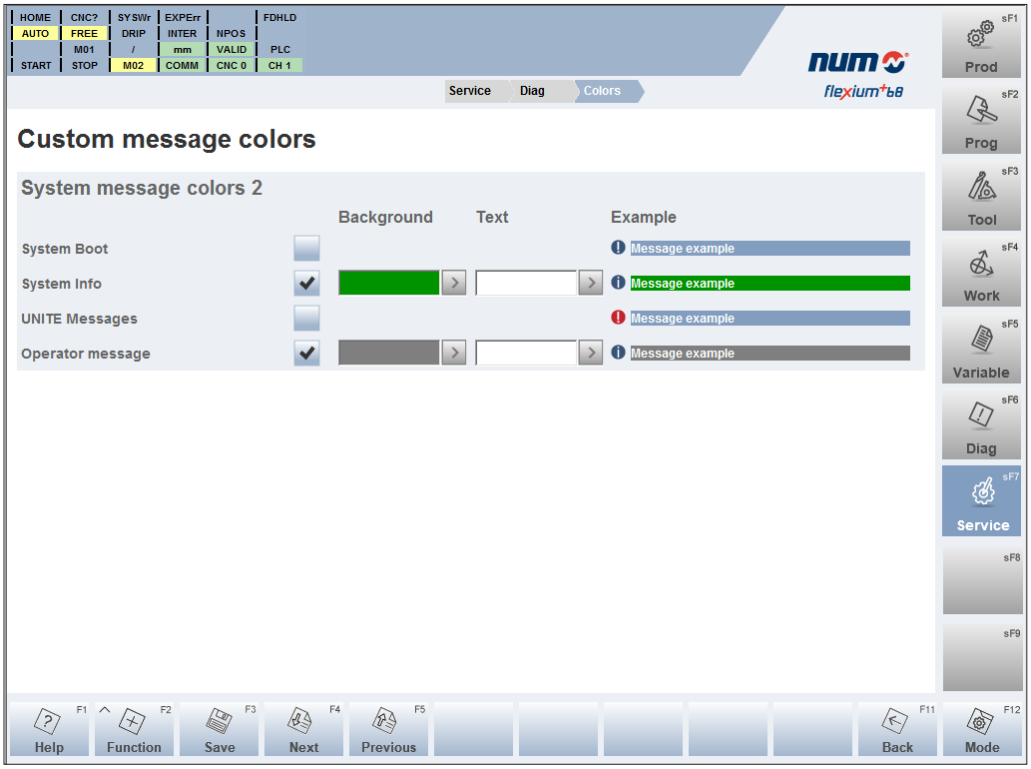
Possible actions

Press F3 to save the colors.
Press F4 to next page (System message 1 or 2 or PLC selections).
Press F5 to previous page (System message 1 or 2 or PLC selections).
Press F10 to the Diagnostic Settings page.

9.3.1.2 System message colors 2

Press “Service” then “ F4 Diagnostic Service” and F10 colors buttons , then F4 : “Next”.

Enable the checkbox to change the background and the text colors of the system message colors.



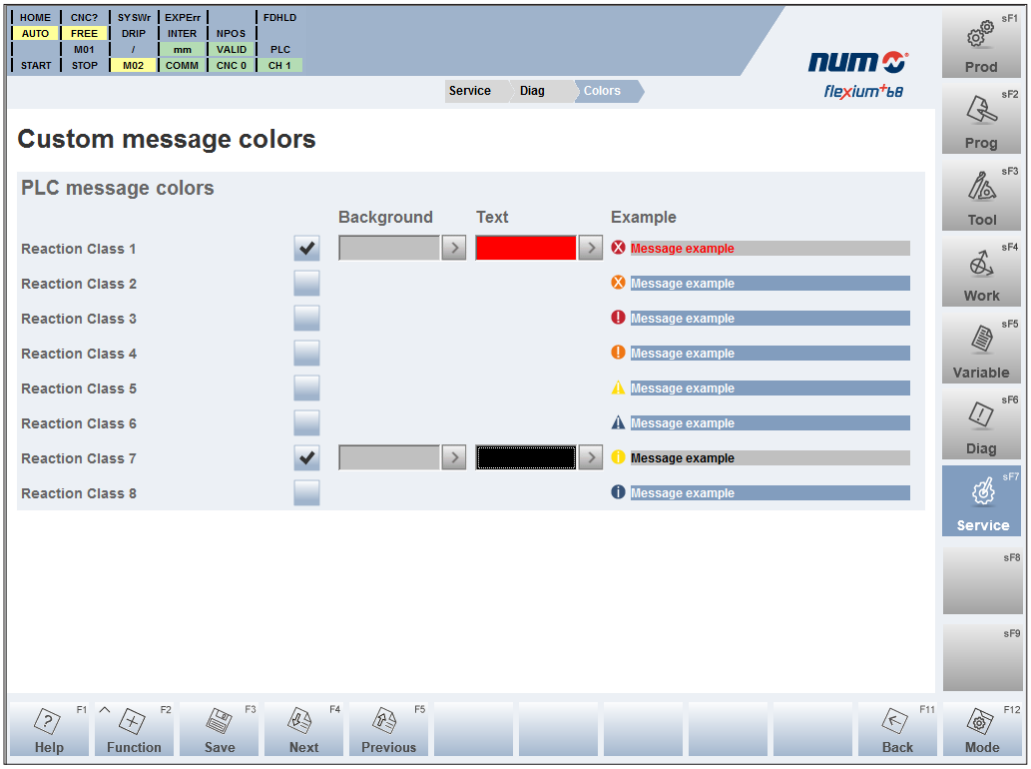
Possible actions

Press F3 to save the colors.
Press F4 to next page (System message 1 or 2 or PLC selections).
Press F5 to previous page (System message 1 or 2 or PLC selections).
Press F10 to the Diagnostic Settings page.

9.3.1.3 PLC message colors

Press “Service” then “ F4 Diagnostic Service” and F10 colors buttons, then twice F4 : “Next”.

Enable the checkbox to change the background and the text colors of the system message colors.



Possible actions

Press F3 to save the colors.
Press F4 to next page (System message 1 or 2 or PLC selections).
Press F5 to previous page (System message 1 or 2 or PLC selections).
Press F10 to the Diagnostic Settings page.

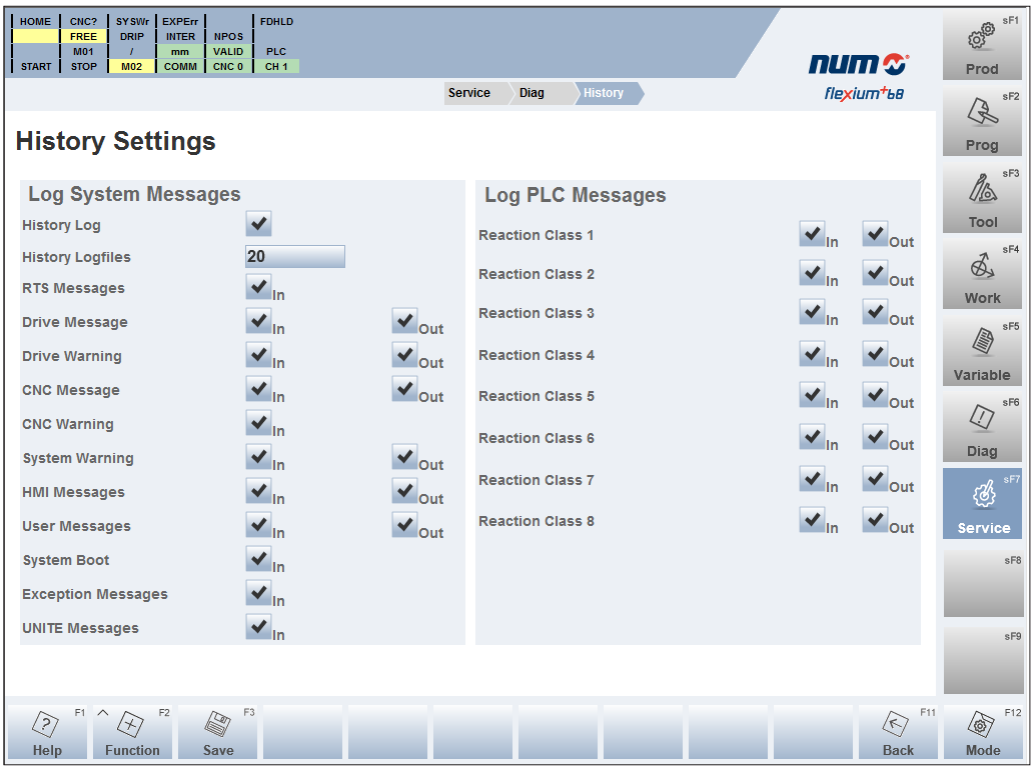
9.3.2 History

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.

9.3.2.1 History setting

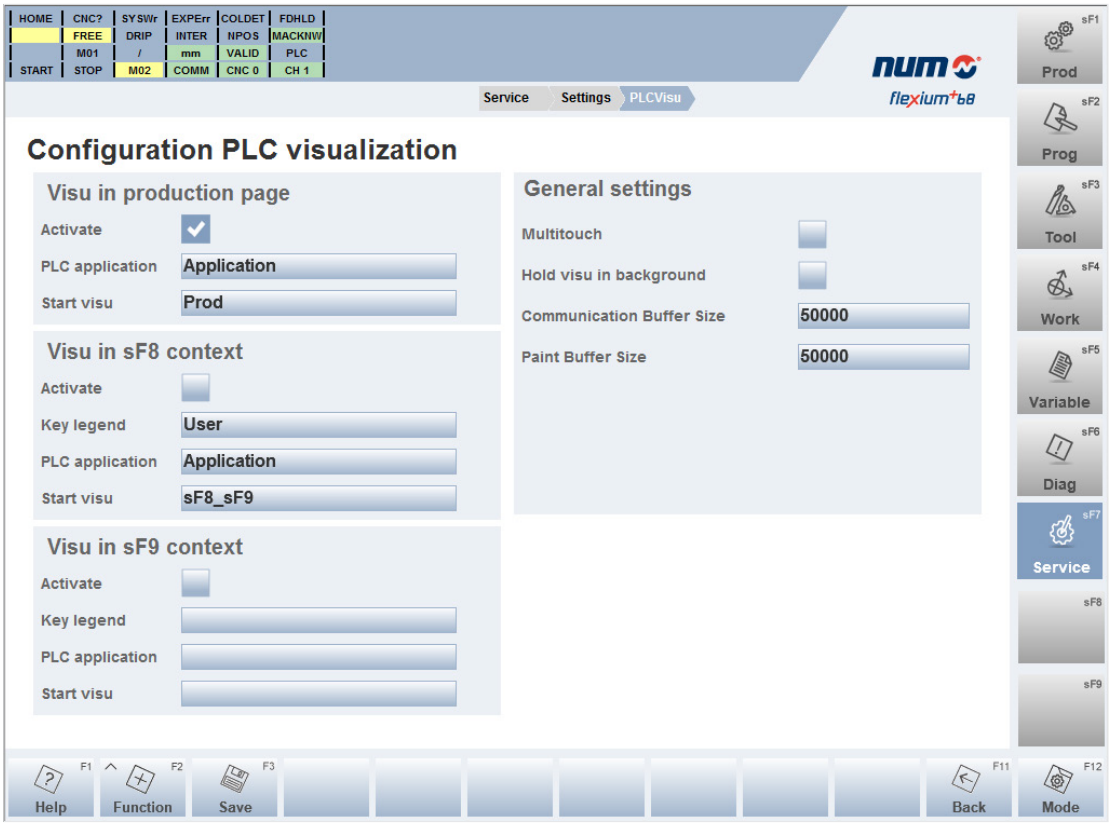


9.3.3 PLC Visu

This page allows for defining three different visualisations that can be displayed

- In the right tab of the PROD context
- In sF8 context (empty by default)
- In sF9 context (empty by default)

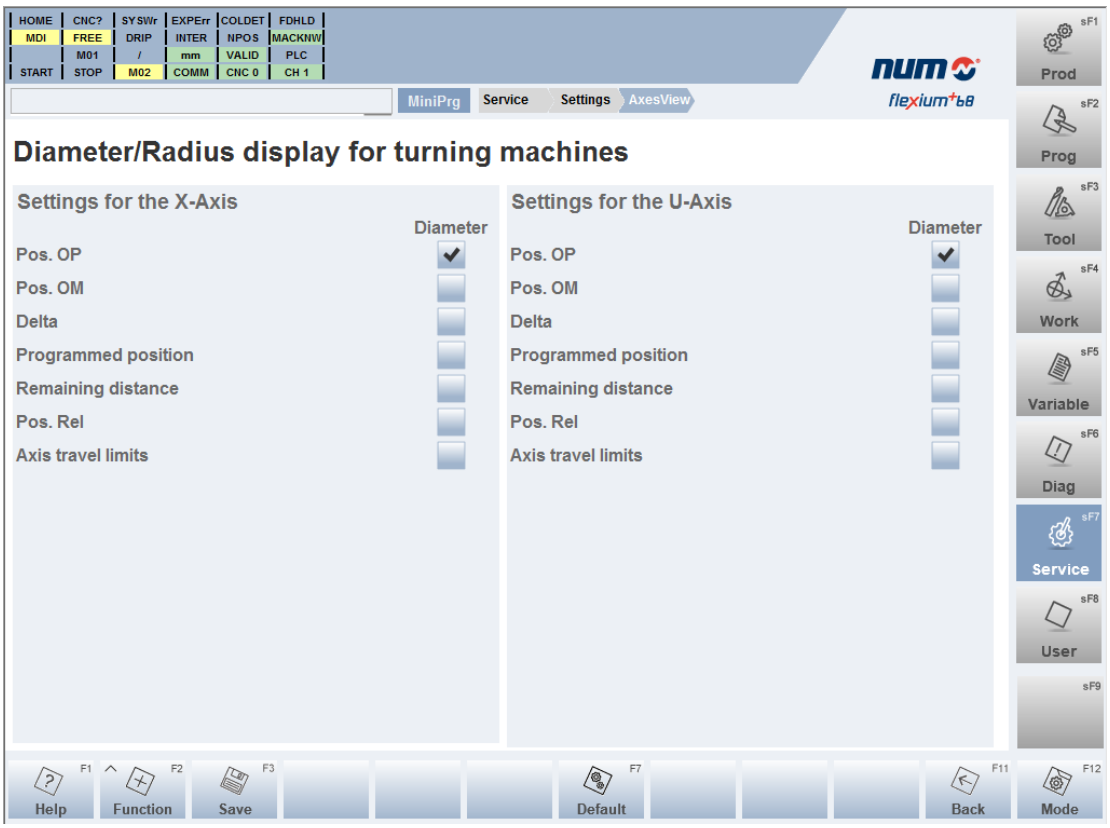
Those graphic visualisations are defined by the machine builder. Generally the settings of this page should not be edited



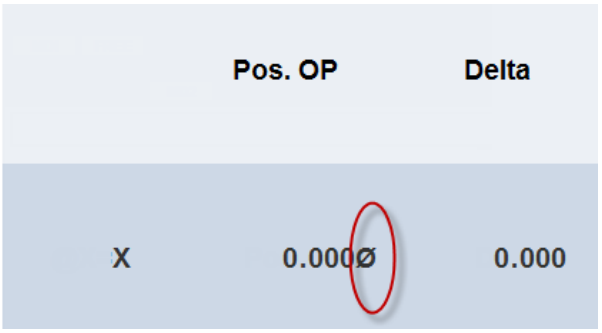
9.3.4 Axes view

This page permits to define the characteristics of axes display setting on turning machine ‘display at radius or diameter).

The settings can be defined for X and U axis. By default they are at the diameter when displayed at OP and the radius in the other situations.



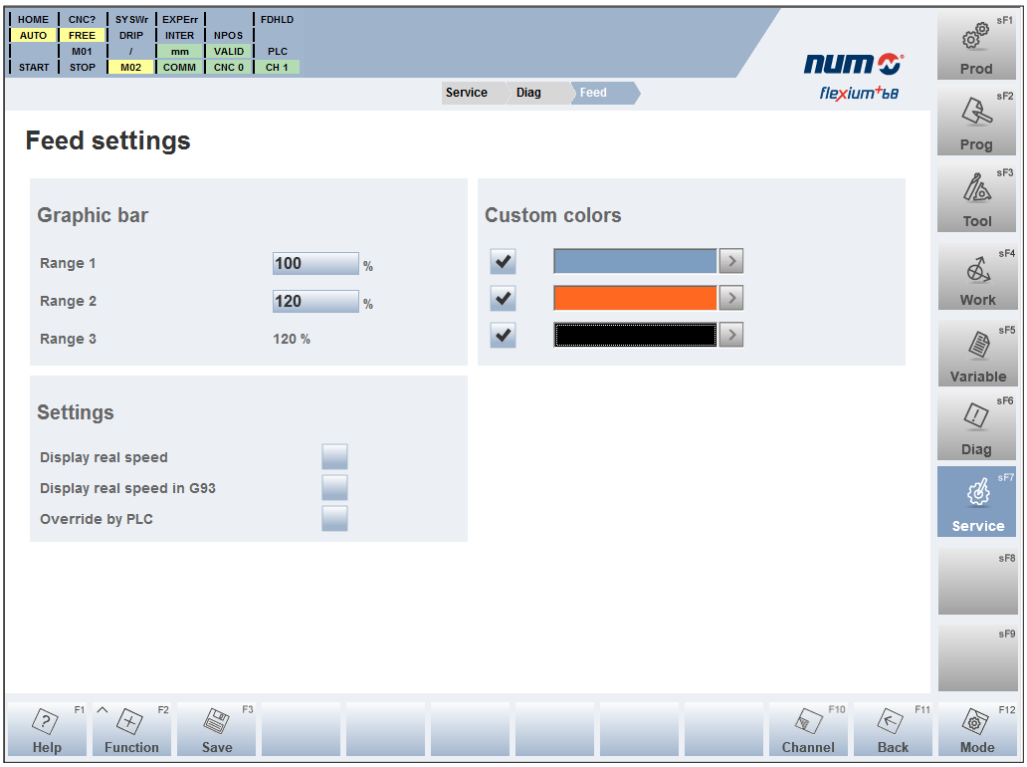
When at the diameter a symbol is shown to confirm this situation.



Feed Rate Settings

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

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



Possible actions

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

Explanation of the setting options
The entire range of the feed rate override can be divided into two sub-ranges.

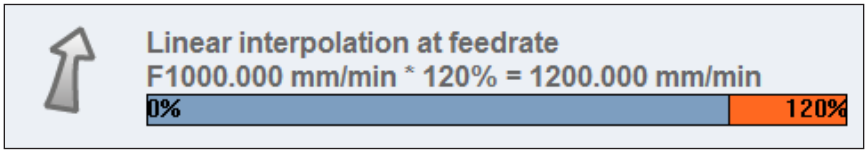
Graphic bar - Range	
Select a setting	☞ Tab
For the % fields: enter the maximum value of the range and validate	☞ Enter

Settings	
Display real speed	Check this box to display the real axes speed in the axes position pages. (for linear axes only)
Display real speed in G93	If this checkbox is selected, the real speed is displayed in the G93.
Override by PLC	If this box is checked, the displayed feed override value will be the one given by the PLC only. If not checked it will be the value given by E79002 parameter. Checking the box prevents erratic values to be displayed in case the override is continuously modified by Dynamic Operators. (e.g. waterjet cutting, glue deposit etc).

Custom colors

Checking the box customer colors is possible to change the colors of the 3 speed areas in the progress bar.

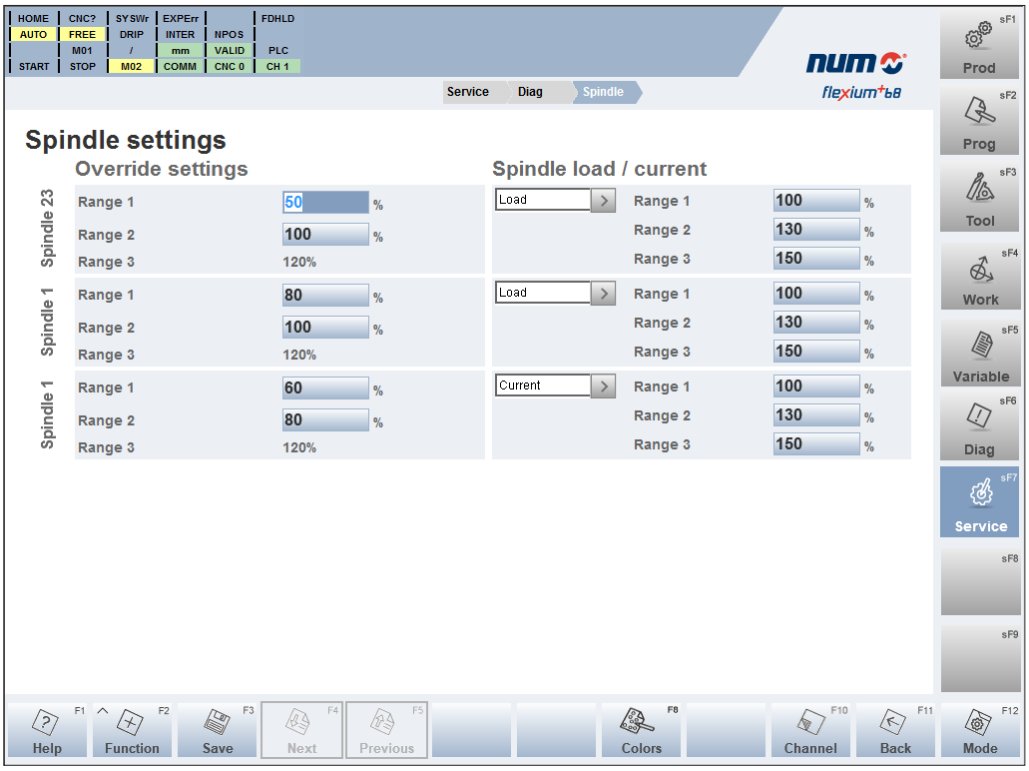
Example



9.3.5 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.

To access this page, press F5 in the *'Diagnostic Settings'* of the *'Service'* context.



Possible actions

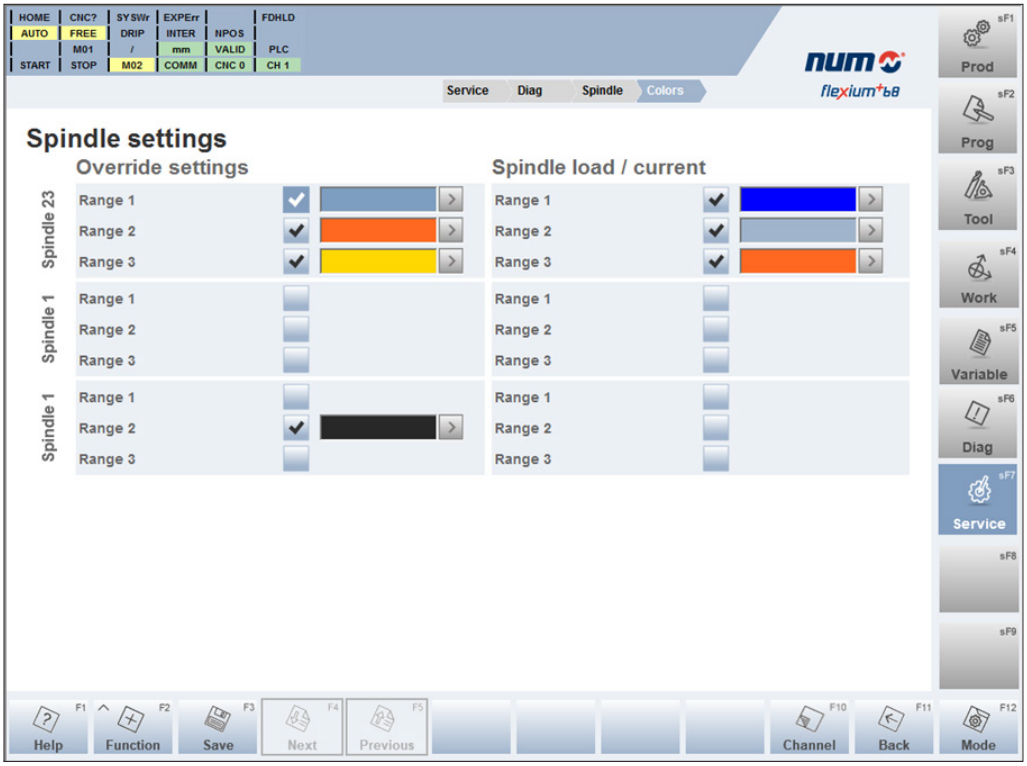
Press F8 to change the colors.
Press F5 to previous page.
Press F4 to next page.
Press F3 to save the settings.
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter ↵

Explanation of the setting options

Setting options for spindle override	The entire range of the spindle override can be divided into two sub-ranges. An own colour is assigned to every sub-range. For explanations of the settings, refer to chapter *‘9.3.3 Feed Rate Settings’* in this manual.
Settings for Spindle Load / Current	The entire range of the spindle load can be divided into two sub-ranges. An own colour is assigned to every sub-range.
Spindle Load	Display the Spindle Load: If the Variable *Flexium_NCK.WCNC.Spindle[x].Load.ByPLC* is true, the Spindle Load is displayed from the PLC Variable *Flexium_NCK.WCNC.Spindle[x].Load.Value*. If in MP 7 is configured in the DISC NT+ Drive, the Spindle Load is displayed from the Drive otherwise, the Spindle Load is not available.
Spindle Current	Display the Spindle Current: If the Variable *Flexium_NCK.WCNC.Spindle[x].Current.ByPLC* is true, the Spindle current is displayed from the PLC Variable *Flexium_NCK.WCNC.Spindle[x].Current.Value.* If in MP 180 is configured in the DISC NT+ Drive, the Spindle current is displayed from the Drive Otherwise, the Spindle Current isn’t displayed.

Spindle color settings

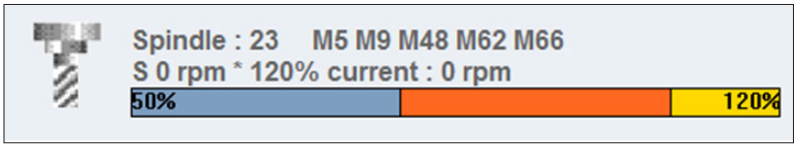
To access this page, press F8 in the *'Spindle Settings'* page of the *'Service'* context.



Possible actions

Press F3 to save settings.
Press F4 to enter next page.
Press F5 to enter previous page.

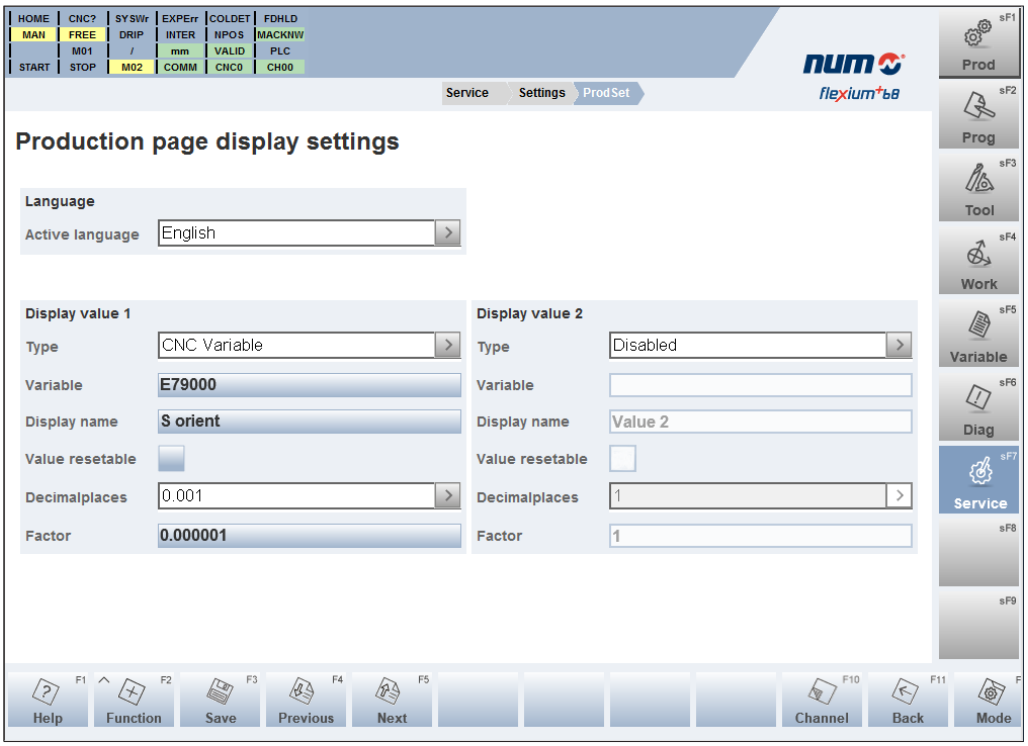
It is possible to change the colors of the 3 ranges in the Spindle progress bar as indicated in the below example:



9.3.6 Production Settings

This page is used to add additional information in the production page.

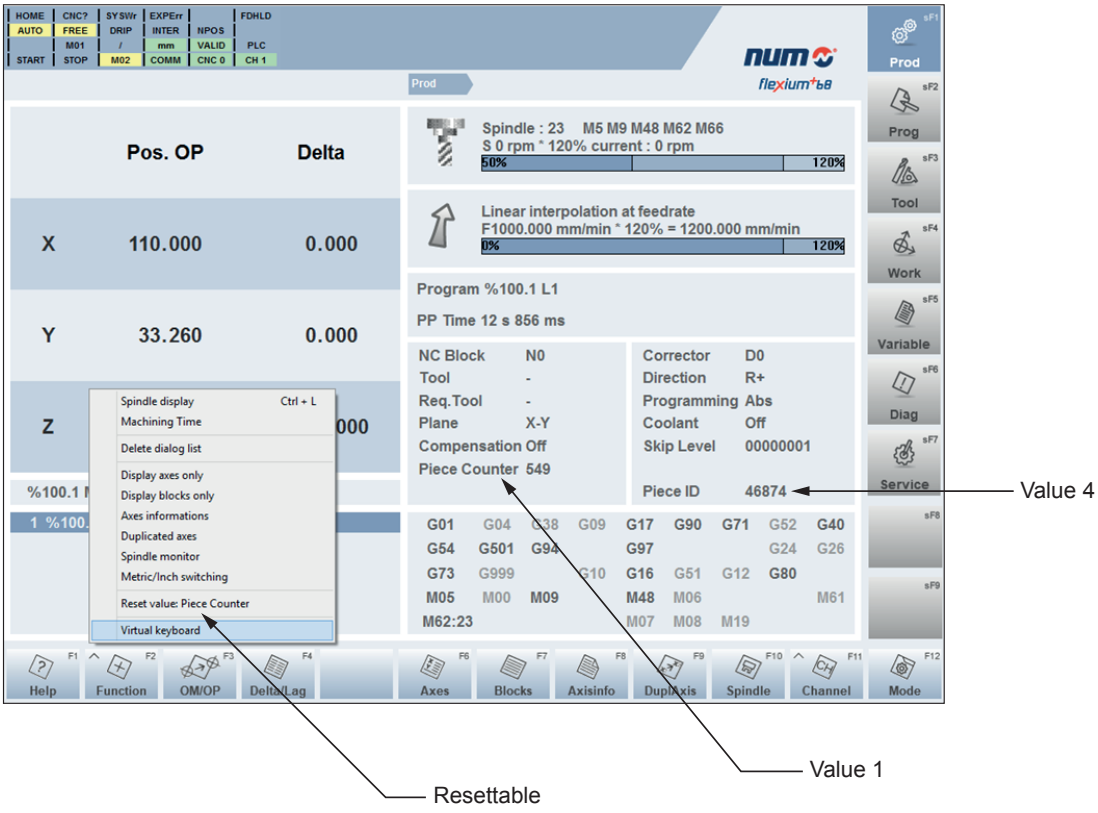
To access this page, press F4 in the *‘Diagnostic Settings’* of the *‘Service’* context.



Language	Allows to select the language
Type	Allows to select whether PLC or CNC variable is used.
Display Name	Allows to displayed the text on the Production page. The text can entered in all language of the HMI. It’s only needed to change the language in the Combo box.
Value resettable	If this checkbox is checked, the value can is resettable over the F2 menu in the production page
Decimal places	Number of decimal places displayed
Factor	Display factor

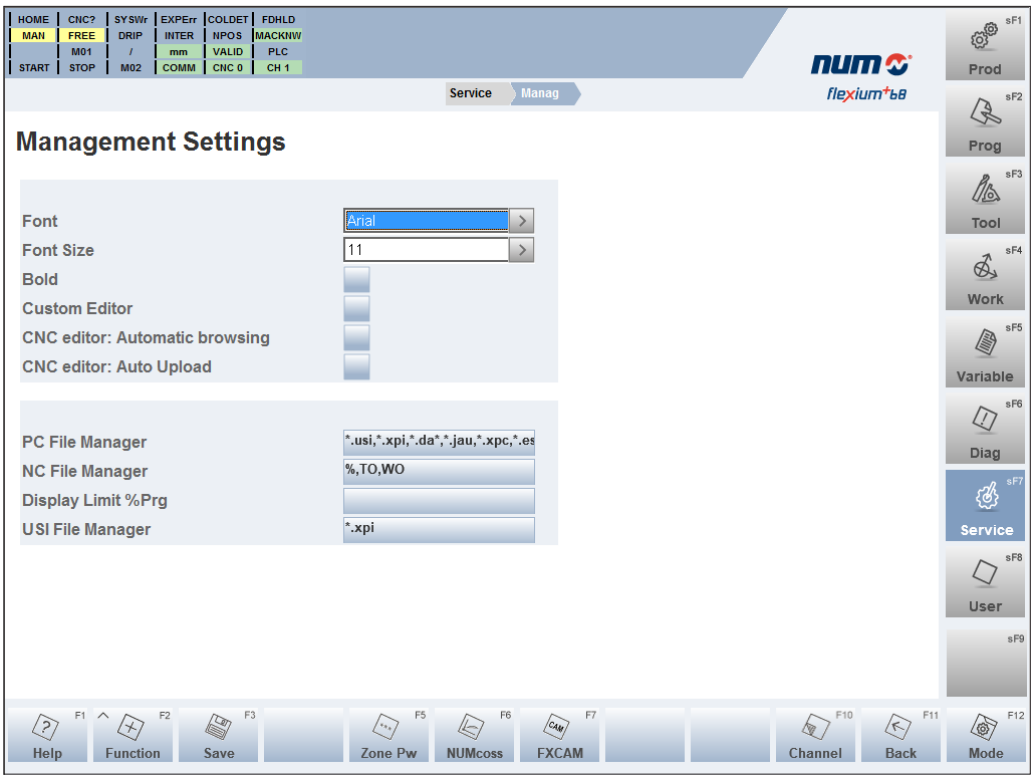
Press F3 to save the selections.

The configured values are displayed in the Production page, see the following picture.



9.4 Management Settings

To access this page, press F5 in the main page of the *‘Service’* context.



Possible actions

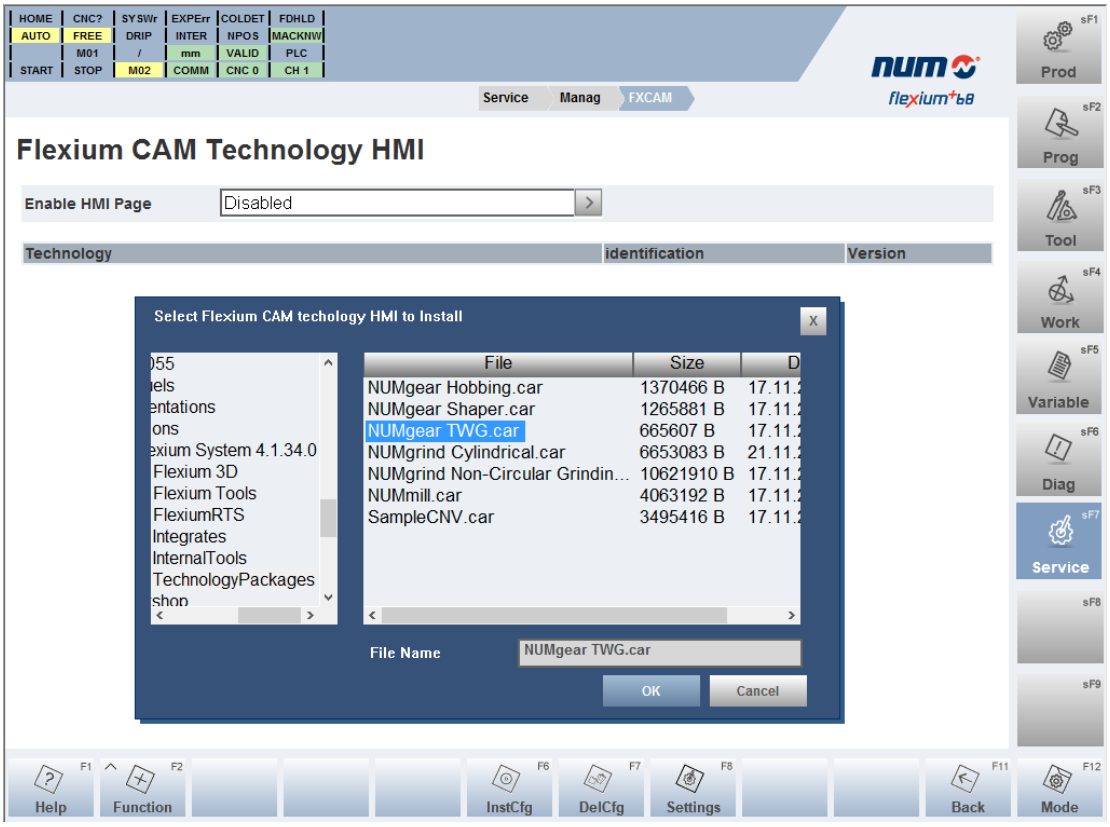
Press F7 to access the FXCAM setting (optional) for information about FXCAM please refer to the FXCAM section in the Additional Functions chapter of this manual.
Press F6 to access the NUMCoss setting (optional) for information about NUMCoss please refer to the NUMCoss section in the Additional Functions chapter of this manual.
Press F5 to manage the password of the different memory zones (Customer or OEM). The NUM Zone password cannot be managed.
Press F3 to save the settings.
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter↵
Press F10 to change the Channel selection. Select the function with ↑ or ↓ and Enter↵

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 *'9.2 Data and File Location Settings'* in this manual for details on how to set the path to this editor.
CNC editor: Automatic browsing	In the CNC editor page: enable automatic scrolling between pages during program execution.
CNC editor: Auto Upload	If the Program has been changed in the CNC Memory, the Editor asks you leaving, if you want to make an upload of the modified part program.
File Manager	
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 % Prg	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.

9.4.1 FXCAM

This page is dedicated to load and customize the FXCAM technologies(NUMGrind, NUMGear, NUMMill, etc.). The setup is described in each of those applications.

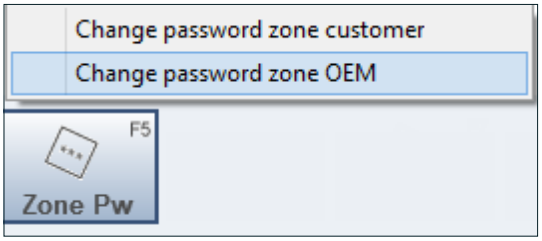


.
Example of loading a technology

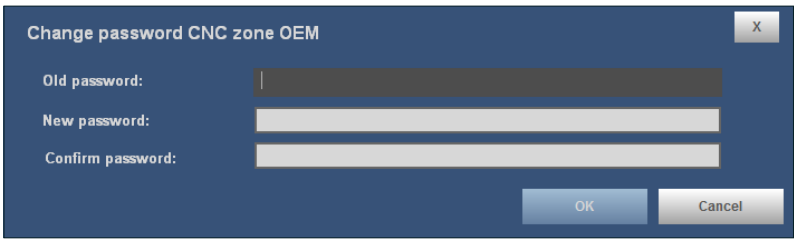
9.4.2 Password management

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.

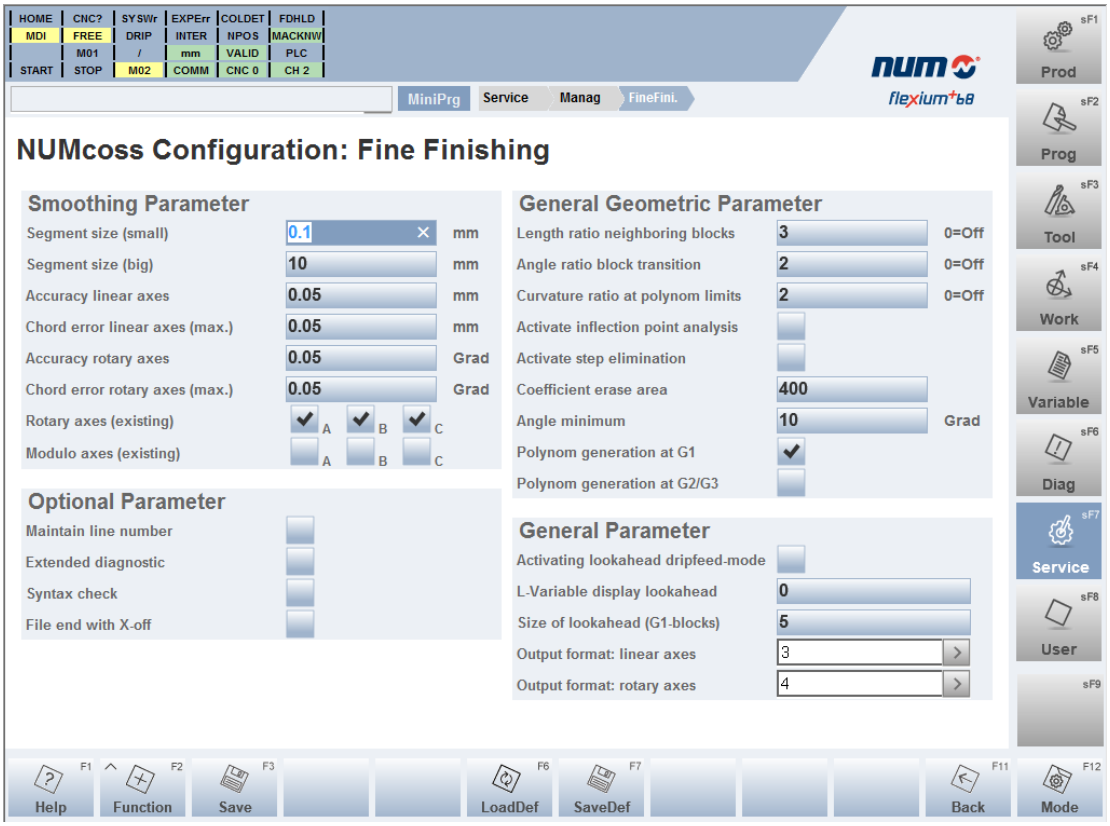


Select the zone password to set or change, the following prompt will be displayed.



9.4.3 NUMCoss

NUMCoss is an application to optimise speed and surface finish performances of part programs programmed by small blocks (tabcycls).



For additional information about the NUMCoss settings, please refer to chapter 12.9

9.5 Machine Data

This page is intended to modify NCK and Drives parameters.

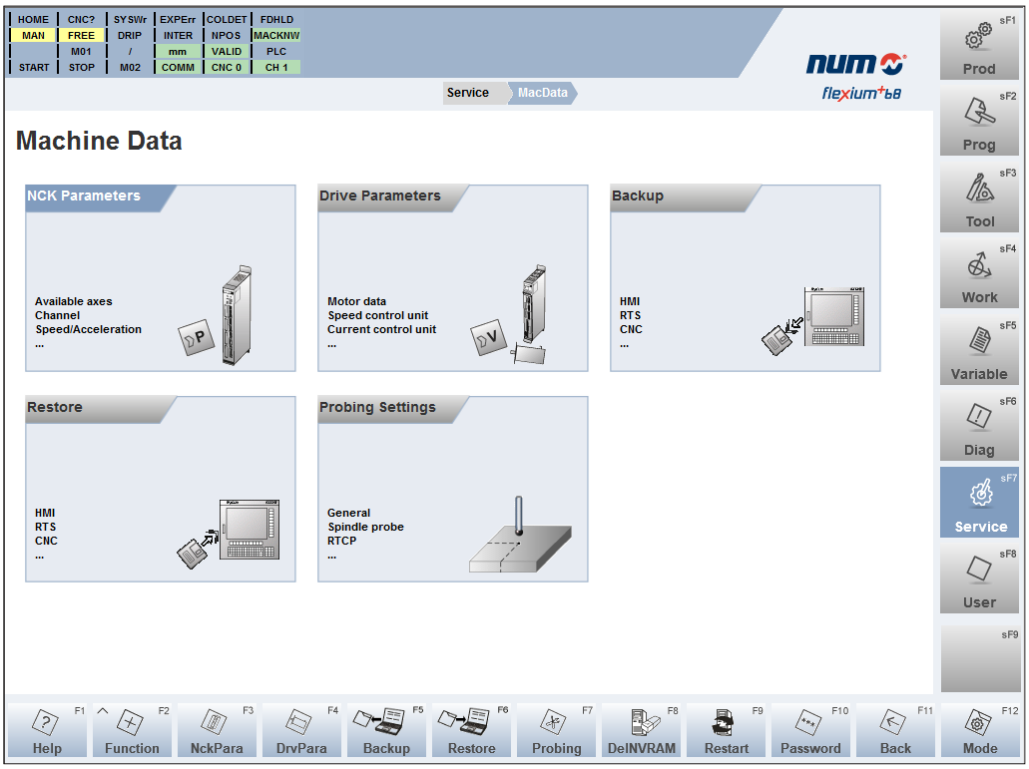
Two levels are authorized:

- *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 F8 to clear the NVRAM on the NCK.
Press F7 to access the Probing setting function
Press F6 to access the Restore function.
Press F5 to access the Backup function.
Press F4 to access the drives parameters.
Press F3 to access the NCK parameters.

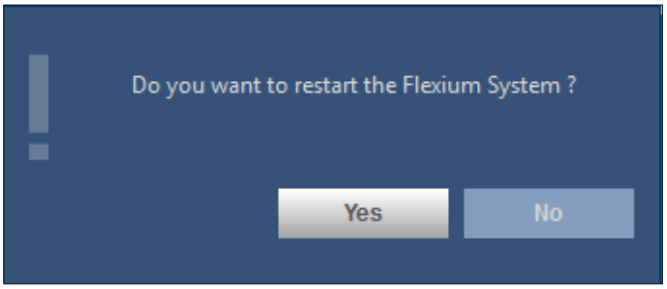


9.5.1 Change Password

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.

9.5.2 Restart



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

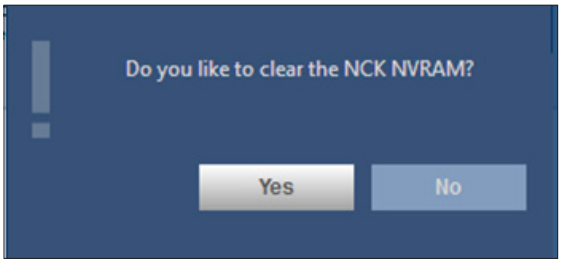
A restart is only possible with *Advanced* access.

To access this command, press F9 in the *‘Machine Data’* page of the *‘Service’* context.

The system will prompt for confirmation and optionally for the advanced access password.

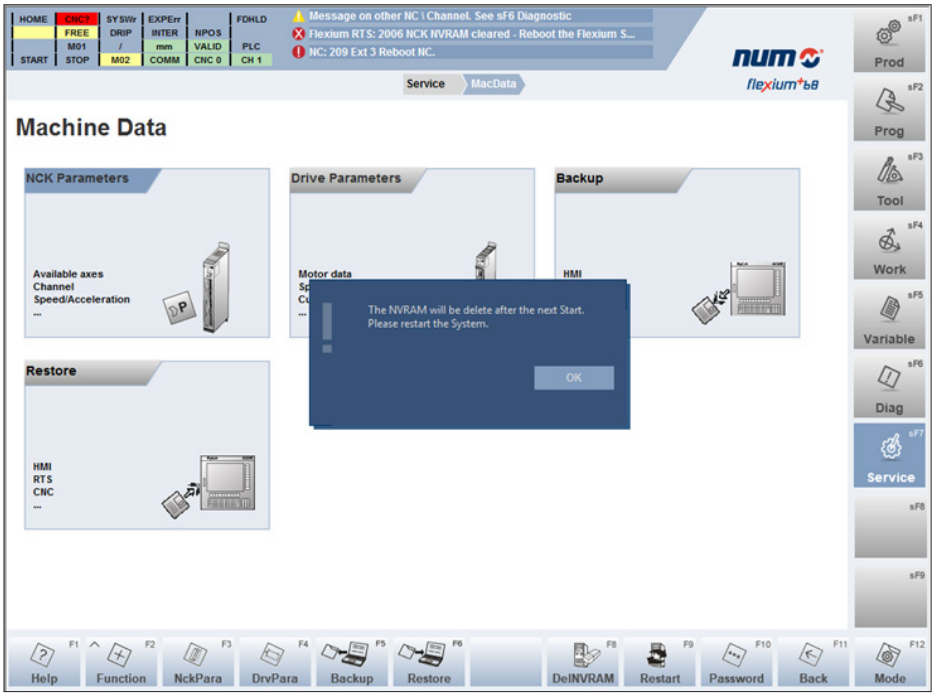
9.5.3 Clear the NVRAM

After clear NVRAM all Persistent data on the CNC are lost!!!



The HMI ask if you like to Clear the NVRAM and ask for the password with advanced access.

A reboot is required at the end of the Clear NVRAM process.



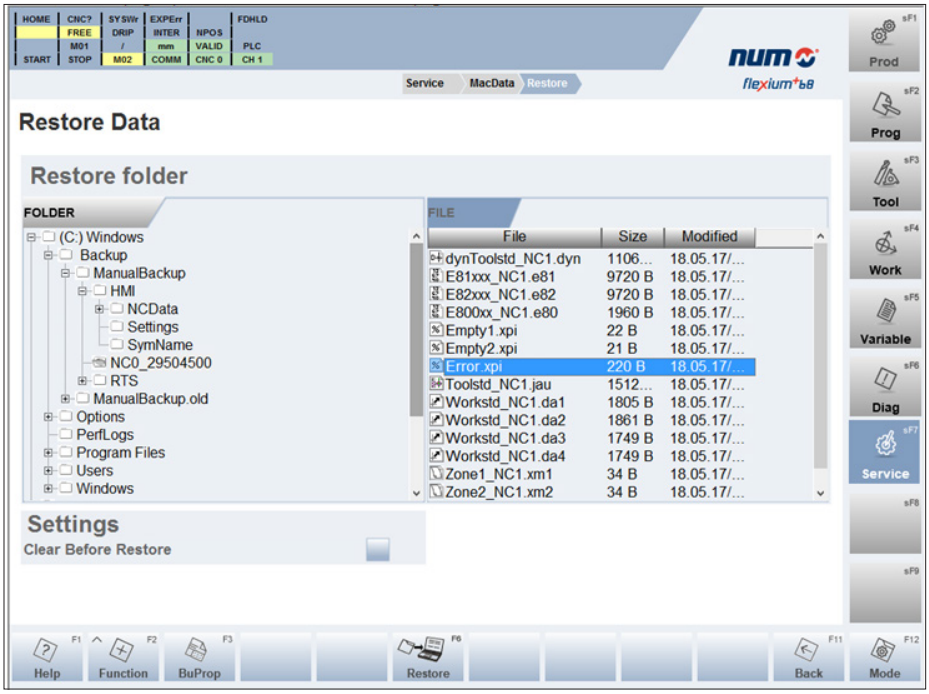
9.5.4 Probing

This page displays and allows for editing the different parameters related to probing, Rtcp and iPlane. It is generally set by the OEM and the values should not be changed .

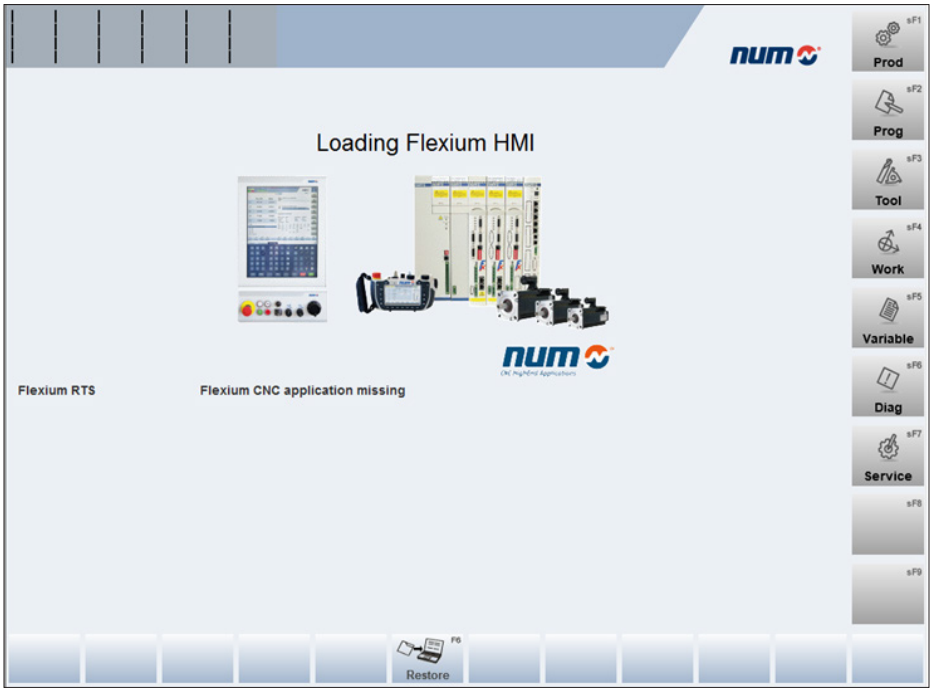
9.5.5 Restore

This page allows restoring the Flexium+ System.

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

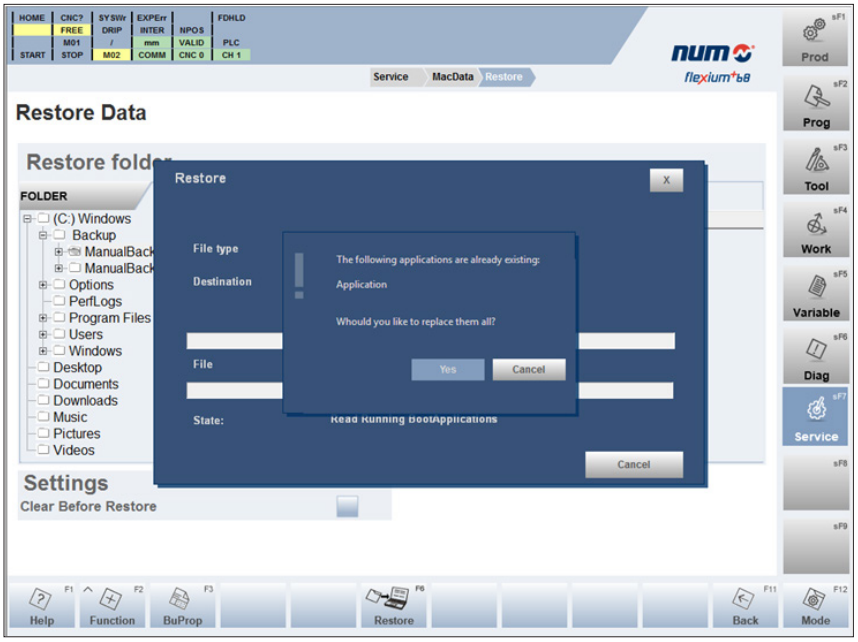


Should the boot project be missing, the loading page will give access to the restore function.

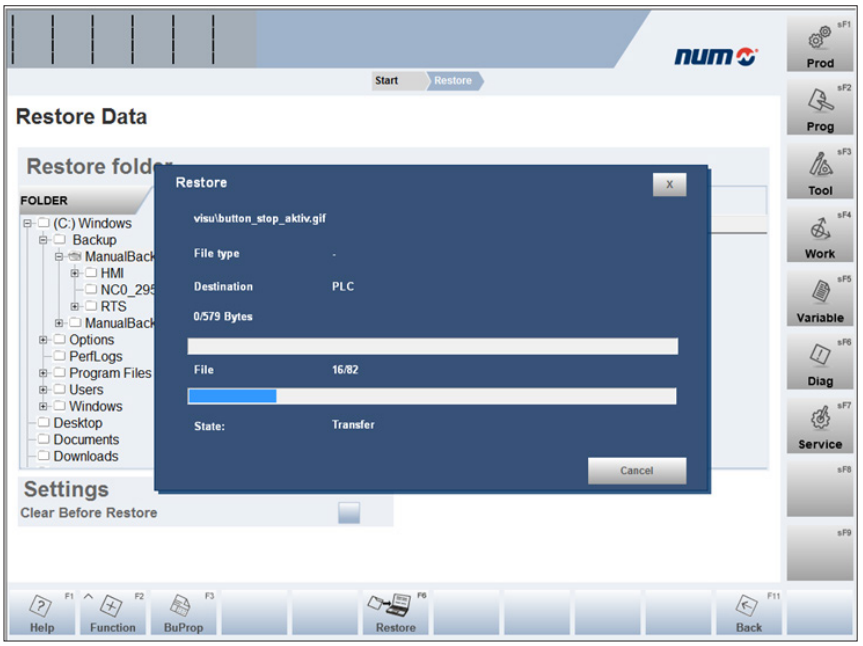


To Restore the Backup, select a backup Folder or a backup file and press F6 .

If a Boot project is present on the Flexium System, the HMI will ask you, if you want to replace it.



It is possible that the HMI restarts several times once the restore is launched.



Possible actions

Press F2 to select Functions.
Press F3 to show the Backup Properties.
Press F6 to Restore a Backup.

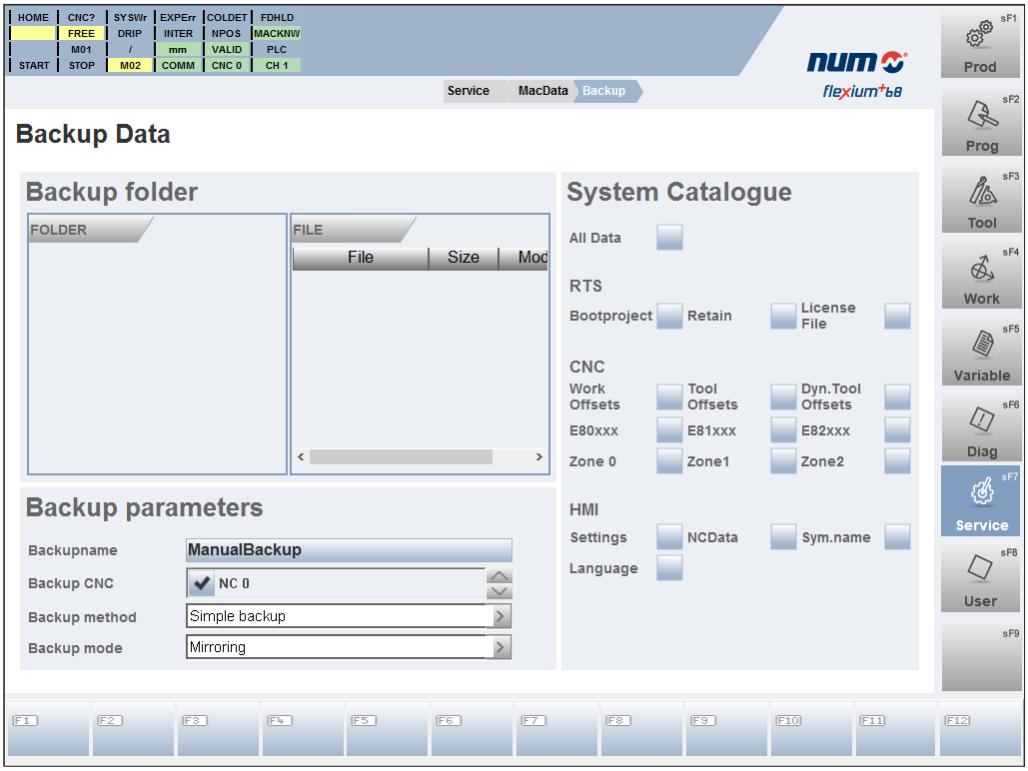
Explanation of the setting options

Settings	
Clear Before Restore	Before the Restore, the NVRAM and all the files on RTS will be deleted.

9.5.6 Backup

This page allows Backing-up of the Flexium+ System.

To access this page, press F5 in the *'Machine Data'* page of the *'Service'* context.



Possible actions

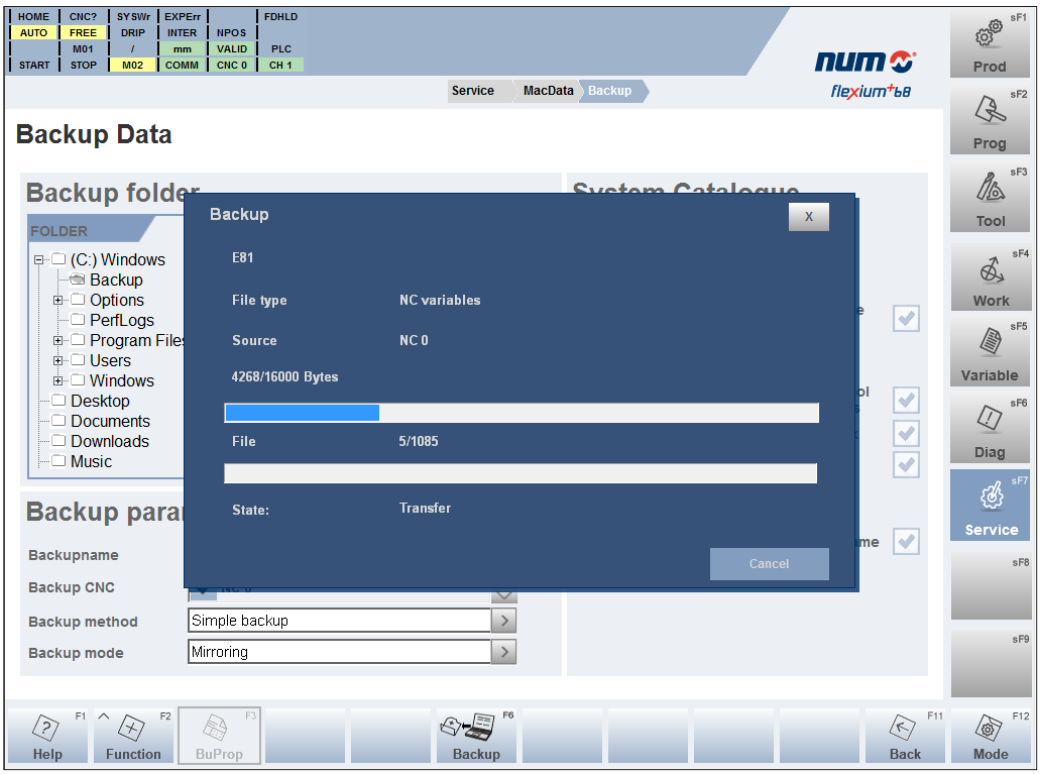
Press F6 to create the Backup.
Press F3 to show the Backup Properties.
Press F2 to select Functions.

To completely backup and restore NUM industrial PC another solution is available called NUMSmartBackup (see M00009 Installation manual). NUMSmartBackup it's a USB stick capable to clone and restore the complete industrial PC mass memory device including:

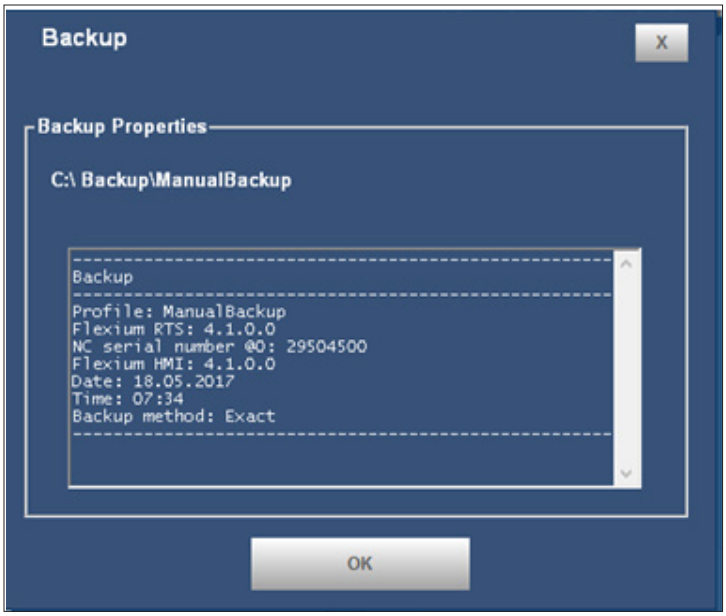
- Windows image and drivers,
- Flexium suite,
- Licence files,
- Part programs etc.

To create a Backup, select the folder to store the Backup and the options in the System Catalogue and Backup parameters blocks.

Press F6 to start the Backup.



When a backup (file or folder) is selected, the function key F3 gives access to a window with the backup information.



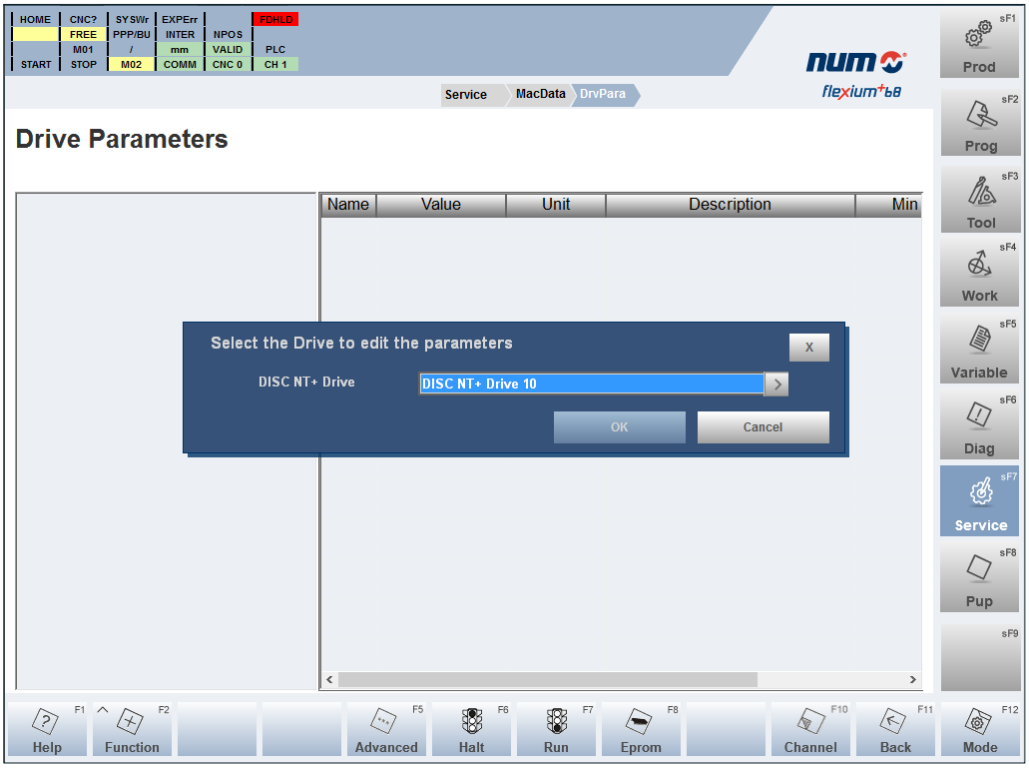
Explanation of the setting options

Backup parameters	
Backup name	Enter the name of the Backup folder / xbu file.
Backup CNC	Select the CNC, where the Backup has to be performed.
Backup method	Simple backup: A previous backup with the same name will be overwritten. Exact backup: A previous backup with the same name will be renamed as *.old at the beginning of the backup process.
Backup mode	Mirroring: A Folder will be created with the Name of the Backup. Backup file (*.xbu): A *.xbu file will be created with the Backup name.
System Catalogue	
All Data	Backup all Data. The settings below aren't considered any-more.
RTS	
Bootproject	Backup the Bootproject. This contains: - Application. - Visualisation. - Project archive.
Retain	Backup the PLC retain variables stored in the File or NVRAM.
License File	Backup the License file.
CNC	
Work Offsets	Backup the Work Offsets.
Tool Offsets	Backup the Tool Offsets.
Dyn.Tool Offsets	Backup the Dynamic Tool Offsets.
E80xxx	Backup all E80 Parameters.
E81xxx	Backup all E81 Parameters.
E82xxx	Backup all E82 Parameters.
Zone 0	Backup the CNC files in the Zone 0.
Zone1	Backup the Macro Zone 1.
Zone2	Backup the Macro Zone 2.
HMI	
Settings	Backup all HMI Settings.
NCData	Backup the NC Data directory on the PC.
Sym.name	Backup the Settings of the symbolic name feature.
Language	Backup all text files

9.5.7 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.



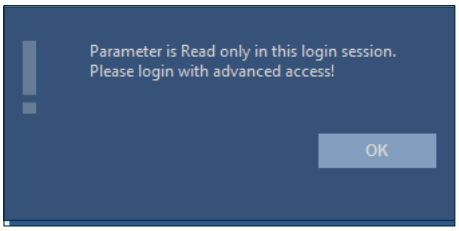
Click on a parameter to get access to its value. If the parameter is with advanced access you will not be able to continue until the advanced password is given.

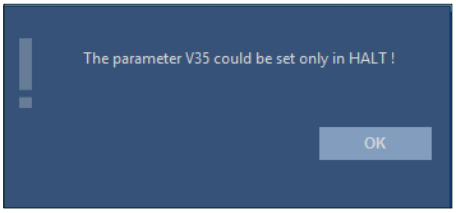
Possible actions

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.

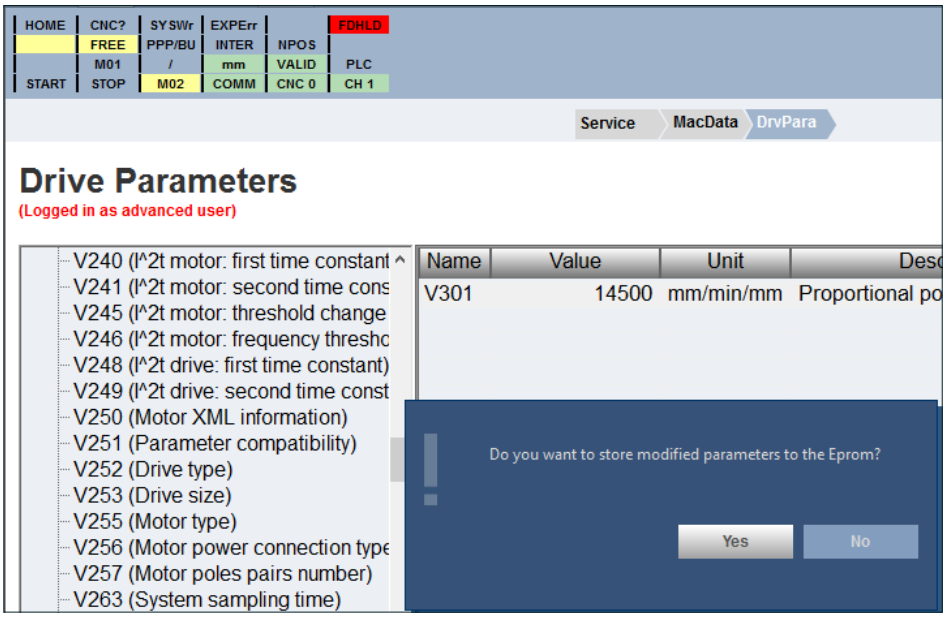
Once the drive selected the system will read and display all drive parameters.
Click on one particular parameter to have edit access.

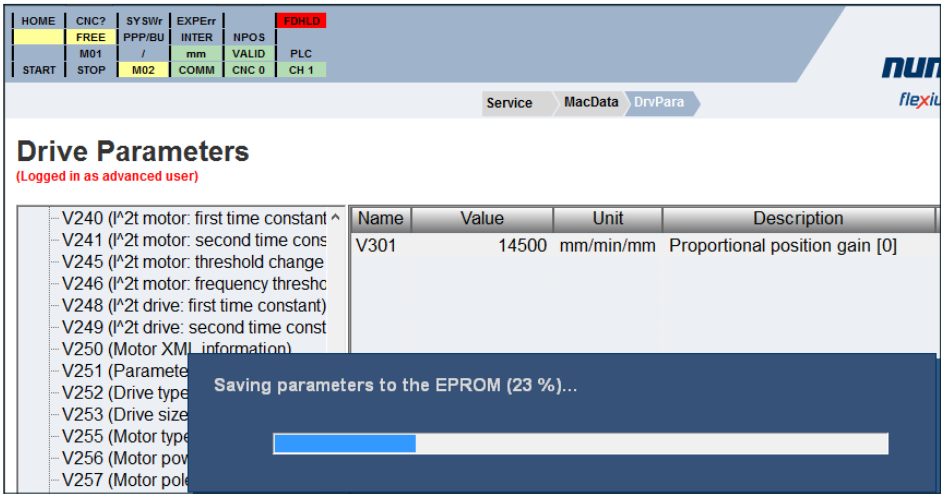
According to the parameter, it may be necessary to have advanced access or to set the drive in Halt.





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.

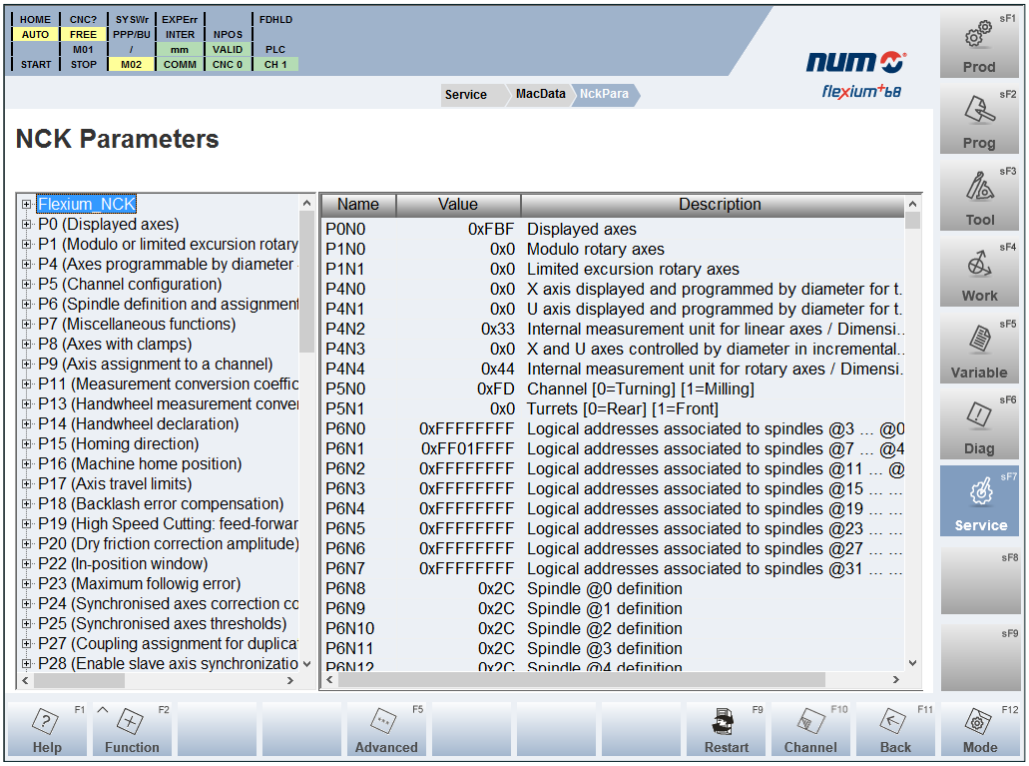




9.5.8 NCK parameters

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

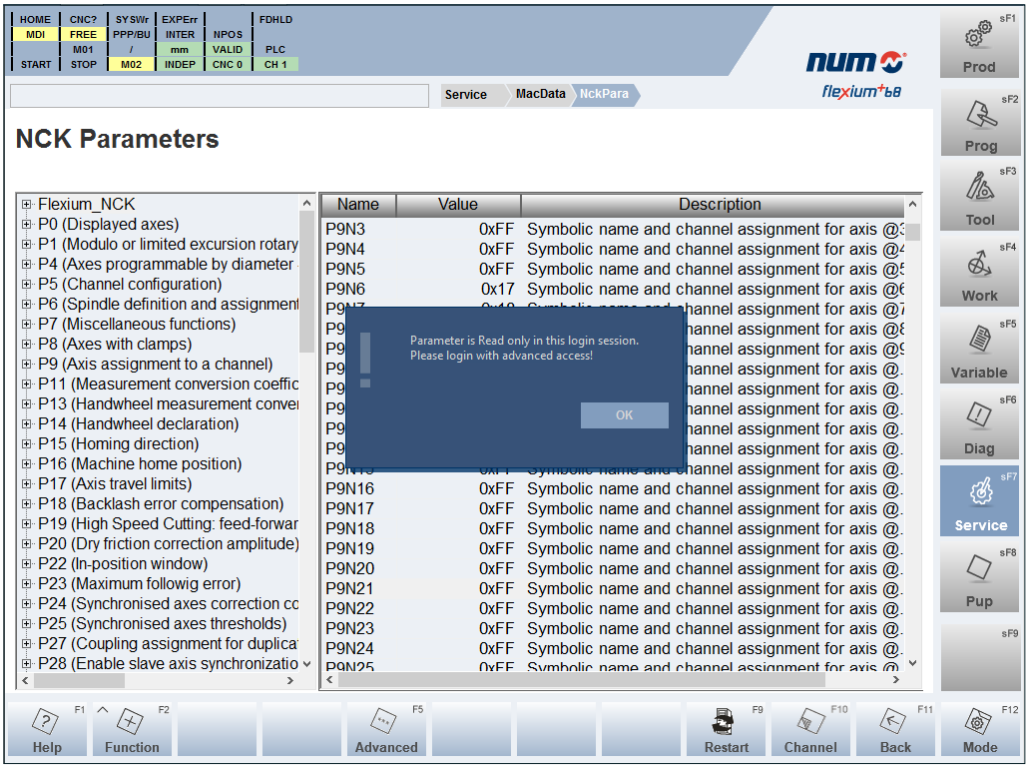
To access this page, press F3 in the *'Machine Data'* page of the *'Service'* context.



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.

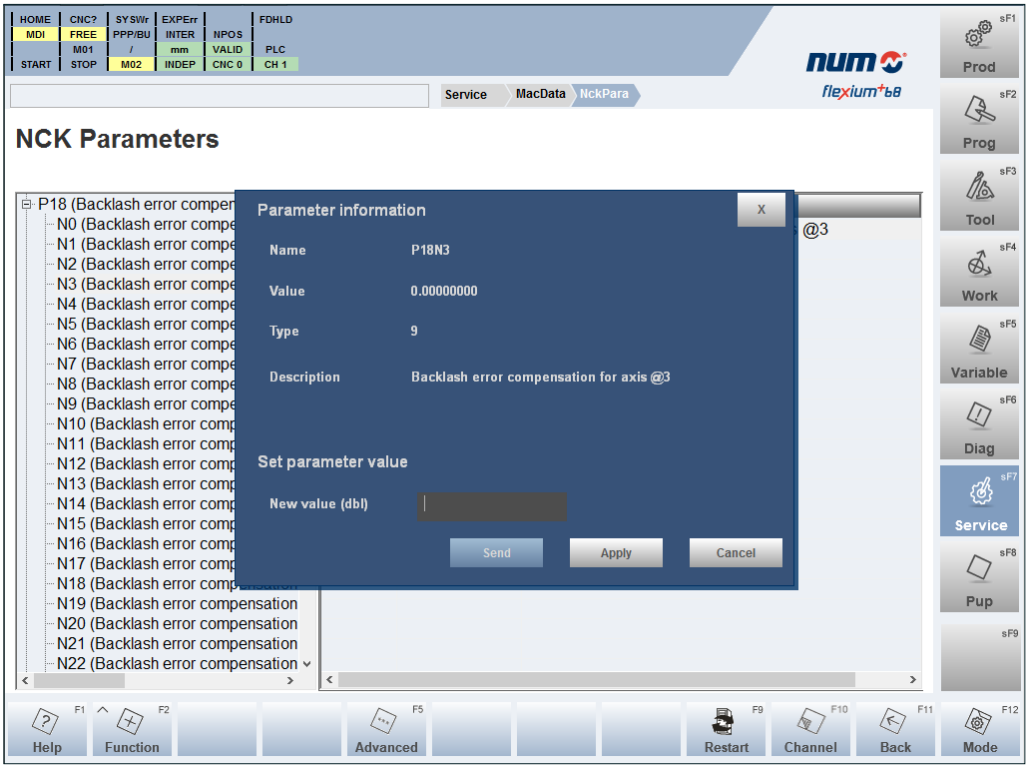
The default password is *EXTOEM*.

It is strongly suggested to modify this password before setting the machine into 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.



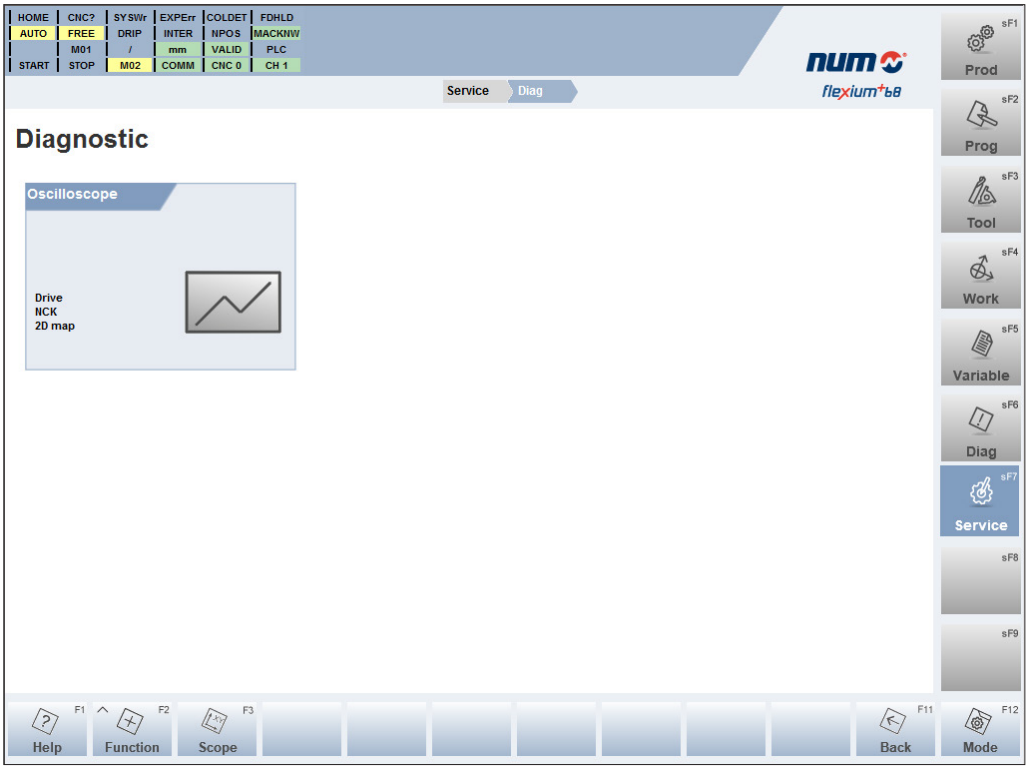
9.6 Diagnostic

For diagnostic purposes the oscilloscope function is used.
The operator can select between different oscilloscope views in the course of time like:

- Drive scope
- NCK scope
- 2D Map with path velocity profile.

The operator selects the sampling rate as well as axes to be inspected. Beside these recording functions traces can be saved, loaded and compared.

To access this page, press F7 in the main page of the *'Service'* context.



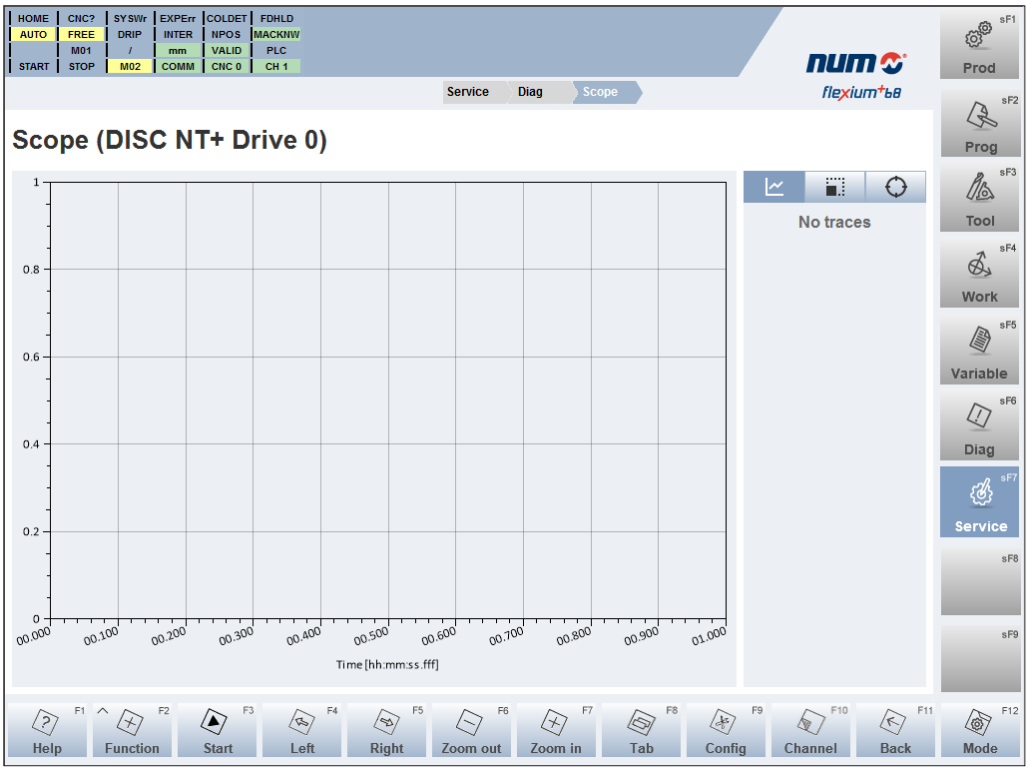
Possible actions

Press F3 to access the oscilloscope.
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter↵.

9.6.1 Oscilloscope

The page of the last selected context is shown. Default context when entering this page for the first time is *“Drive scope”*.

To access this page, press F3 in the *‘Diagnostic’* page of the *‘Service’* context.

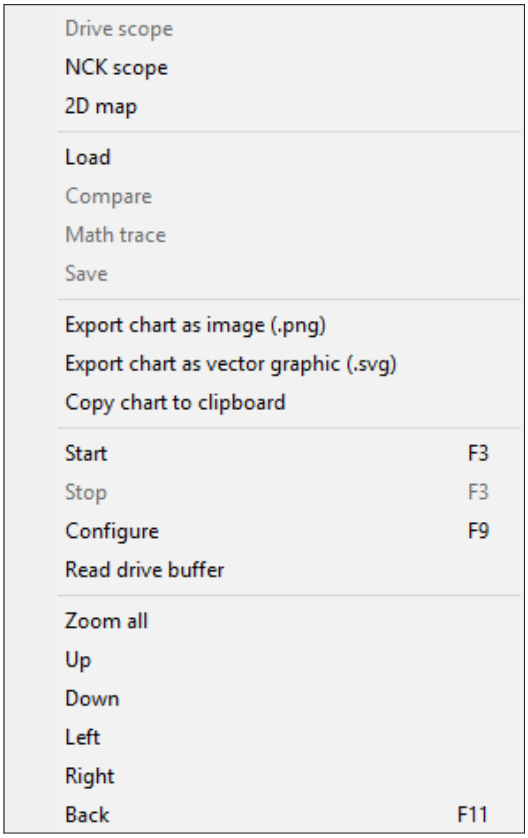


Possible actions

Press F9 to access the “*Configure*” page.
Press F8 to change the shown tab on the right side.
Press F7 to zoom in. Note that this function could be disabled dependent on the auto zoom settings.
Press F6 to zoom out. Note that the maximum view range is 60 seconds. Note that this function could be disabled dependent on the auto zoom settings.
Press F5 to move the plot to the right. In context *‘2D map’* the feed rate chart is moved. Note that this function could be disabled dependent on the auto zoom settings.
Press F4 to move the plot to the left. In context *‘2D map’* the feed rate chart is moved. Note that this function could be disabled dependent on the auto zoom settings.
Press F3 to start or stop the oscilloscope recording.
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter↵.

9.6.1.1 Oscilloscope functions

To access this menu, press F2 in the *'Oscilloscope'* page.



Possible actions

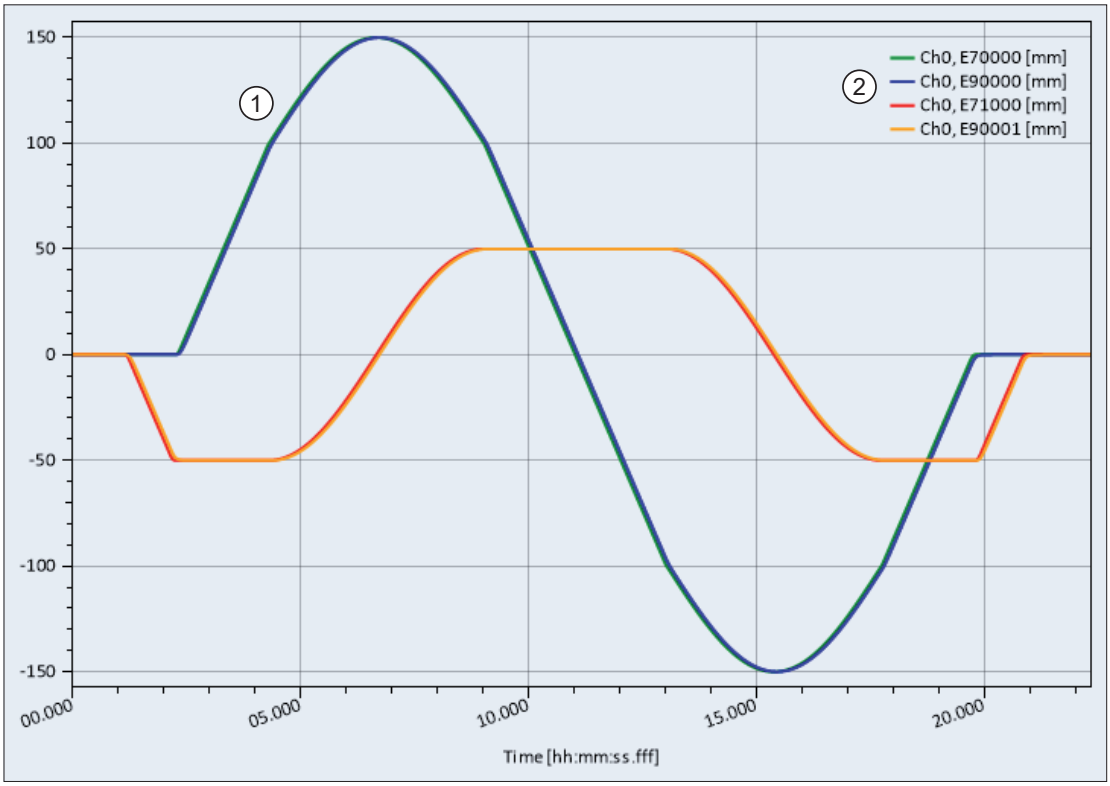
Drive scope	Change context to *'Drive scope'*.
NCK scope	Change context to *'NCK scope'*.
2d map	Change context to *'2D map'*.
Load	Open a dialog to load a saved recording.
Compare	Open a dialog to load an additional saved recording to the chart. This function is only available if there is already a recording shown.
Math trace	Open a dialog to add a trace, which is mathematically dependent from an already shown one. The available math functions are mirror, derivation and integration. This function is only available if there is already a recording shown. In the context *'2D map'* this function is not available.
Save	Open a dialog to save the recording.
Export chart as image (.png)	Open a dialog to export the chart as a portable network graphic.
Export chart as vector graphic (.svg)	Open a dialog to export the chart as a scalable vector graphic.
Copy chart to clipboard	Copy the chart as an image to the clipboard. It can then be pasted with Ctrl + [V] to any suitable program.

Start	Start recording. In context *‘NCK scope’* and *‘2D map’* data is recorded and shown in the chart until ***‘Stop’*** is pressed. In context *‘Drive scope’* the data is recorded but shown only when all data is collected (the recording is stopped by itself). This function is only available when a valid configuration is applied and the recording is stopped.
Stop	Stop recording. This function is only available when the recording is running.
Configure	Access the *‘Configure’* page to change and apply the configuration.
Read drive buffer	Read the data which is stored in the selected drive. This fails if there is no data stored in the drive yet. This function is only visible when in *‘Drive scope’* context.
Zoom all	Zoom the plot in or out to show all data. Note that the maximum view range is 60 seconds.
Up	Move the plot up. Note that this function could be disabled dependent on the auto zoom settings.
Down	Move the plot down. Note that this function could be disabled dependent on the auto zoom settings.
Left	Move the plot to the left. Note that this function could be disabled dependent on the auto zoom settings.
Right	Move the plot to the right. Note that this function could be disabled dependent on the auto zoom settings.

9.6.1.2 Chart

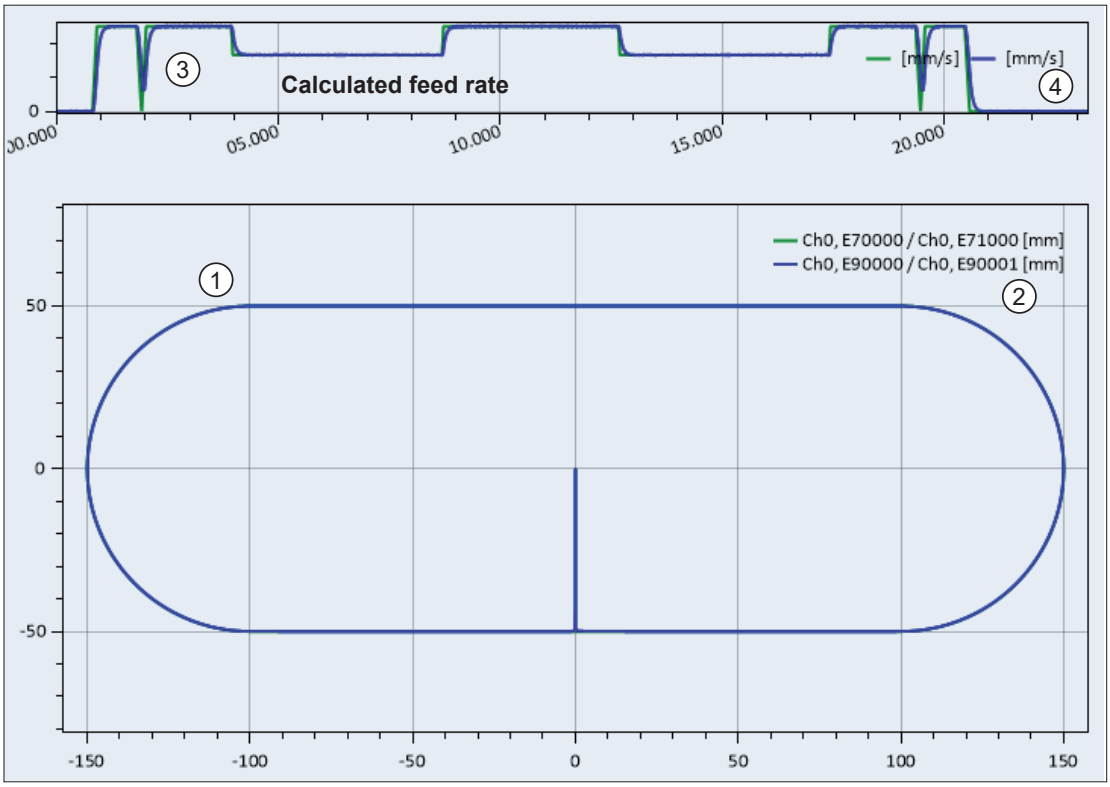
On the left side, a chart shows the recorded data. It is dependent on the selected context.

Drive scope and NCK scope context



1	Visualized data
2	Legend

2D map context



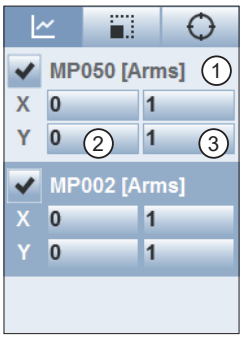
1	Visualized data
2	Legend
3	Calculated feed rate
4	Legend (feed rate)

9.6.1.3 Tabs

On the right side, there is additional information shown. It is divided into three tabs.
To change the tab, press F8 to toggle through them or click on a particular tab.

Traces

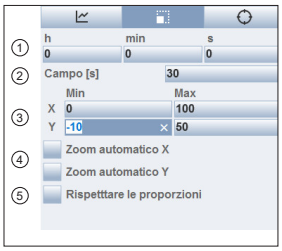
This tab shows information about the traces.



1	Show or hide a trace
2	Offset (manually move trace in according axis)
3	Scale (manually scale trace in according axis)

Zoom

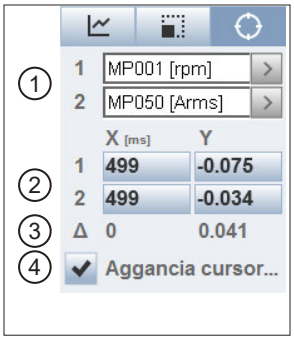
This tab allows to precisely move and zoom the chart.



1	Fill in the hours, minutes and seconds and then press Enter in any of these input fields to move the middle of the chart to this position
2	Fill in the range in seconds and then apply with Enter to change the visible range. Note that the maximum view range is 60 seconds.
3	Fill in the minimum and maximum and then press Enter in any of these input fields to set the visible range of the according axis. Note that these inputs are only available if associated “*Auto zoom*” is off.
4	Turn on or off auto zoom. Note that these inputs are not visible in certain contexts.
5	Fix the aspect ratio (1:1) between the abscissa and the ordinate. Note that this input is only available in *‘2D map’* context.

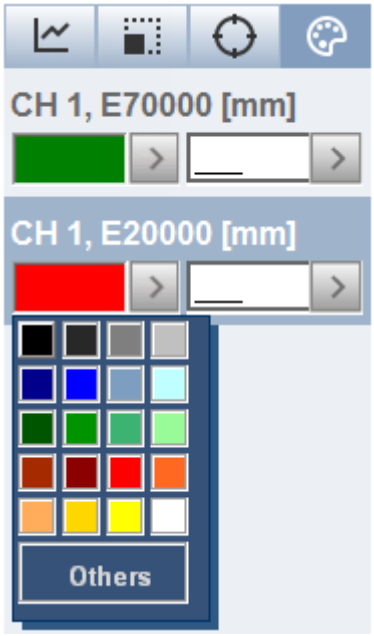
Cursor

This tab allows to add, move and remove cursors. Two cursors are available.



1	Selection of the trace. Select *“No cursor”* to remove the cursor.
2	Cursor values. To move the cursor there are three options: - Fill in the value and apply with Enter - Select input field and press ← or → Hold. - Shift for fast mode or Ctrl for slow mode. - Use mouse to click and drag the cursor.
3	Difference (delta) between the cursor values (cursor 2 – cursor 1) Note that these values are only shown if both cursors are active.
4	Change cursor mode. If selected, the cursor snaps only to recorded data points (no interpolation). Note that this input is not available in *‘2D map’* context (snap is always on).

Starting with version 4.2.0.0 it is possible to change the colors of the traces.



When in context *'2D map'* additional information about the line number and block number is shown.

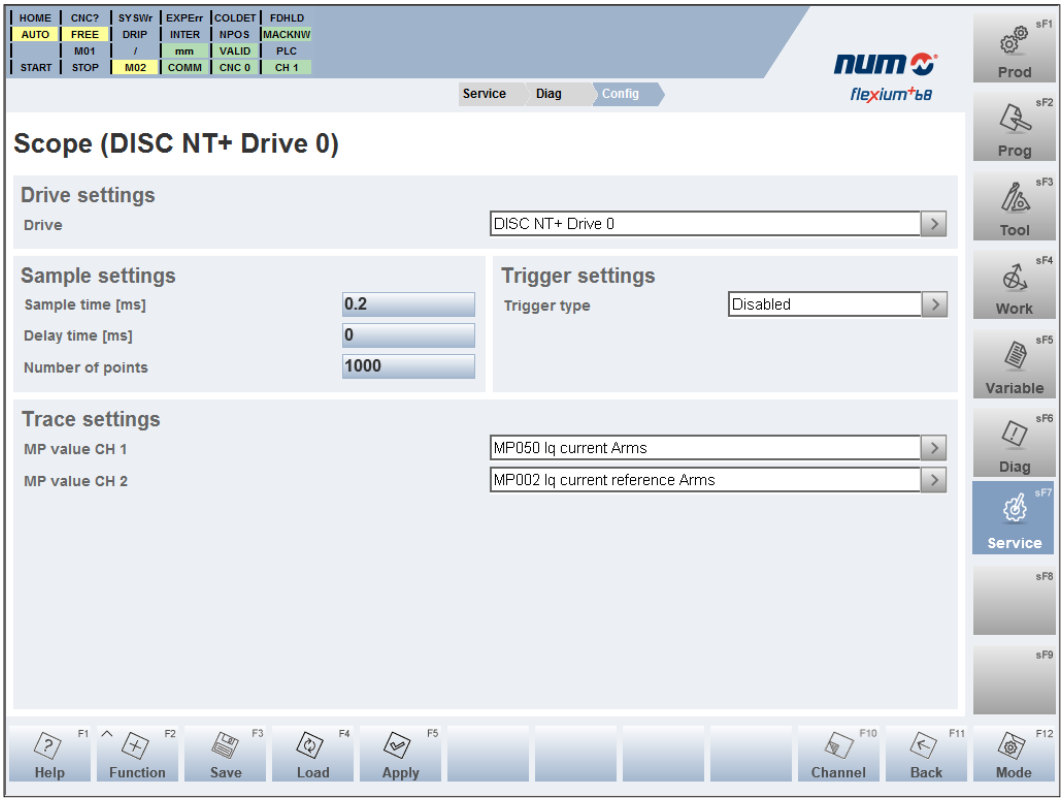
9.6.1.4 Configure page

To access this page, press F9 in the *‘Oscilloscope’* page.
The shown page is dependent on the selected context.

Possible actions

Press F5 to apply the configuration. Without applying, the configuration is not transferred to the machine!
Press F4 to open a dialog to load a configuration.
Press F3 to open a dialog to save the configuration.
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter↵.

Drive scope context configuration



Explanation of the configuration options

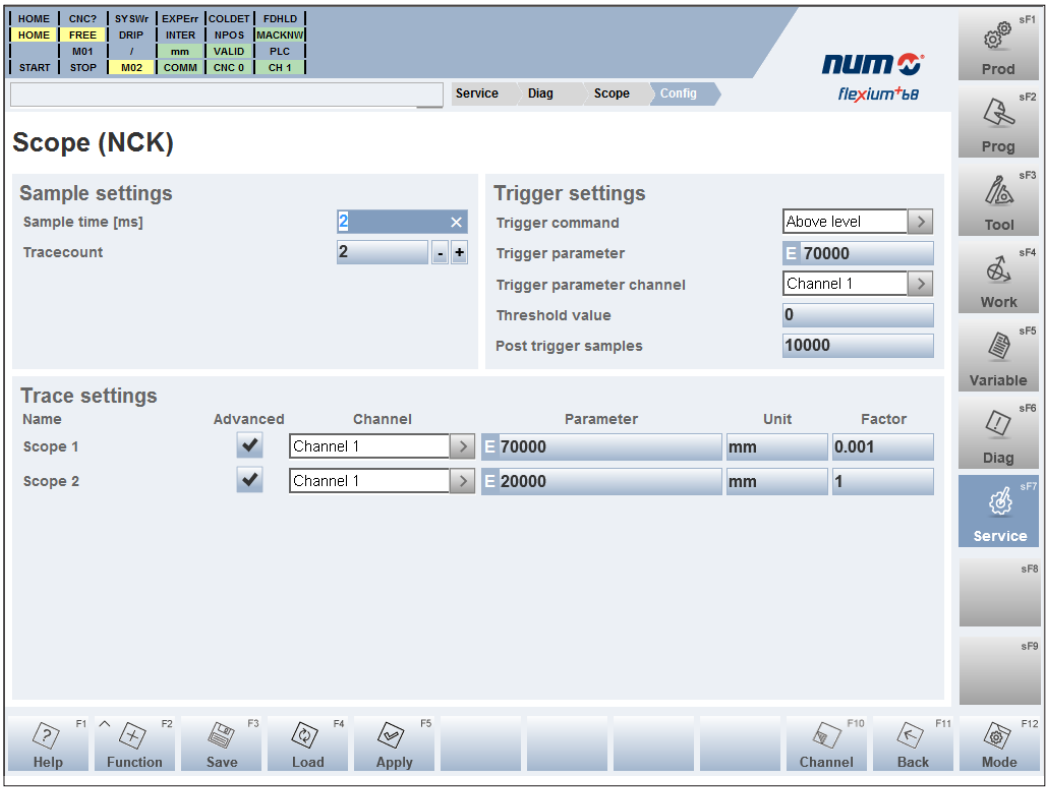
Drive settings	
Drive	Select the drive that will be used. When the selection is changed, the configuration below is read from the drive.
Sample settings	
Sample time	Rate in milliseconds at which the data is recorded (0.1 - 6553.5). Minimum sample time is current P50 sample time value.
Delay time	How many milliseconds the recording is delayed after recording start (0 - 6553.5). Note that this value is only visible if no trigger is selected.
Number of points	The number of recorded points. (2 - 1024 / 2 - 2048) Note that the maximum is dependent on the selected measurement points.
Trigger settings	

Trigger type	Select the trigger type from following list: - Disabled: No trigger, the recording starts immediately and stops when all samples have been recorded. - Alarm: Recording stops when a drive error occurs and all post trigger samples have been recorded. - Below level: Recording stops when the trigger value is becoming smaller than the threshold value and all post trigger samples have been recorded. - Above level: Recording stops when the trigger value is becoming bigger than the threshold value and all post trigger samples have been recorded. If the number of points is bigger than the amount of post trigger samples, then there is also data before the trigger (trigger is at time 0).
Trigger MP value	The measurement point that is used as trigger. Note that this value is only visible if trigger type is *"Below level"* or *"Above level"*.
Threshold value	The value that is used as threshold for the trigger value. Note that this value is only visible if trigger type is *"Below level"* or *"Above level"*.
Post trigger samples	The number of recorded points after the trigger. (0 - (number of points - 1)) Note that this value is only visible if a trigger type is selected.
Trace settings	
MP value CH 1	The first drive measurement point to record.
MP value CH 2	The second drive measurement point to record.

Possible actions

Press F2 and choose *"Read drive settings"* to manually read the configuration from the selected drive.

NCK scope context configuration



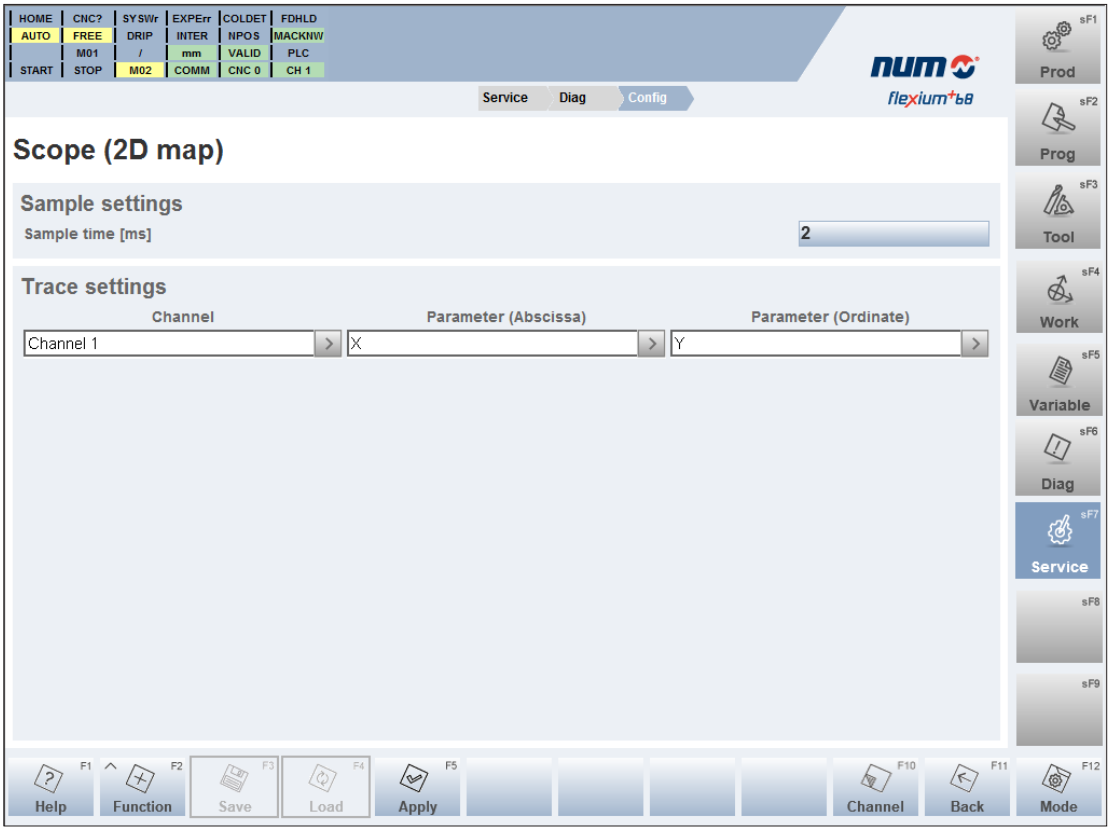
Explanation of the configuration options

Sample settings	
Sample time	Rate in milliseconds at which the data is recorded (1 - 65535).
Trace count	Amount of traces to record (1 - 9).
Trigger settings	
Trigger command	Select the trigger type from following list: - Disabled: No trigger, the recording is started immediately and stopped by the user - Alarm: Recording starts immediately. It stops when a drive error occurs and all post trigger samples have been recorded. - Below level: Recording starts immediately. It stops when the trigger value is becoming smaller than the threshold value and all post trigger have been recorded. - Above level: Recording starts immediately. It stops when the trigger value is becoming bigger than the threshold value and all post trigger samples have been recorded.
Trigger parameter	The parameter that is used as trigger. Note that this value is only visible if trigger command is *“Below level”* or *“Above level”*.
Trigger parameter channel	The trigger parameters channel. Note that this value is only visible if trigger command is *“Below level”* or *“Above level”*.
Threshold value	The value that is used as threshold for the trigger value. Note that this value is only visible if trigger command is *“Below level”* or *“Above level”*.
Post trigger samples	The number of recorded points after the trigger. Note that this value is only visible if a trigger command is selected.

Trace settings	
Name	A numbered trace name.
Advanced	Turn on and off advanced configuration mode for a trace.
Channel	Choose the channel to trace. It is also possible to select the existing drives (if *'Advanced'* is unselected).
Parameter	Choose the parameter to trace. If in *'Channel'* a drive is selected, this list shows the first and the second drive measure point. If *'Advanced'* is selected, the parameter can be freely selected by typing a number in.
Unit	Choose the unit to show. Note that this input field is only available in advanced mode.
Factor	Choose the correction factor to correctly show the parameter value in the selected unit. Note that this input field is only available in advanced mode.

2D map context configuration

The operator selects the channel dependent axes. 2D scope traces automatically reference and real position of selected axes.



Explanation of the configuration options

Sample settings	
Sample time	Rate in milliseconds at which the data is recorded (1 - 65535).
Trace settings	
Channel	Choose the channel to trace.
Parameter (Abscissa)	Choose the axis to trace for the abscissa. The selection has to be different from the ordinate parameter.
Parameter (Ordinate)	Choose the axis to trace for the ordinate. The selection has to be different from the abscissa parameter.

Page deliberately empty

Summary

10 CopyExplorer and Virtual keyboard..... 229
10.1 Access to CopyExplorer 229
10.2 Save.....230
10.2.1 Save one file only 230
10.2.2 Save several successive files.....230
10.2.3 Save several non successive files.....230
10.2.4 Save all files.....230
10.3 Restore 230
10.4 Virtual Keyboard 231
10.4.1 Activating the keyboard 231
10.4.2 Closing the keyboard display.....231

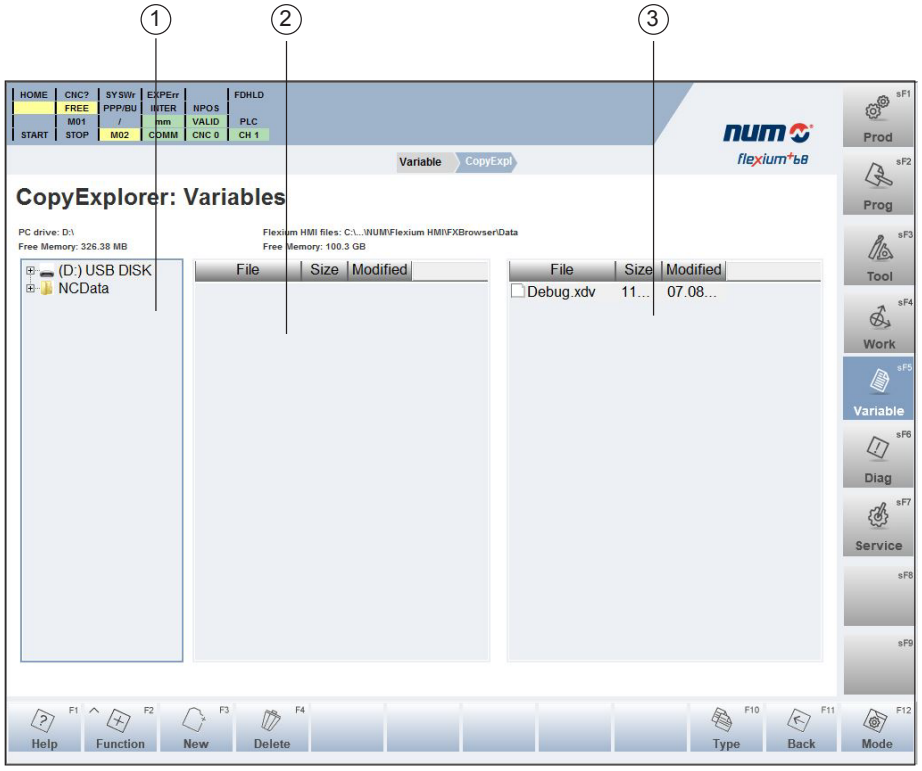
Page deliberately empty

10 CopyExplorer and Virtual keyboard

10.1 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.



1	List of drives and folders for save and restore.For the selection of drives and folders, refer to chapter *"9.4 Data and File Location Settings"* in this manual.
2	List of files present in the selected folder.
3	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↵

10.2 Save

10.2.1 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

10.2.2 Save several successive files

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

10.2.3 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

10.2.4 Save all files

Select all files	☞ F5
Save the files	☞ F6

10.3 Restore

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

10.4 Virtual Keyboard

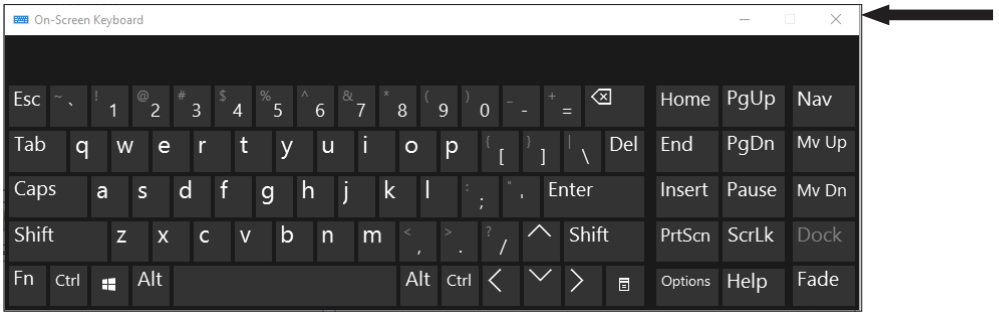
The virtual keyboard offers the same possibilities for data input as a normal keyboard.
It can be activated in every context or page allowing an input of data.

Some Operating System configuration may not offer the virtual keyboard. The application FXKeyboard covers this functionnality.

10.4.1 Activating the keyboard

Press F2 "Function".
Select *"Virtual keyboard"*.

10.4.2 Closing the keyboard display



Page deliberately empty

Summary

11 CNC Operation 235

11.1 Declaration of Axes Assignment 235

11.2 Execution of a Program 236
11.2.1 Program Test 236

11.3 Execution on the machine 237
11.3.1 Optional Stop and Block Skip Enable 237
11.3.2 Program Execution - General 237
11.3.3 Execution of a Program Step by Step 239
11.3.4 High Speed Execution of a Program 240
11.3.5 Execution of a Program in Automatic Mode 240

11.4 Operator Interventions 241
11.4.1 Unplanned Interventions 241
11.4.2 Interruption of Machining 241
11.4.3 Retracting the Axes 242
11.4.4 Manual Axis Recall 242
11.4.5 Automatic Axes Recall 243
11.4.6 Sequence Number Search 245
11.4.7 Search Mode Dialog wizard 247
11.4.7.1 Search Block number 247
11.4.7.2 Search Line number 248
11.4.7.3 Search String 249
11.4.7.4 Restart at Interrupted position 250
11.4.8 Extended possibilities in Search mode 251
11.4.9 Backtrack along Path 252
11.4.10 Return along Path 255
11.4.11 Emergency Retraction 256
11.4.12 Program Resumption after a Following Error 256

11.5 Programmed Interventions 257
11.5.1 Machining Stop Function M00 (or M01 if enabled) 257
11.5.2 Forcing an intervention by M12 257

Page deliberately empty

11 CNC Operation

11.1 Declaration of Axes Assignment

Axes assignment is used to modify the Xm Ym Zm physical reference system of the machine with respect to the Xp Yp Zp part program reference system.

The part program reference system is then adapted to the workpiece orientation on the milling machine. Axes 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:
Axes 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 axes 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...).

- It is applied downstream of programmed geometric transformations such as angular offset (ED...), programmed offset (G59), mirroring (G51...) and scaling factor (G74),
- It is not applied to the dimensions programmed with respect to the measurement origin (G52...)
- Does not affect offsets DAT1, DAT2 and DAT3.

11.2 Execution of a Program

11.2.1 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.

11.3 Execution on the machine

11.3.1 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.

11.3.2 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.

11.3.3 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.

11.3.4 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.

11.3.5 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.

11.4 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.

11.4.1 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 push buttons to activate them.

By convention, the push buttons 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).

11.4.2 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 according to the PLC program defined by the machine builder, spindle stop can be enabled. Please refer to the machine instruction manual.
The axes can be retracted (see 11.4.3).
Restart machining by pressing the *"START"* button on the machine panel.

11.4.3 Retracting the Axes

Axis retraction is related to interruption of machining (see 11.4.2).
The axes are retracted manually one by one 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 11.4.4), • Automatically (see 11.4.5).
Restart machining by pressing the *"START"* button on the machine panel.

11.4.4 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 axes 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 axes 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.

11.4.5 Automatic Axes Recall

Automatic axes 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 in the next page).

Automatic axes recall can be combined with the functions "Backtrack along path" (see 11.4.9) and "Return along path" (see 11.4.10).

Requirements
Machining interrupted on the system; axes retracted (*"INTER"* indicator flashing).
Press the *"Automatic Axes 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 Axes Recall

Axes 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 Flexium+ Parameter Manual M00041). If the number of points is equal to zero, the axes are recalled by linear interpolation.

Number of retraction points stored

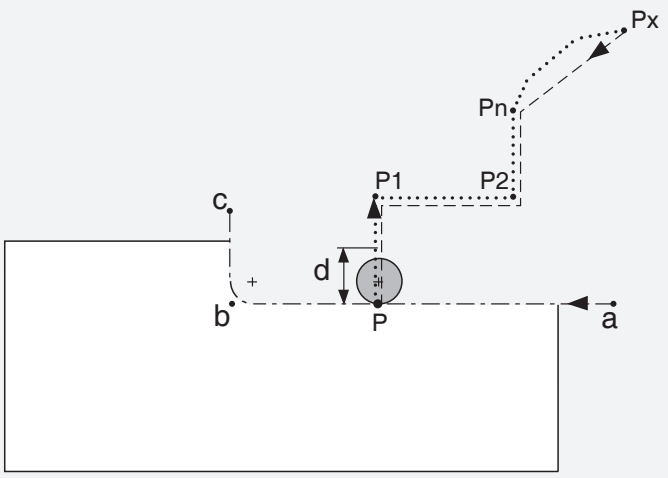
The number of retraction points that can be stored (maximum 10) is declared in word N2 of machine parameter P114 (see Flexium+ Parameter Manual M00041).

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 along each axis up to the starting point.

Speed of movement and approach distance “d”

Automatic axes 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 Flexium+ Parameter Manual M00041).

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.

11.4.6 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 wind.	
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 11.4.4).
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”*.

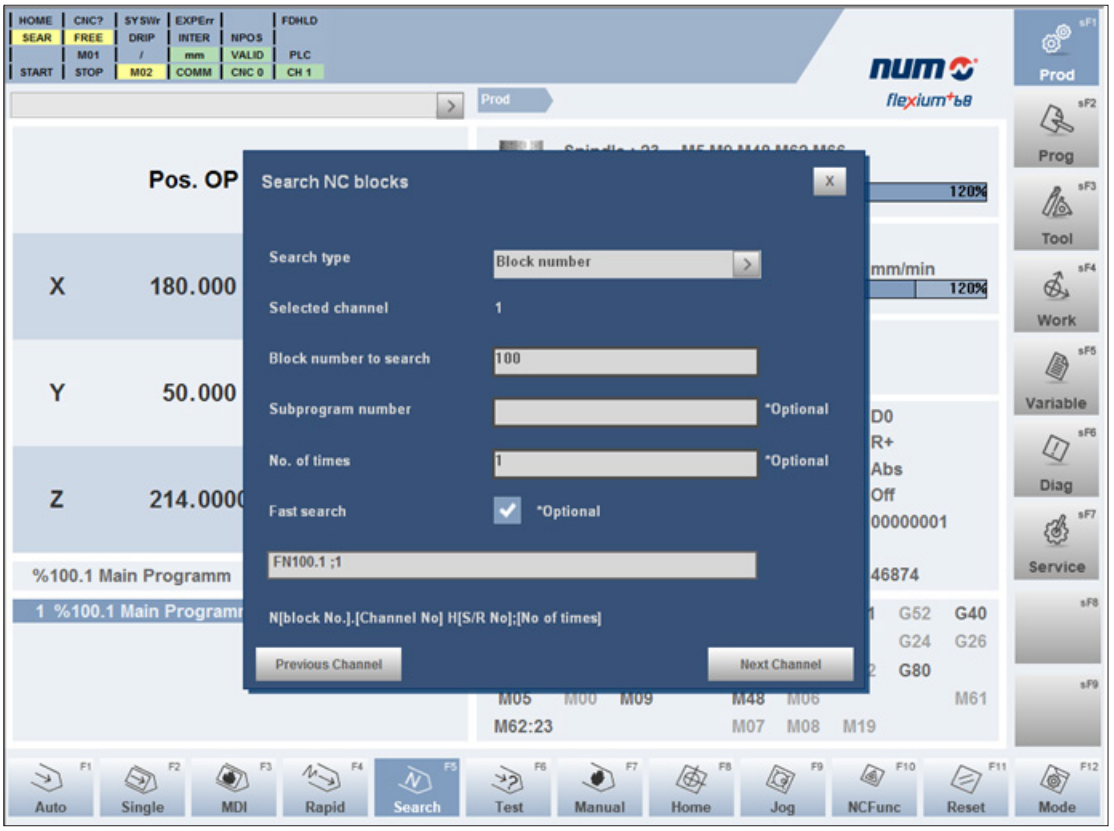
11.4.7 Search Mode Dialog wizard

Open the Wizard by pressing on the [>] or by pressing *Cursor* down.

To finish the search, it's needed to enter the *Search value* for all Channels.
In each channel it is possible to search in a different way.

11.4.7.1 Search Block number

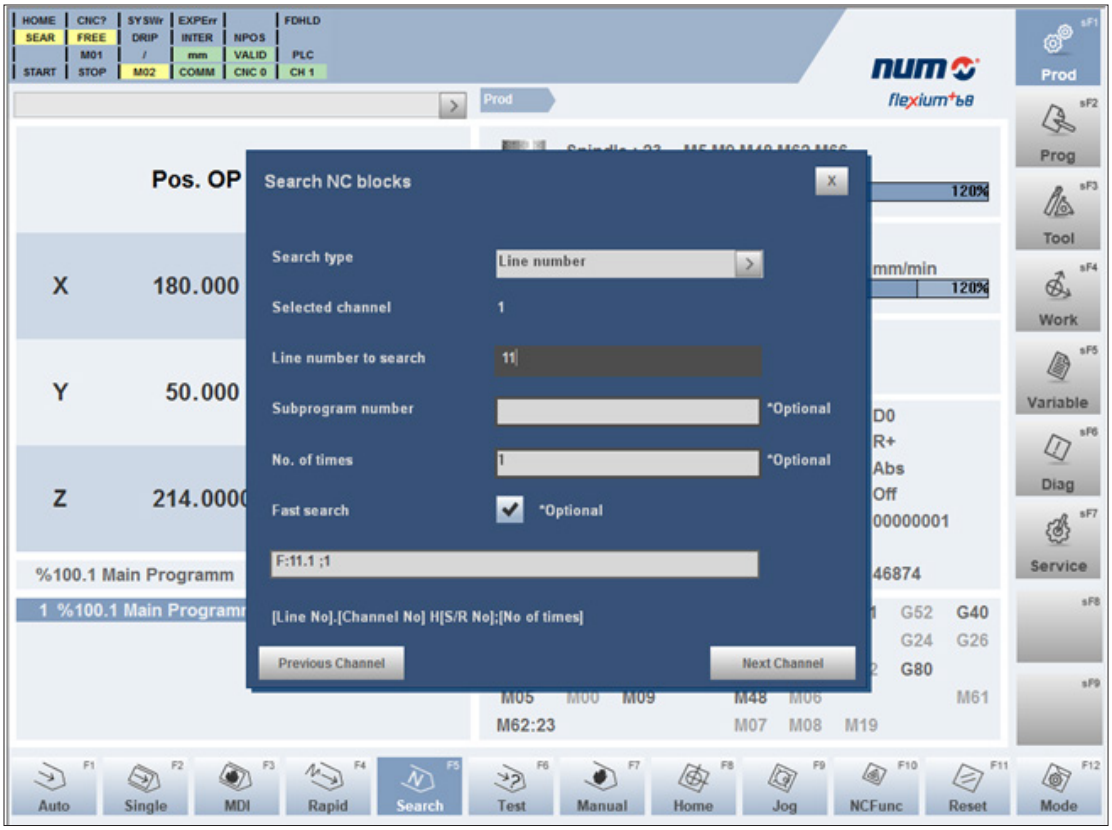
Search a *Block number* in the active CNC program.



Block number to search	Block number in the CNC program N100.
Subprogram number	Subprogram to search. This input is optional.
No. of times	Number of times where the Blocknumber is found (for loops). This input is optional.
Fast search	Fast search mode. Fast Search Mode limits the computation and exchange of data during the first phase of the search. This results in an about 10 folds improvement of the Search procedure. They are however some drawbacks to this speed improvement: • Back on track on the blocks previous to the search point will not be available • Coded M functions are not sent to the PLC during the first phase • Synchronization between channels is not handled as well during this first phase Fast or regular Search can be used during different steps of the execution regardless of which one was used previously.

11.4.7.2 Search Line number

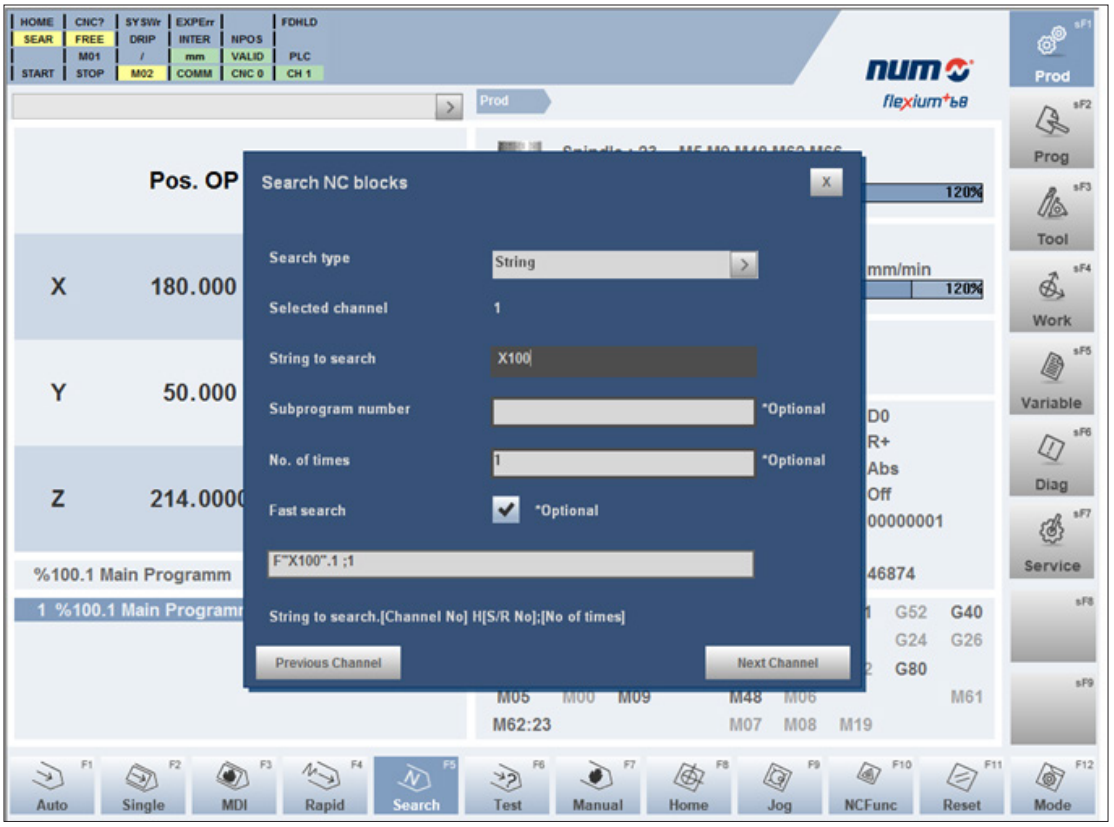
Search a *Line number* in the CNC program.



Line number to search	Source line number in the CNC Program.
Subprogram number	Subprogram to search. This input is optional.
No. of times	Number of times where the block number is found (for loops). This input is optional.
Fast search	Fast search mode. Please refer to chapter 11.4.7.1.

11.4.7.3 Search String

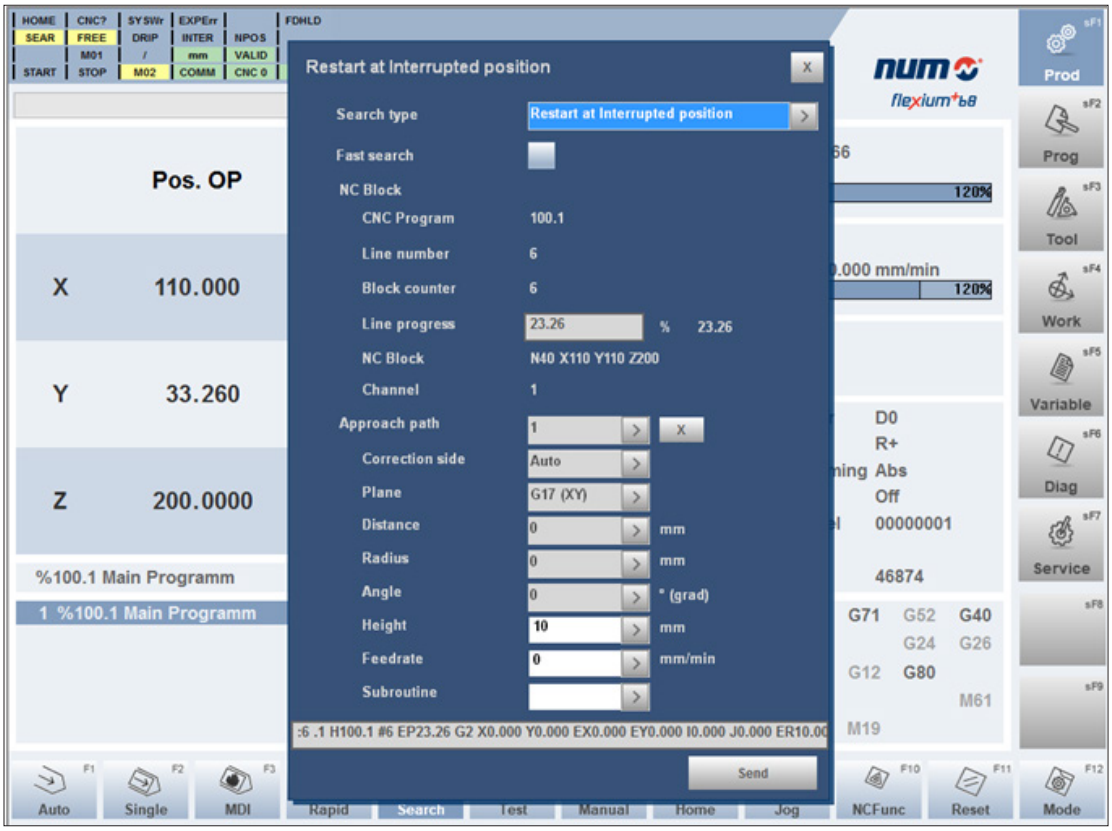
Search a *String* in the CNC program.



String to search	Search string inside CNC program.
Subprogram number	Subprogram to search. This input is optional.
No. of times	Number of times where the block number is found (for loops). This input is optional.
Fast search	Fast search mode. Please refer to chapter 11.4.7.1.

11.4.7.4 Restart at Interrupted position

This specific search mode seeks for the position on the trajectory where the CNC program is interrupted.
It is mandatory to set the PLC Variable *WCNC.General.StoreEnable*.



Fast search	Fast search mode. Please refer to chapter 11.4.7.1.
Line progress	Percentage value in current movement/line to resume.
Approach path	Vertical approach to the re-engage position of the trajectory. The operator can select the current circle or compensation plane (G17/G18/G19)
Feedrate	Feedrate for the approach path.
Height	Height of the approach path small a / 2 times.
Subroutine	Subprogram which has to be executed before the approach contour is started.

11.4.8 Extended possibilities in Search mode

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).

11.4.9 Backtrack along Path

After an interruption in machining (see 11.4.2), 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 11.4.9).

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 11.4.9) 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 dry run (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 dry run 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.

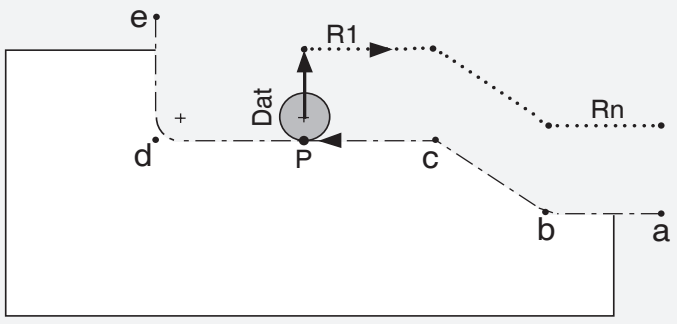
Tool/Workpiece offset

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 tool head (for further information, see Sec. 11.4.3 Retracting the Axes), press the *"AXRCL"* button on the machine panel (the *"AXRCL"* indicator is lit in the status window) and retract the tool head using the axes 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 Flexium+ Parameter Manual M00041).

Restriction on use of backtrack along path

It should be noted that:

- Only the functions related to axes 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 channel.

11.4.10 Return along Path

The return along path function is indissociable from the backtrack along path function (see 11.4.9). After a backtrack along path, the return along path returns the tool head 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 placed 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 11.4.9).

11.4.11 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 Flexium+ Programming manual M00040).
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 axes 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 11.4.3).
Restart the machining procedure by pressing the *"START"* button.

11.4.12 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.

11.5 Programmed Interventions

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

11.5.1 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.

11.5.2 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 axes 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.

After an M12, the program will resume from where the axes are.
Use caution to prevent any collision.

Page deliberately empty

Summary

12 Additional functions 261

12.1 Introduction 261

12.2 Teach-in 262
12.2.1 Access to the Teach-in function 262
12.2.2 Teach-in editor window for part programmes 264
12.2.3 Explanation of the fields and functions 265
12.2.4 Teach-in procedure 266
12.2.5 Teach-in directly in NC Memory 267

12.3 Extended tool table 271
12.3.1 Displays 271
12.3.2 Channel display 272
12.3.3 Channel assignment and comments 273
12.3.4 Assigning a tool to a channel 273
12.3.5 Assigning a name and a comment 273

12.4 Symbolic names 274
12.4.1 Set up of the symbolic names 274
12.4.2 Axes configuration 275
12.4.3 Entering axes names 275
12.4.4 Giving a name to an axis address assignment 276
12.4.5 Deleting the name given to an axis address assignment 276
12.4.6 Display of axis name of address 276
12.4.7 Axes channel configuration 277
12.4.8 Display of symbolic names 280
12.4.9 Display in the status window 281
12.4.10 Channel selection 282
12.4.11 Particularities of the channel selection 283
12.4.12 Particularities with the function Multi-CNC 284
12.4.13 Specific features of the extended tool table 284

12.5 Extensions for NUMROTOplus® 285
12.5.1 Particularities in the context bar 285
12.5.2 Setting in the context “Service” 286
12.5.3 Wheel compensation 288

12.6 PLC visualization 289
12.6.1 Integrated PLC visualization 289
12.6.2 Configuration – Softkey sF8 / sF9 293
12.6.3 Sample of PLC Visu Pages 294

12.7 FXKeyboard 295
12.7.1 Settings 297

12.8 Flexium CAM 301
12.8.1 Settings 302
12.8.2 Install a configuration 303
12.8.3 Operation 304
12.8.3.1 Block number 306

12.9 NUMcoss 308
12.9.1 Access to the NUMcoss Configuration page 309
12.9.2 NUMcoss Configuration 310
12.9.2.1 Fine Finishing 310
12.9.2.2 Theoretical Background and Smoothing 311
12.9.2.3 Explanation of all NUMcoss Settings 312

12.9.3 NUMcoss in the Context Programming 315
12.9.4 Programmable NUMcoss parameter 317
12.10 Tool management 318
12.10.1 Tool view 319
12.10.1.1 Search a tool 320
12.10.1.2 Load a tool 321
12.10.1.3 Unload a tool 321
12.10.1.4 Create a new tool 322
12.10.1.5 Function list menu 323
12.10.2 Magazine View 324
12.10.2.1 Function list menu 326
12.10.2.2 Buffer View 327
12.10.2.3 Spindle View 327
12.10.3 Tool data View 328
12.10.3.1 Function list menu 330

12 Additional functions

12.1 Introduction

This chapter describes the various additional functions which can be integrated in the Flexium+ HMI. These functions are activated by their respective licences.

At the time, the following additional functions are available:

Designation	Description
Teach-in	The editor in the context “Programming” (sF2) can be enhanced by teach-in functions. This offers the possibility to transfer actual axes positions into the part programme.
Extended tool table	This function is an enhancement of the tool table in the context “Tools” (sF3). It is possible to give a name and a comment to any tool and to assign it to an axes channel. These information are saved in a file together with the tool dimensions.
Symbolic names	Allows to assign names to the axes and axes channels. These are displayed instead of usual address numbers, for example on the axes position page or in the channel selection window.
Extensions for NUMROTOplus	These extensions adapt the HMI for tool grinding machines using NUMROTOplus®.
Integrated PLC visualization	Display of PLC visualization elements in the contexts sF1, sF8 or sF9.
Flexium CAM	Interactive programming system for different technologies.
NUMCoss	Complement of High Speed Cutting dedicated in particular to smoothing trajectories.
Tool Management	PLC function to manage a tool database.

12.2 Teach-in

This function offers the possibility to transfer and record actual axes positions into the part programme.

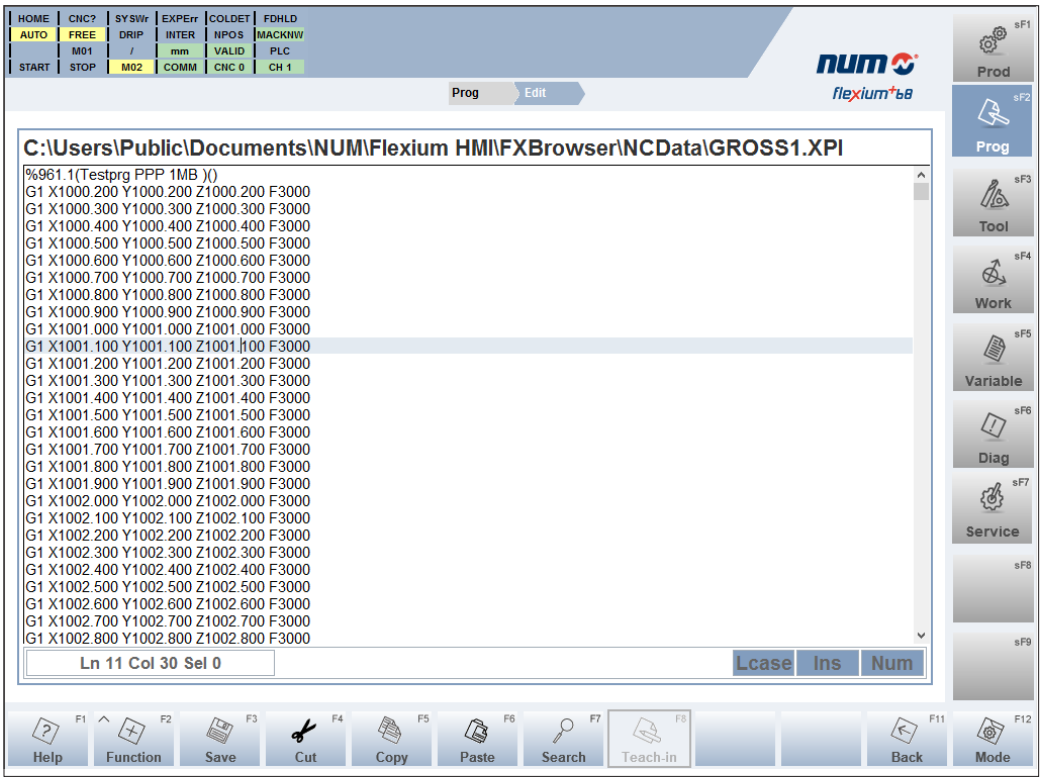
In addition to axes positions, CNC functions like G, M, F etc. may be entered in the programme. Teach-in is possible in part programmes (*.xpi) as well as in usinage files (*.usi).

To activate this function, the CNC must be in “MANUAL” mode.

12.2.1 Access to the Teach-in function

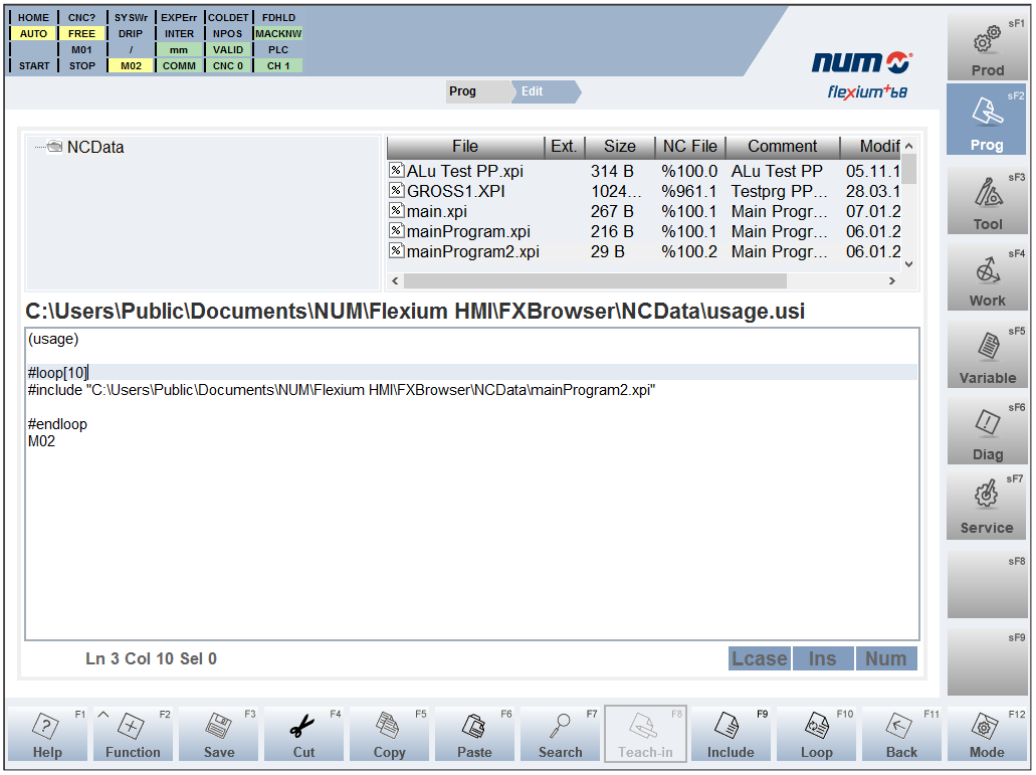
Select the context “Programming	☞ Shift + F2
Select the right hand window of NC Memory	☞ Tab...
Select the desired file	☞ ↑ or ↓
To move the cursor one window at a time	☞ PgUp or PgDn
To move directly to the top or the bottom of the list	☞ Home or End
Press	☞ F5

This opens the usual editor for part programmes:



The access to the Teach-in can be forbidden by the Settings Lock offline Teach In in the Service -> General Settings -> Security page.

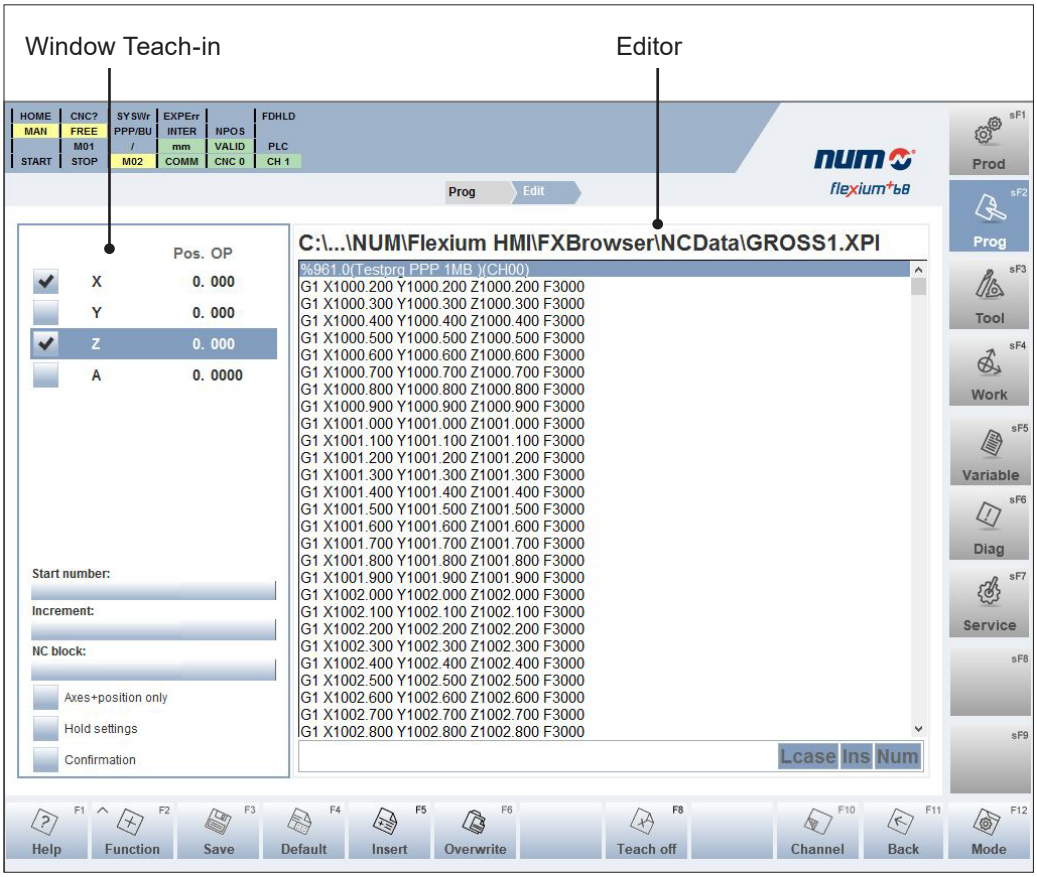
or for usinage files:



To activate the teach-in function	☞ F8

12.2.2 Teach-in editor window for part programmes

The following description applies to the teach-in editor for part programmes (*.xpi) as well as for usinage files (*.usi).



To toggle between teach-in window and editor	☞ Tab

Possible actions

To toggle between teach-in window and editor.
Press F10 to select the axes channel (with multichannel only).
Press F8 to quit teach-in and return to the editor.
Press F6 to overwrite data in a CNC block.
Press F5 to insert a CNC block.

The function keys F6 and F5 are not active if the teach-in procedure is under PLC control.

Press F4 to activate the default settings.
The default settings are as follows • Axes selection: no axes selected • Field “Start number”: empty • Field *“Increment”*: empty • Field *“NC block”*: empty • *“Axes and positions only”*: not active • *“Hold settings”*: not active • *“Confirmation”*: not active.
Press F3 to save the file
Press F2 to access all existing functions. Select the function with ↑ or ↓ and Enter ↵

12.2.3 Explanation of the fields and functions

Axes addresses	To select the desired axes for the recording of their positions.
Start number	Number of the first block for the automatic block numbering.
Increment	Increment for the automatic block numbering.

The automatic block numbering will become active as soon as the fields *“Start number”* and “*Increment”* contain a value.
If one of these fields is empty, no automatic numbering will occur.
The maximum allowed block number is N32767.

NC block	Used to enter functions and values in the CNC block.
Axes and positions only	Is effective in overwrite mode only. Functions already present in the CNC block will not be modified. Moreover, the HMI will ask for a confirmation if the number of axes or the axes positions differ from the values already present in the block.

STOP

The function *“Axes and positions only”* will be deactivated automatically after the recording of values. As a result, no further confirmation will be asked for at the following recording! To prevent this deactivation, check the box *“Hold settings”*.

Hold settings	After every recording of values in the programme, the contents of the field *“NC block”* will be erased and the function *“Axes and positions only”* will be deactivated. Check this box to hold the settings for the following recordings.
Confirmation	If this box is checked, a confirmation will be required for every recording of values.

12.2.4 Teach-in procedure

Insert or overwrite CNC block

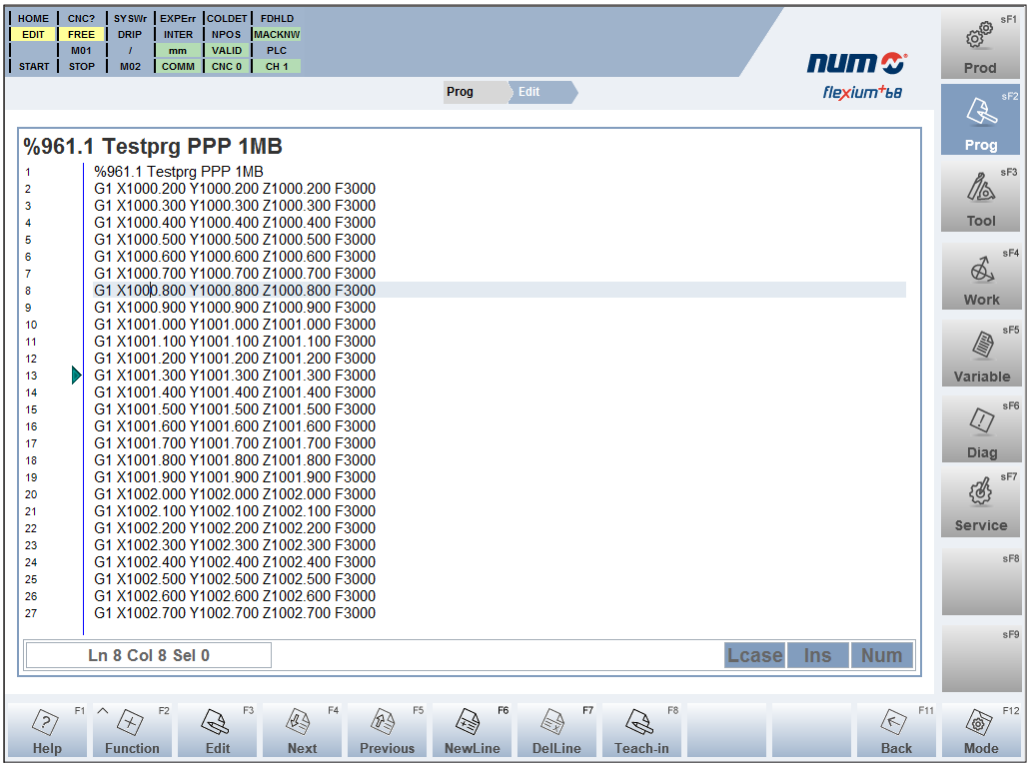
Select the right hand editor window	☞ Tab
Move the cursor to the desired line	☞ ↑ or ↓
Select the left hand teach-in window	☞ Tab
Check the desired axes	☞ ↑ or ↓ and Enter
Select and open the field “Start number” if applicable	☞ ↑ or ↓ and Enter
Enter the desired block number and validate	☞ Enter
Select and open the field “Increment” if applicable	☞ ↑ or ↓ and Enter
Enter the desired block number increment and validate	☞ Enter
Select and open the field “NC block” if applicable	☞ ↑ or ↓ and Enter
Enter the needed functions and values and validate	☞ Enter
Activate the function “Axes and positions only” if applicable	☞ ↑ or ↓ and Enter
Activate the function “Hold settings” if applicable	☞ ↑ or ↓ and Enter
Activate the function “Confirmation” if applicable	☞ ↑ or ↓ and Enter
Move the axes in the desired positions	
Insert or overwrite a CNC block	☞ F5 or F6

Switching between teach-in and editor mode is possible at any time:
☞ Tab .
To enter a tabulator in editor mode: Ctrl + Tab .

12.2.5 Teach-in directly in NC Memory

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.



The access to the Teach-in can be forbidden by the Settings Lock offline Teach-In in the Service -> General Settings -> Security page.

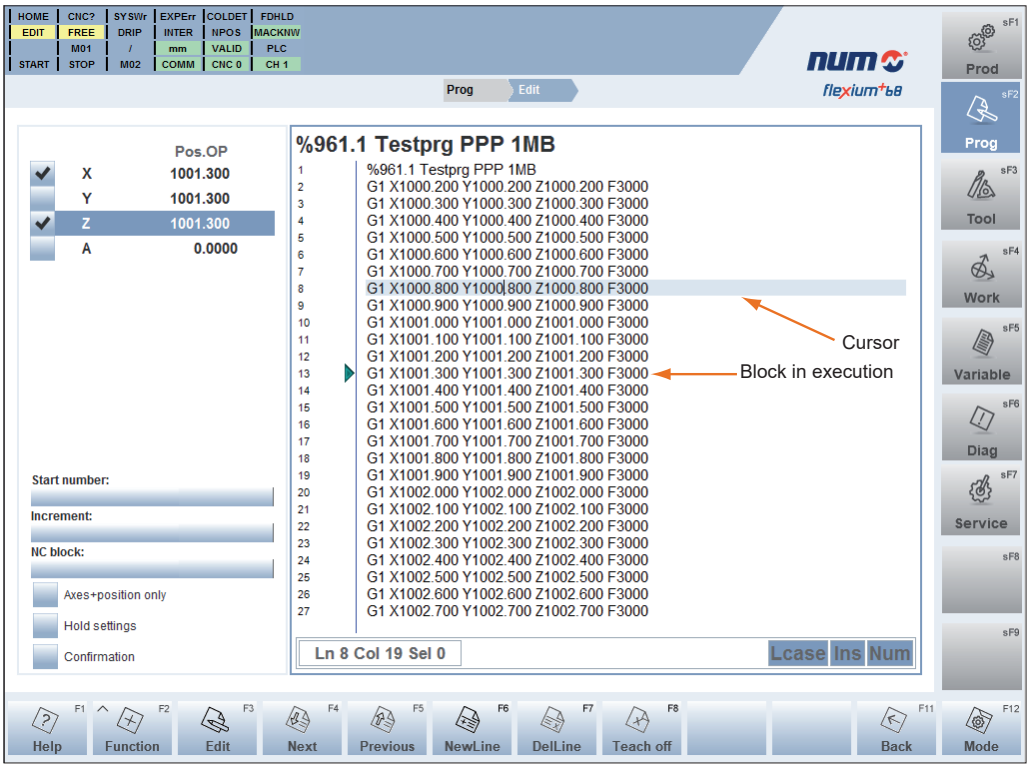
Procedure:

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)

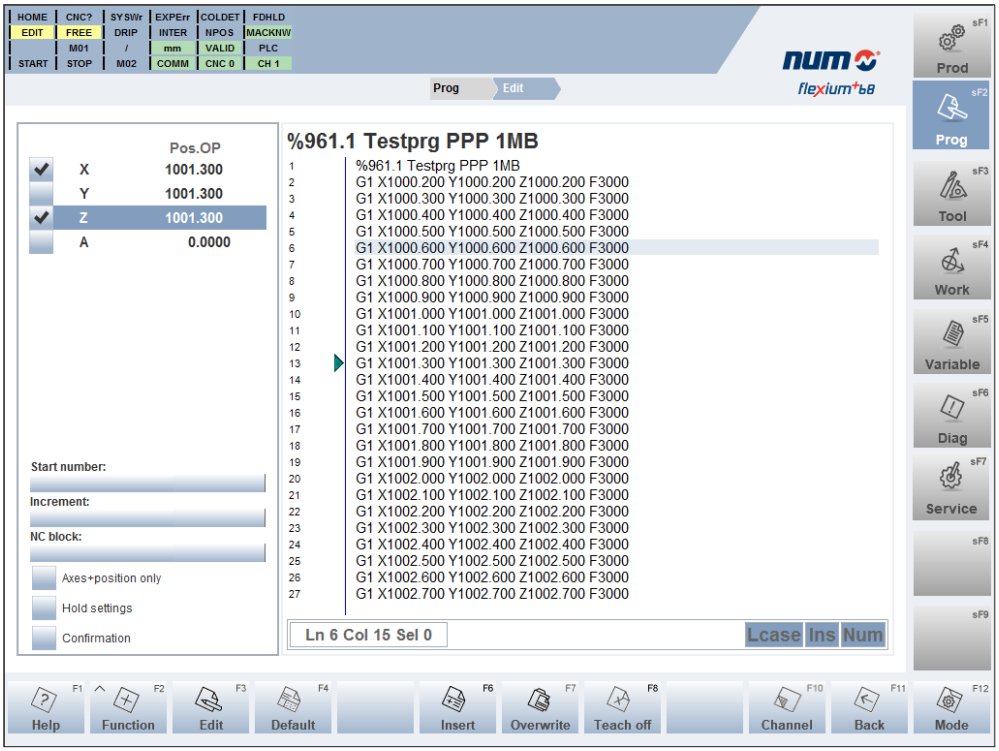


Possible actions

Press F11 to leave the editor.
Press F8 to access the Teach-in 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 to access all existing functions. Select the function with ☞↑ or ↓ and Enter↵

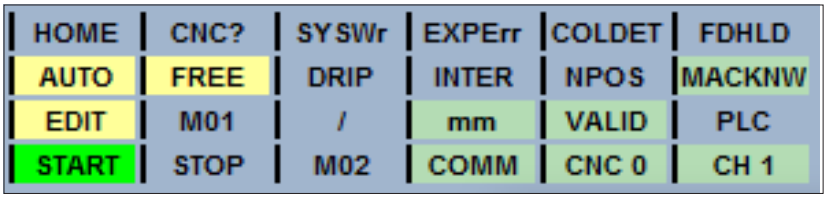
To enable an automatic scrolling between pages during program execution, the checkbox *"CNC editor: autoscroll"* in the context *"sF7 Service, Management"* must be activated.

To access the teach-in editor	☞ F8



It is now possible to enter the 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. The Teach-in functions are the same as in the normal Teach-in editor.

This is a line editor. Each modification on a line should be validated with the key Enter which does not create a new line, contrary to Page editors. Creating a new line is performed by the function key F6.

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

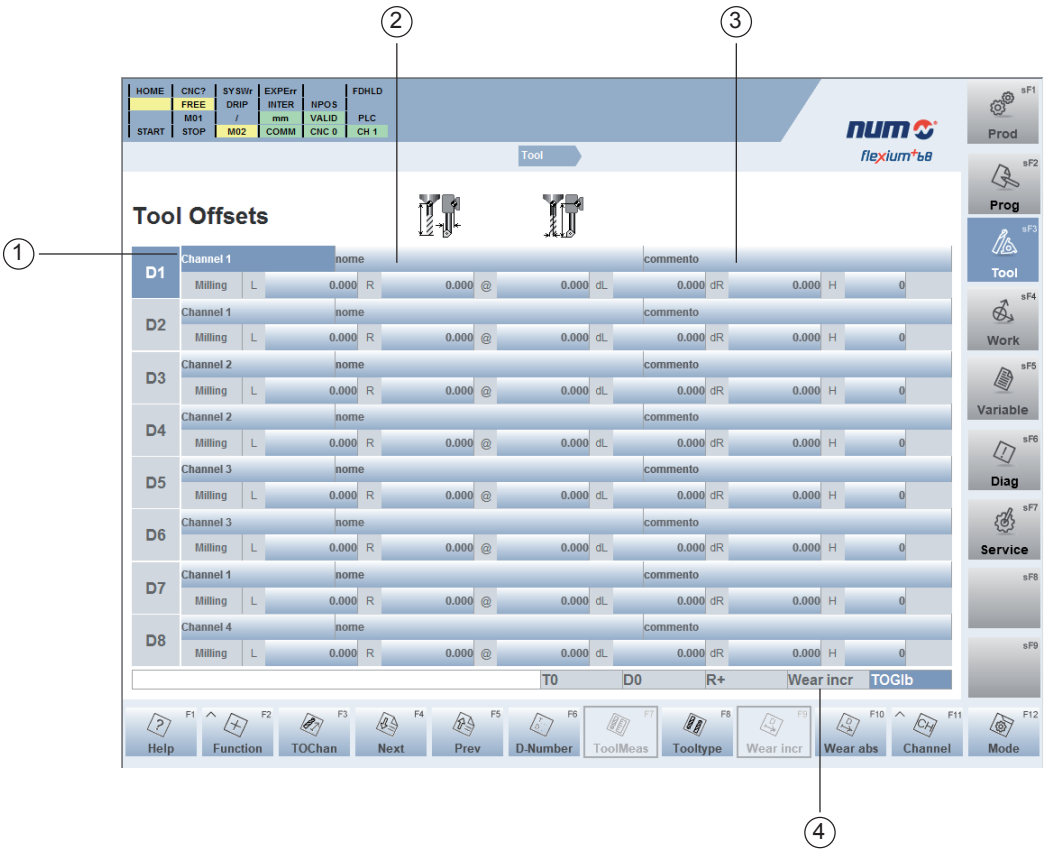
12.3 Extended tool table

This function allows an extension of the tool descriptions.
Each tool can be assigned to an axes channel. If the function *"Symbolic names"* is active, this assignment will take place using the name *"Channel long symbolic"*.

Besides, a name and a comment may be assigned to each tool. Up to 23 letters and numbers may be used in the name and comment fields.

12.3.1 Displays

There are two kinds of tool table display.



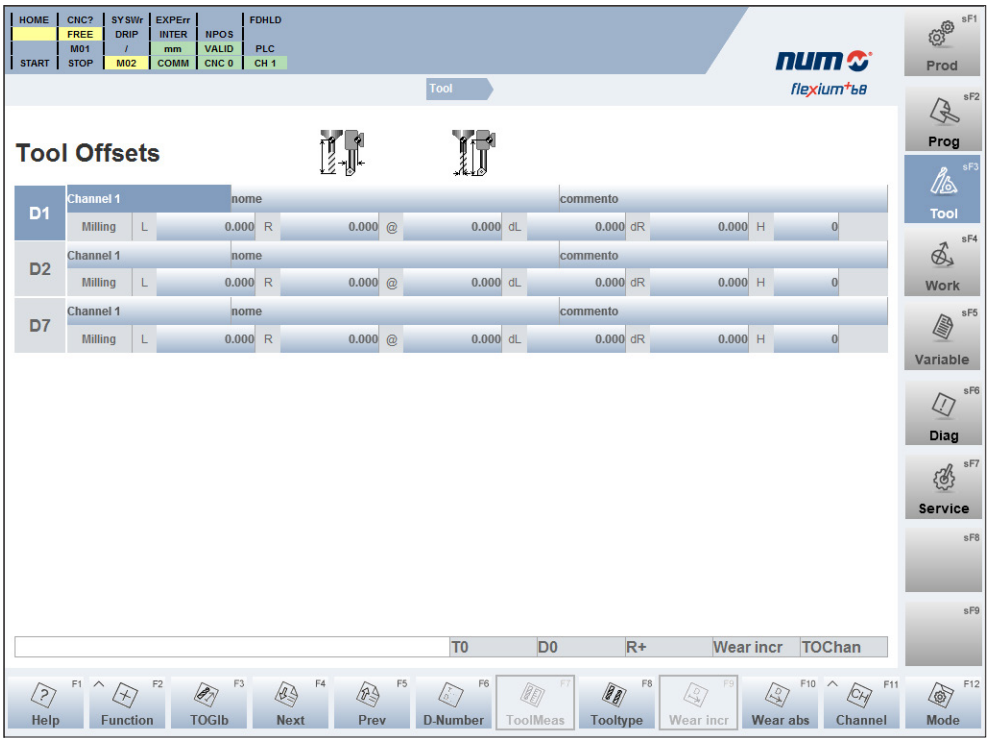
1	Channel selection.
2	Name or comment field.
3	Name or comment field.
4	Kind of display.

The global display shows all tools.

To toggle between displays	☞ F3

12.3.2 Channel display

This kind of display shows only the tools assigned to the currently selected channel.



12.3.3 Channel assignment and comments

For additional information on entering tool dimensions, refer to chapter *"Context Tool"*.

12.3.4 Assigning a tool to a channel

Select the channel field	☞ ← or →; ↑ or ↓
Open the channel list	☞ Enter
The channel list opens only if the display is set to *"Global"*.	
Select a channel	☞ ↑ or ↓ and Enter

12.3.5 Assigning a name and a comment

Select the field	☞ ← or →; ↑ or ↓
Open the field	☞ Enter
Enter the name and validate	☞ Enter

12.4 Symbolic names

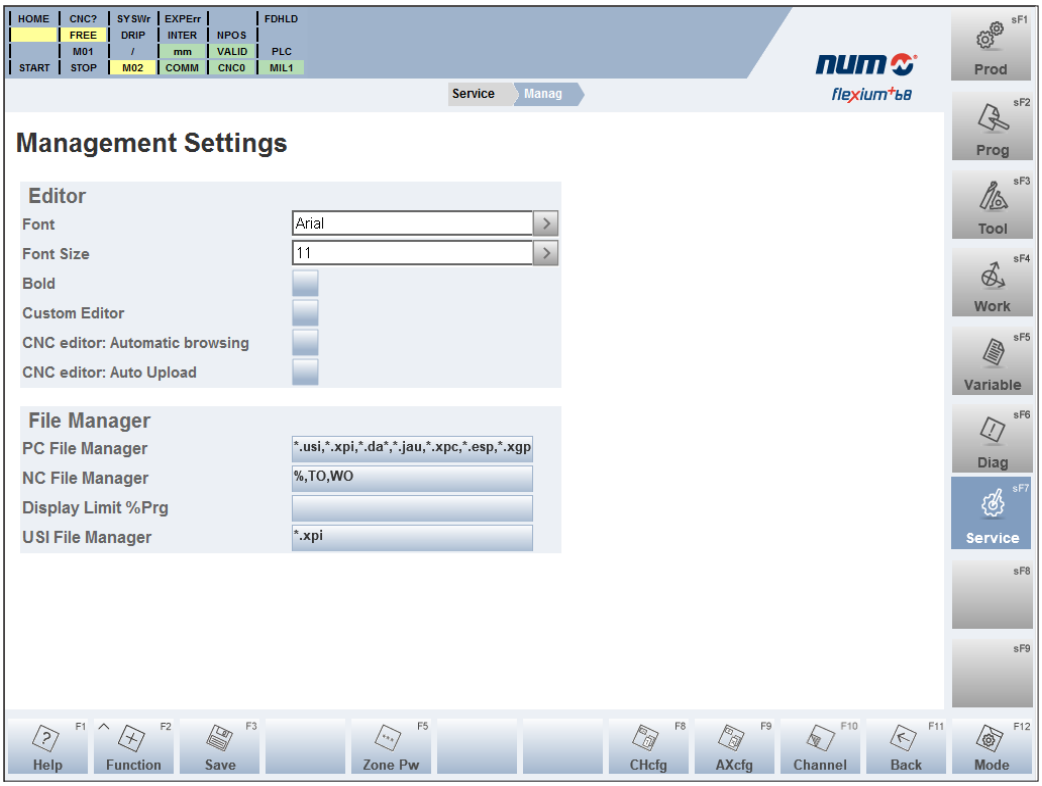
Gives the possibility to assign names to the axes, axes channels and CNC. These are displayed instead of usual address numbers.

12.4.1 Set up of the symbolic names

Two pages are available for the set up of symbolic names:

Select the context “Service”	☞ Shift + F7
Enter the password and validate	☞ Enter
Select the page “Management”	☞ F5

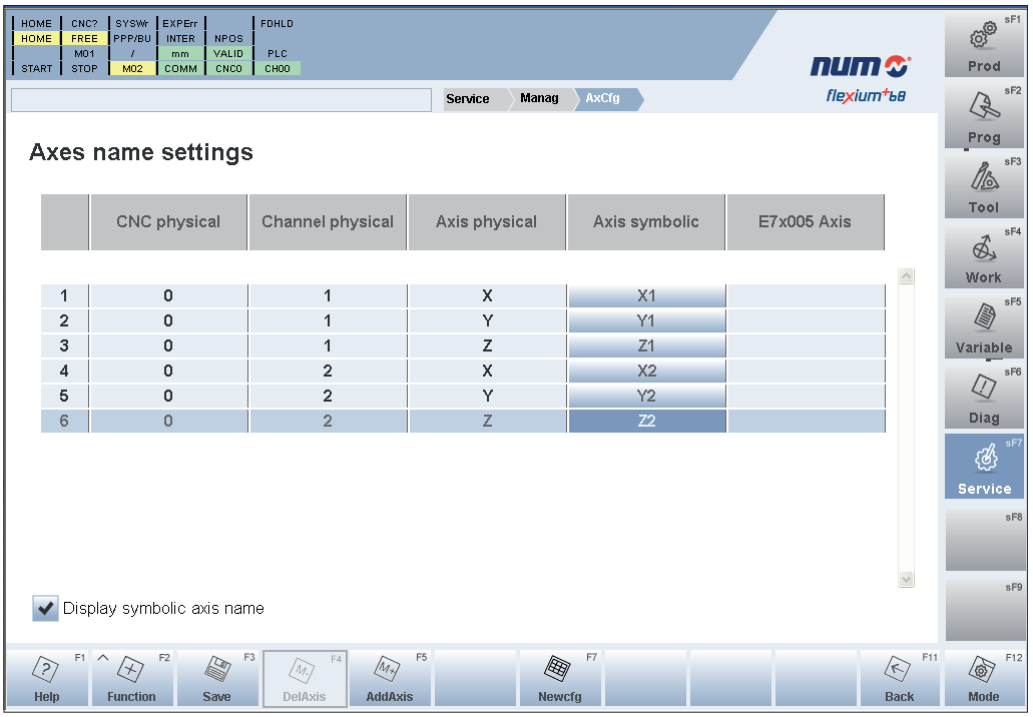
This opens the management settings page



Possible actions

Press F9 to access the axes configuration.
Press F8 to access the channel configuration.
Press F5 to enter memory zone password.

12.4.2 Axes configuration



Possible actions

Press F7 to set default symbolic axes names.
Press F5 to give a symbolic name to an axis address assignment.
Press F4 to delete the name given to an axis address assignment.
Press F3 to save the settings.

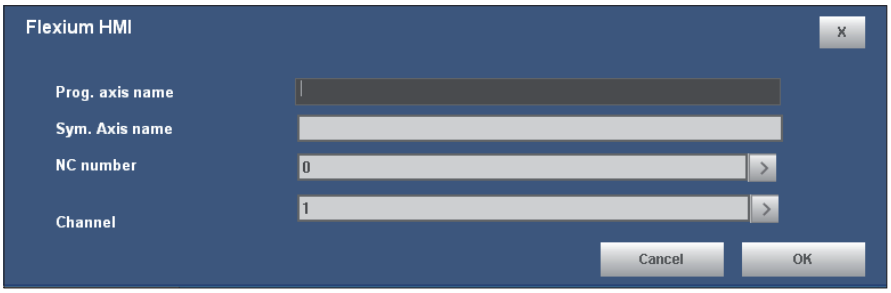
12.4.3 Entering axes names

Up to 6 letters or numbers may be used for a symbolic axis name.	
Select the desired axis	☞ ↑ or ↓
Open the field	☞ Enter
Enter the axis name and validate	☞ Enter

12.4.4 Giving a name to an axis address assignment

For additional information on axis address assignment, refer to the Flexium+ CNC Programming manual M00040.

Select the function *“Add axis”*	☞ F5



Move the cursor to the field “NC number”	☞ Tab
Select the CNC	☞ ↑ or ↓
Move the cursor to the field “Axes channel”	☞ Tab
Select the channel	☞ ↑ or ↓
Move the cursor to the field “Programmed axis name”	☞ Tab
Enter a valid axis name and validate	☞ Enter
Move the cursor to the field “Symbolic axis name”	☞ Tab
Enter a symbolic name and validate (6 characters max.)	☞ Enter

12.4.5 Deleting the name given to an axis address assignment

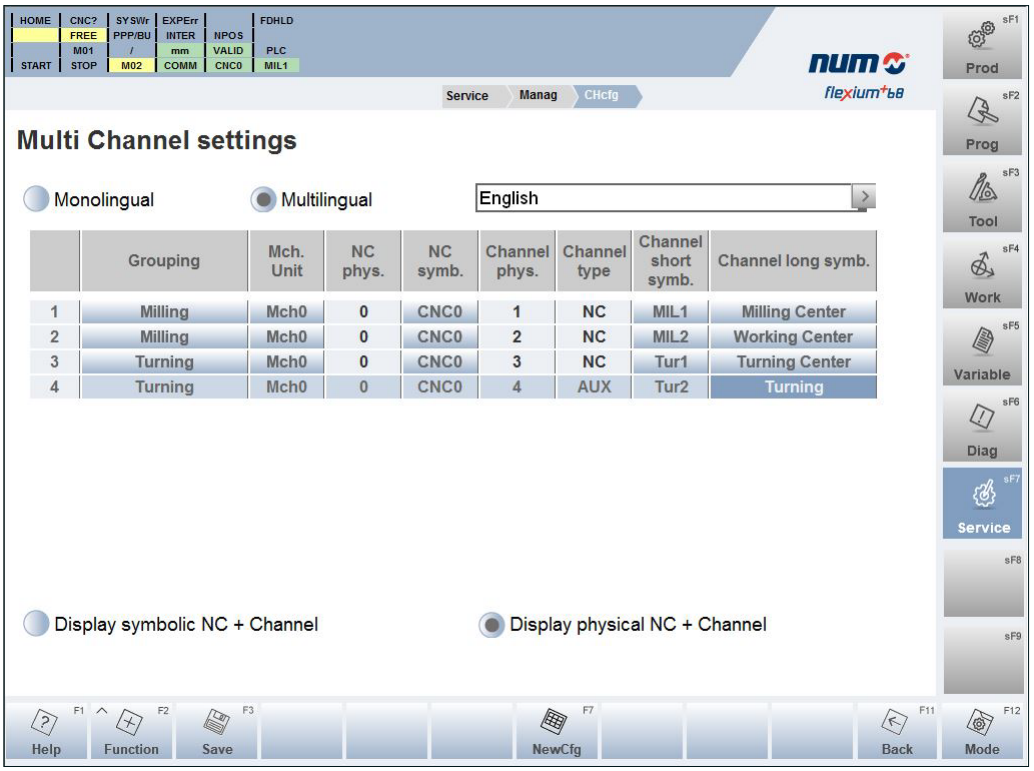
Move the cursor to the axis to be deleted	☞ ↑ or ↓
Delete the assignment	☞ F4

12.4.6 Display of axis name of address

It is possible to toggle between axis address and symbolic name display at any moment.
In the “Axes name settings” page:

Move the cursor to the checkbox “Display symbolic axis name”	☞ Tab
Toggle the display mode	☞ Space or Enter

12.4.7 Axes channel configuration



Possible actions

Press F7 to set default symbolic axes channel names.
Press F3 to save the settings.

The settings of the channel and axes configuration are stored in the configuration files *Channel.cfg and Axis.cfg.*
These files are located in the folder *c:\Program Files\NUM\Flexium RTS\FlexiumHMI.*

Explanation of the setting options

Monolingual	If this box is activated, only the symbolic names of the English configuration file are used.
Multilingual	Activate this box to be able to use the long symbolic names of the various languages.The selection of the languages can be made in the field at the right.

The activation of the box *"Multilingual"* and the subsequent selection of the language allows the entry of names and assignments in the corresponding configuration file. This makes it possible to display the long symbolic names in the language chosen for the HMI in the various contexts.

The language selection in this page allows only to enter the names into the appropriate configuration file.The displayed symbolic names in the HMI will only be adapted to the language chosen after it is selected in the *"General settings"* of the context *"Service"*.

After entering, the names *"CNC symbolic"* and *"Channel short symbolic"* will be active for all languages. A subsequent modification in any language will thus be effective for all other languages. In such a case, it will not be possible to load already existing programmes in the CNC!

Grouping	Allows to join together any axes channel under a common name.Up to 15 letters and numbers may be used.
Machine Unit	Name of an entire machining unit. This designation is mainly used together with the function Multi-CNC. It allows to join together all CNC of a machine under a common name. As a result, the management and assignment of part programmes common to all CNC is made possible in an easy way. Up to 5 letters and numbers may be used.
NC physical	Shows the physical number of the CNC. This number is given automatically by the HMI.
NC symbolic	Allows to assign a symbolic name to each CNC. Up to 5 letters and numbers may be used.
Channel physical	Shows the physical number of the axes channel. This number is given automatically by the HMI.
Channel type	Shows the type of the axes channel. This description is given automatically by the HMI.
Channel short symbolic	This field allows to assign a short symbolic name to the channel. Up to 5 letters and numbers may be used.
Channel long symbolic	Entry field for the long symbolic name of the channel. Up to 15 letters and numbers may be used.
Display symbolic NC and axes channel	To display the symbolic names.
Display physical NC and axes channel	To display the physical addresses.

Navigation and selection

Select a setting	☞ Tab
Open a setting list	☞ Enter
Select an entry in an open list	☞ ↑ or ↓ and Enter
Activate or deactivate a checkbox	☞ Space or Enter

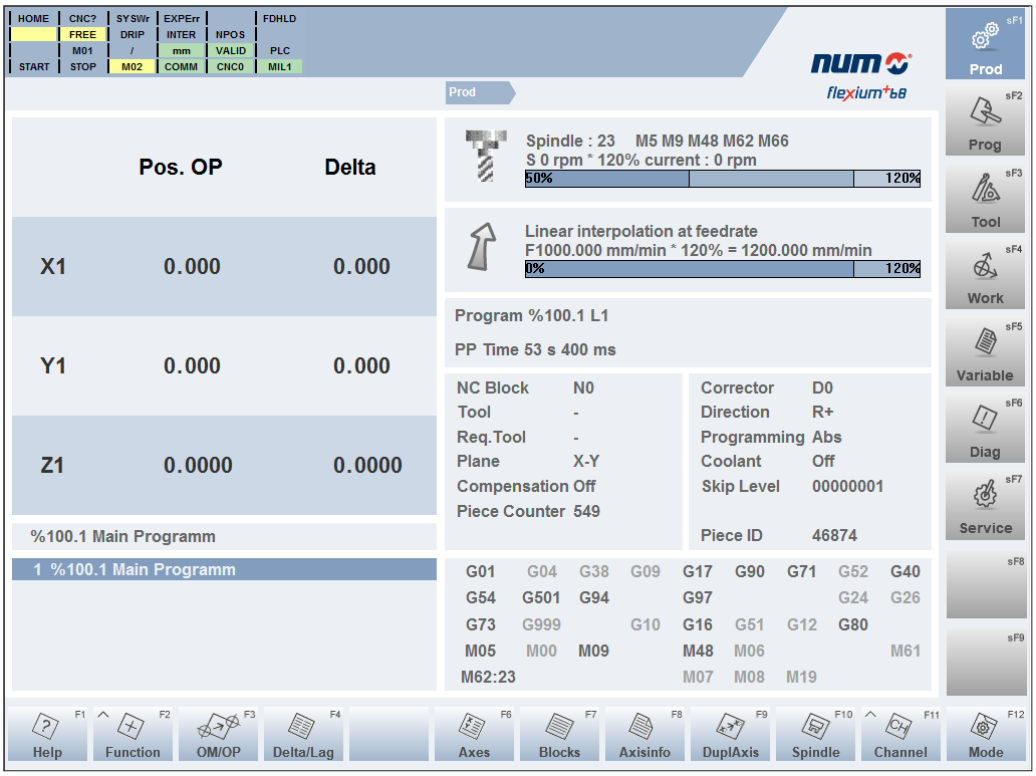
Entering names

Move the cursor to a description field	☞ Tab
Select a field	☞ ← or →; ↑ or ↓
Open the field	☞ Enter
Enter a name and validate	☞ Enter

12.4.8 Display of symbolic names

Symbolic names are displayed in all contexts and pages where the usual physical addresses are normally shown.

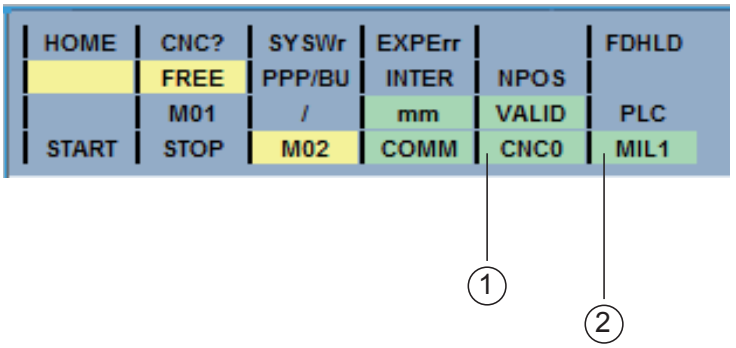
As example, the main page of the context “Production”:



12.4.9 Display in the status window

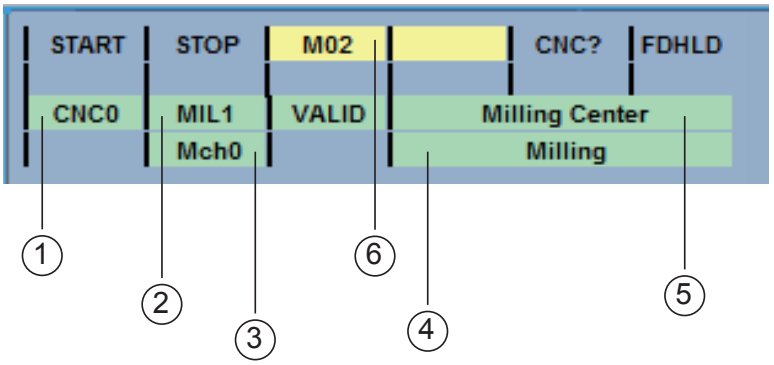
The status window can be displayed in two different ways.

Status window with short symbolic names



1	Symbolic name of the CNC.
2	Short symbolic name of the channel.

Status window with long symbolic names



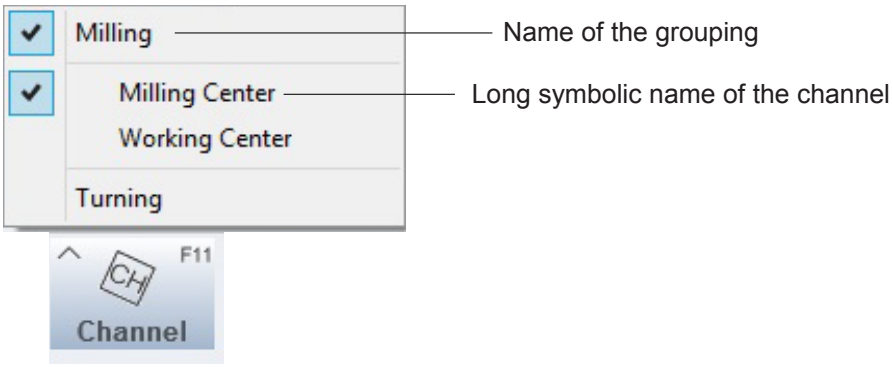
1	Symbolic name of the CNC.
2	Short symbolic name of the channel.
3	Name of the machine unit.
4	Name of the grouping.
5	Long symbolic name of the channel.
6	Mode waiting (selected, but not yet active).

Selecting the status window display

Open the function list	☞ F2
Select “Toggle status display”	☞ ↑ or ↓ and Enter↵
or Toggle with	☞ Ctrl + W

12.4.10 Channel selection

The channel selection is available on every relevant page of the HMI.
The procedure is the same as with physical addresses.



Channel selection procedure

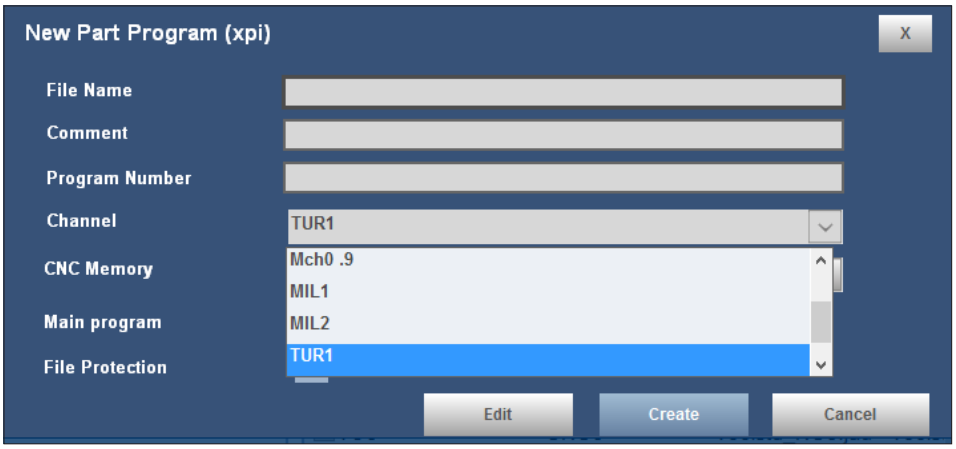
Open the channel selection	☞ F11
Move the cursor to the desired grouping	☞ ↑ or ↓ and Enter
Move the cursor to the desired channel	☞ ↑ or ↓ and Enter

12.4.11 Particularities of the channel selection

When creating new programmes and files, an assignment to axes channels or CNC will be required. This can also be done at a later time with the function *"Property"*.

A file whose assignment does not correspond to the settings of the channel configuration cannot be loaded in the CNC.

Example of creation of part programme

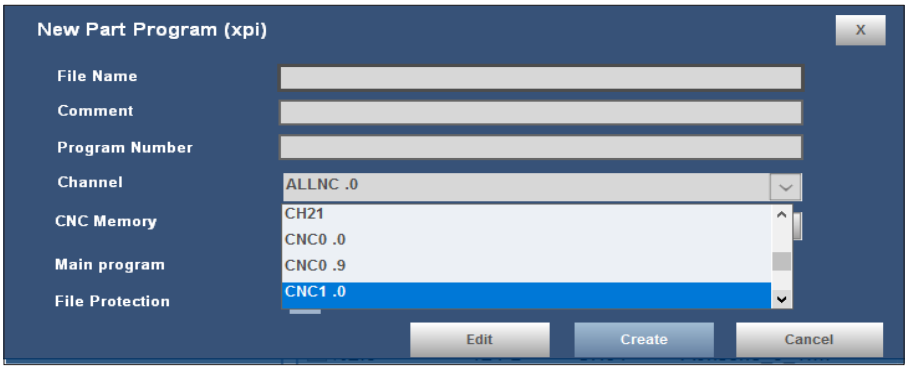


Channel assignment	Assigns the file to the settings of the axes channel configuration. The file gets automatically the number of the corresponding channel.
CNC assignment	This assignment can be used to create so-called global programmes for the selected CNC. These programmes can be used by any axes channel. When they are created, they are given a number of the type nnnn.0 or nnnn.9.
Assignment to all CNC	Used to create global programmes for all CNC. They can be used by any axes channel of any CNC and are given a number of the type nnnn.0 or nnnn.9.

12.4.12 Particularities with the function Multi-CNC

If the function Multi-CNC is activated in the system, an additional assignment to the Machine Units can be carried out.

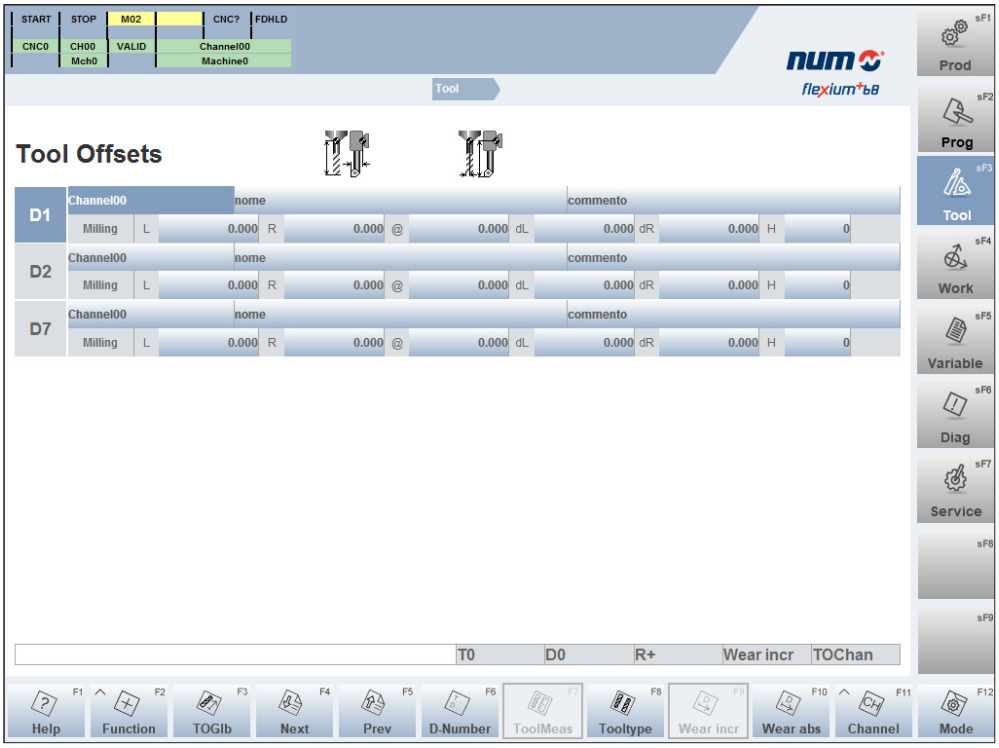
It is thus possible to load part programmes (*.xpi) as well as compact part programmes (*.xpc) automatically into all CNC belonging to a defined Machine Unit.



Assignment to Machine Units	For part programmes (*.xpi): they are given a number of the type nnnn.0 or nnnn.9. For compact part programmes (*.xpc): these programmes are assigned to a Machine Unit without specific number.

12.4.13 Specific features of the extended tool table

The function *“Extended tool table”* allows a precise specification of the tools.
The names *“Channel long symbolic”* are used here to assign the tools to a specific channel.



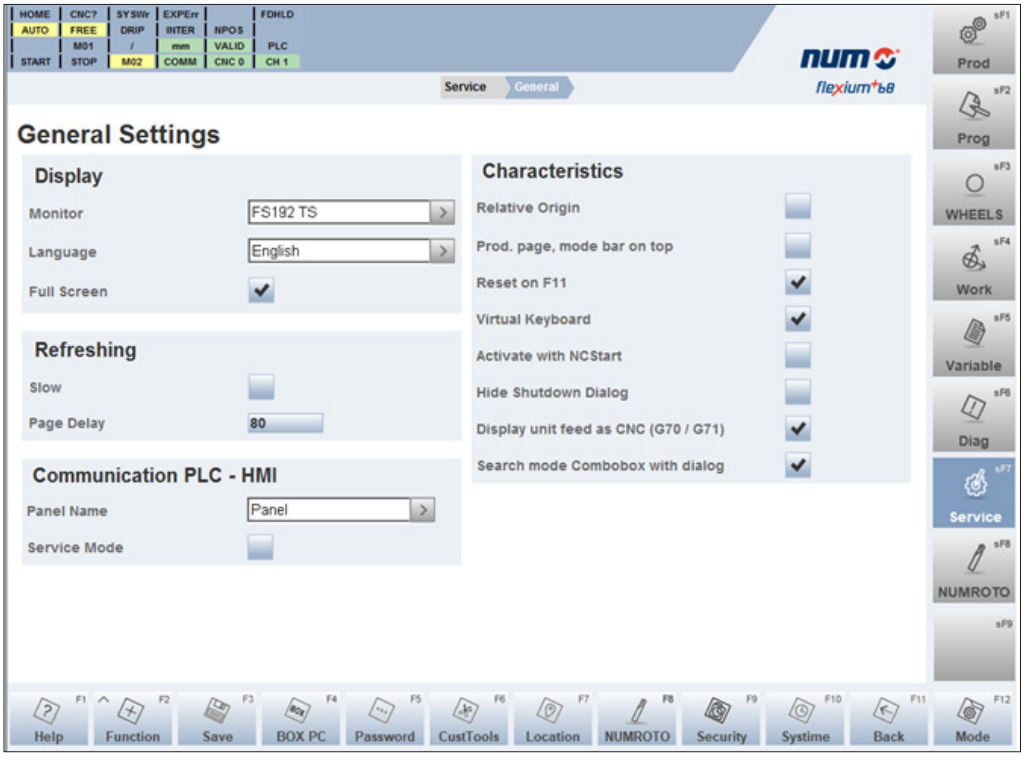
12.5 Extensions for NUMROTOplus®

12.5.1 Particularities in the context bar

1	Access to the context “Wheel compensation”.
2	Access to NUMROTO*plus*®.

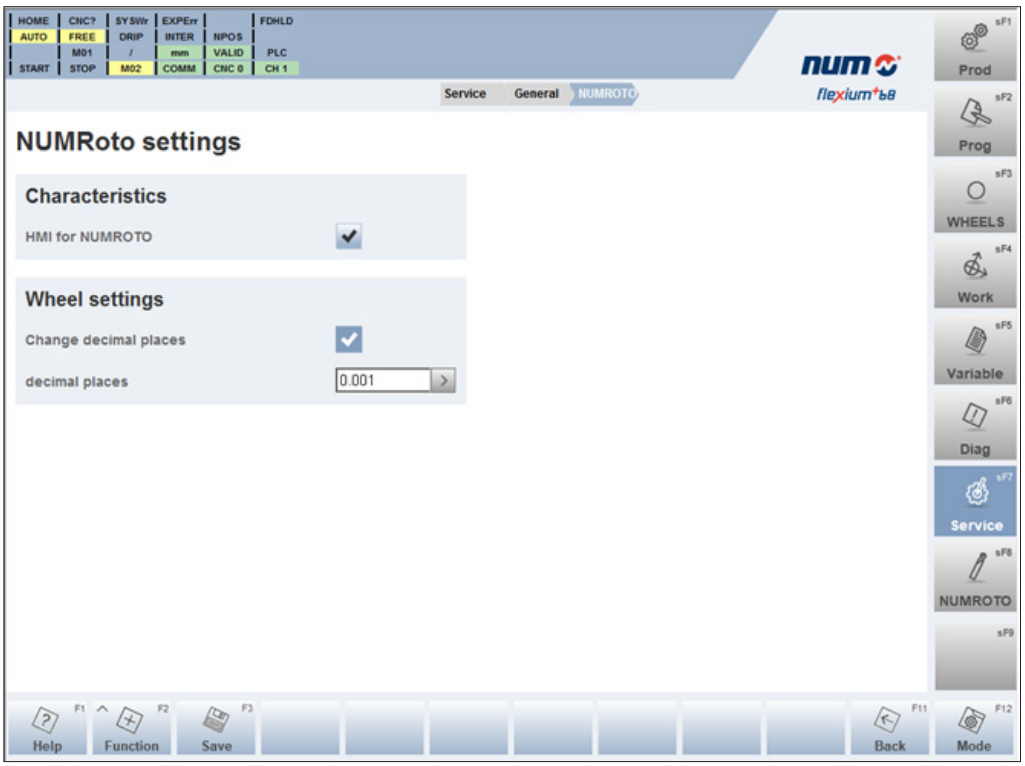
12.5.2 Setting in the context “Service”

A page is available for set up the NUMROTO settings.
The settings page is in the *“General Settings”* page of *“Context Service”* on the F8 button.



Possible actions

Press F3 to save the settings.



Explanation of the setting options in the field “Display”

Characteristics	
HMI for NUMROTO	If this box is checked, Flexium will be used as the standard HMI in the NUMROTOplus® environment. Activating the function key F12 on the main page of NUMROTOplus® will switch over to Flexium HMI.

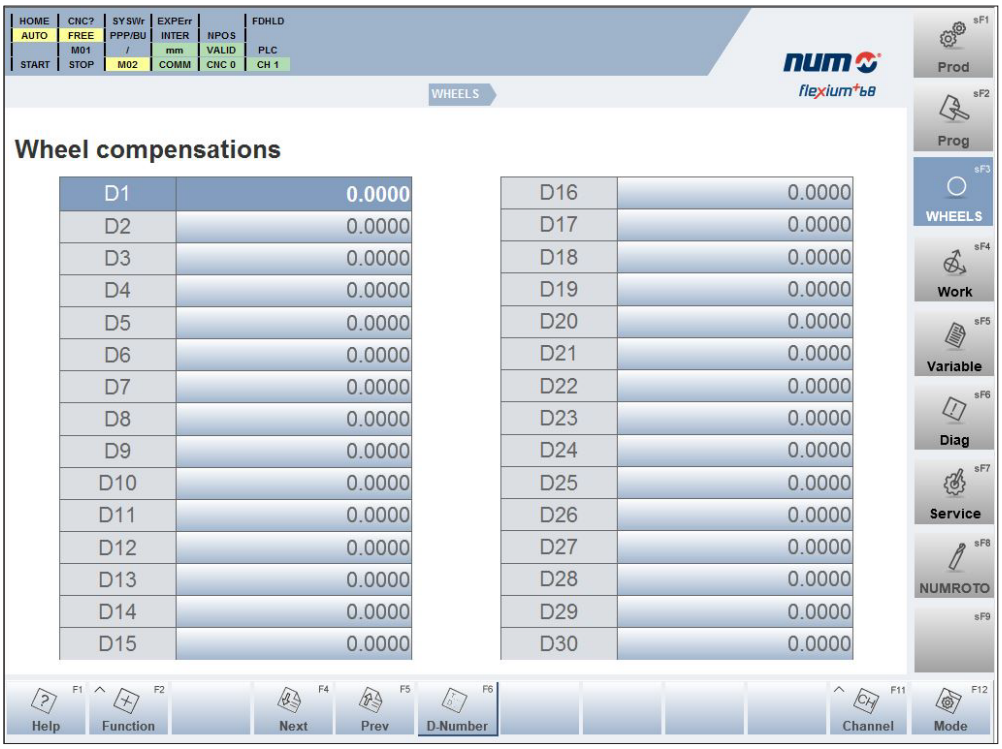
Wheel settings	
Change decimal places	Activate the options to change the decimal places in the Wheels page.
Decimal places	Select the decimal places of the Wheels page.

Possible actions

Press F3 to save the settings.

12.5.3 Wheel compensation

This context allows to enter and modify wheel compensations.
255 compensation values are available.



Possible actions

Press F6 to position the cursor directly on the desired compensation 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 ↵

Entering compensation values

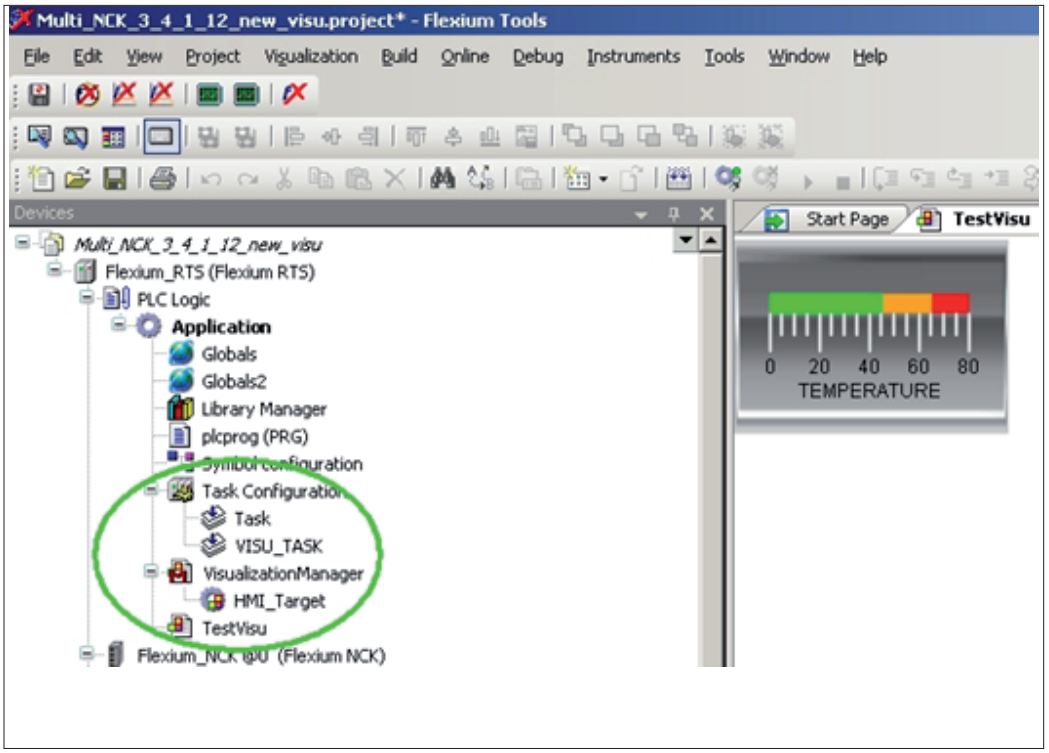
Select a compensation using number D, enter the appropriate number and validate or Select a compensation using the list	☞ F6 , ☞ Enter ☞ F4 , F5 , ↑ , ↓
Open the field	☞ Enter ↵
Enter the desired value and validate	☞ Enter ↵

12.6 PLC visualization

12.6.1 Integrated PLC visualization

This function gives access to main pages generated by the PLC application and embedded in the HMI. Two pages can be accessed by the free keys sF8 and sF9 a third one is integrated in the right side of the “PROD” page.

These pages are created by the OEM with Flexium Tools.

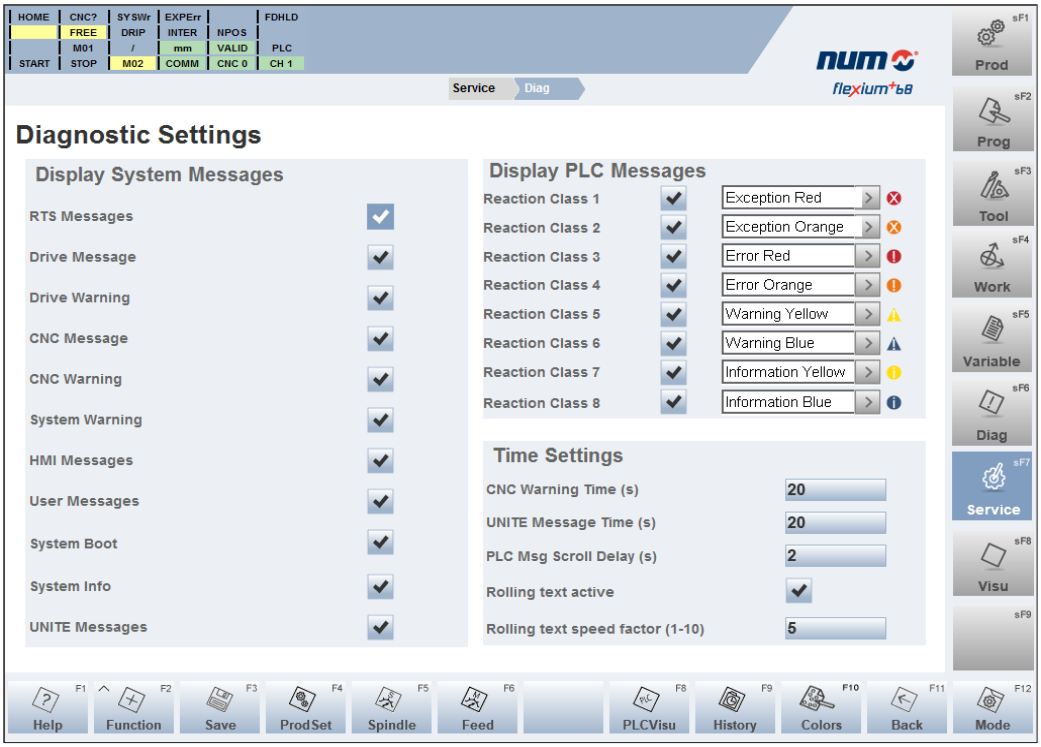


Possible actions

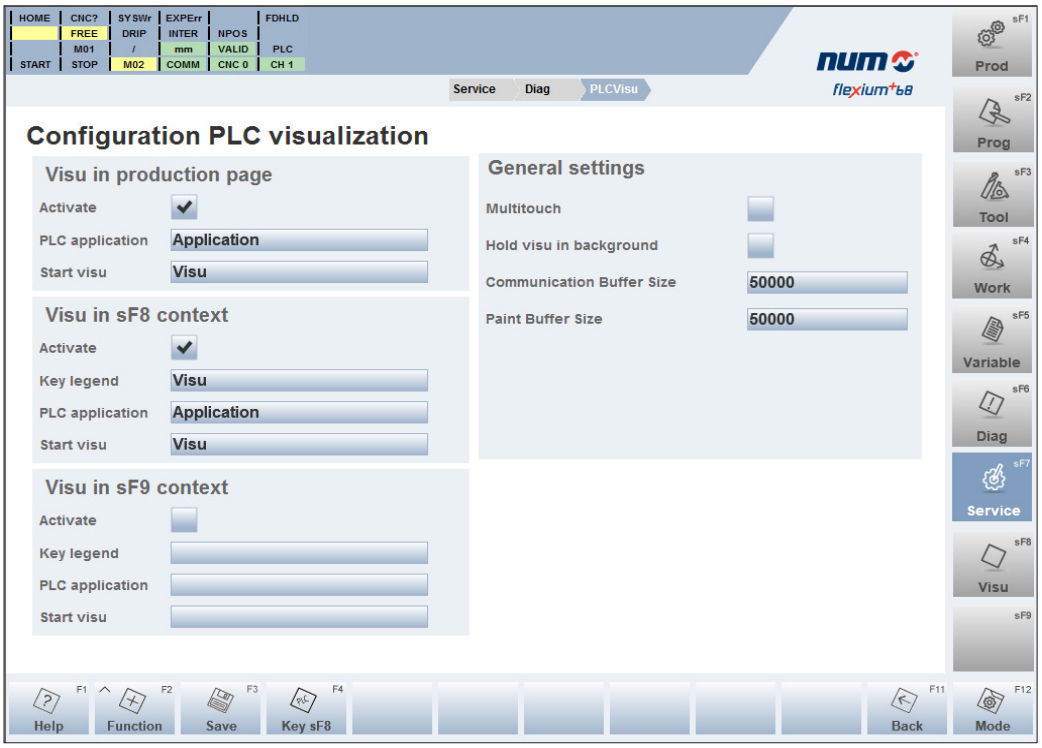
Press F2 to select the functions.
Press F3 to save the settings.
Press F4 to access the sF8 Key page. This is only available, if the visu in sF8 is enabled.
Press F5 to access the sF9 Key page. This is only available, if the visu in sF9 is enabled.

Settings

Select the context *“Service”*	☞ ⇧ Shift + F7
Enter your password	
Open the page *“Diagnostic settings”*	☞ F4



Press F8 to access the visualization settings page.



You may now select the desired location for your visualization display.
Enter a key legend, the PLC application and the visualization names. To activate the display, check the appropriate box.

The check box Multi touch allows two fingers operations on those visualizations.

You have to close and restart Flexium⁺ HMI for the changes to take effect.

The visualization displays are now ready to be used.

Explanation of the setting options

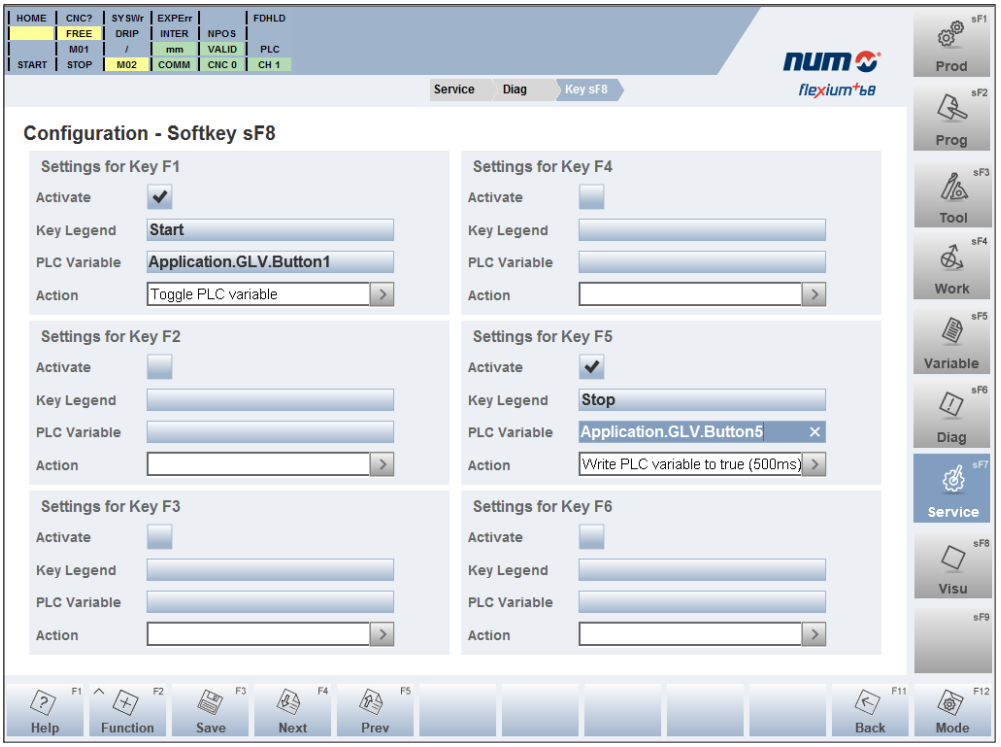
Visu in production page	
Activate	The Visu is enabled.
PLC Application	Name of the PLC Application containing the said visualization page.
Start Visu	Name of the Start Visu page.

Visu in sF8 / sF9 Context	
Activate	The Visu is enabled.
Key legend	Legend on the sF8 / sF9 Button.
PLC Application	Name of the PLC Application containing the said visualization page.
Start Visu	Name of the Start Visu page.

General Settings	
Multitouch	Enables multi touch mode. The Visu use the Multitouch functionality
Hold Visu in background	Keeps the visualization in background when another page is selected. This allows for a faster display but only one page (sF8 or sF9) can be managed. Note:This option works only, if one Visu is configured.
Communication buffer size	Size of the Communication buffer between PLC and HMI.
Paint buffer size	Size of the Paint buffer in the HMI.

12.6.2 Configuration – Softkey sF8 / sF9

This page configures the Softkey sF8 / sF9.



Possible actions

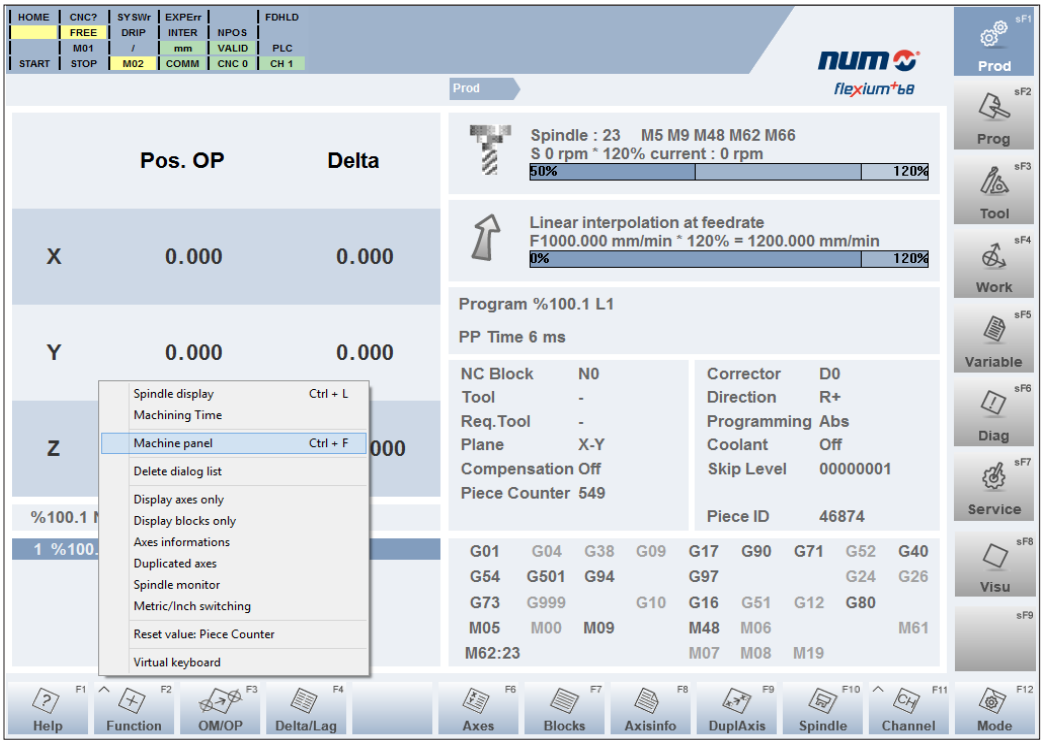
Press F3 to save the settings.
Press F4 to next page.
Press F5 to previous page.

Explanation of the setting options

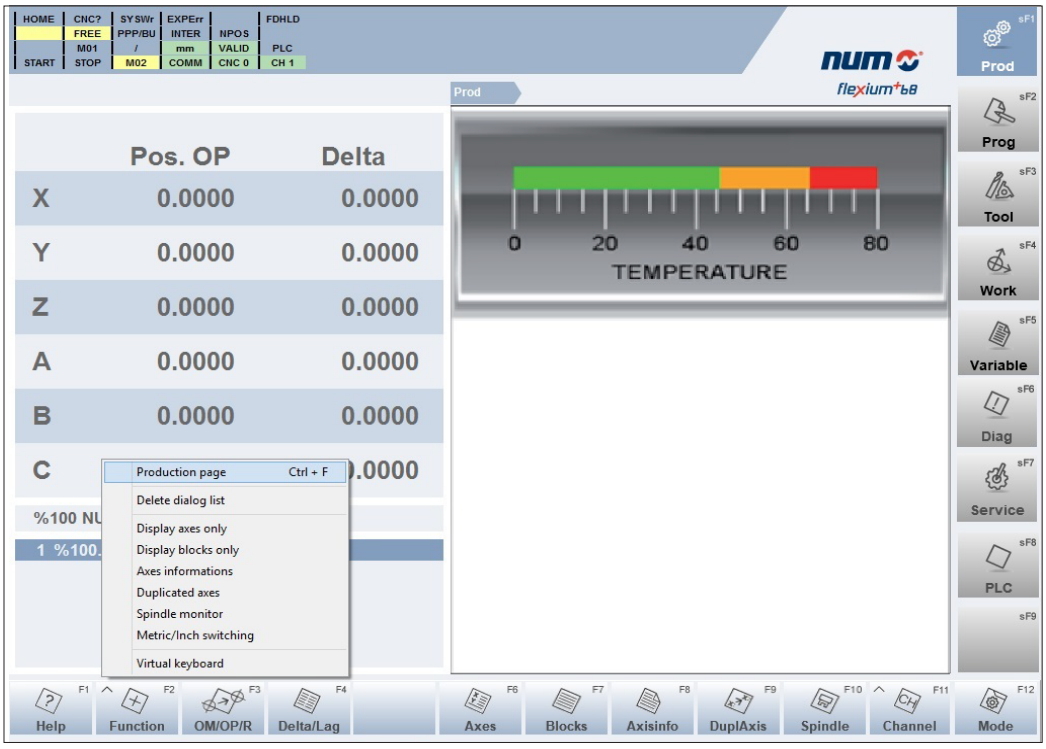
Settings for Key Fx	
Activate	The Button is enabled.
Key Legend	Text to display on the button.
PLC Variable	PLC Variable for the action.
Action	Action of the PLC Variable: - Toggle PLC Variable: the PLC Variable will toggle between true and false each time the button is pressed. - Write PLC variable to true (500mS): The PLC Variable is True, for 500ms.

12.6.3 Sample of PLC Visu Pages

Particularity of the display in the context “Production”



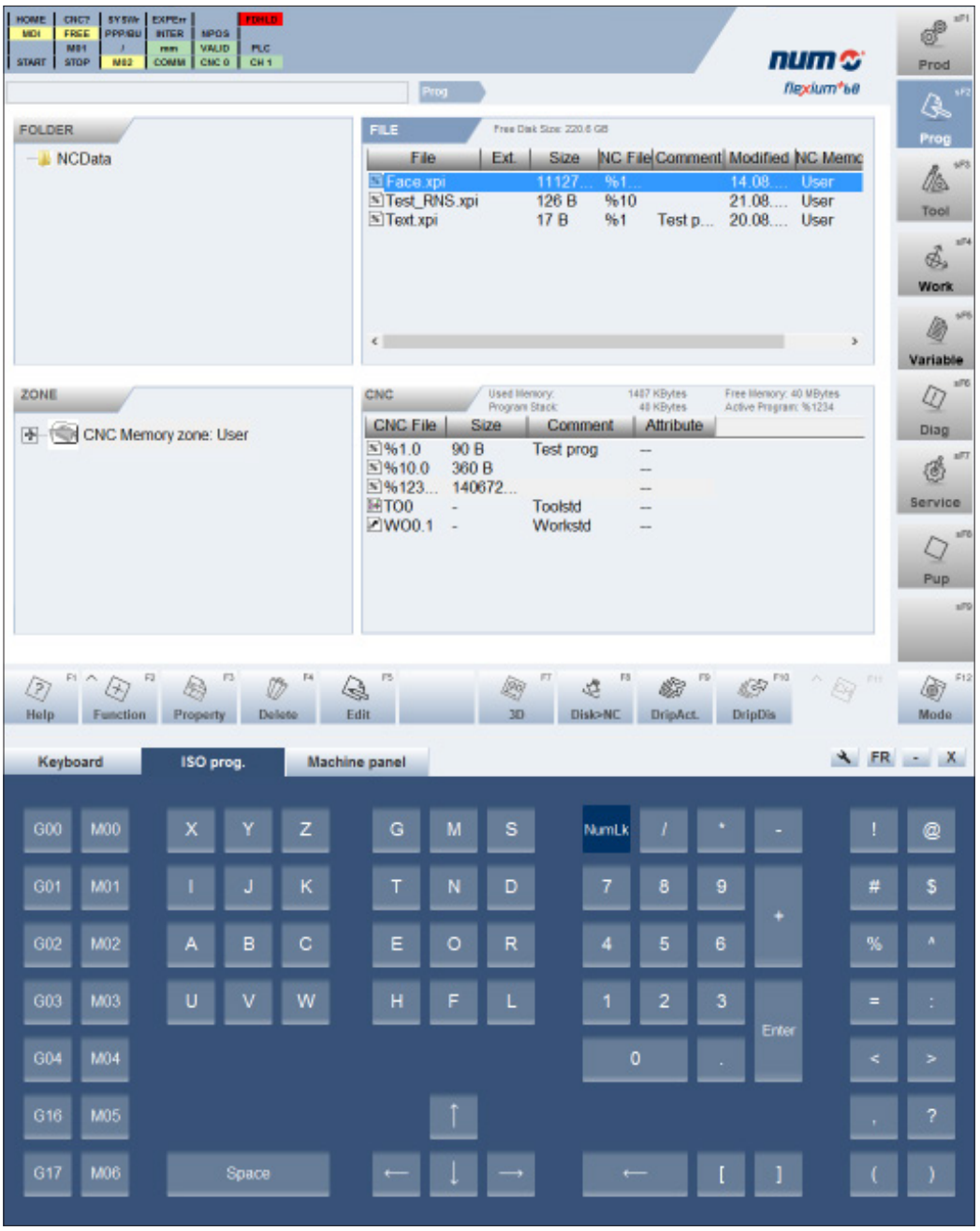
Press F2 and “Machine panel” or Ctrl + F to activate the display.



Press F2 and “Production page or Ctrl + F to activate the display.

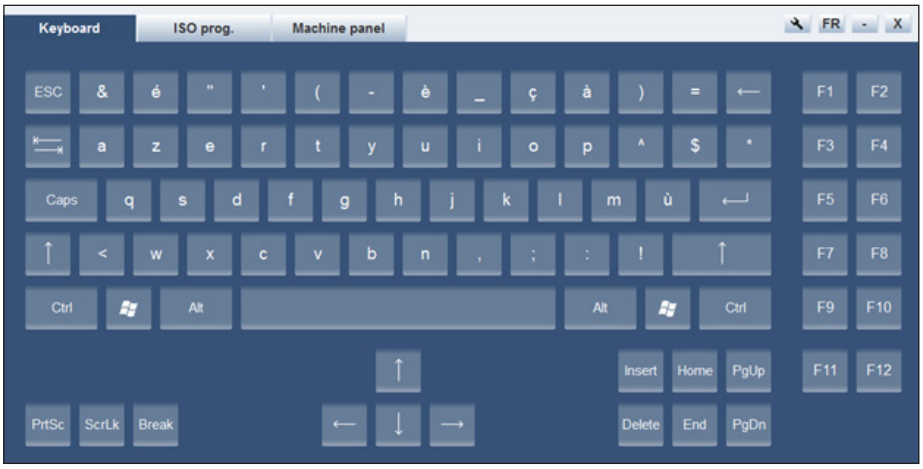
12.7 FXKeyboard

Some systems equipped with a 19” vertical display can feature an additional application: FXKeyboard.

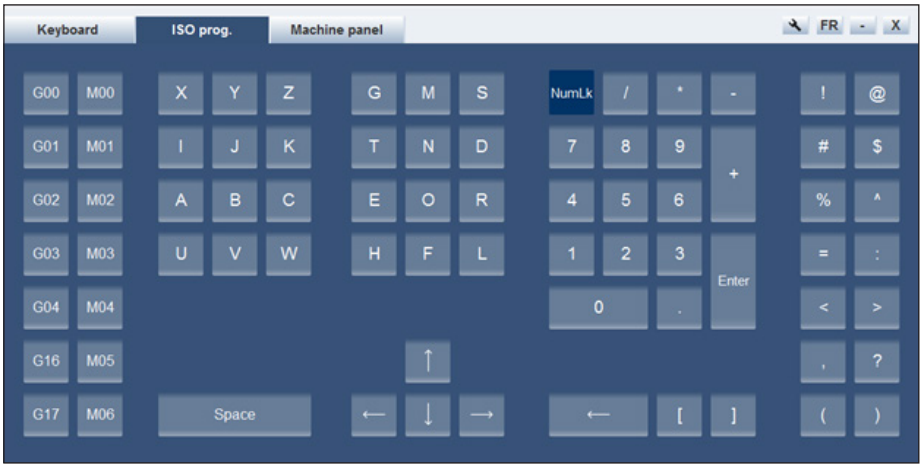


This application contains up to three tabs:

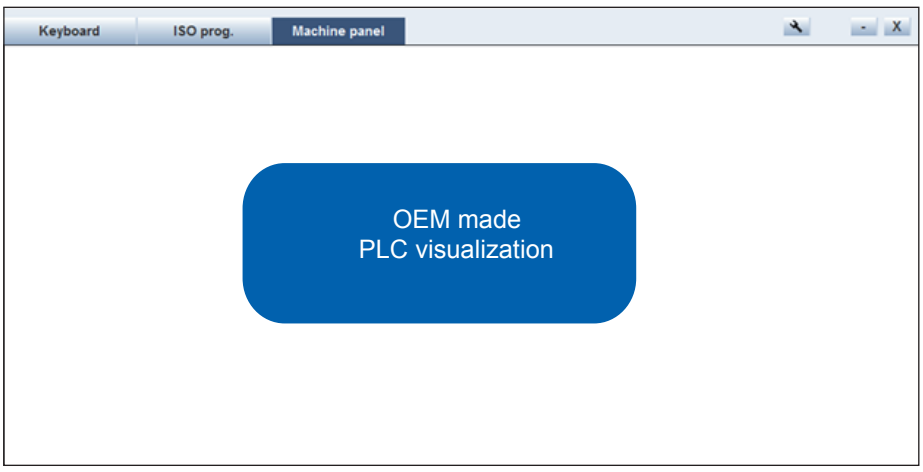
Regular Keyboard



Dedicated keyboard for ISO programming

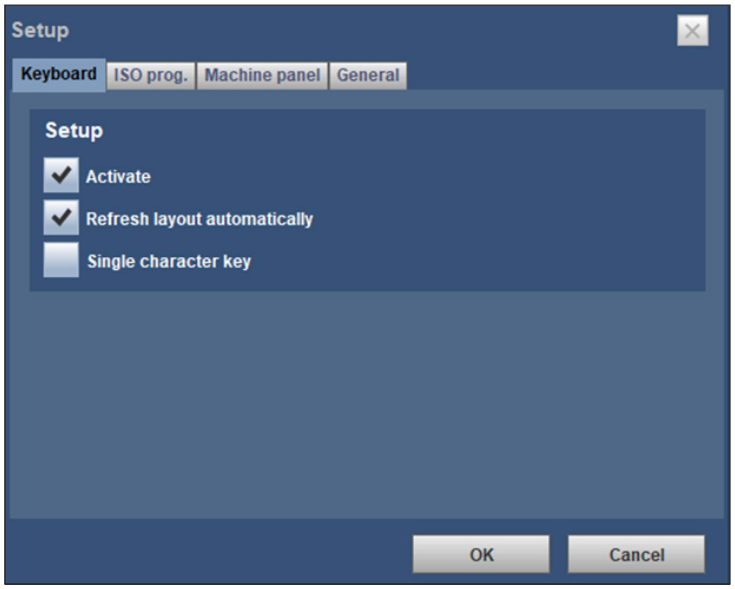


Machine panel (optional – created by the OEM)

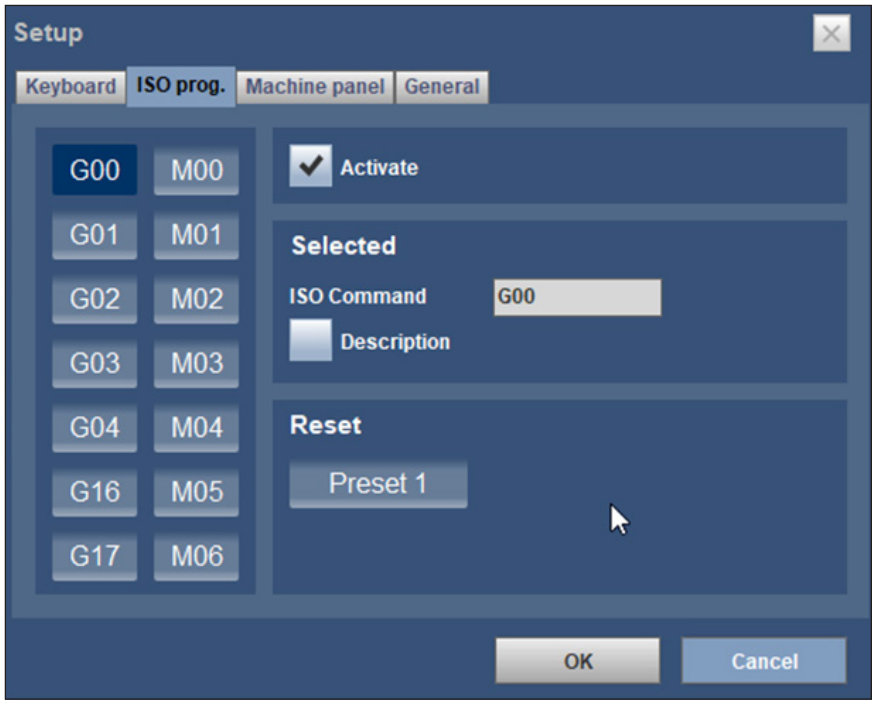


12.7.1 Settings

Settings
The settings are accessed by the Setup button.

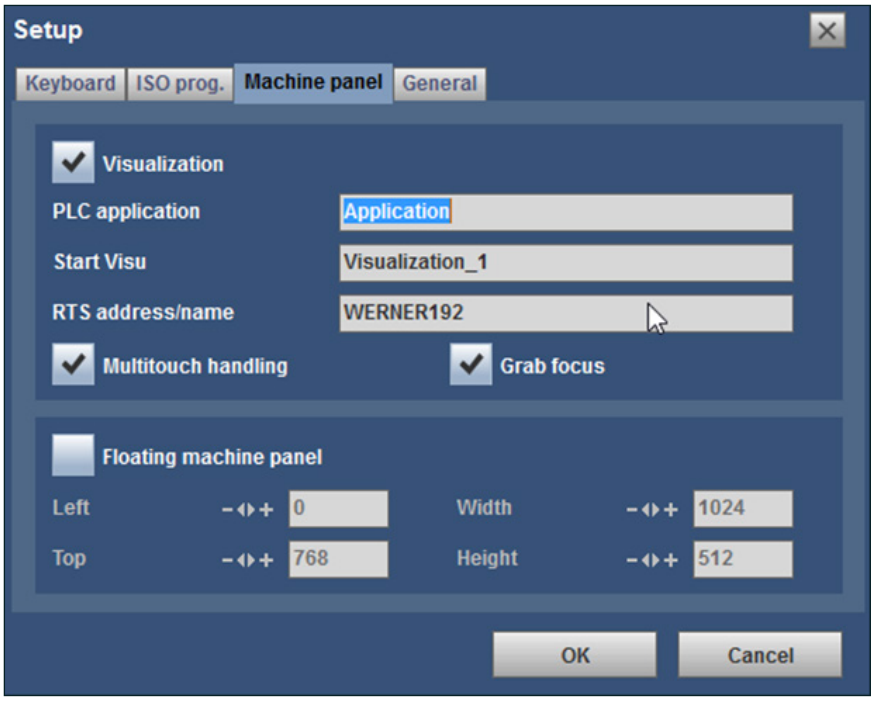
Keyboard Setup


Activate	Activate Virtual Machine panel.
Refresh layout automatically	The Key layout automatically follows the Windows Keyboard layout.
Single character key	Unique character shown on the keys. It is adjusted in relation with ⇧ Shift, Alt and Ctrl.

ISo Prog Layout


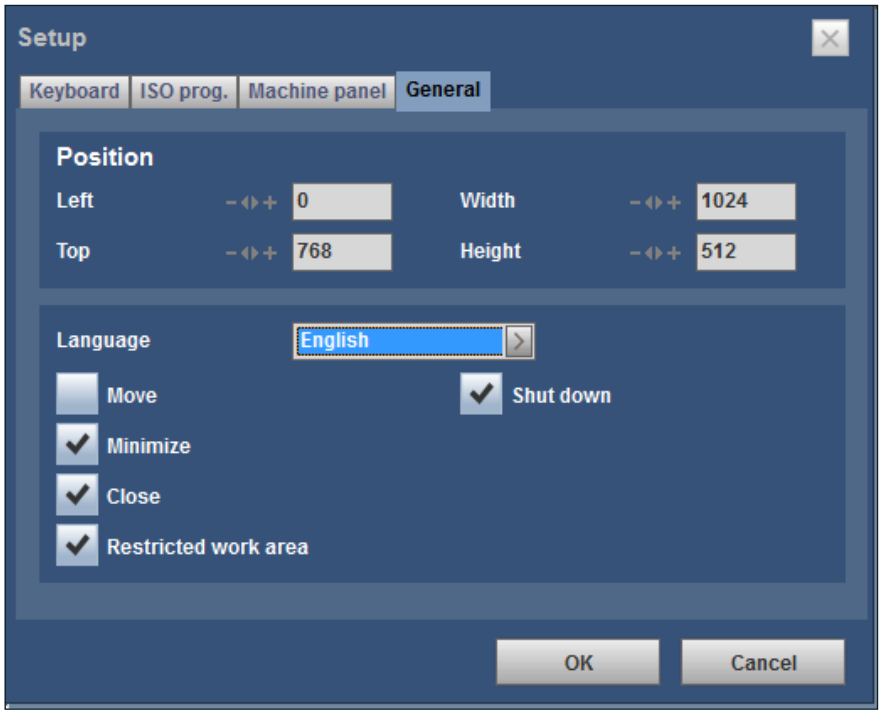
ISO Command	The command sent to the HMI each time the corresponding key is pressed.
Description	The identification of the button.

Machine Panel



PLC application	The name of the PLC application (generally Application).
Start Visu	The initial page to display.
RTS address/name	Name or IP Address of the RTS containing the application and the page (case of Box PC).
Floating machine panel	Possibility to move the machine panel around (not recommended).
Left, Top, Width, Height	Dimensions (standard values shown).
Multitouch handling	When checked, this permits two fingers operation on the machine panel.
Visualization	Enable PLC application in Virtual Keyboard representing the Machine panel.
Grab focus	This is option is needed to edit some text fields inside the machine panel. Usually the virtual keyboard does not take the focus from other applications nor windows.

General



Position	Position of FXKeyboard (standard values shown).
Language	Legend of the tabs.
Move	Possibility to move FXKeyboard (not recommended).
Minimize	Possibility to Move and Expand (button -).
Close	Possibility to Close FXKeyboard (button X).
Restricted work area	The screen position of the machine panel is forbidden to other applications (recommended).
Shut down	Triggers the presence of a shutdown or reboot button on FXKeyboard.

All the settings are recorded automatically in the following file:
C:\Program Data\NUM\Flexium HMI\FXKeyboard\Config\FXKeyboard.ini

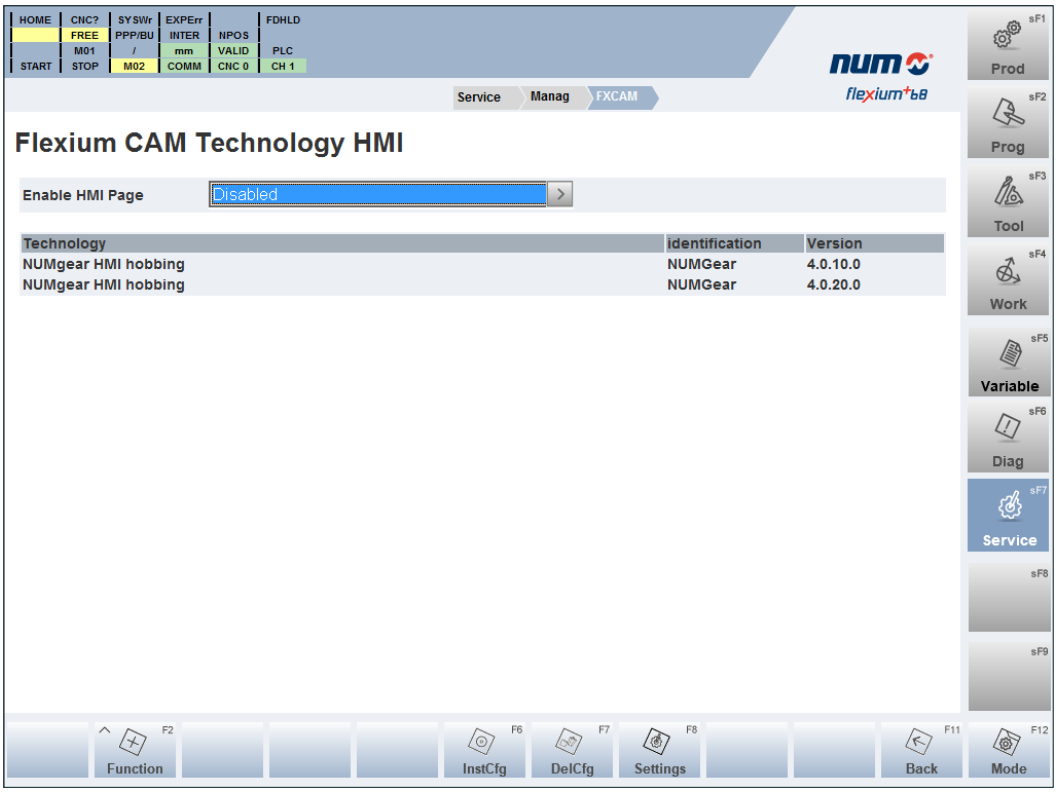
12.8 Flexium CAM

Flexium CAM is an interactive environment to create part programs using a graphic approach. A dedicated suite (Flexium CAM Designer) is used to create such programming environments (configurations).

The *Service* context of the HMI allows for integrating these configurations into the HMI.

Access to Flexium CAM integration is granted via the key F7 available under the management settings page of the *Service* context.

This opens the following page:

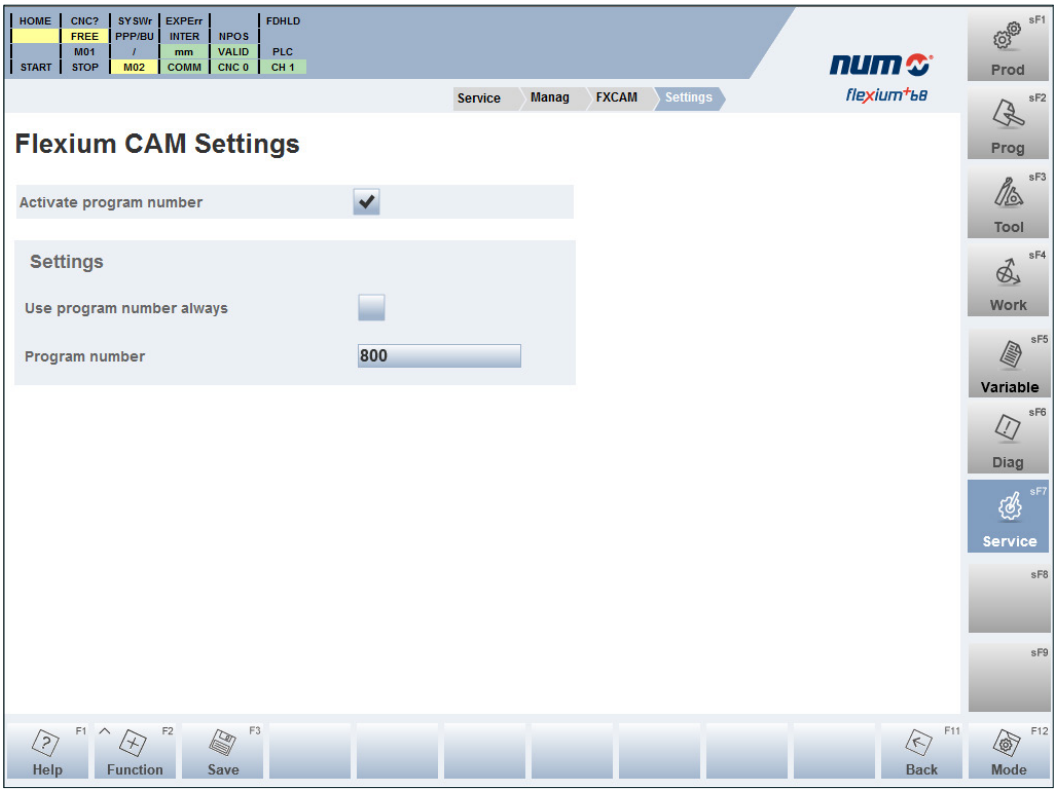


This page displays all configurations installed. Configurations are delivered under the form of an *".car file"*. Several configurations for different technologies can be installed.

Possible actions

Press F6 to install configuration : Installs a configuration created by Flexium CAM Designer.
Press F7 to remove an already installed configuration.
Press F8 to defines the possible settings.

12.8.1 Settings



Activate program number	Predefines a program number when a new file is created.
Use Program number always	The program created always gets the indicated number.
Program number	Default number of program.

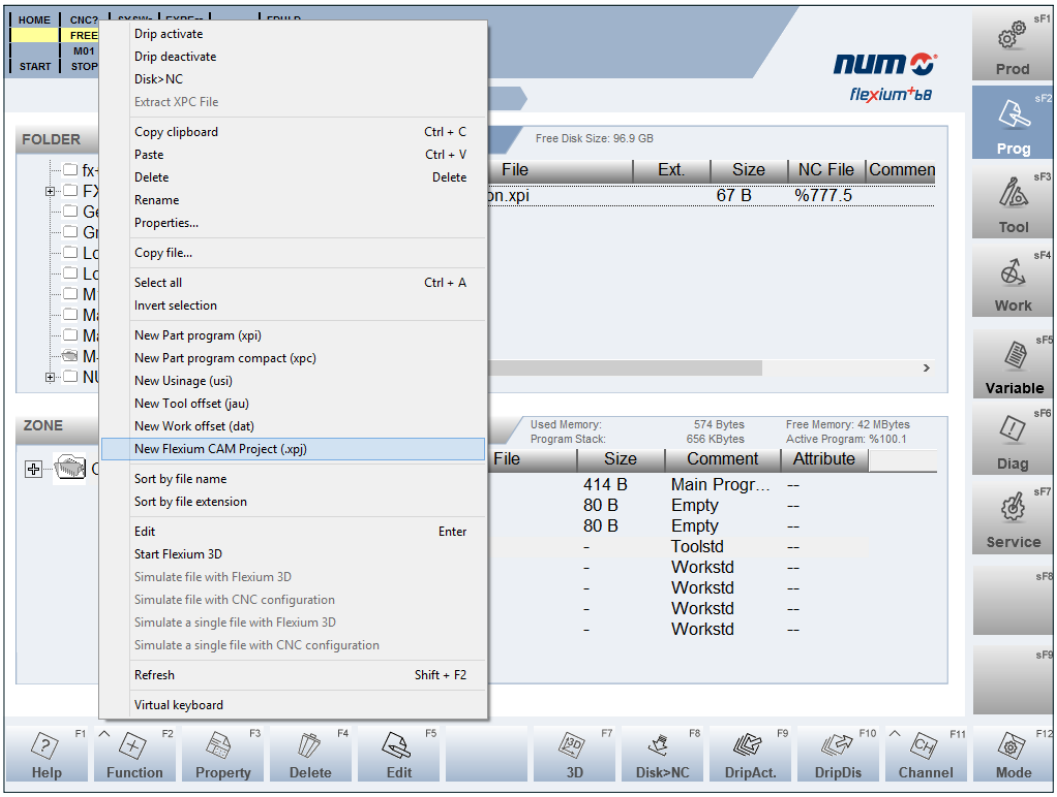
12.8.2 Install a configuration

Select a *".car"* file and press OK.
The file will be expanded and installed. This programming technology is now available.

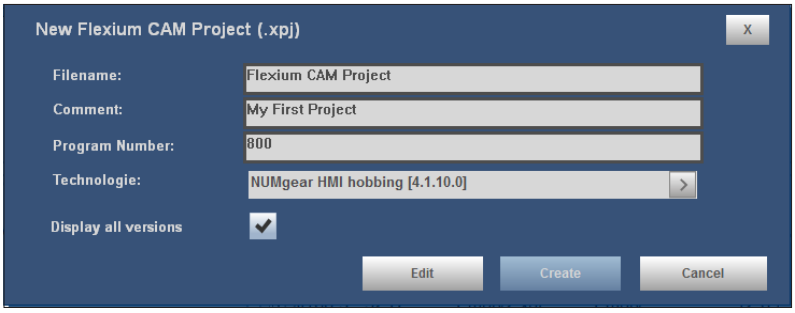
12.8.3 Operation

Once the new configuration is installed it is available in the *PROG* context.

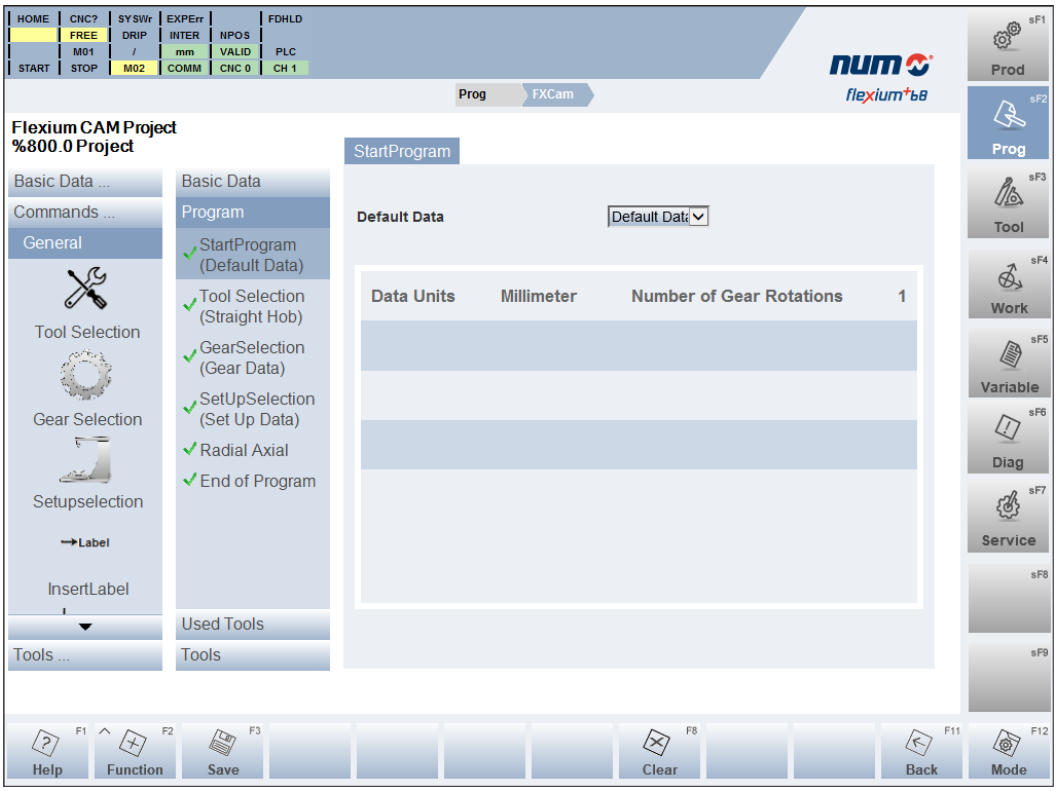
A right click in the Upper right windows will show the following menu. All configurations installed are used to create a Project *(*.xpj)*.



Selecting *'New Flexium CAM project'* will open the following window.



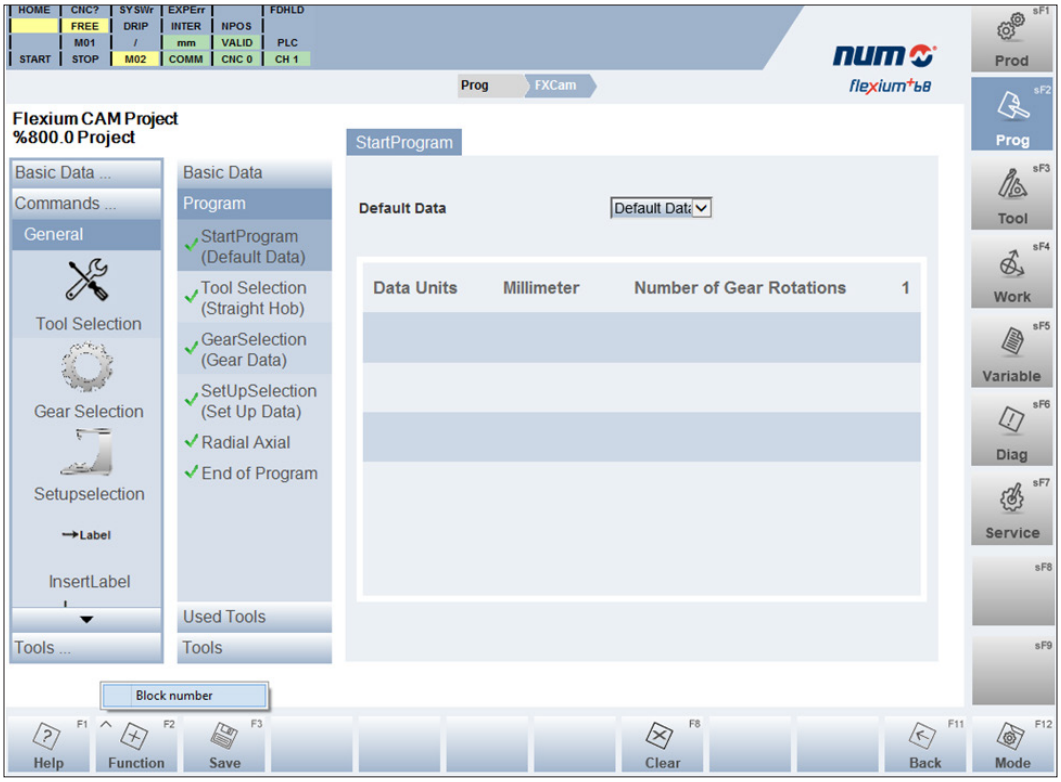
Fill in the required data and press OK. According to the configuration the system will show a page similar to the one below:

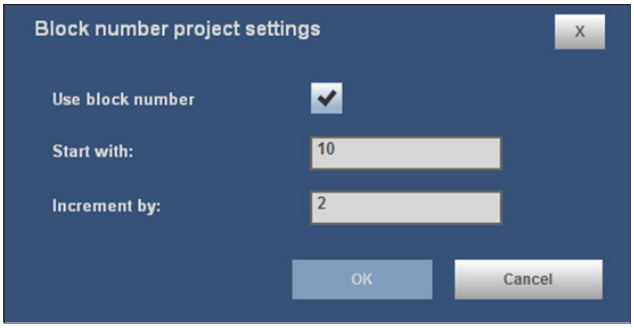


Example of a configuration for Gear Hobbing.

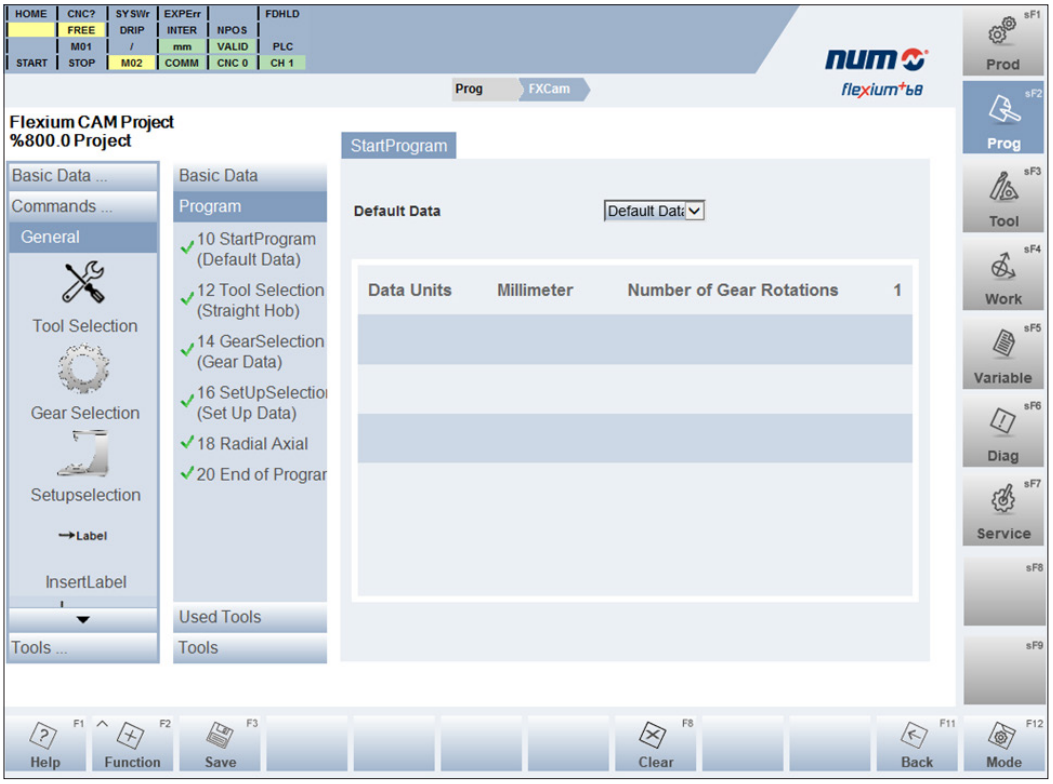
12.8.3.1 Block number

To use and configure the *Block number*, open the F2 menu and select *Block number*.





Use block number	To enable the block number.
Start with	First block number in the CNC output.
Increment by	Increment the block number with the value from one block to the next.



When the block number is use, the block number is displayed at begin from each command.

12.9 NUMcoss

NUMcoss is an additional component of High Speed Cutting (HSC) functionality in NUM Flexium CNC and is integrated in Flexium+ HMI Panel software.

Word background:
NUMcoss is an acronym for the words:

co	*Data compress* (decrease file size with transition to polynomial Interpolation data).
s	*Geometrical smoothing* of linear data (elimination of geometric Discontinuity regarding velocity, acceleration and jerk).
s	*Speed-up* (usage of polynomial data results in higher execution velocity).

With *NumCoss*, the customer will get an excellent tool to smooth the trajectory, reduce the file size and speed-up program execution for cutting and milling applications in NUM CNC´s.

Prior to data transfer to *NC-kernel NumCoss* analyses, smooths and converts G01- path conditions in ISO-programs, generated from CAM-systems with integrated post processors, into polynomial data. Thus this conversion happens on Flexium+ HMI (PC-side), NC-kernel is not stressed and whole kernel performance is free for quick interpolation and the ramp algorithm.

Main criteria inside the geometric transformation of *NUMcoss* are modification tolerance and chord error for linear and rotary axes. That means in which dimension the given linear data (polygon path) can be modified during transition to polynomial data.

Addition criteria are specific treatment of different segment lengths as well as correct filtration of spikes and geometric gaps.

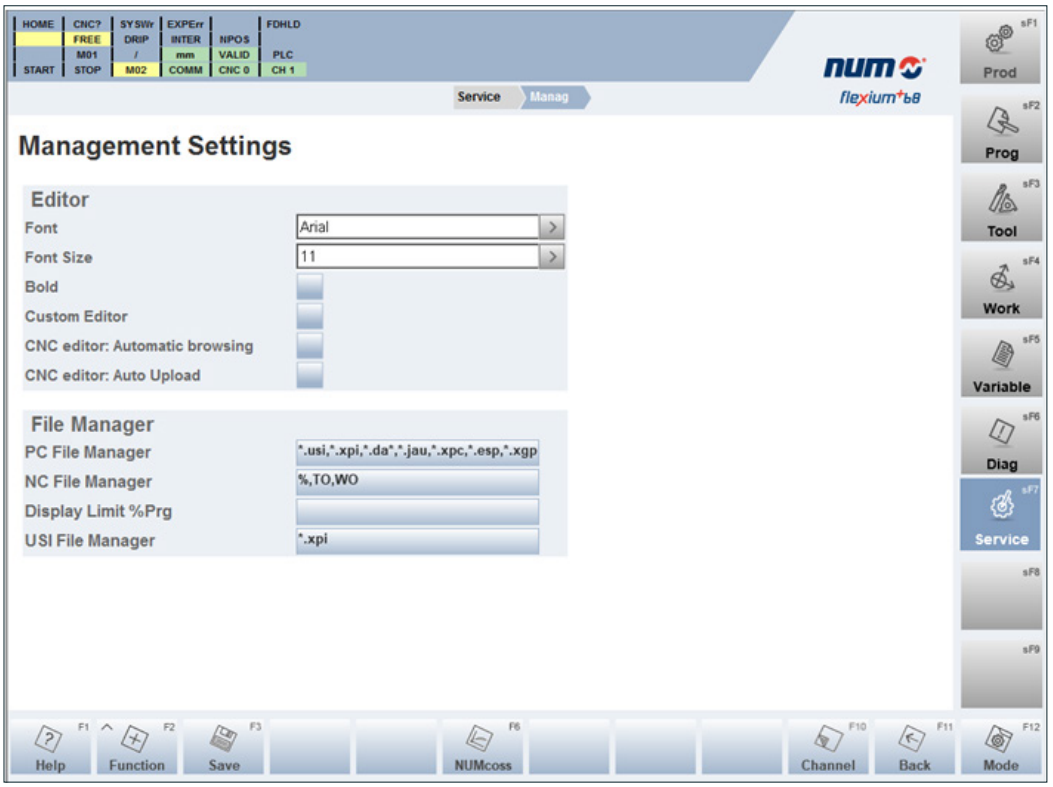
NUMcoss provides a standard set of smoothing configuration parameters for the machine or application. These parameters can be changed inside ISO-program with specific commands. In addition to this the user can work with smoothing parameter dependent the type of the current machining cycle like roughing, finishing and fine finishing.

12.9.1 Access to the NUMcoss Configuration page

Access to NUMcoss parameter settings is in *Service* context *Management* via key F6 ↑ ↓, ← → or End .

On this HMI page all necessary NUMcoss parameter can be pre-defined as default settings.

There are certain commands to modify, disable and enable each parameter inside the part program (see chapter 12.9.3).



12.9.2 NUMcoss Configuration

12.9.2.1 Fine Finishing

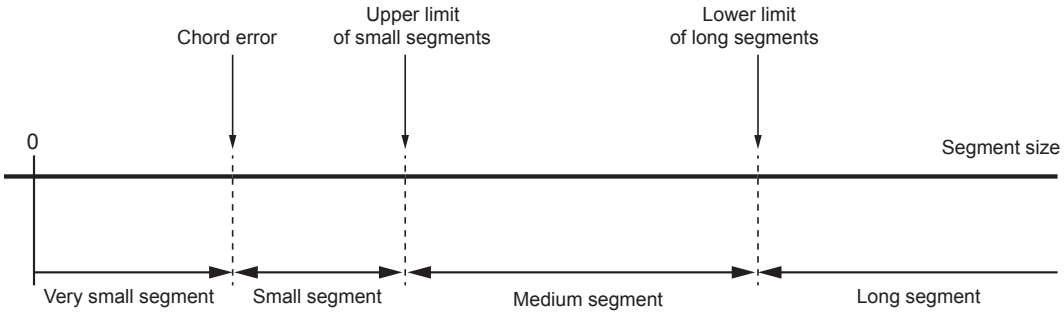
Possible actions

Press F3 to save the settings.
Press F6 to To Load the default settings.
Press F7 to To Save the settings as default.

12.9.2.2 Theoretical Background and Smoothing

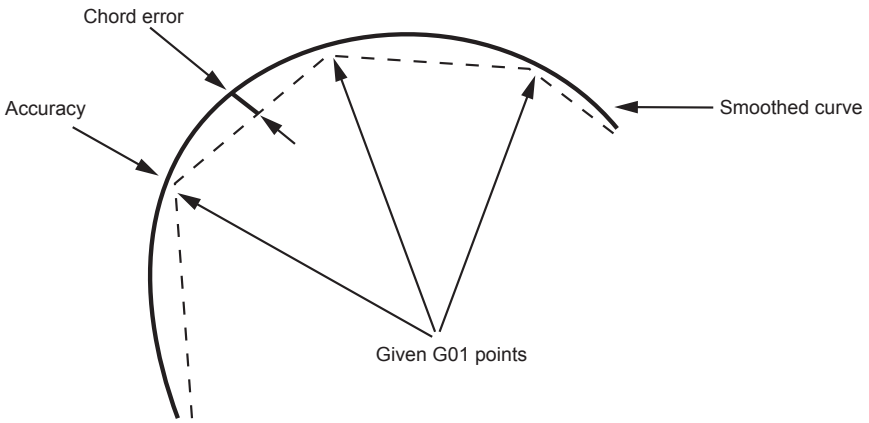
Background
There are three parameter (chord error, upper limit of small segments, and lower limit of big segments), which allow you to classify the segments related to their lengths.

Please refer to the picture below.



Smoothing in general
Smoothing is an approximation of a less sequence of polynomial data instead of a huge number of linear data (G01). This approximation generates 2 types of geometric errors.
- Accuracy
- Chord error.

Please refer to the picture below.



Possible actions

Press F3 to save the settings.
Press F6 to To Load the default settings.
Press F7 to To Save the settings as default.

12.9.2.3 Explanation of all NUMcoss Settings

Geometric and Smoothing Parameter	
Segment size (small)	Sometimes the points are only a few hundredths of a millimetres apart. To avoid inaccuracies in the calculation of such segments, they are regarded as “not essential” in the algorithm. They are nevertheless verified and limited by the accuracy module in the software (dimension = mm).
Segment size (big)	To avoid the risk of distortion in flat surfaces, long segments will never be included in the generated polynomials. (dimension=mm).
Accuracy linear axes	The accuracy / tolerance of linear axes is the approximation error on the programmed point (dimension=mm).
Chord error linear axes (max.)	The maximum chord error of linear axes is the generated error when approaching between the center of the initial segment and the (approximated) curve (dimension=mm).
Accuracy rotary axes	The accuracy / tolerance of rotary axes is the approximation error on the programmed point (dimension=degree).
Chord error rotary axes (max.)	The maximum chord error of rotary axes is the generated error when approaching between the center of the initial segment and the (approximated) curve (dimension=degree).
Rotary axes (existing)	Define rotary axes (e.g. A, B and/or C axis), if the kinematic of your machine is equipped with rotary axes.
Modulo axes (existing)	Define Modulo axes (e.g. A, B and/or C axis), if the kinematic of your machine is equipped with rotary axes defined as Modulo.

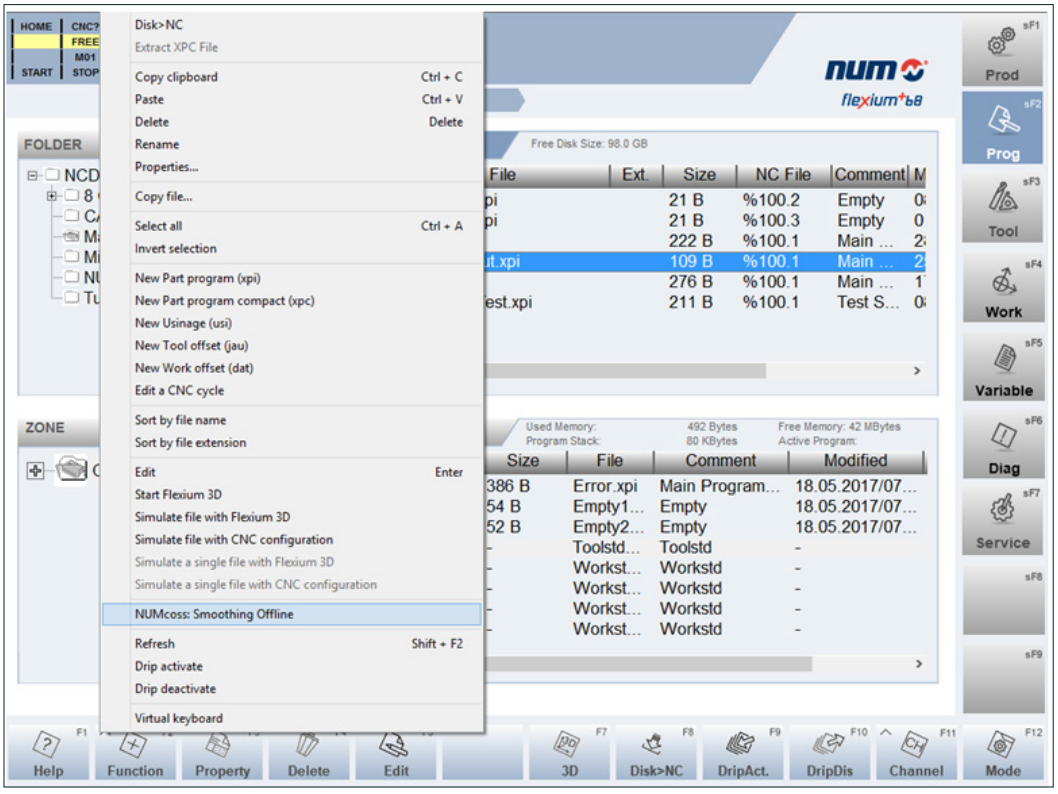
General Geometric Parameter	
Length ratio neighbouring blocks	0 = Inactive 1 = 0 Limit ratio for consecutive blocks.
Angle ratio block transition	0 = Inactive 1 = 0 Limit ratio angle change.
Curvature ratio at polynomial limits	0 = Inactive 1 = 0 Limit ratio curvature change.
Activate inflection point analysis	Checkbox Inflection point analysis.
Activate step elimination	Checkbox Step elimination analysis.
Coefficient erase area	Coefficient factor for step length.
Angle minimum	Minimum angle in degrees.
Polynomial generation at G1	Convert G1 in Polynomial data checkbox.
Polynomial generation at G2/G3	Convert G2/G3 in Polynomial data checkbox.

Optional Parameter	
Maintain line number	If this option is activated, the order of the block numbers in the part program is retained and the respective blocks are NOT smoothed. They are reproduced identically in the output file.
Extended diagnostic	While an ISO file is being analysed, this option also allows the generation of file information regarding the geometry analysis of the segments (such as the angle between two consecutive segments, the length of the segments, inflection points). In the “smoothing” of a part program, this option also allows the output of a smoothing report at the end of the part program.
Syntax check	Generally this smoothing software recognizes only syntactic forms of the most frequently used commands. If this option is activated, the execution of the processing occurs at the beginning of each character and is interrupted if an unknown Syntactic form occurs in the part program (analogous to the NCK).
File end with X-off	This option is intended only to allow online sending of part programs to NCK (through the serial port of previous CNC generations). Currently not supported.

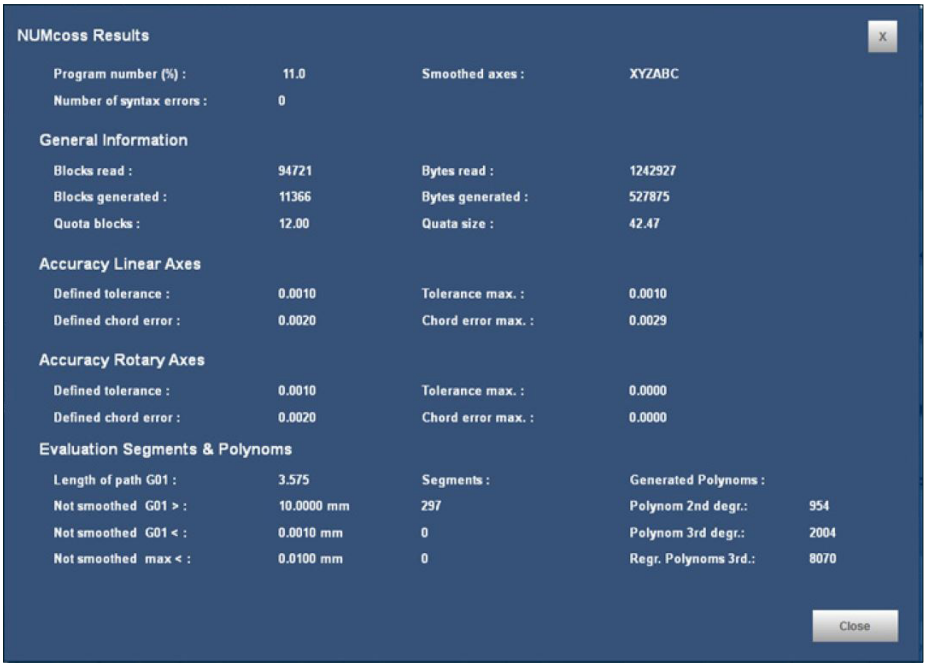
Optional Parameter	
Activating lookahead dripfeed-mode	Checkbox. Not yet supported (Needed for online transformation).
L-Variable display lookahead	Not yet supported (Needed for online transformation) L variable number.
Size of lookahead (G1-blocks)	Not yet supported (Needed for online transformation) Evaluation range (number of NC-Blocks for lookahead).
Output format: linear axes	Number of digits after decimal point for linear position values in output file.
Output format: rotary axes	Number of digits after decimal point for rotary position values in output file.

12.9.3 NUMcoss in the Context Programming

Select an *xpi file* and press F2 to open the context menu and select *NUMcoss: Smoothing Offline*.



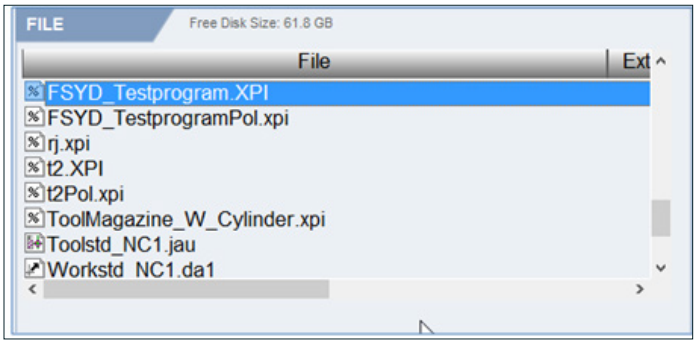
If the part program is successfully converted, this NUMcoss result dialog is displayed with current data regarding input and output file.
After closing this dialog the filename will be extended x *"Pol"* and part program with polynomial data is stored on the same PP folder.



The result dialog is logical grouped. The operator receives information for:

- CNC part program number, involved axes and number of detected syntax errors.
- File information regarding file size in bytes and blocks and the changing quota.
- Accuracy and chord error: predefined and after analysis and transformation.
- Total path length in meter of converted trajectory.
- Listing of generated and classified polynomials.

PcFolder view with already converted part programs



12.9.4 Programmable NUMcoss parameter

The Mnemonic decides which parameter the programmer wants to change.

Programmable NUMcoss Smoothing Parameter	Mnemonic	Default-Value
Segment size (small) (mm)	SB	0.1
Segment size big (mm)	LB	10.0
Accuracy linear axes (mm)	T	0.0015
Chord error linear axes (mm)	C	0.01
Accuracy rotary axes (degree)	TR	0.0015
Chord error rotary axes (degree)	CR	0.01
Activate Inflection point analysis (0: inactive)	DI	1
Length ratio neighboring blocks (0: inactive)	DL	3
Angle ratio block transition (0: inactive)	DR	2
File end with XOFF (0: No)	EOF	0
Curvature ratio at polynomial limits	CV	2
Filtration of small segments	EL	1
Coefficient for elimination area	KEL	400
Angle minimum for elimination	AEL	10
Maintain Block numbers (0: No)	NSC	0
Syntax check (0: No)	NSY	0
List of rotatory axes	LR	0
List of modulo axes	LM	0

The values of the column default are oriented of course to your Setup configuration.

Programming procedure and remarks

- The comment key *"(//)"* theoretically allows all syntactic forms to be accepted as recognized by the smoothing software immediately. Inserted in the programming header, the comment key allows the corresponding block to be copied identically into the output stream.

- The separator *("//OFF")* temporarily deactivates the scanning and smoothing of the blocks. All blocks after these separators are then copied into the output stream as a separator without analysis.

- *"(// ON)"* is generally not found in the part program file, except smoothing is deactivated temporarily. For example, this delimiter can be used to disable anti-aliasing in the design phase.

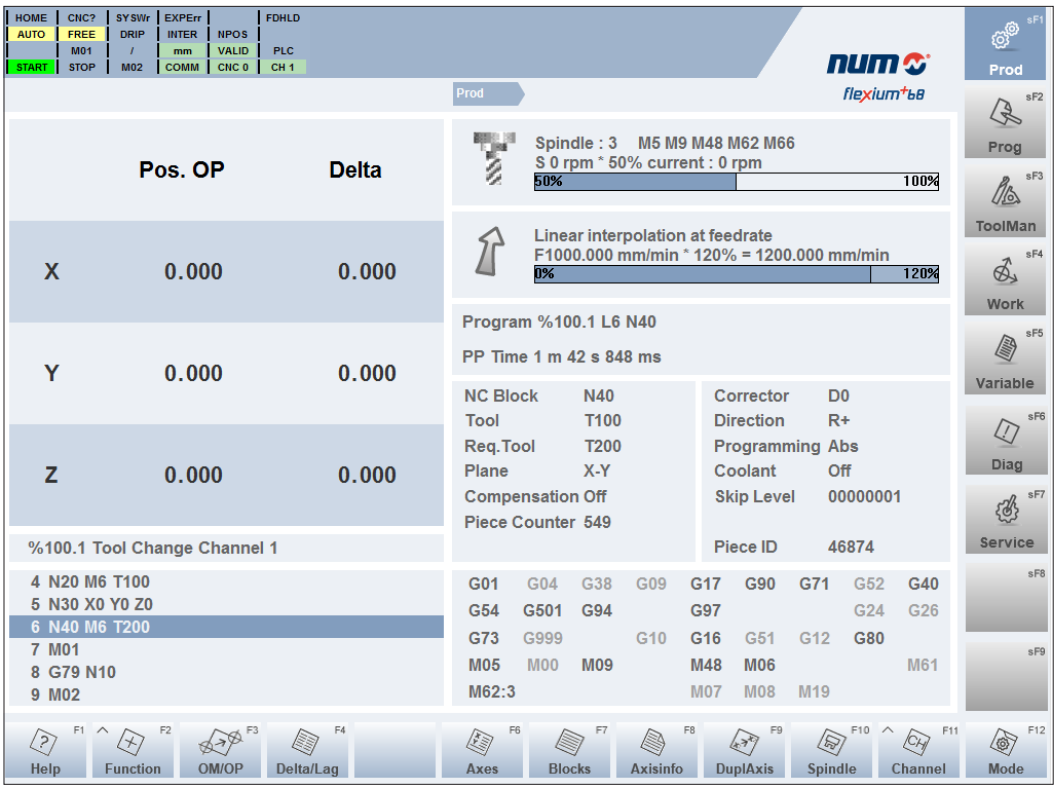
Example

……
(// DL=0 T=0.001 TR=0.001 C=0.002 CR=0.002 SB=0.001)

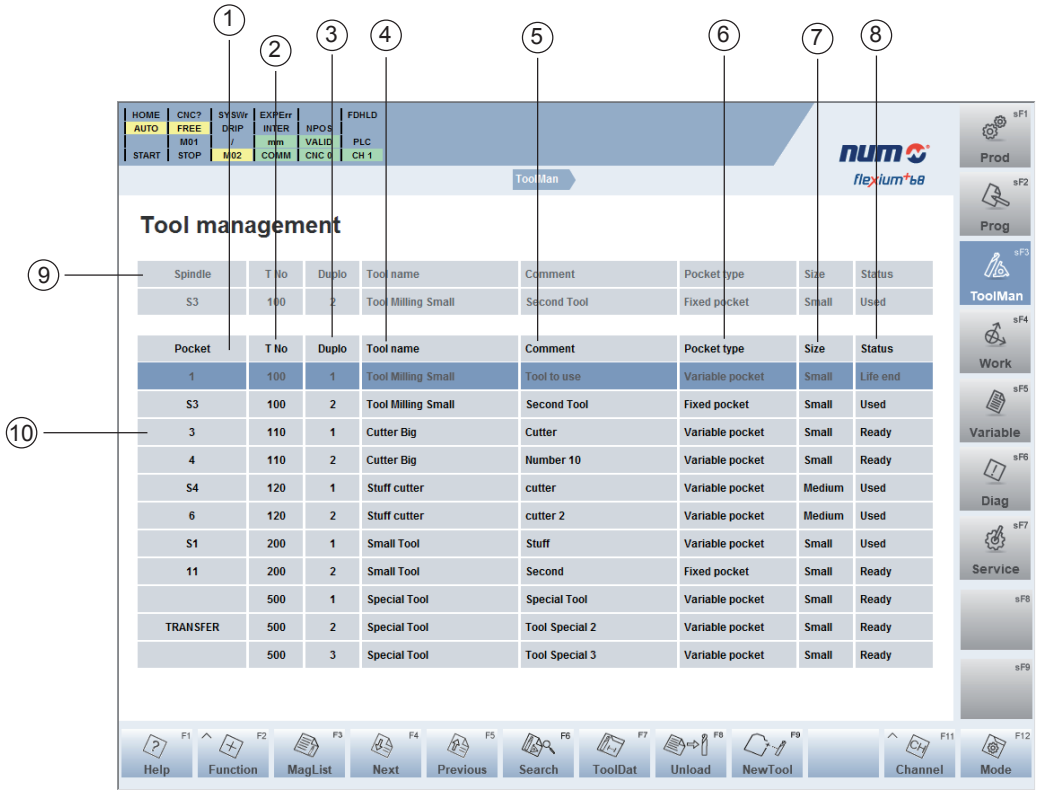
Means:
DL=0: *Length ratio neighbouring blocks deactivated*.
T/TR=0.001: Accuracy and chord error of linear axes = 1 micron.
C/CR=0.002: Accuracy and chord error of rotary axes = 2 micron.
SB=0.001: Upper limit of small segments = 1 micron.

12.10 Tool management

The *Tool management* function is used for connection the Machine with a Tool database.



12.10.1 Tool view



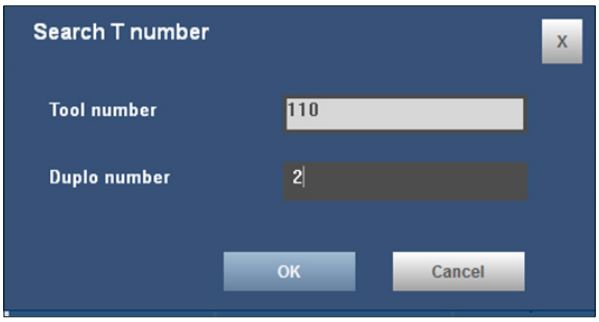
1	Pocket: - Information, where the Tool is loaded - Number: Pocket place in the Magazine - Sxx: Spindle - Buxx: Buffer - Name: The Tool is loaded on another Machine, Name of the Machine.
2	T Number of the Tool.
3	Duplo number of the Tool.
4	The name of the Tool. All Tools with the same T-Number has the same name.
5	A comment of the Tool.
6	Pocket type Variable : the tool is managed in variables positions Fixes: The tool is managed in fixed position.
7	Size: - Small: the Tool use 1 Pocket Place - Medium: The Tool use 1 ½ Pocket Place in all direction - Big: The tool use 2 Pocket place in all direction.
8	Tools status : New, In use, Warning, In error, Broken, ...
9	Display of the Active Spindle.
10	Display of all tools in the database.

Possible actions

Press F2 to enter the function menu.
Press F3 to enter the magazine List.
Press F4 to go on the next Page.
Press F5 to go on the previous Page.
Press F6 to search a tool.
Press F7 to change the tool data.
Press F8 to Load / Unload a tools.
Press F9 to create a new Tool.

12.10.1.1 Search a tool

Press F6 to search a tool. Then the following dialog is open:

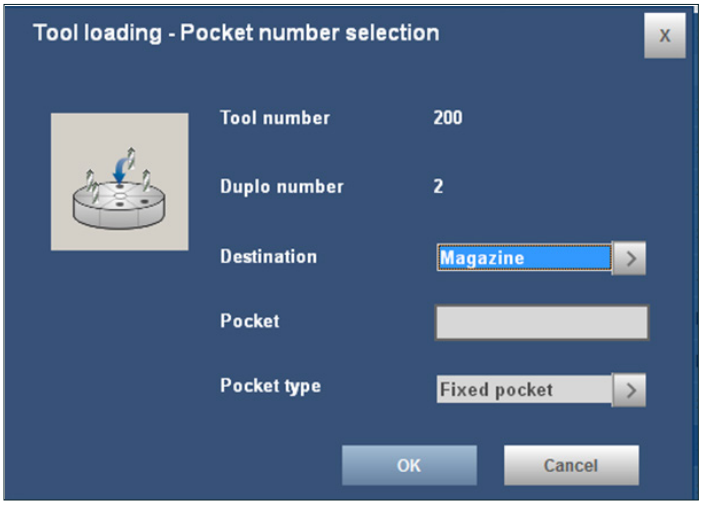


Enter in the Dialog the Tool and Duplo number.
After press OK, the tool will be selected in the list.

12.10.1.2 Load a tool

Select a tool in the list, which is not loaded and press F8.

Then the following dialog is open:



Select the Destination: *Magazine / Buffer / Spindle* and enter the pocket number.

Selected the Pocket type: *Fixed pocket / Variable Pocket*.
With Fixed pocket, the pocket will be reserved for the tool in the magazine, if it in the buffer / Spindle.

12.10.1.3 Unload a tool

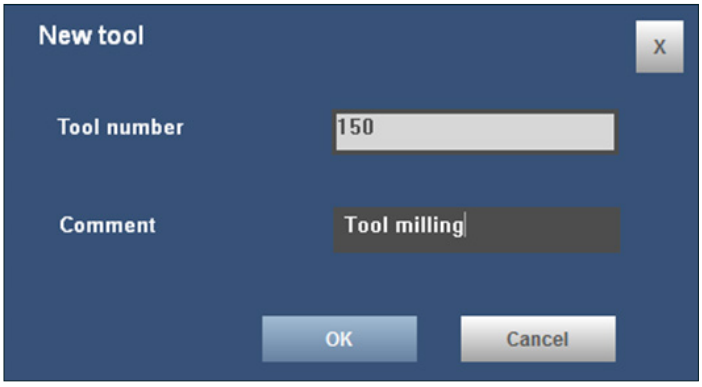
Select a *Loaded Tool* and press and press F8.

After this, the Tool will be unloaded from the *Magazine / Buffer / Spindle*.

12.10.1.4 Create a new tool

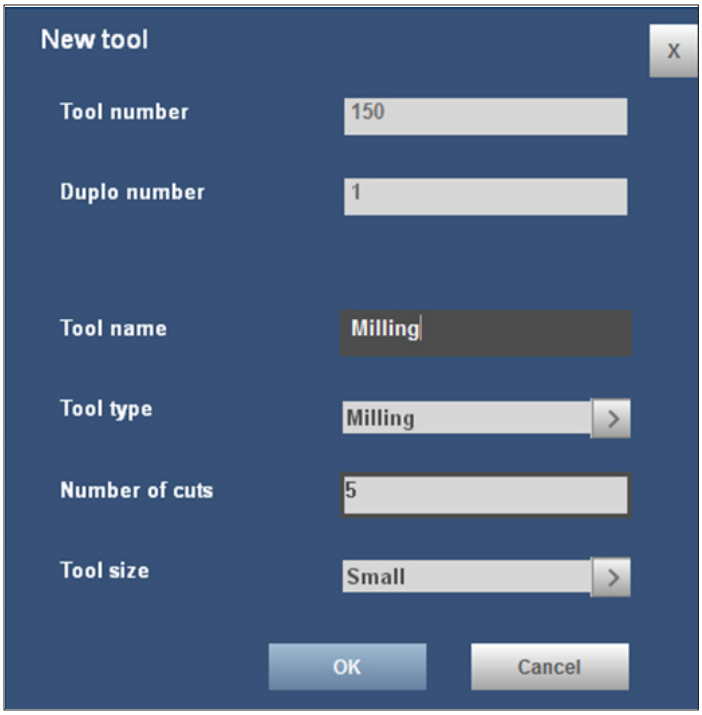
Press F9.

Then the following dialog is open:



Enter the *Tool number* and the *Comment*.
If a Tool with the same *Tool number* is already present, a new Tool will be created with a new *Duplo number* and the same parameter like the existing tool with this Tool number.

If the Tool is not existing, this dialog will be opened:

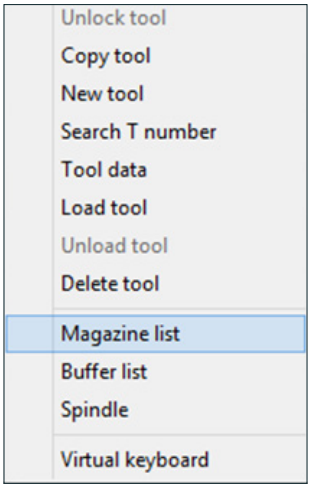


Press OK to create the Tool.

Tool name	Name of the tool.
Tool type	Milling / Turning / Boring.
Number of cuts	Enter the number of Correction that the tool has.
Tool size	Small 1 Pocket, medium 1 ½ Pocket, Big 2 Pockets.

12.10.1.5 Function list menu

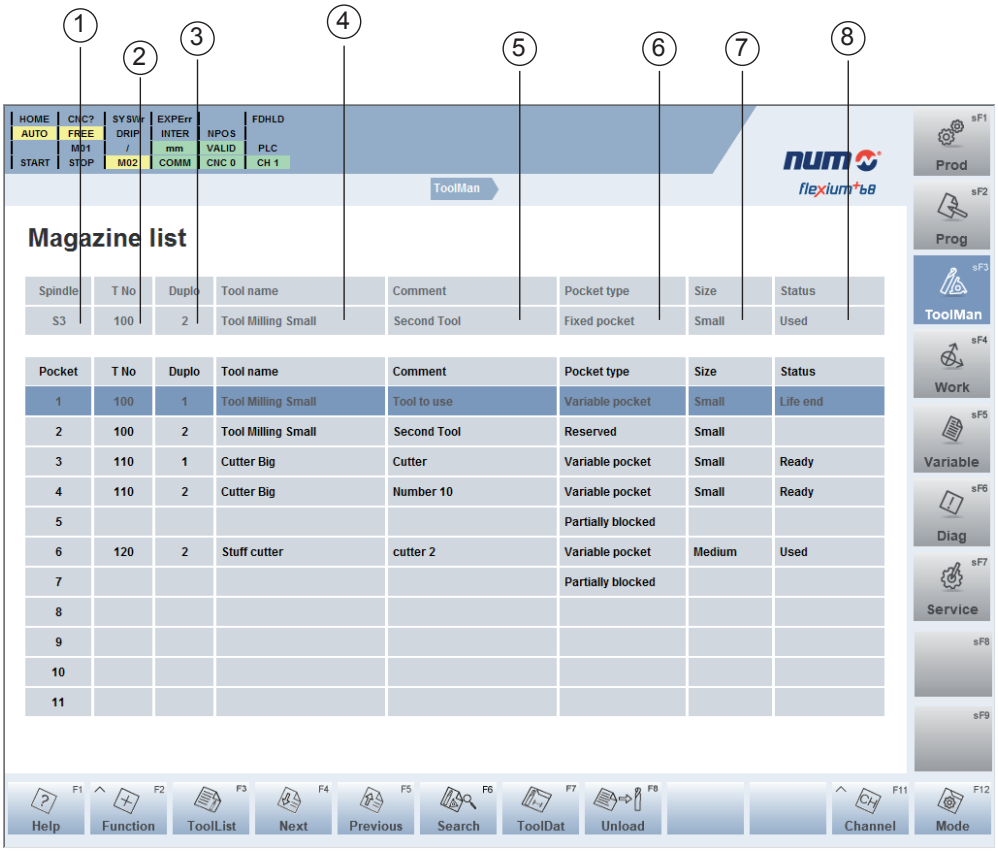
Press F2 to enter the function list menu



Unlock tool	The Tool is marked as loaded, but not loaded in the Magazine. With this function the tool can be unlocked and marked as unloaded.
Copy tool	A new Tool will be created with a new T-Number and the same parameters.
New tool	Create a new tool.
Search T Number	Search a tool with the T number.
Tool data	Enter the Tool data.
Load tool	Load a tool in the Magazine / spindle / buffer.
Delete tool	Delete a tool from the database.
Magazine list	Enter to the Magazine list.
Buffer list	Enter to the buffer list.
Spindle	Enter to the Spindle list.

12.10.2 Magazine View

This page display all tool loaded in the machine in the magazine view.



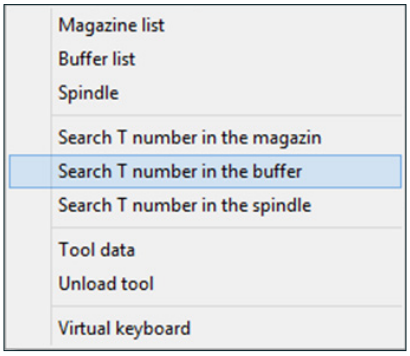
1	Pocket: - Information, where the Tool is loaded - Number: Pocket place in the Magazine - Sxx: Spindle - Buxx: Buffer - Name: The Tool is loaded on another Machine, Name of the Machine.
2	T Number of the Tool.
3	Duplo number of the Tool.
4	The name of the Tool. All Tools with the same T-Number has the same name.
5	A comment of the Tool.
6	Pocket type: - Variable pocket: The Tool has a variable Pocket place. - Fixed pocket: The Tool has a fixed Pocket place. - Partial blocked: A tool with the size medium us this place partial. - Blocked: a tool with the size big use this place.
7	Size: - Small: the Tool use 1 Pocket Place - Medium: The Tool use 1 ½ Pocket Place in all direction - Big: The tool use 2 Pocket place in all direction.
8	Status - Ready: The tool has never been used - Used: The tool has been used already - Life warning: The tool is in status life warning - Life end: The tool I is in the status life end - Blocked: The tool is broken.

Possible actions

Press F2 to enter the function menu.
Press F3 to enter the magazine List.
Press F4 to go on the next Page.
Press F5 to go on the previous Page.
Press F6 to search a tool.
Press F7 to change the tool data.
Press F8 to Unload a tools.

12.10.2.1 Function list menu

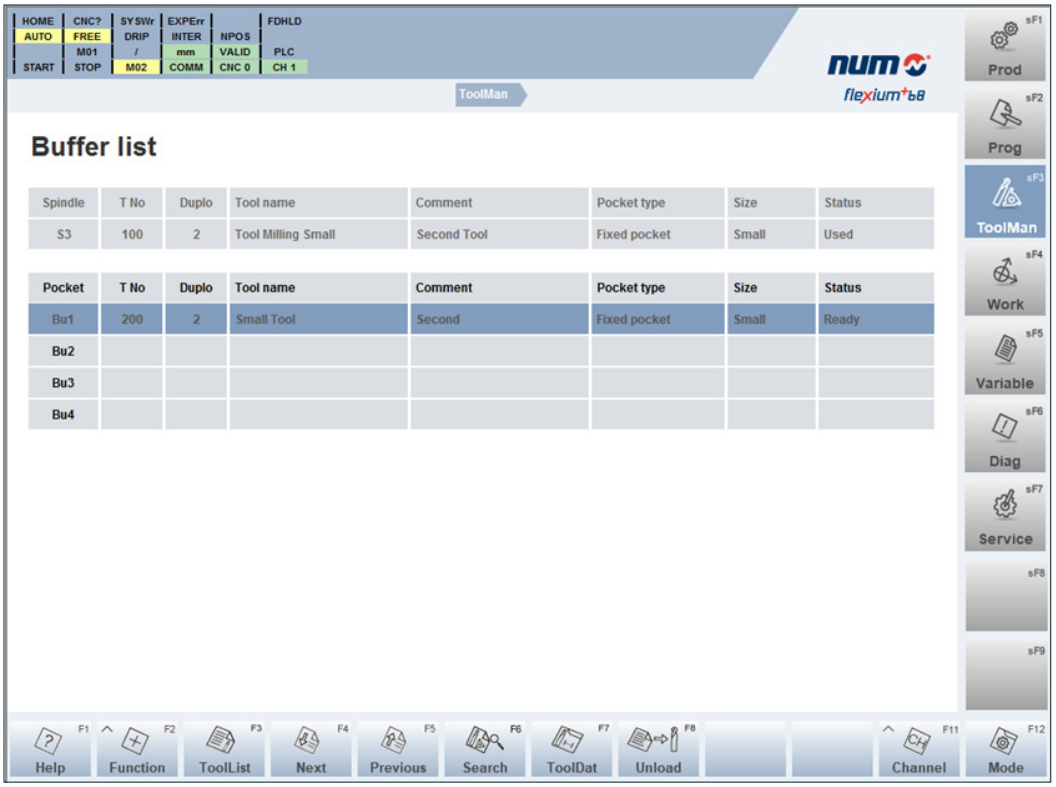
Press F2 to enter the function list menu.



Magazine list	Change to the Magazine list.
Buffer list	Change to the Buffer list.
Spindle	Change to the Spindle list.
Search T Number in the magazine	Search a tool with the T number in the Magazine.
Search T Number in the buffer	Search a tool with the T number in the buffer.
Search T Number in the spindle	Search a tool with the T number in the spindle.
Tool data	Enter the Tool data.
Unload tool	Unload a tool from the machine.

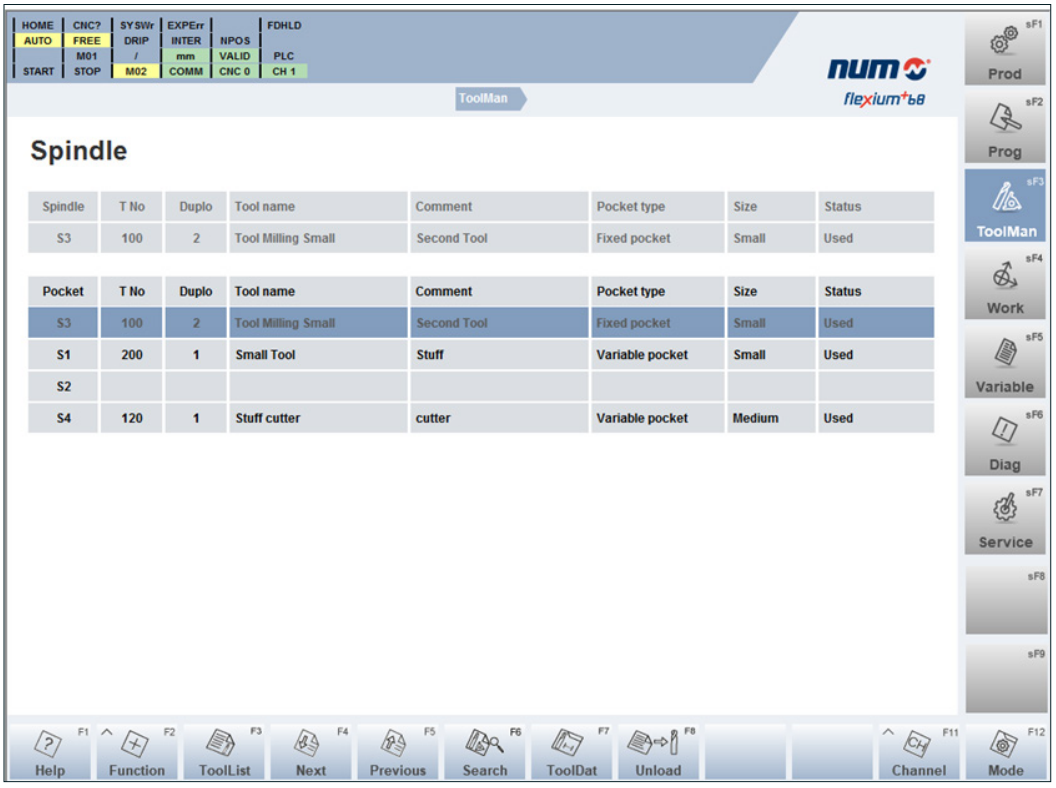
12.10.2.2 Buffer View

This page displays the Buffer if the View is selected as Buffer View.



12.10.2.3 Spindle View

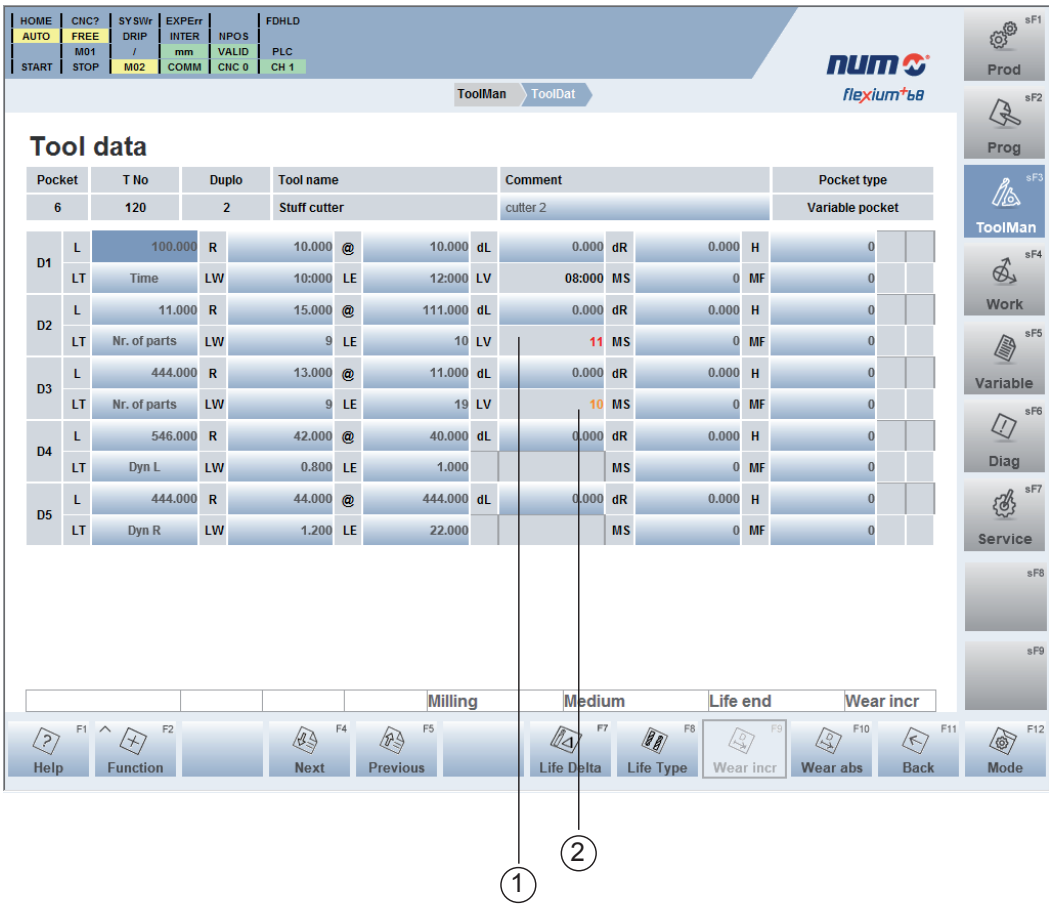
This page displays the Spindle if the View is selected as Spindle View.



12.10.3 Tool data View

This page allows to edit the tool parameter.

Press F7 in the magazine view or tool list to enter this page.



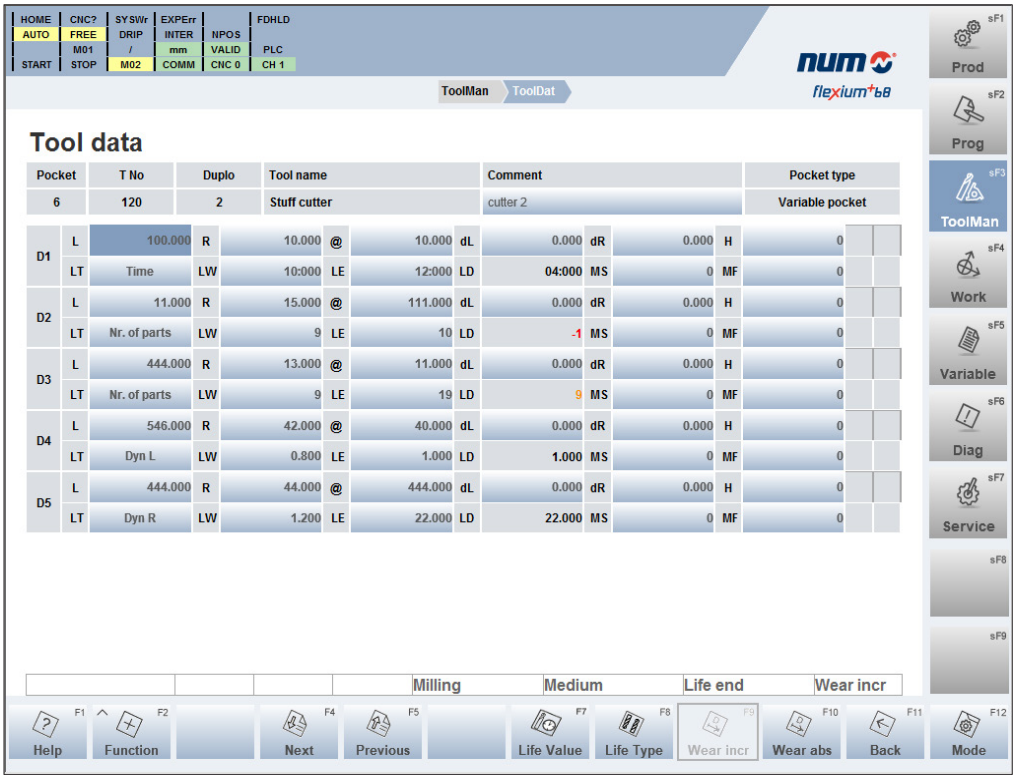
1	Cut in Error LV > LE The Life value is displayed red.
2	Cut in Warning LV > LW and LV < LE The Life value is displayed orange.

Description of the special Tool manager Parameter	
LT	- Time: machining time of the tool. - Nr. of parts: number of programming of this cut. - Dyn L: Limit of the value ABS(dL). - Dyn R: Limit of the value ABS(dR).
LW	Limit life Warning.
LE	Limit life Error.
LV	Count of the current life Value.
MS	Max spindle speed.
MF	Max feed.

Possible actions

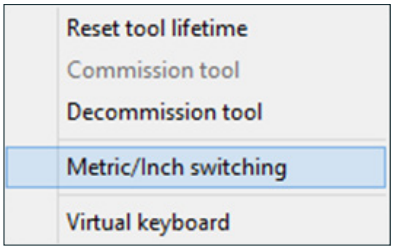
Press F2 to enter the function menu.
Press F4 to go on the next Page.
Press F5 to go on the previous Page.
Press F7 toggle between life Delta / Life value
Press F8 to change the life type.

If you toggle to Life Delta F7, the value of the Life Value is displayed as Life Delta.
This is the remain value, until the tool life is reached.



12.10.3.1 Function list menu

Press F2 to enter the function list menu.



Reset tool lifetime	The LV, dR, dL value are reset to 0.
Commission tool	Set the tool status from blocked in used.
Decommission tool	Set the tool status to blocked.

Page deliberately empty

Page deliberately empty

Summary

13 Review.....335

13.1 System Overview..... 335
13.1.1 Operating Mode 335
13.1.2 Defining a Program..... 335
13.1.3 Creating a Program 336

13.2 Review of Program Coordinate Systems.....337
13.2.1 Review of Axes Definition and Direction 337

13.3 Machine Overview..... 338
13.3.1 General..... 338
13.3.2 Turning and Milling Machines..... 338

13.4 Turning Machine - Specific Features of Front and Rear Turrets..... 339
13.4.1 View of a Turning Machine with Rear Turret..... 339
13.4.2 View of a Turning Machine with Front Turret..... 339

13.5 Definition of Travels and Origins..... 340
13.5.1 Homing with Incremental or Combined Encoders Using Absolute Dimensions 340
13.5.2 Schematic Representation of Travels (Turning and Milling machine) 341

13.6 Offset Definitions 342

13.7 Turning Machine - Offsets Represented on the Z and X Axes..... 343
13.7.1 Offsets on the Z Axis (Solution 1 with DAT2) 343
13.7.2 Offsets on the Z Axis (Solution 2 with DAT2) 343
13.7.3 Offsets on the X Axis (Solution 1 with DAT2) 344
13.7.4 Offsets on the X Axis (Solution 2 without DAT2) 344
13.7.5 Schematic Representation of the Offsets on X and Z 345

13.8 Milling machine - Offsets Represented on the Z, X and Y Axes 346
13.8.1 Offsets on the Z axis..... 346
13.8.2 Offsets on the X axis 346
13.8.3 Offsets on the Y axis 347
13.8.4 Schematic Representation of All Offsets 348
13.8.5 Special Case of Milling Machines Equipped with Rotary Tables..... 349

13.9 Tool Axis Orientation 351
13.9.1 Using the G16 P+ function on turning or milling machine 352

13.10 Definition of Tool Dimensions and Offsets 353
13.10.1 Turning Machine - Dimensions, Tool Tip Radius and Orientation 353
13.10.2 Tool Tip Radius and Orientation 354
13.10.3 Milling Machine - Definition of Tool Dimensions 355

13.11 Definition of Tool Wear 356
13.11.1 Turning Machine 356
13.11.2 Milling Machine.....357

Page deliberately empty

13 Review

13.1 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 ‘13.6 Offset Definitions’ the aim is to define the offsets and corresponding origins or zero points rather than give a method for measuring the offsets.

13.1.1 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.).

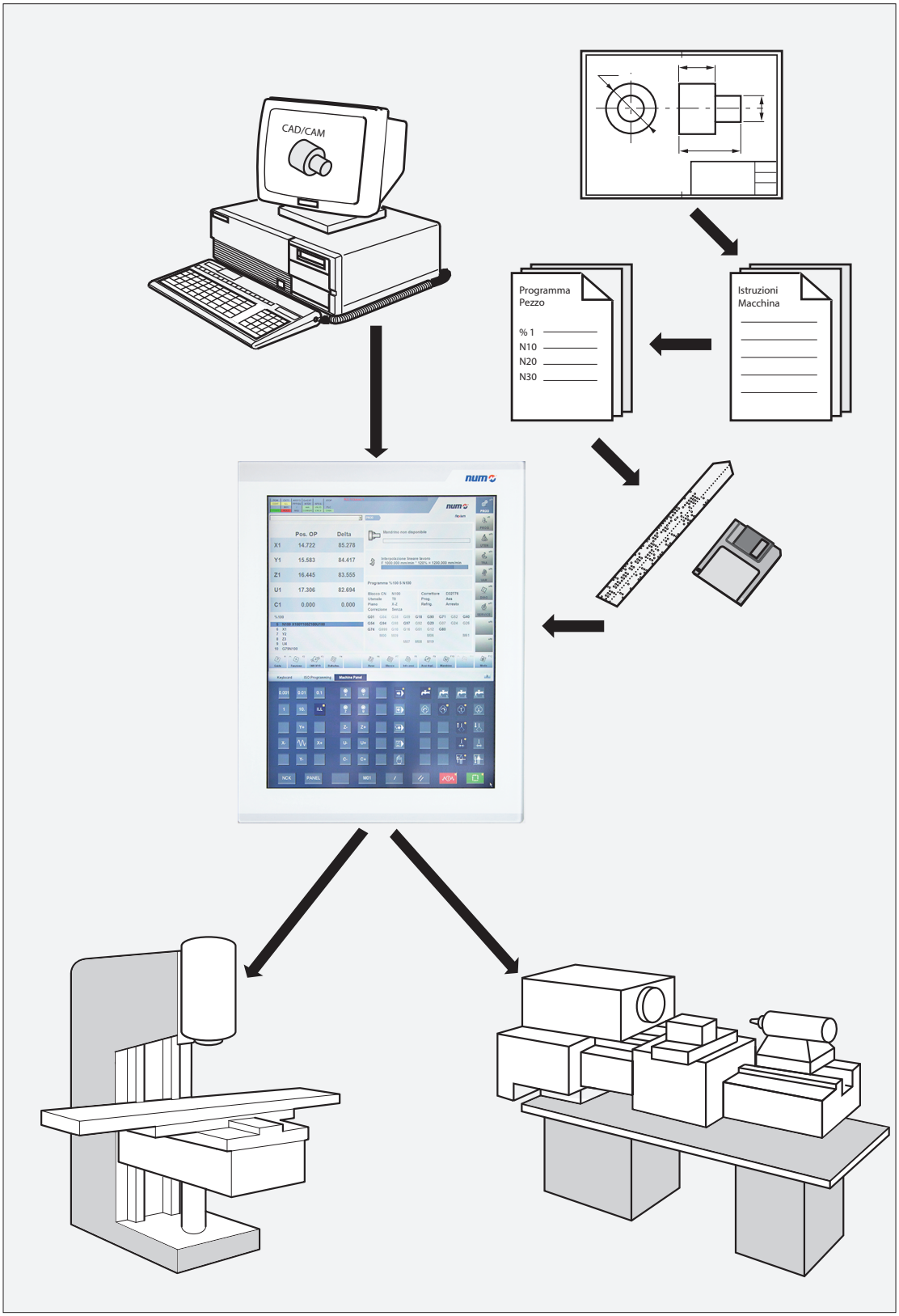
13.1.2 Defining a Program

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.

13.1.3 Creating a Program

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

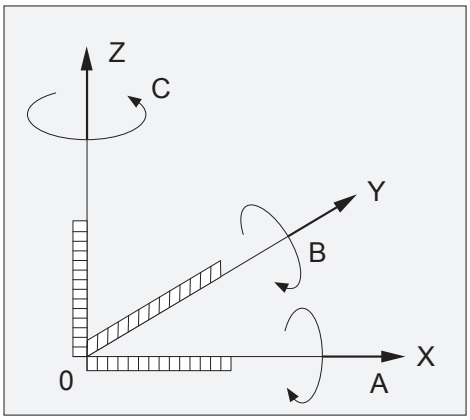


13.2 Review of Program Coordinate Systems

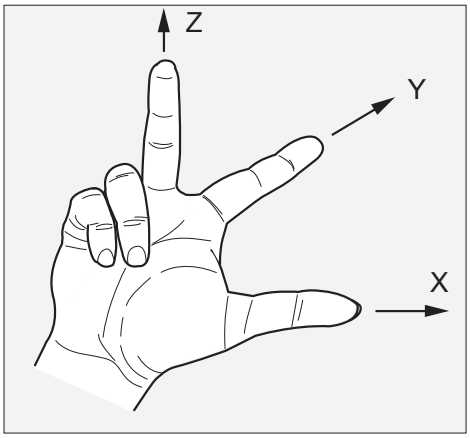
13.2.1 Review of Axes Definition and Direction

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

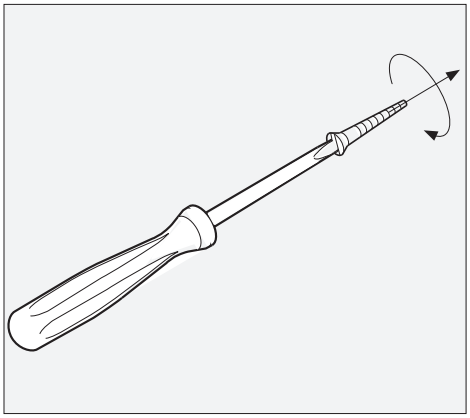
A rectangular Cartesian coordinate system is a direct three-axes system of three linear axes, X, Y and Z, with which are associated three rotary axes, A, B and C.



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.



13.3 Machine Overview

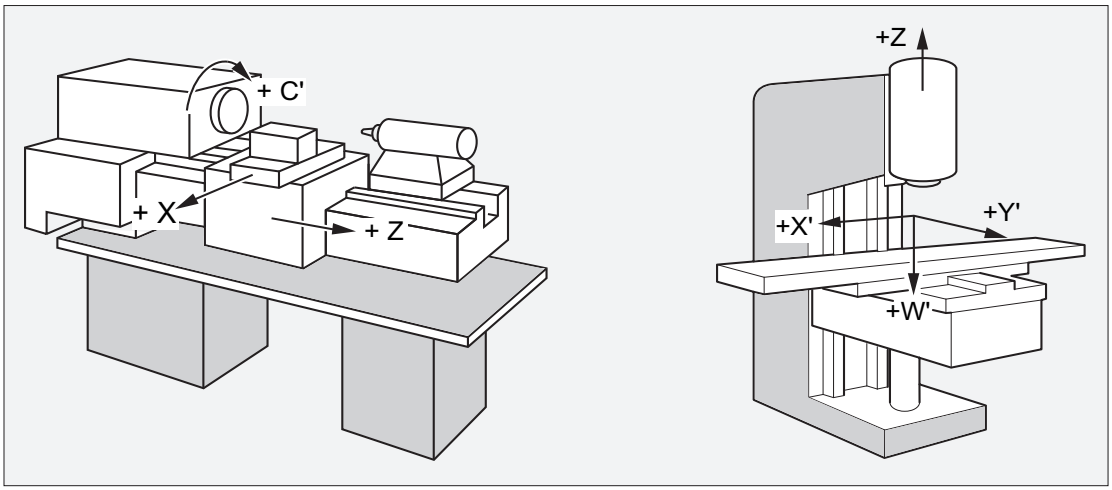
13.3.1 General

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.

13.3.2 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.

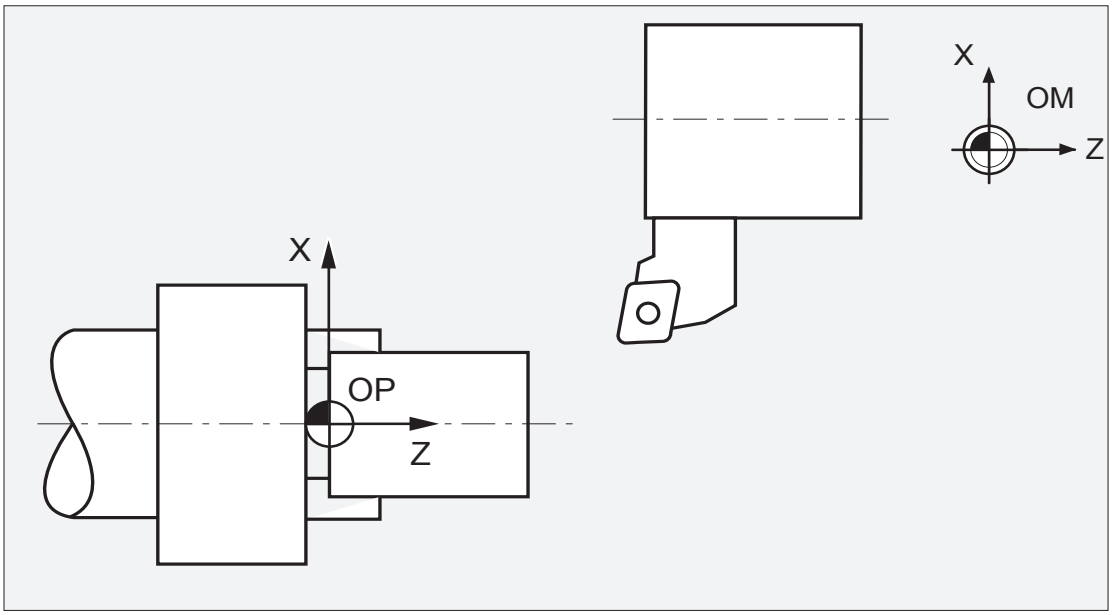
Milling machine
For a milling machine:
- The Z axis is the axis of the main spindle when this axis is parallel to one of the slide ways,
- Positive movement on the Z axis increases the distance between the part and tool,
- The X axis is perpendicular to the Z axis and corresponds to the largest excursion,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

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

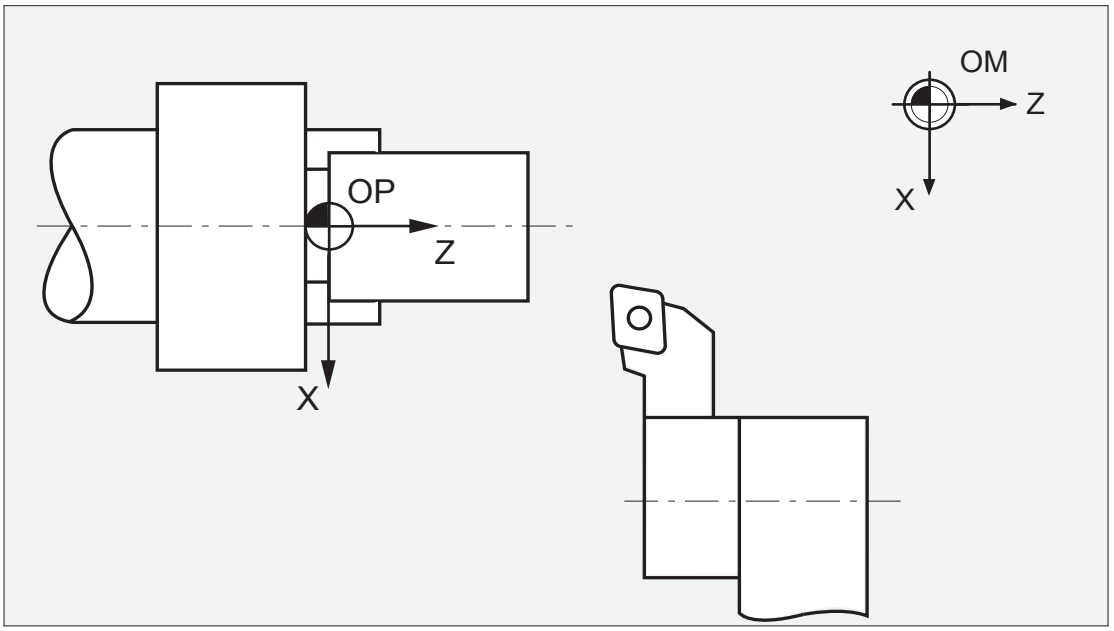
13.4 Turning Machine - Specific Features of Front and Rear Turrets

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 ‘13.10.1 Turning Machine - Dimensions, Tool Tip Radius and Orientation’).

13.4.1 View of a Turning Machine with Rear Turret



13.4.2 View of a Turning Machine with Front Turret



13.5 Definition of Travels and Origins

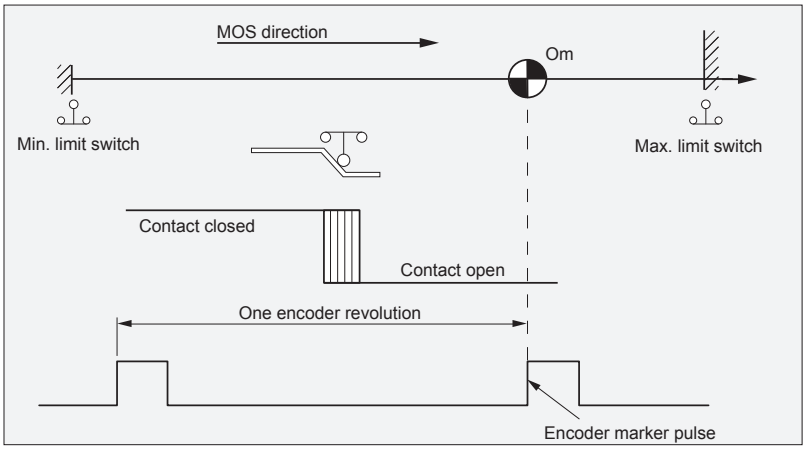
All the CNC movements are defined with respect to the measurement origin or zero point of the machine.
When the system is turned on, it does not know the measurement origin. The mechanical travel on each machine axis is limited by maximum and minimum limit switches.

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).

13.5.1 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.

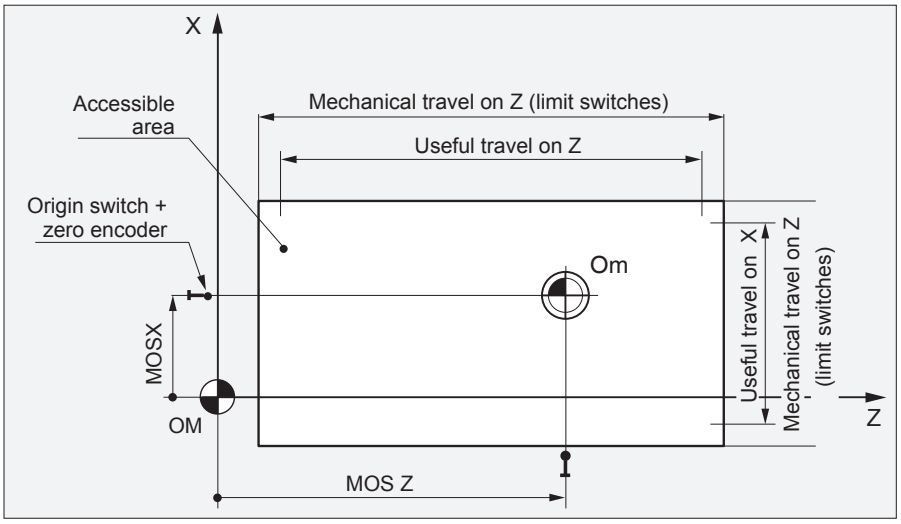
13.5.2 Schematic Representation of Travels (Turning and Milling machine)

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).

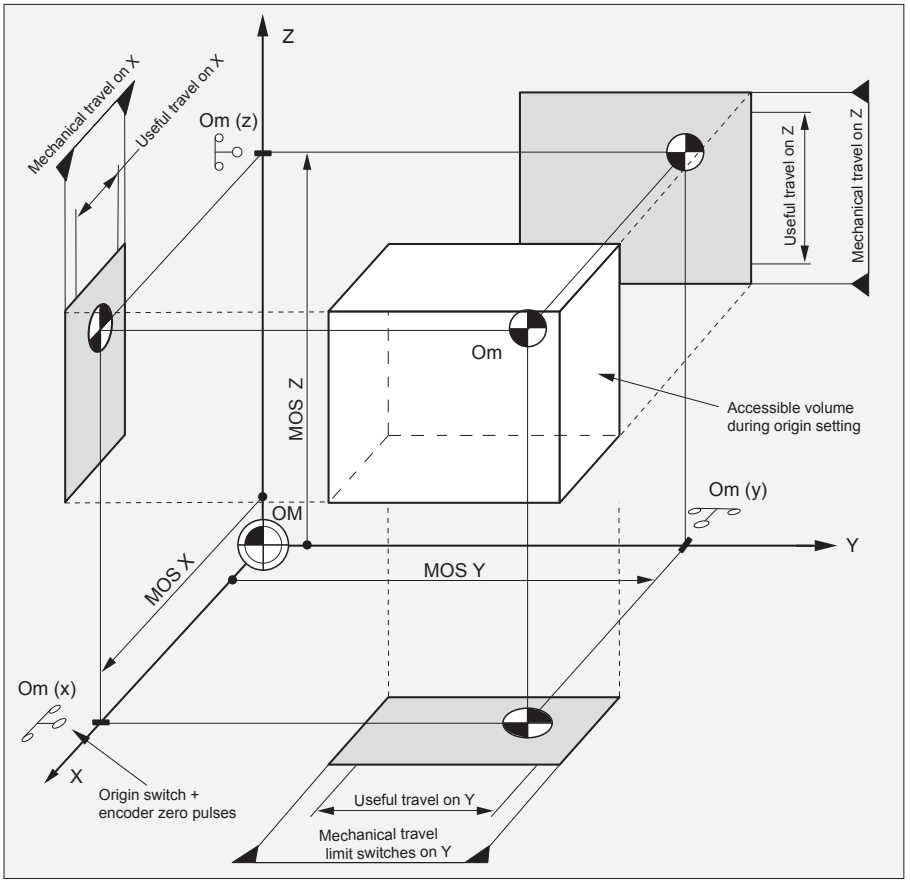
Measurement origin offset (OM/Om) = MOS

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

Turning machine



Milling machine



13.6 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.

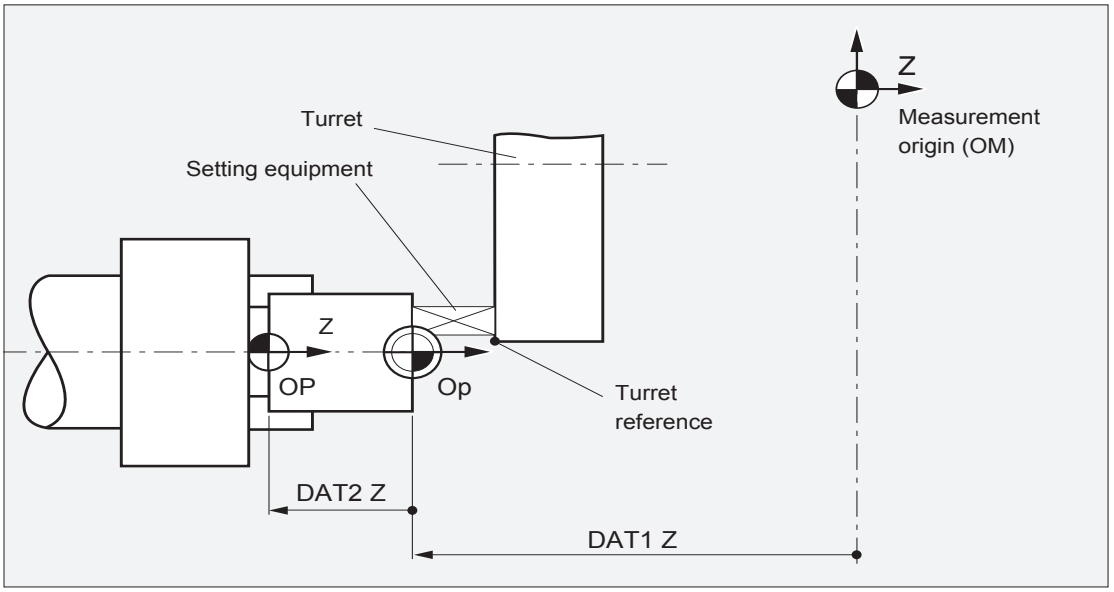
Part origin offset (Op/OM) = DAT1

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

Program origin offset (OP/Op) = DAT2

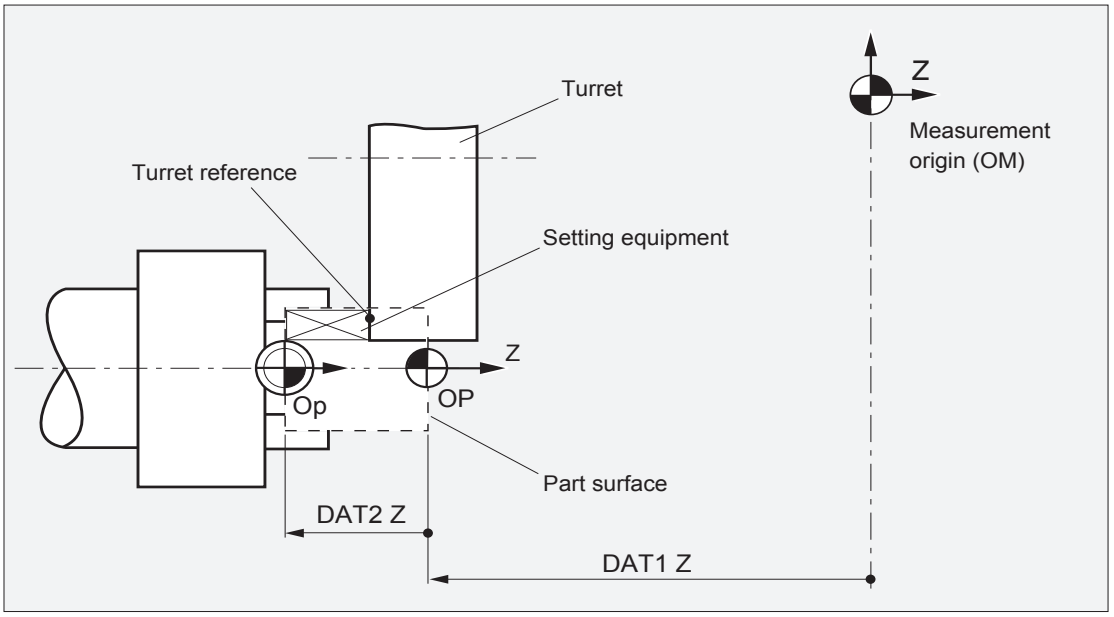
13.7 Turning Machine - Offsets Represented on the Z and X Axes

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

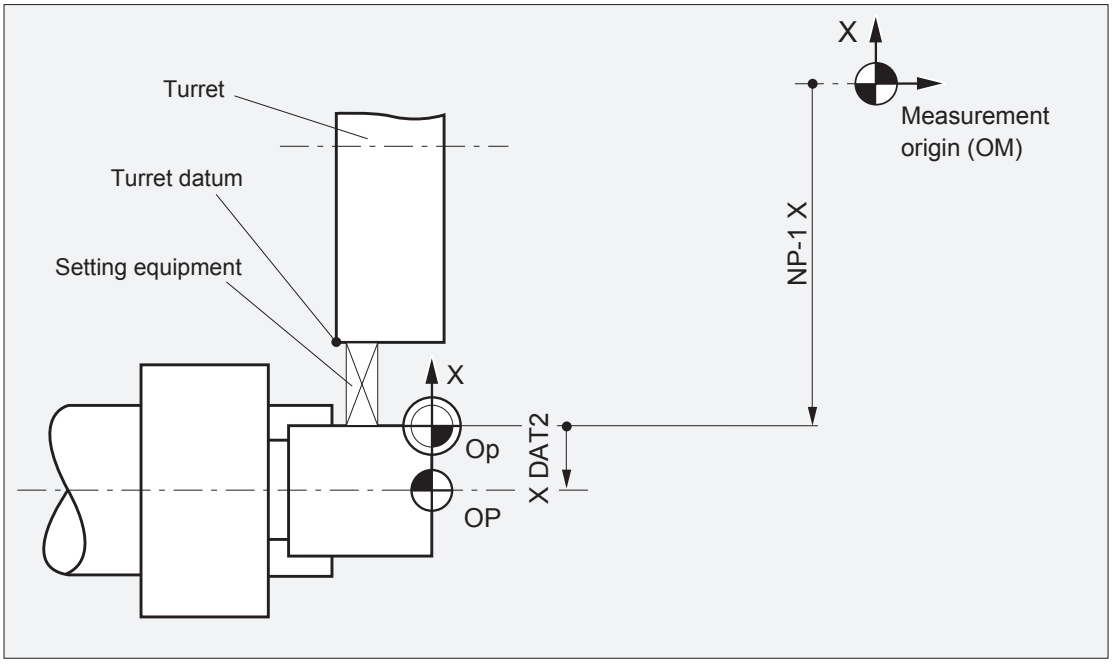


13.7.2 Offsets on the Z Axis (Solution 2 with DAT2)

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

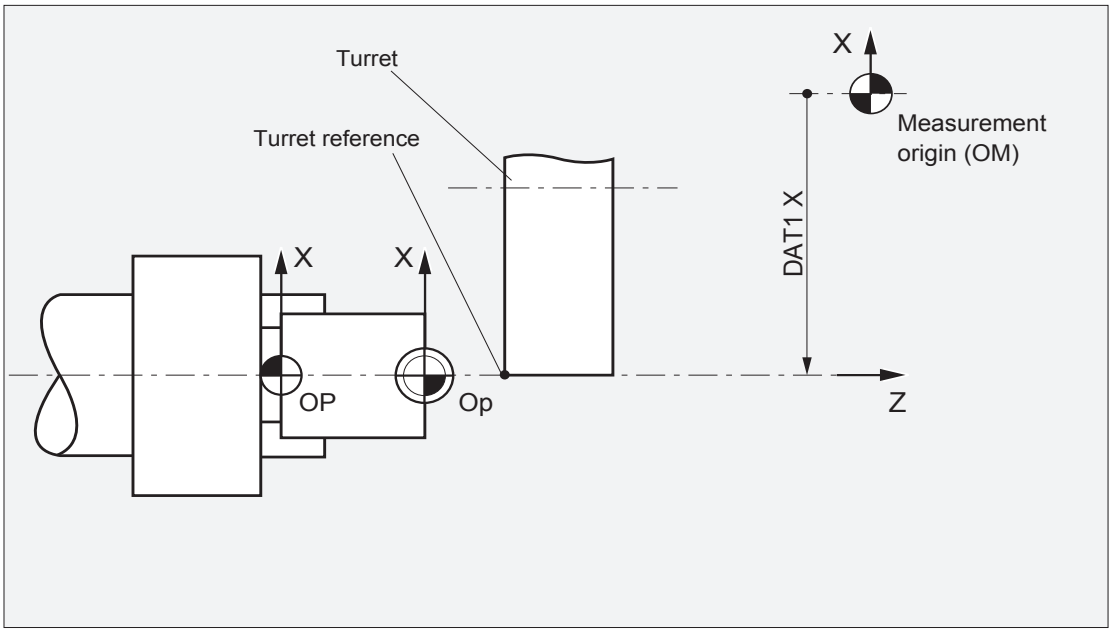


13.7.3 Offsets on the X Axis (Solution 1 with DAT2)



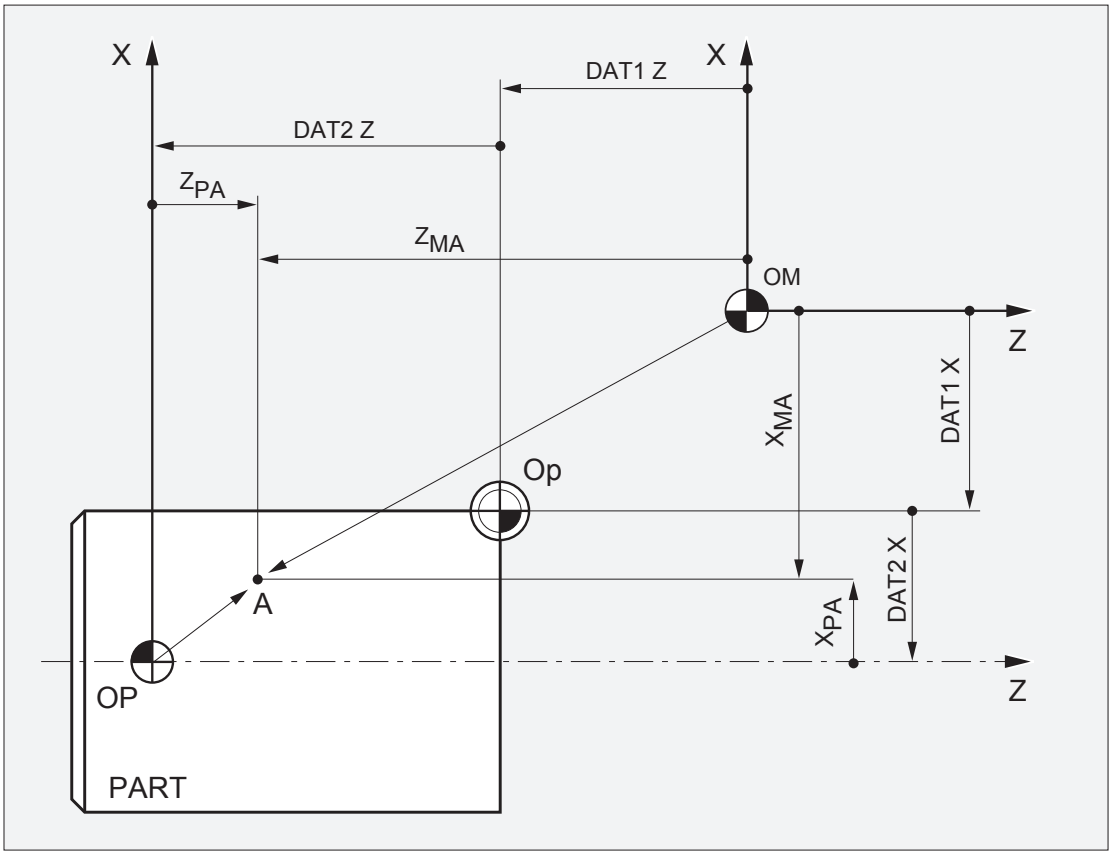
13.7.4 Offsets on the X Axis (Solution 2 without DAT2)

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



13.7.5 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):

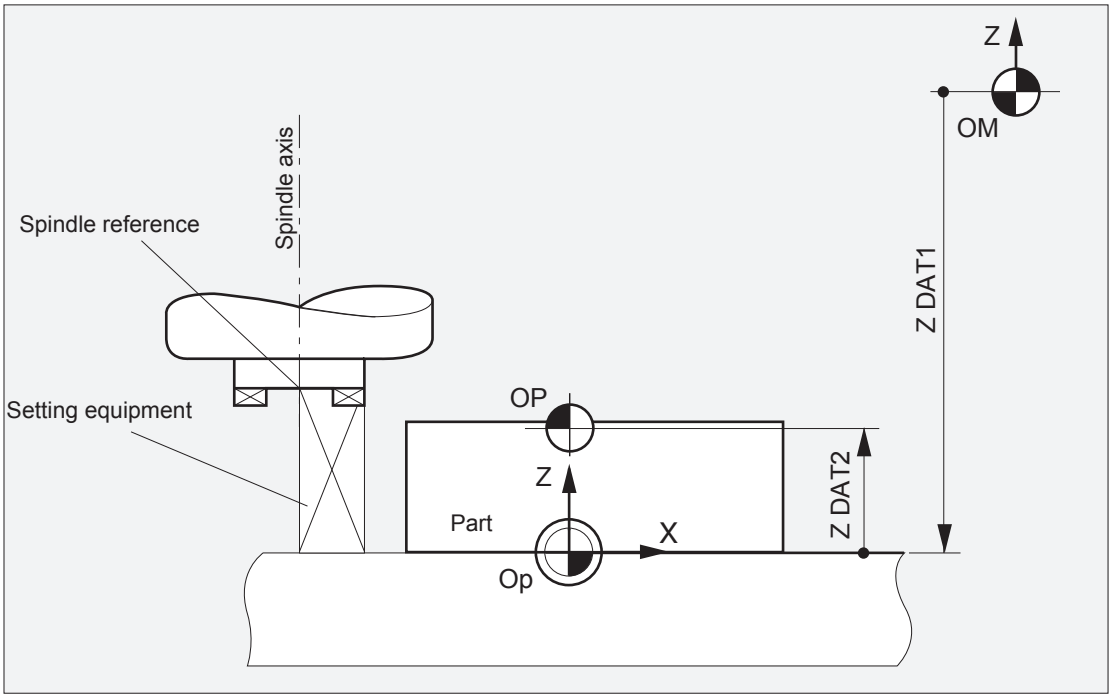


Program dimensions (with respect to OP)	Measurement dimensions (with respect to OM)
X_{PA}	$X_{MA} = X_{PA}$ + DAT1 X + DAT2 X
Z_{PA}	$Z_{MA} = Z_{PA}$ + DAT1 Z + DAT2 Z

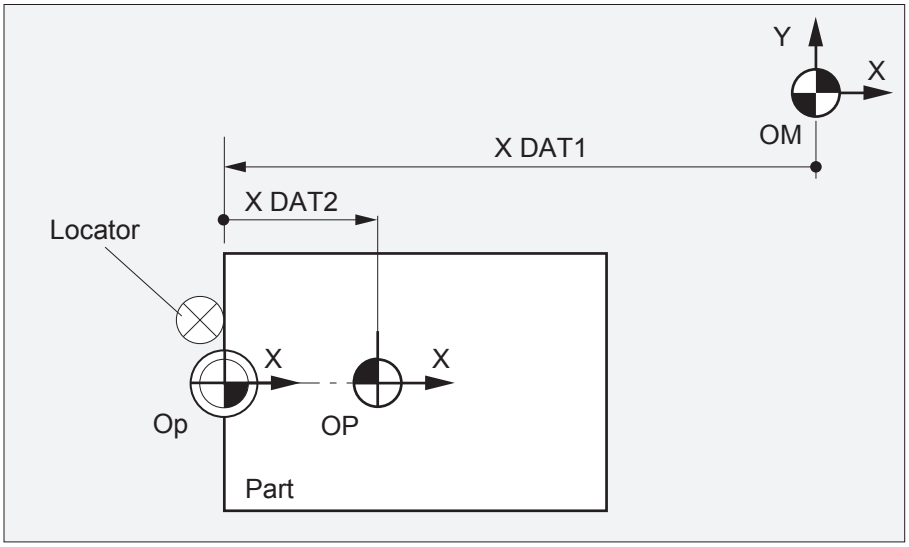
The dimensions are algebraic values.
Programmed shifts can be added to the measurement dimensions.

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

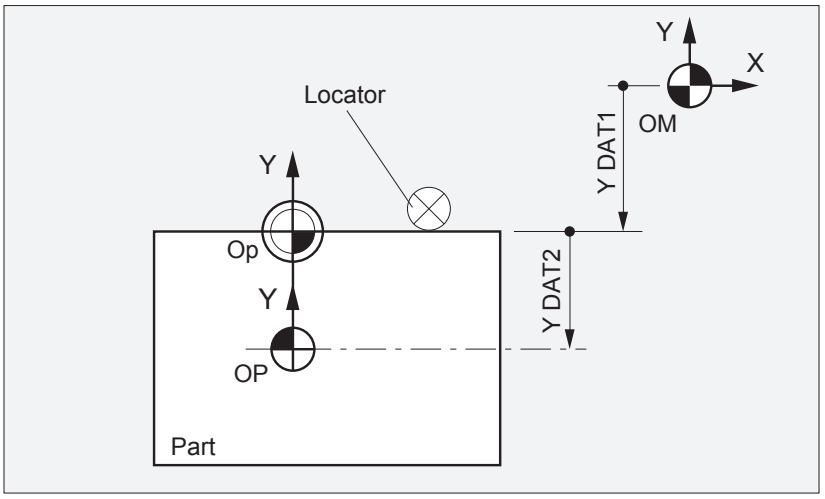
13.8.1 Offsets on the Z axis



13.8.2 Offsets on the X axis

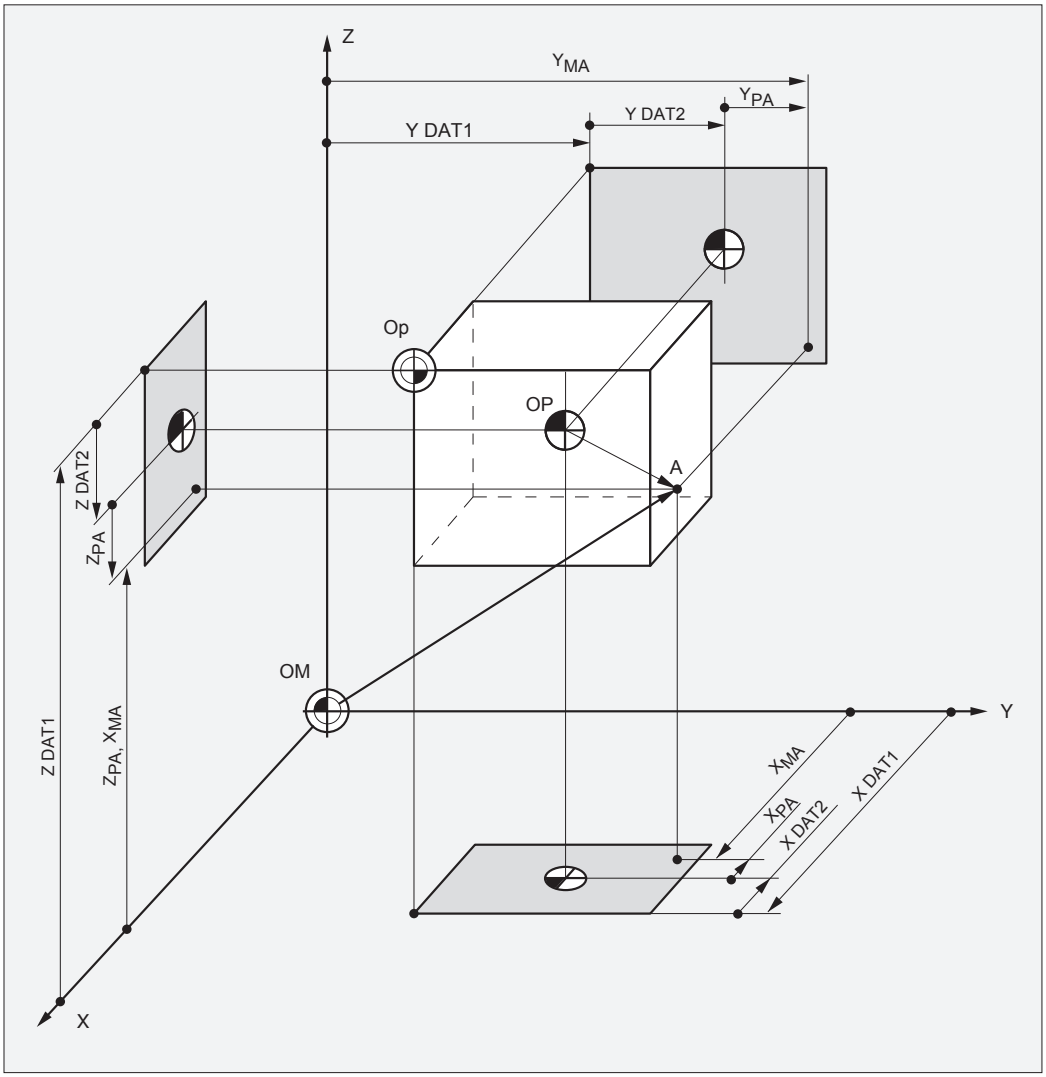


13.8.3 Offsets on the Y axis



13.8.4 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 DAT1 + X DAT2
Y_{PA}	$Y_{MA} = Y_{PA}$ + Y DAT1 + Y DAT2
Z_{PA}	$Z_{MA} = Z_{PA}$ + Z DAT1 + Z DAT2

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

13.8.5 Special Case of Milling Machines Equipped with Rotary Tables

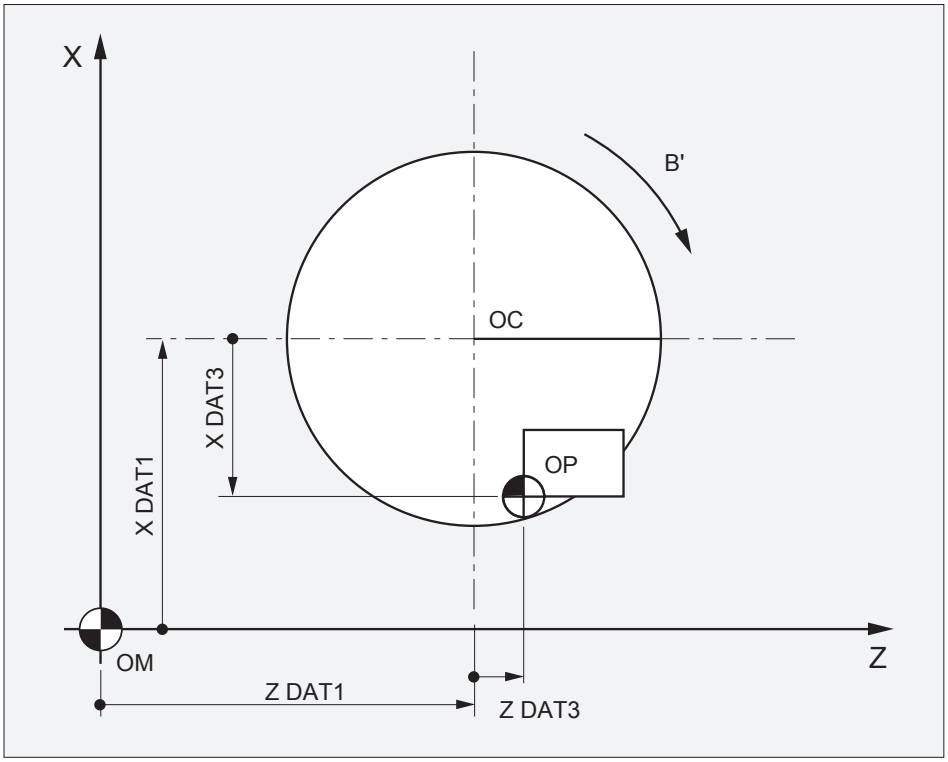
The concept of part origin does not apply to the two axes affected by the rotation.
The centre of rotation of the table (OC) has a special role.

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

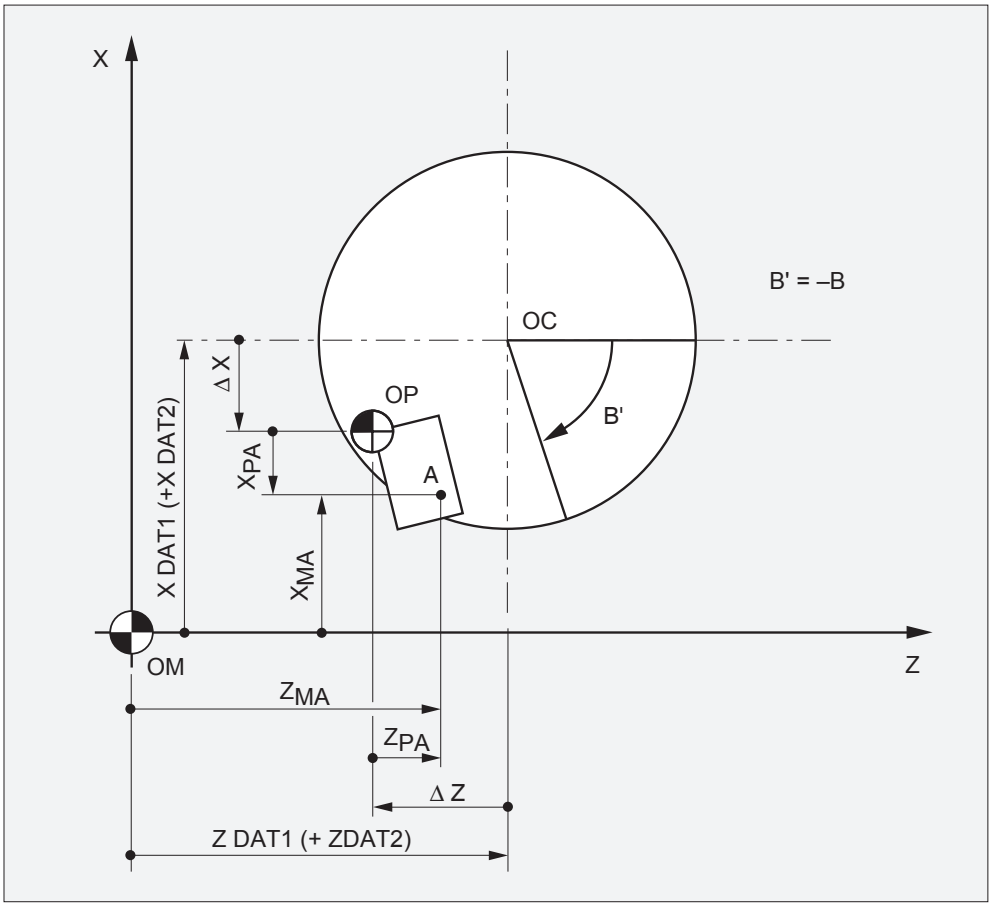
Offset of the part (OP/OC) = DAT3 (axes affected by rotation)

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):

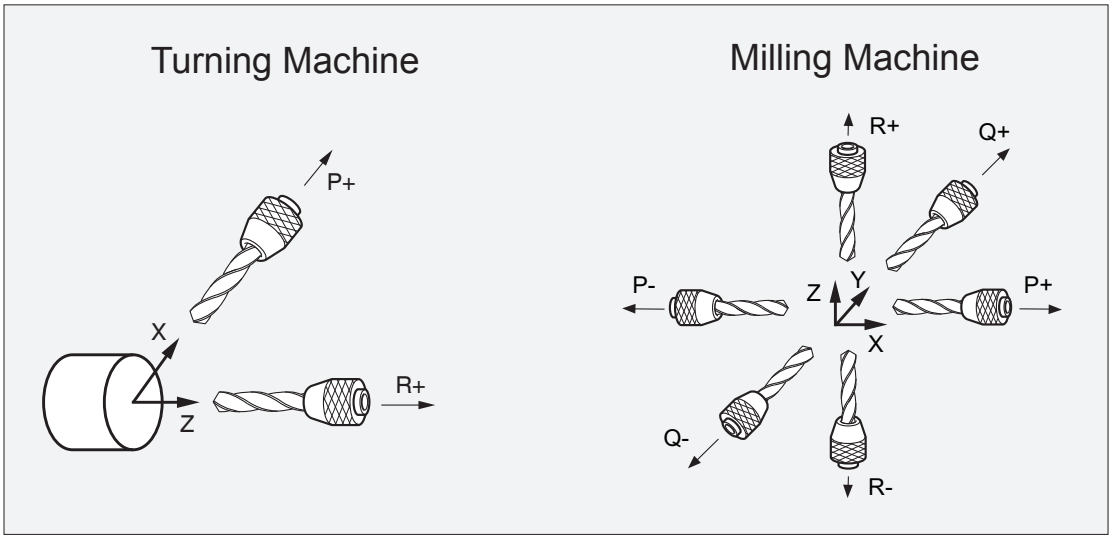


Program dimensions (with respect to OP)	Measurement dimensions (with respect to OM)
X_{PA}	$X_{MA} = X_{PA}$ + X DAT1 (+ X DAT2) + ΔX with ΔX = X DAT3 x cos B – Z DAT3 x sin B
Y_{PA}	$Y_{MA} = Y_{PA}$ + Y DAT1 + Y DAT2
Z_{PA}	$Z_{MA} = Z_{PA}$ + Z DAT1 (+ Z DAT2) + ΔZ with ΔZ = Z DAT3 x cos B + X DAT3 x sin B

13.9 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.

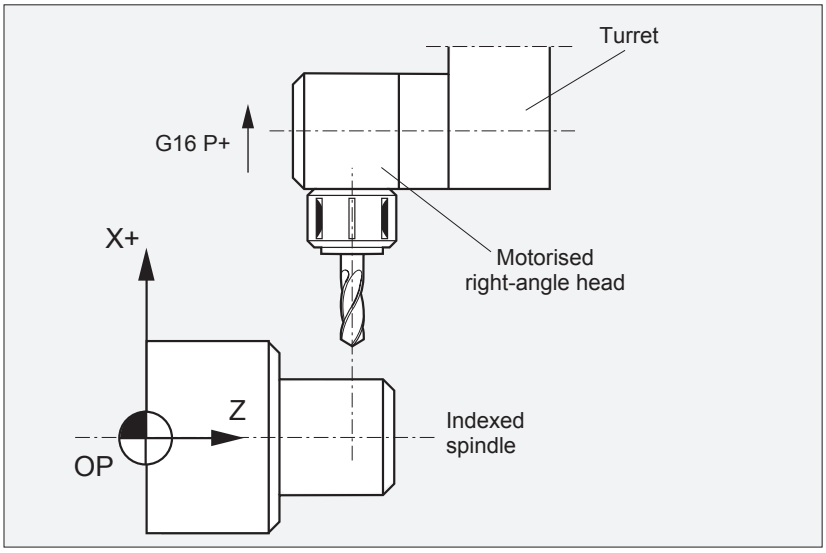
To perform radial machining when turning or milling (on X+), it is necessary to set a new tool axis orientation by programming function G16 P+. (Refer to the figures below and the complete description of the function G16 ... given in the programming manual).



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

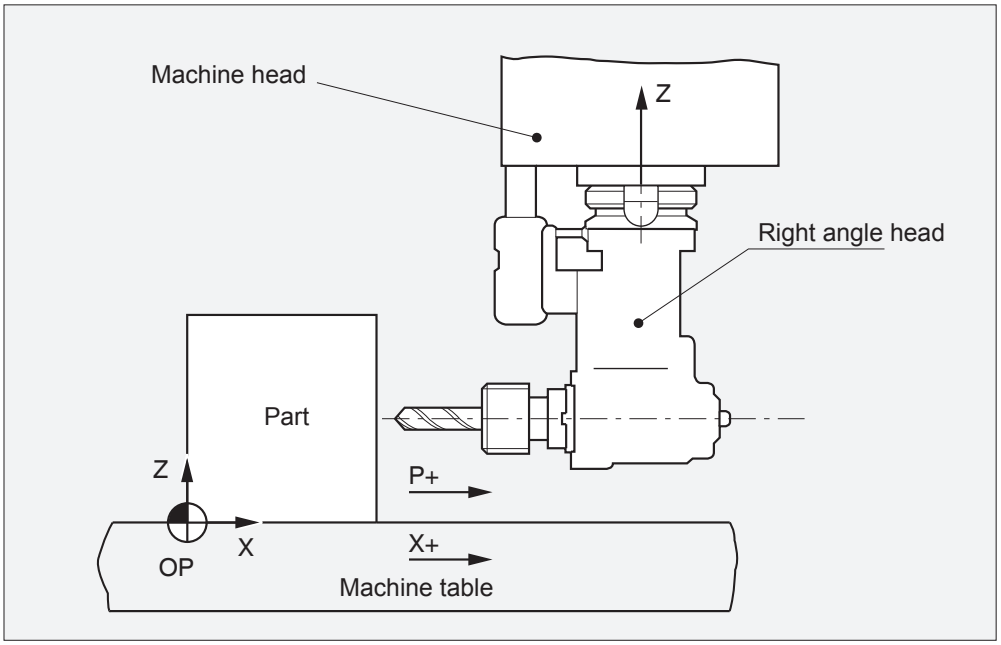
Use on Turning Machine

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



Use on Milling Machine

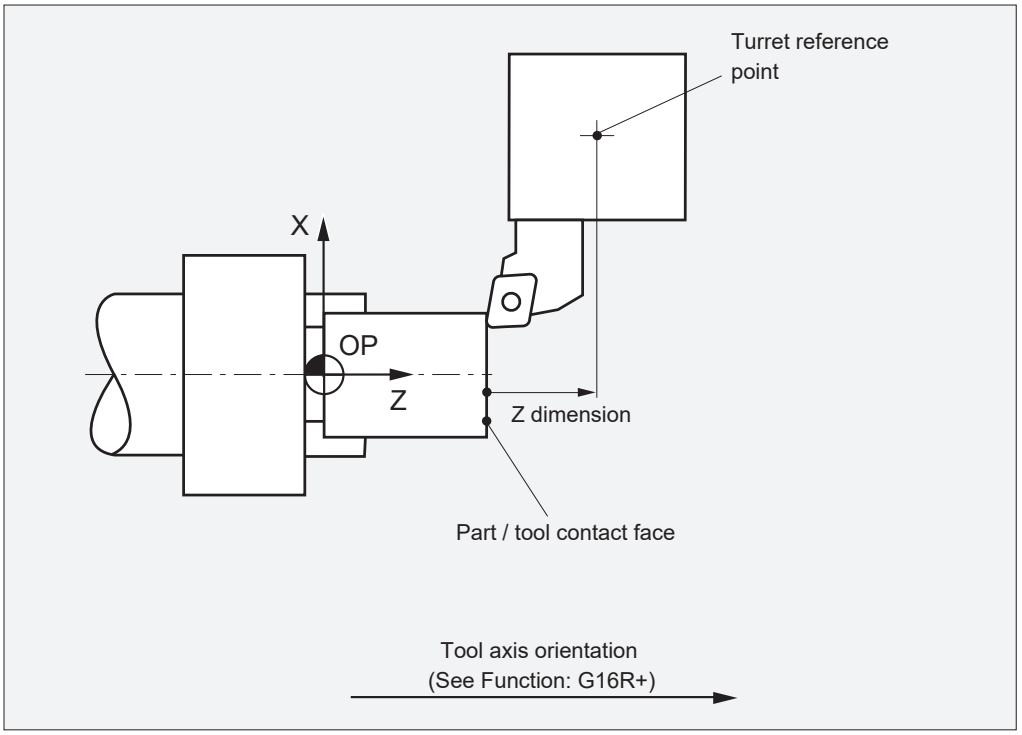
Machine equipped with a right-angle head (X+ orientation).



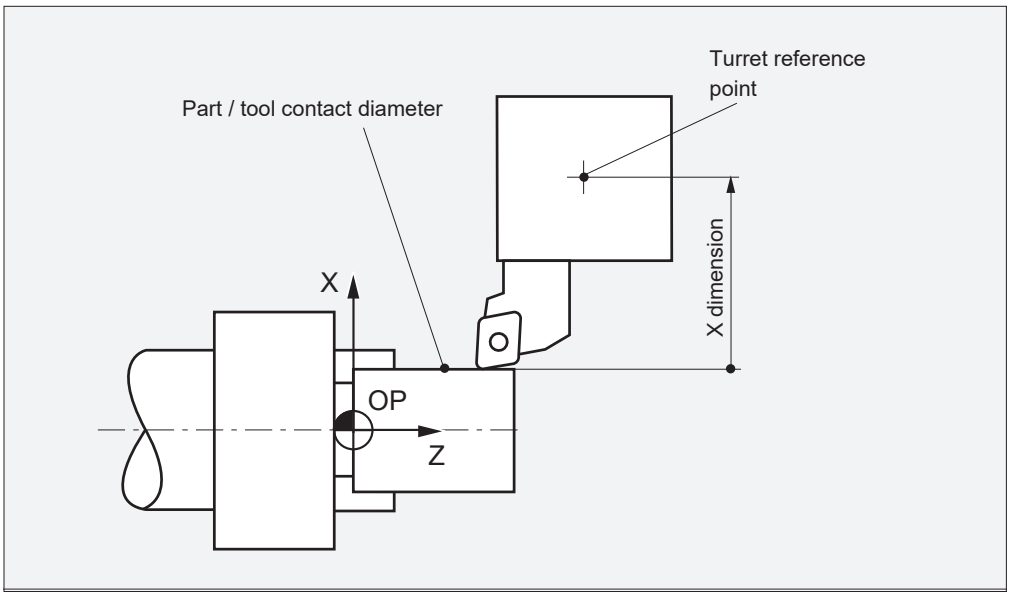
13.10 Definition of Tool Dimensions and Offsets

13.10.1 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

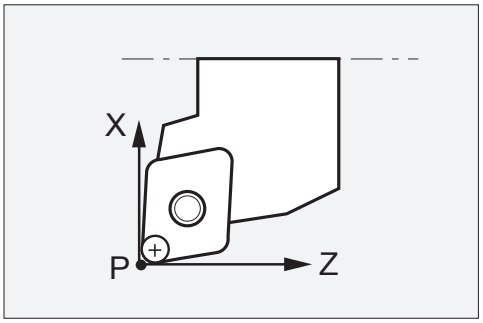


13.10.2 Tool Tip Radius and Orientation

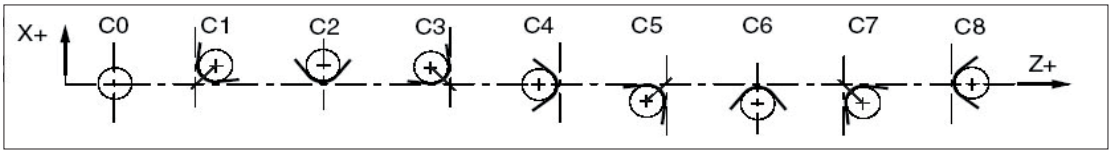
Tool tip orientation = code C0 to C8

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).

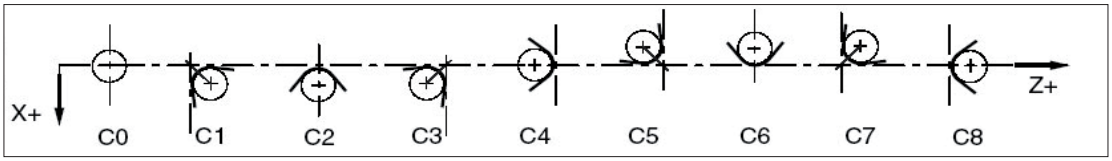
Example:



Tool Tip Orientation for a Rear Turret

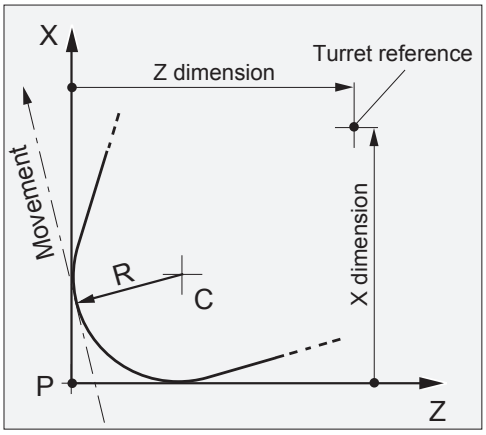


Tool Tip Orientation for a Front Turret



Tool cutting edge radius = R

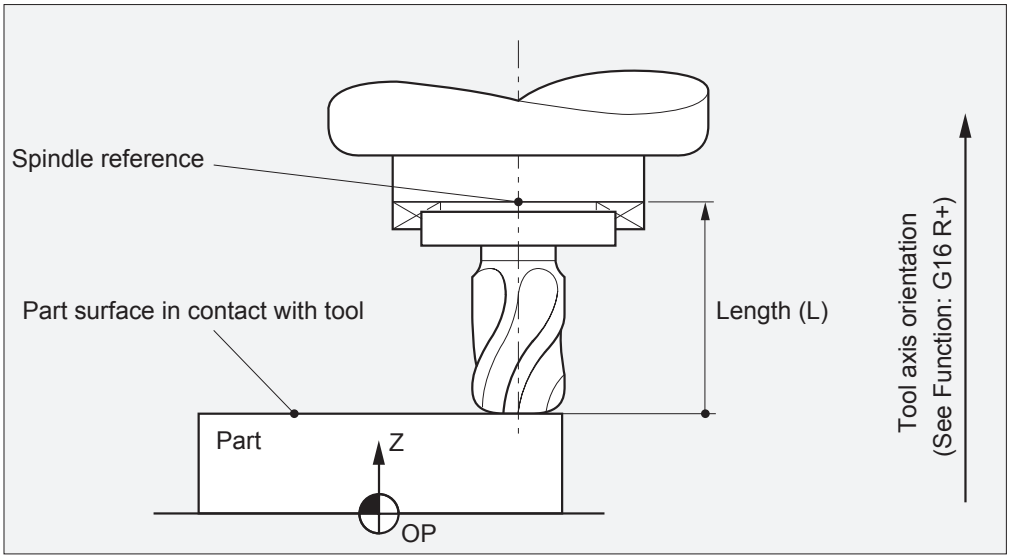
The real cutting point of the tool is obtained by applying a vector from centre “C” with length R perpendicular to the direction of movement.

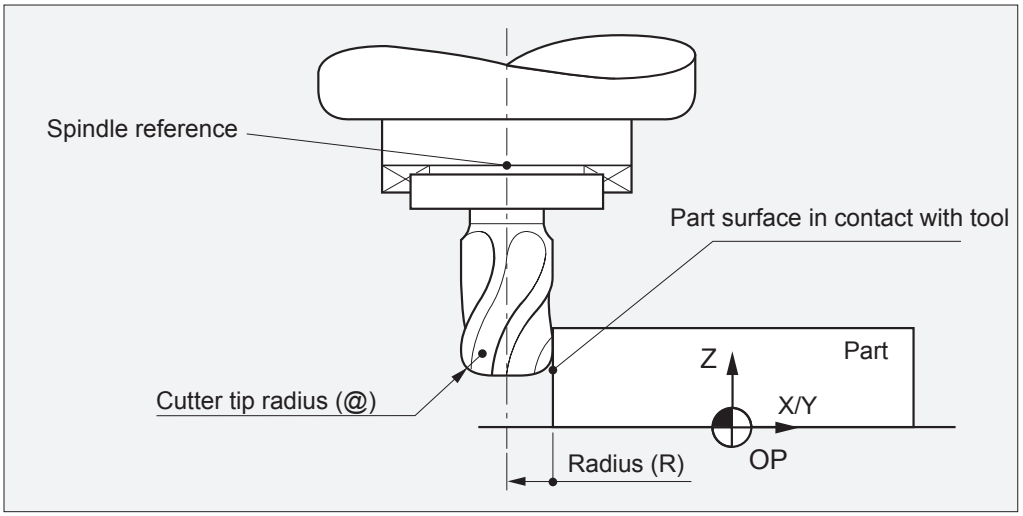


13.10.3 Milling Machine - Definition of Tool Dimensions

Tool dimensions are defined as follows:

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





Tool radius = R
Tool length = L

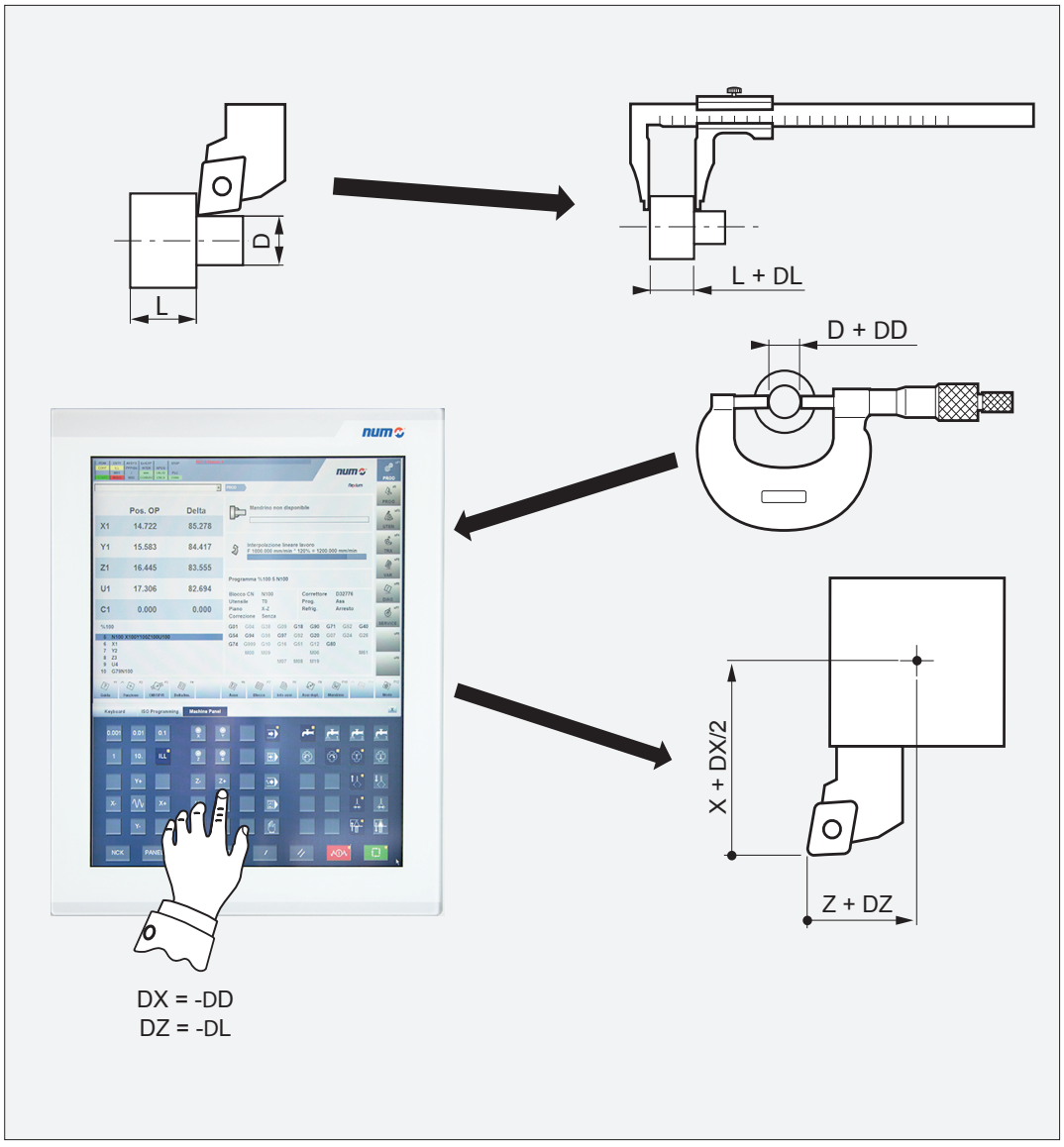
Cutter tip radius = @

13.11 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).

13.11.1 Turning Machine

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

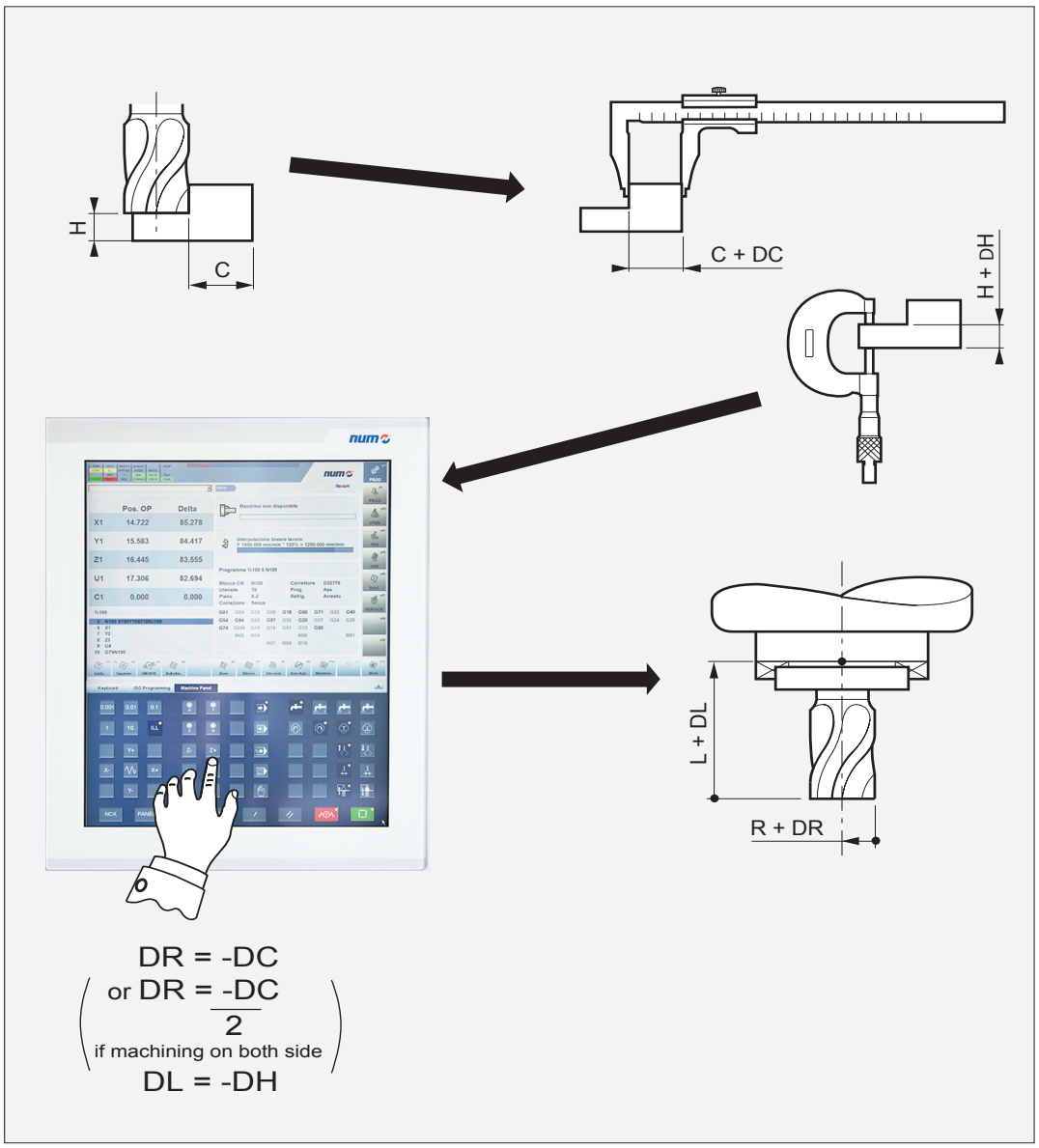


The system uses these offsets to correct the tool dimensions:

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

13.11.2 Milling 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

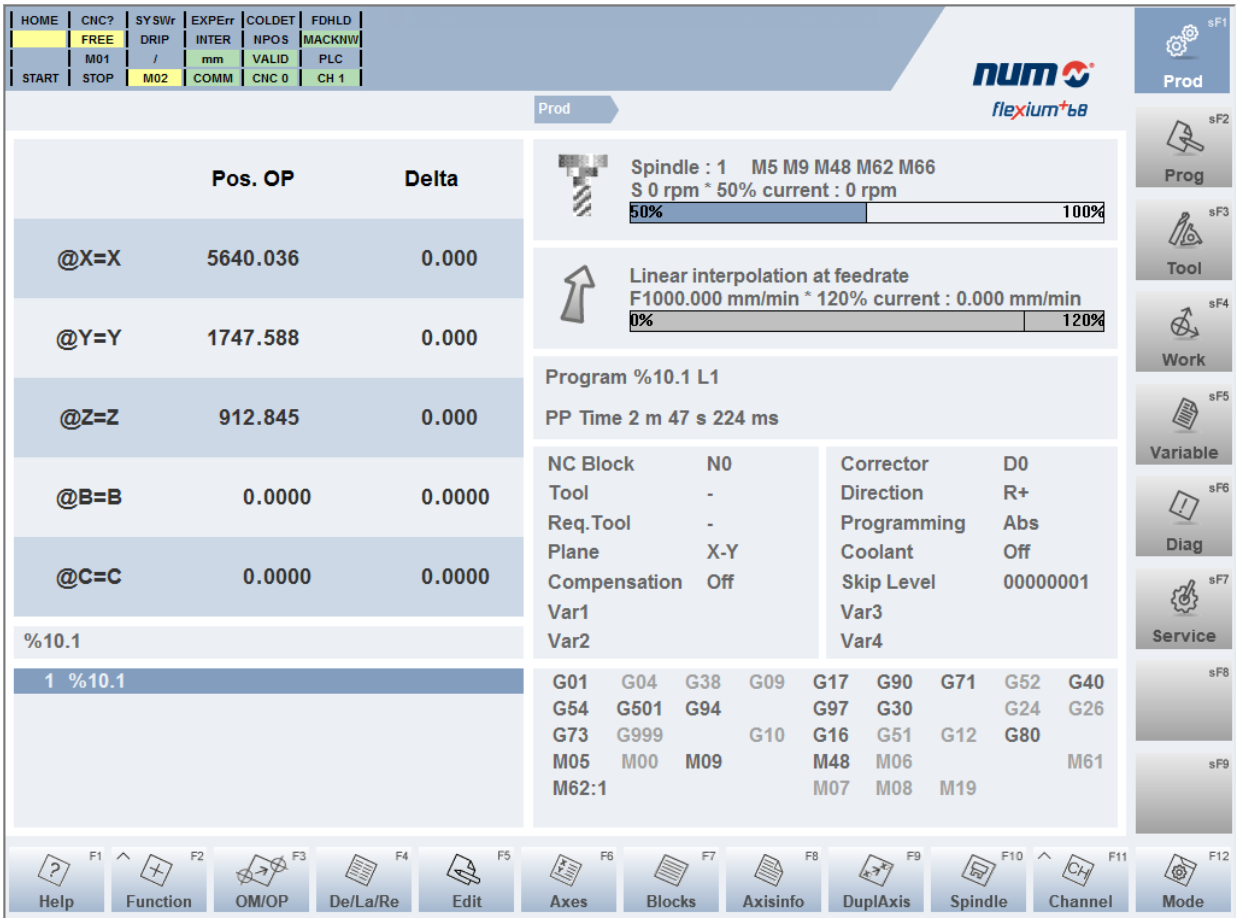
ANNEX 1 - HMI Structure.....361

ANNEX 2 - HMI functions 381

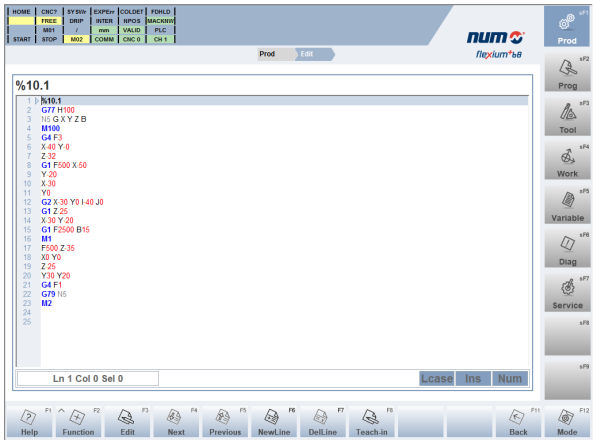
Page deliberately empty

ANNEX 1 - HMI Structure

PROD Context F5 to F8



F5



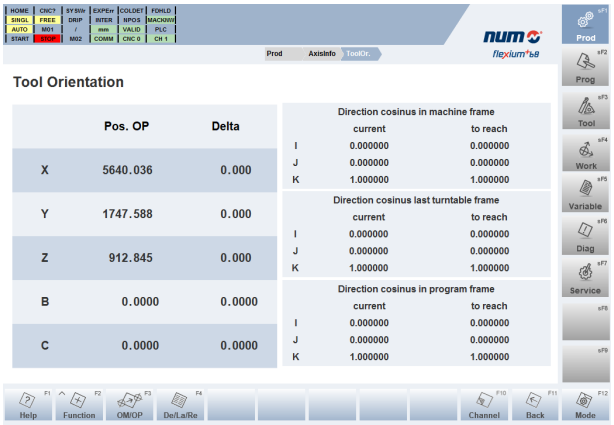
F6

PROD Context F5 to F8

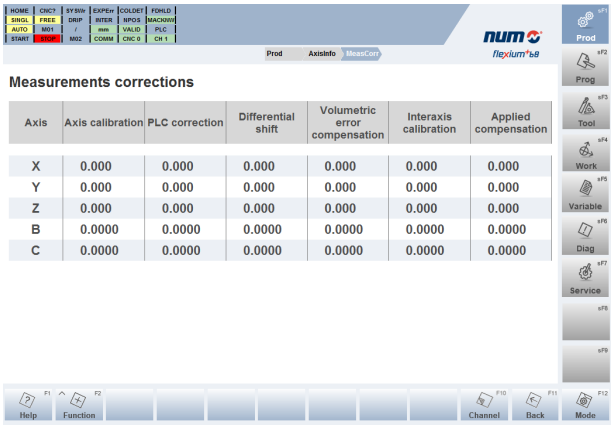
F7

F8

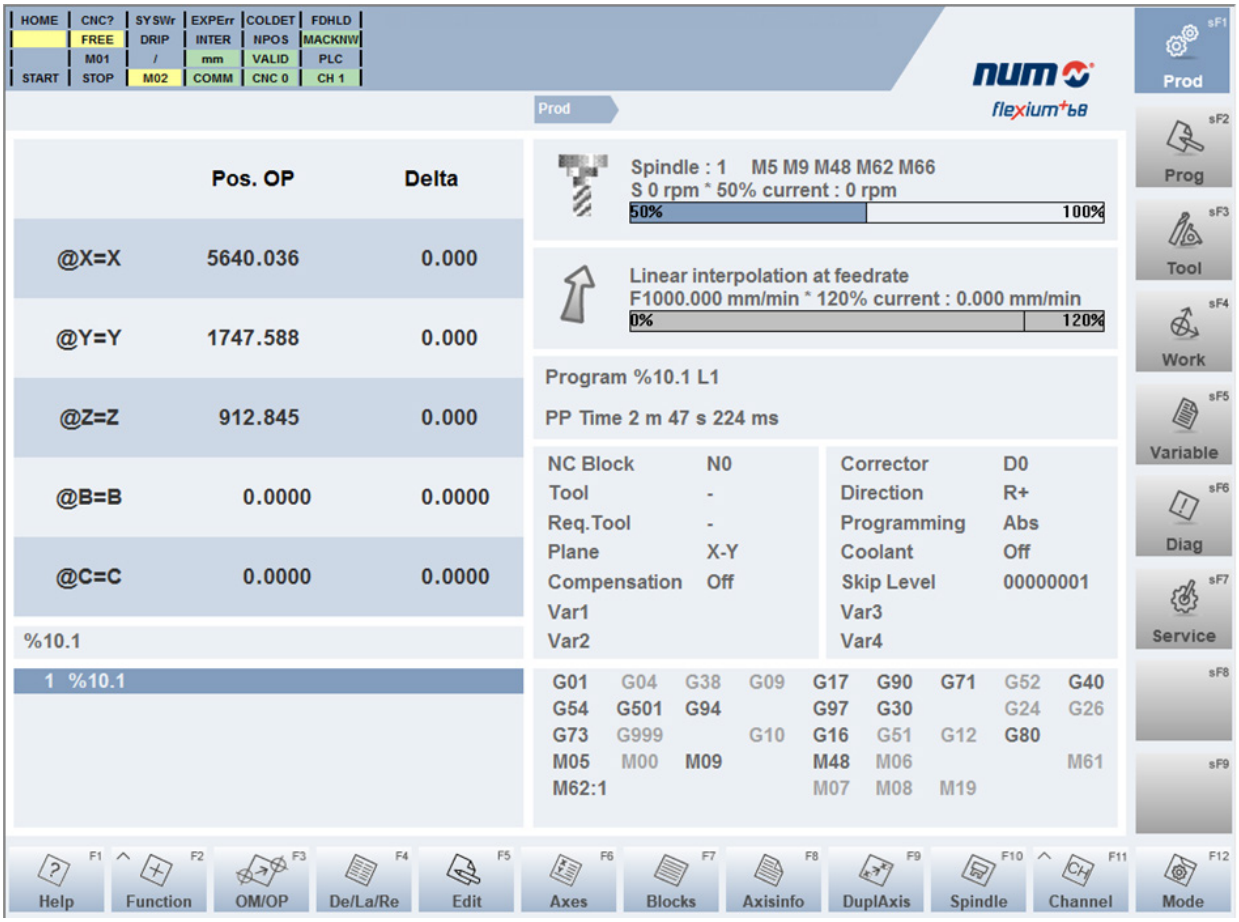
F8 > F7



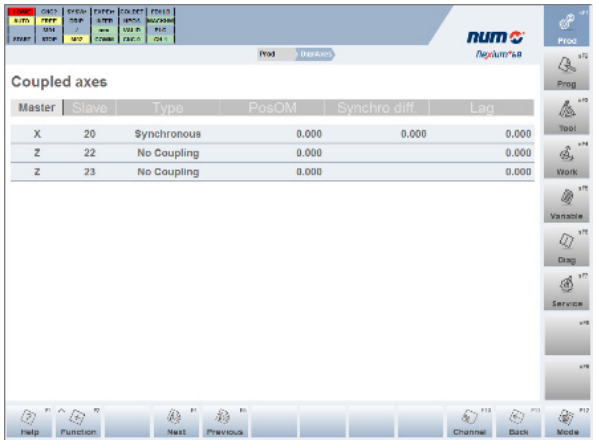
F8 > F8



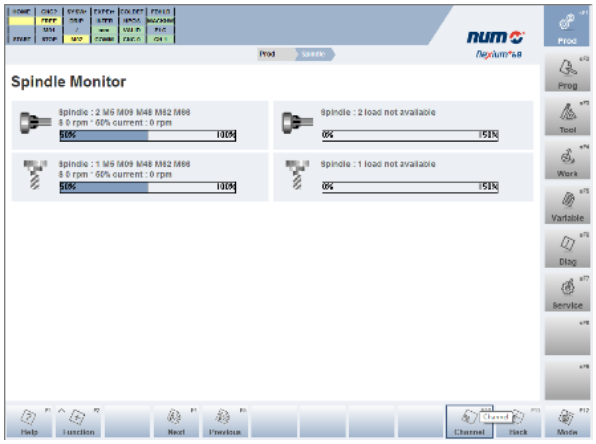
PROD Context F9 to F10



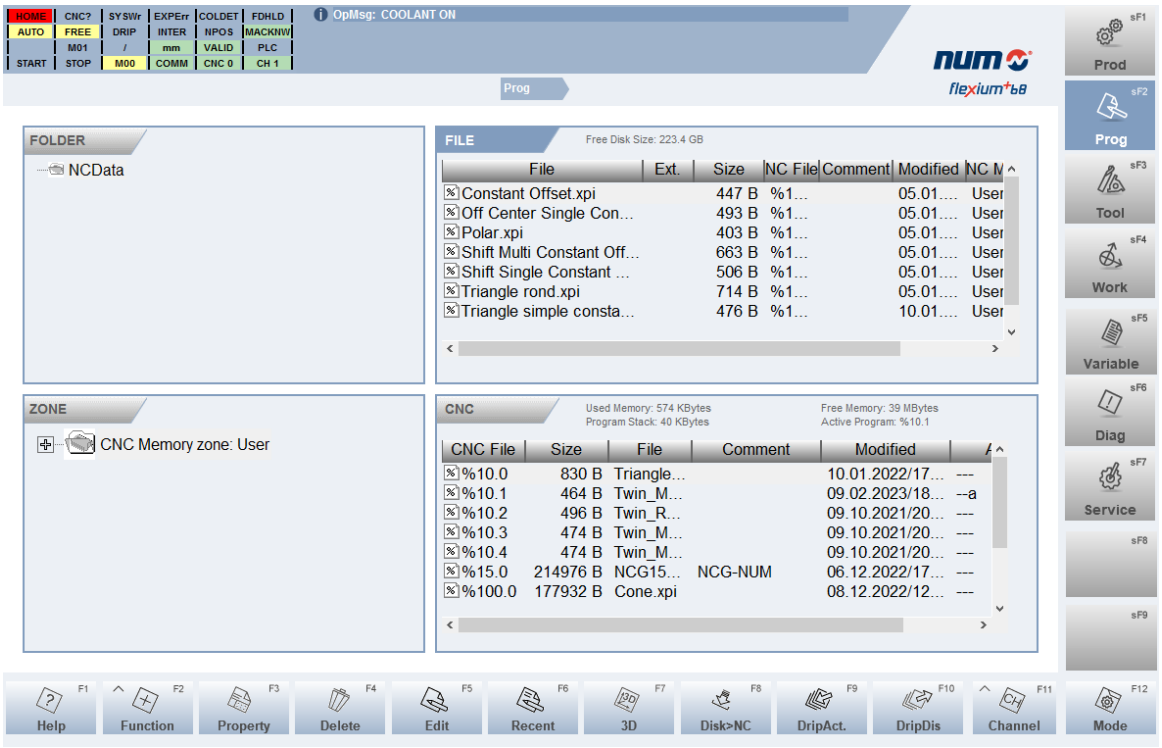
F9



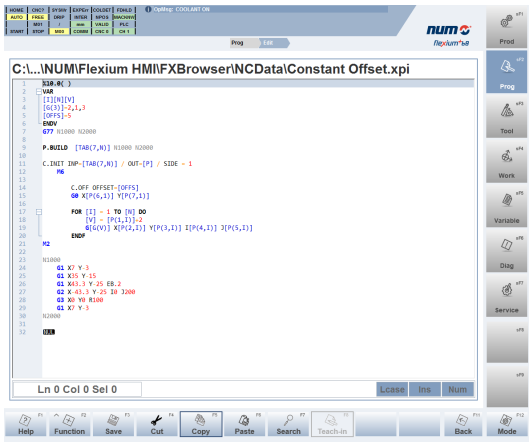
F10



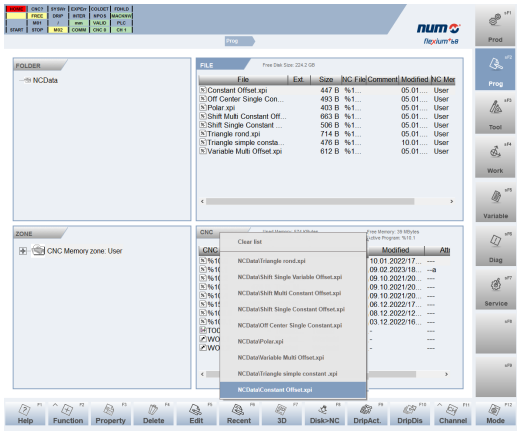
PROG Context F5 to F7



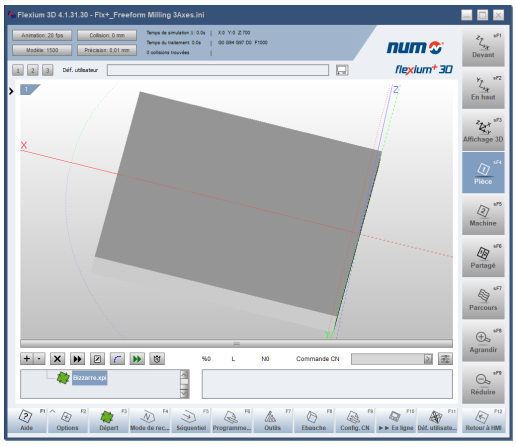
F5



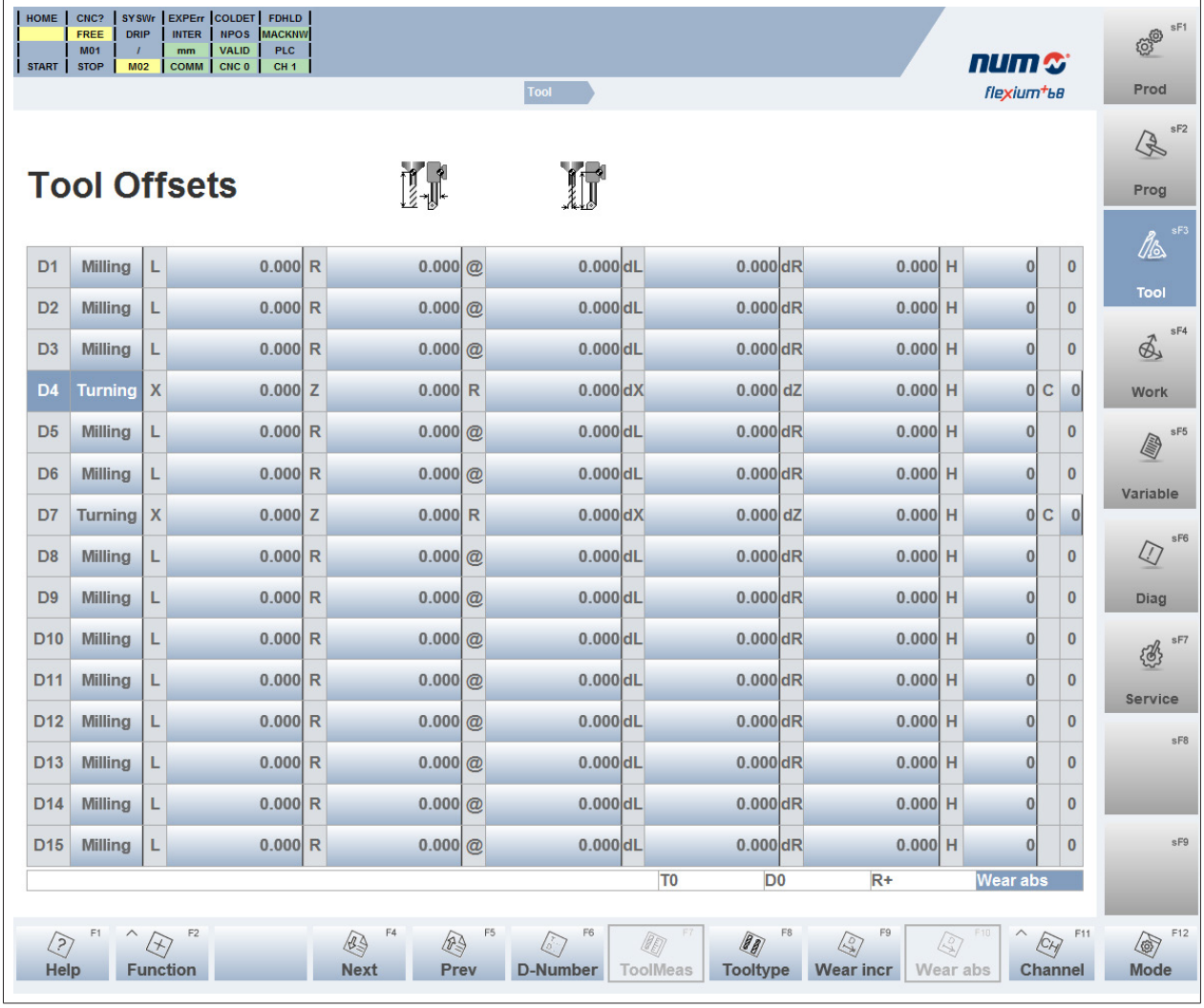
F6



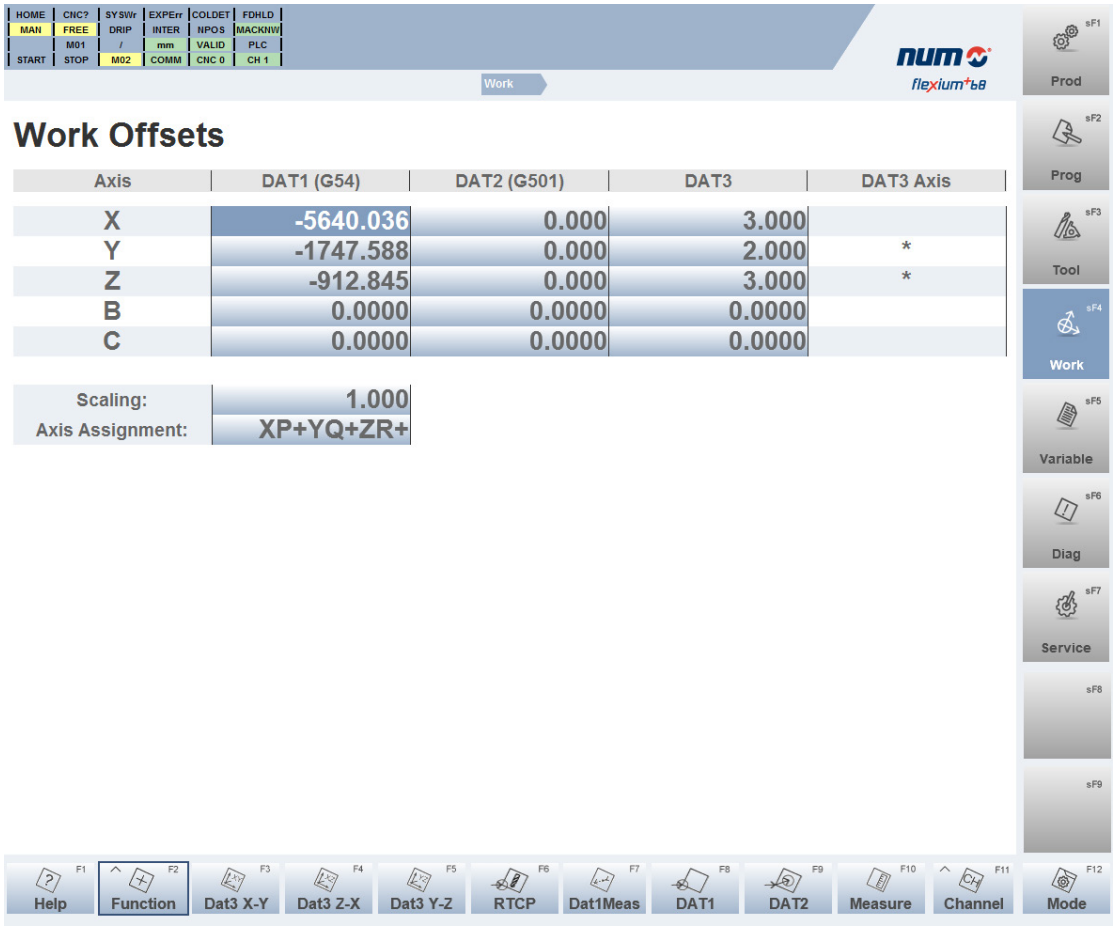
F7



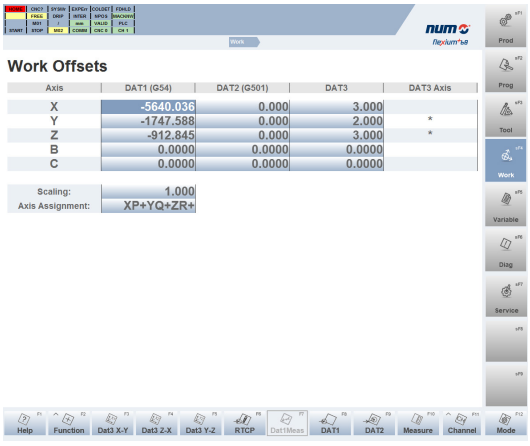
TOOL Context



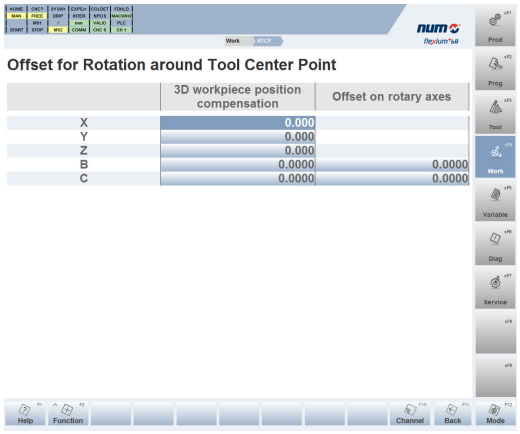
WORK Context F3 to F10



F3 - F5

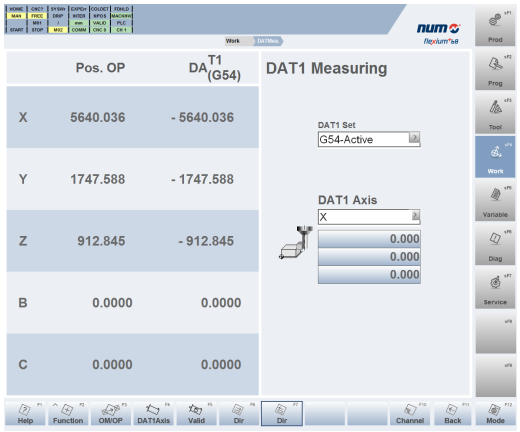


F6

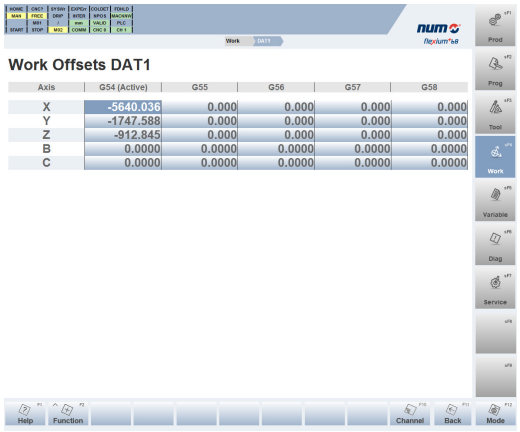


WORK Context F3 to F10

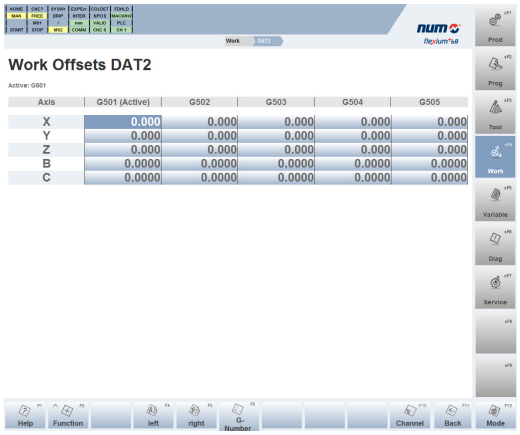
F7



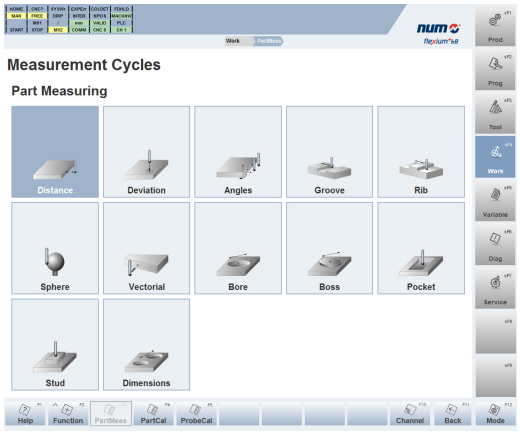
F8



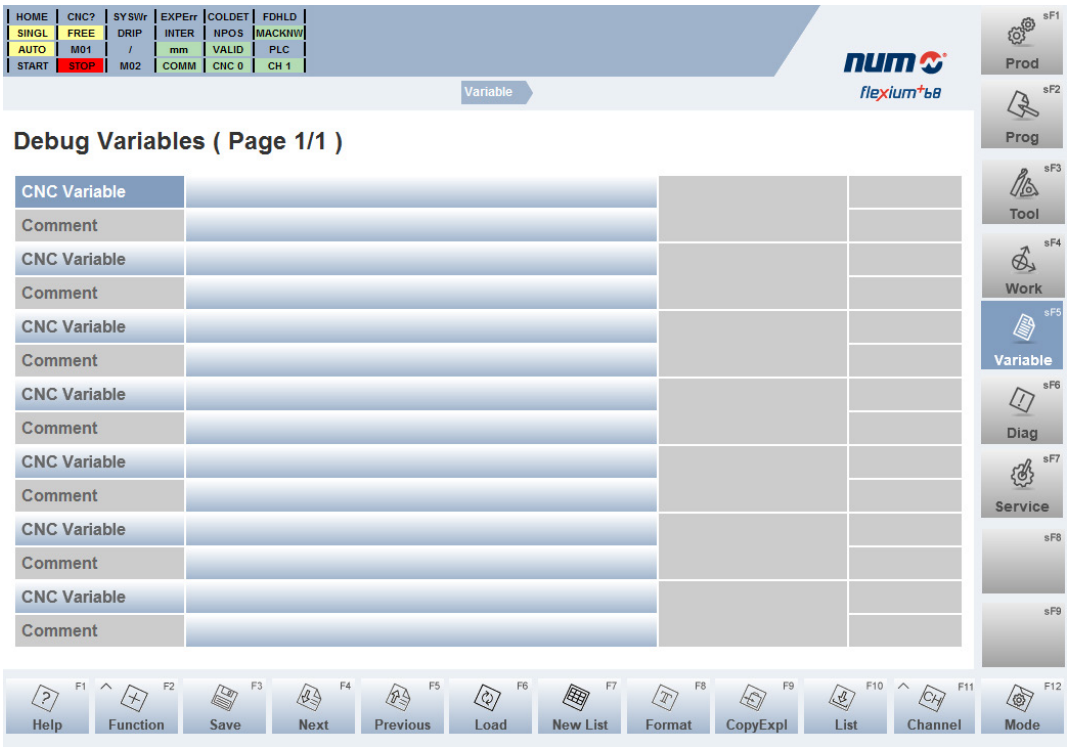
F9



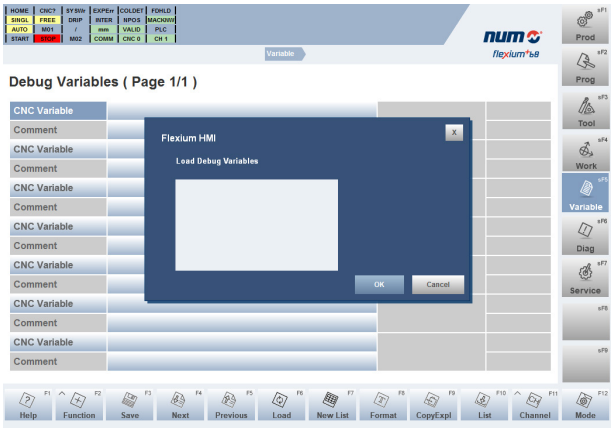
F10



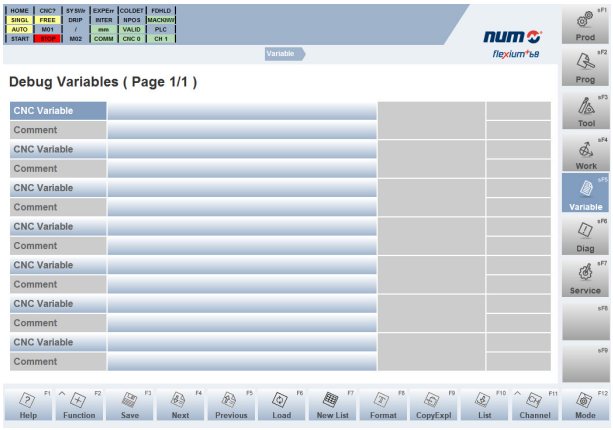
VARIABLE Context F6 - F9 - F10



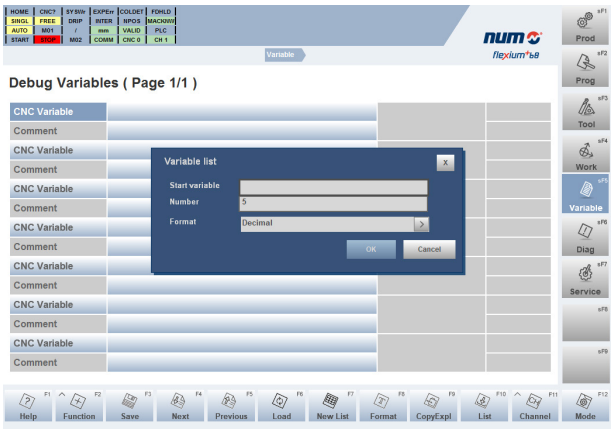
F6



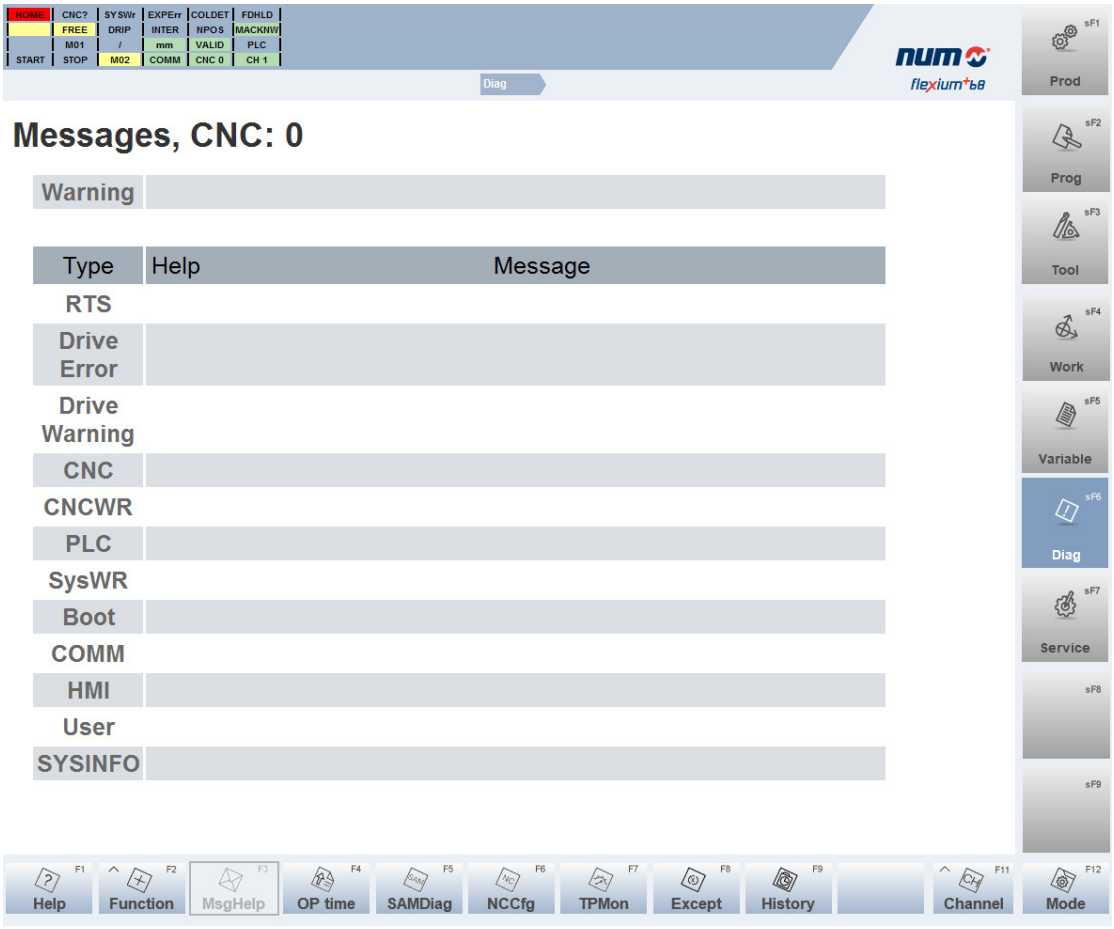
F9



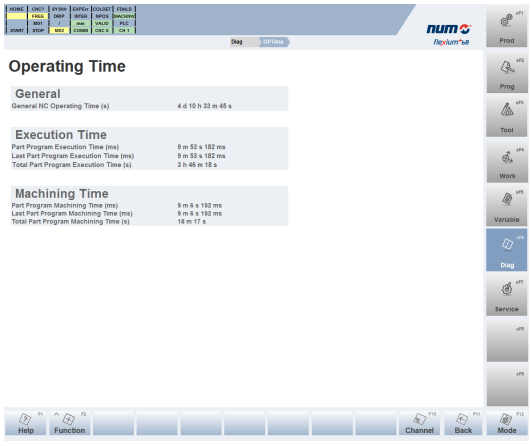
F10



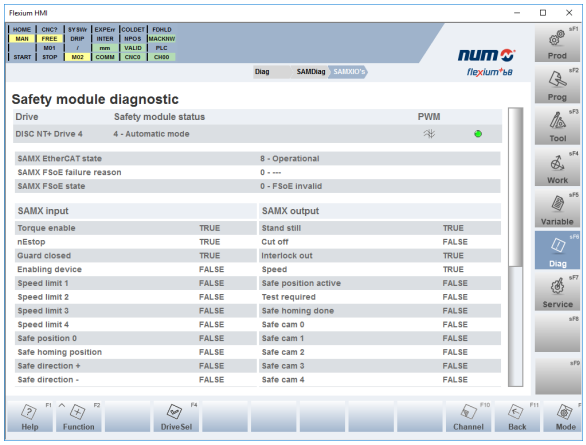
DIAG Context (except NCCFG - F6) F4 to F9



F4

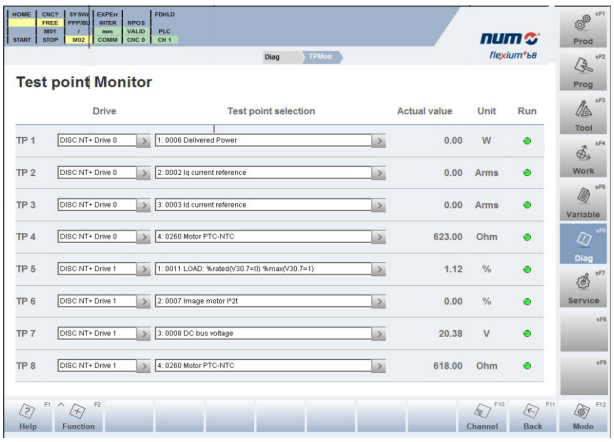


F5

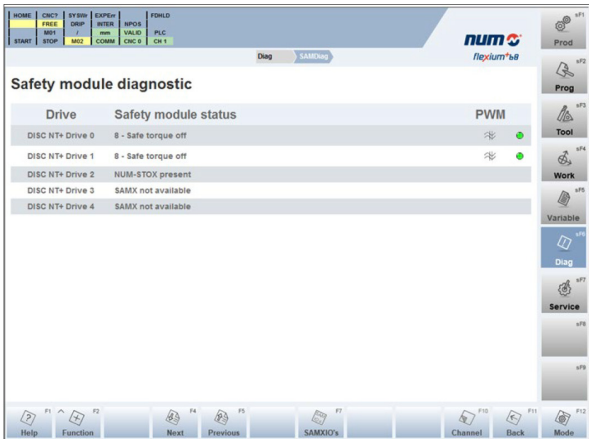


DIAG Context (except NCCFG - F6) F4 to F9

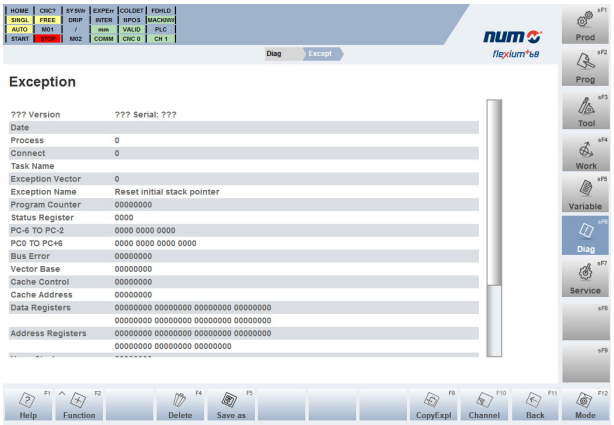
F7



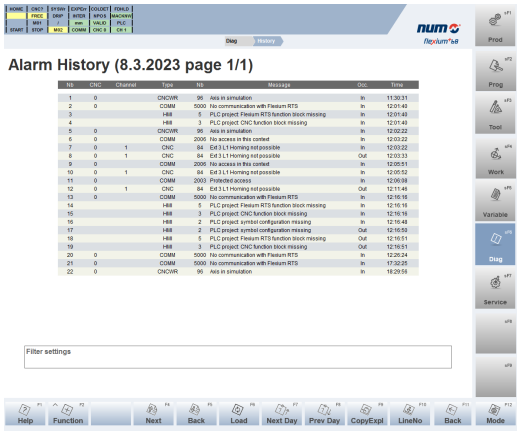
F5 > F7



F8

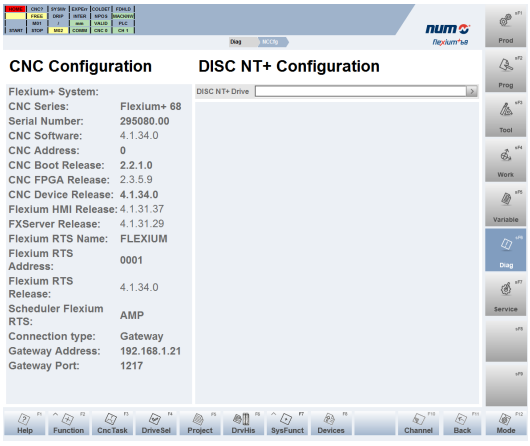


F9

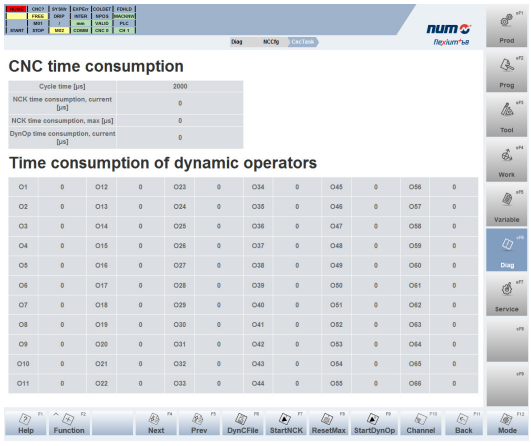


DIAG Context NCCFG F6

F6



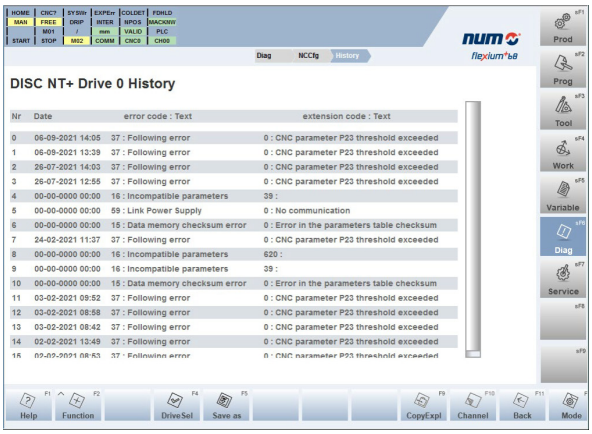
F6 > F3



DIAG Context NCCFG F6

F6 > F5

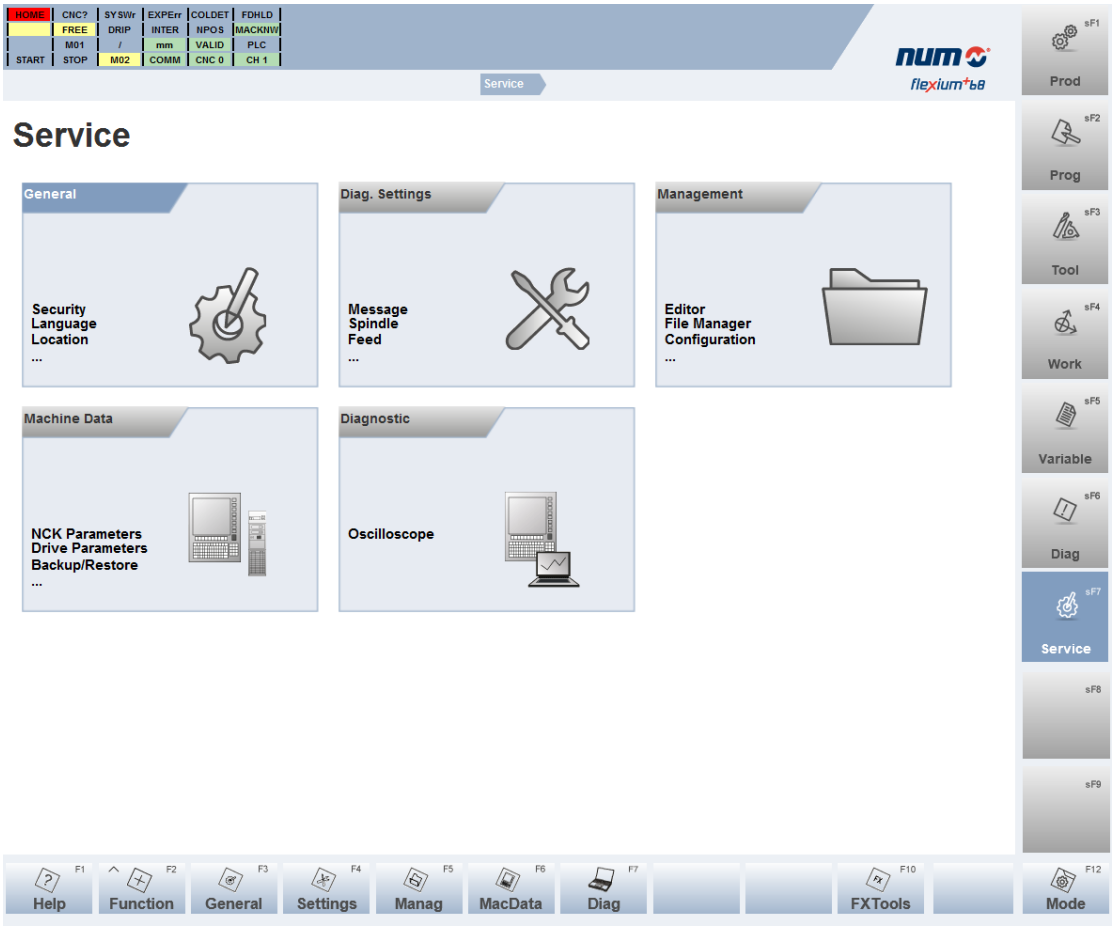
F6 > F6



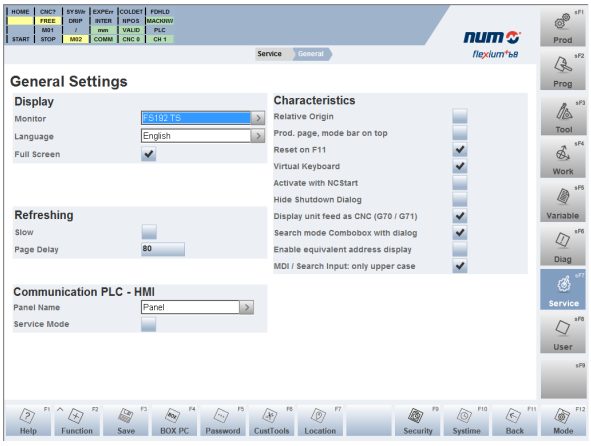
F6 > F7

F6 > F8

SERVICE Context - General F3



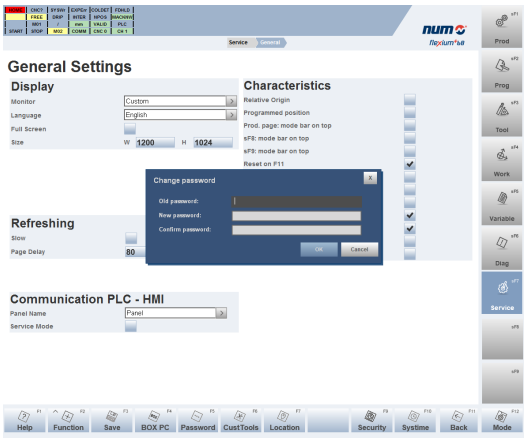
F3



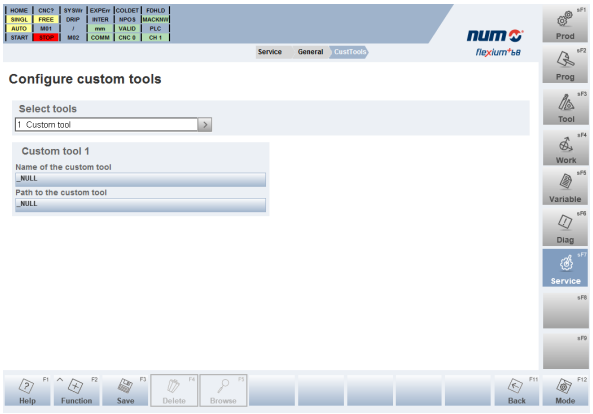
F3 > F4

SERVICE Context - General F3

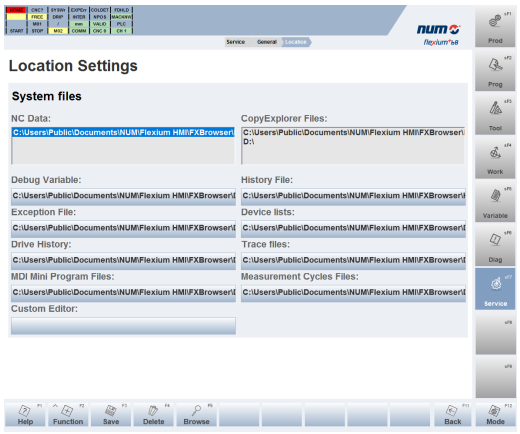
F3 > F5



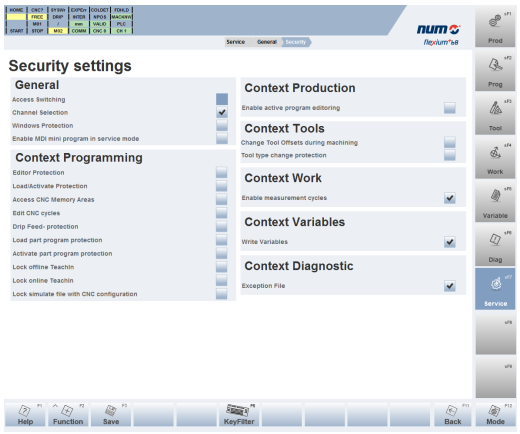
F3 > F6



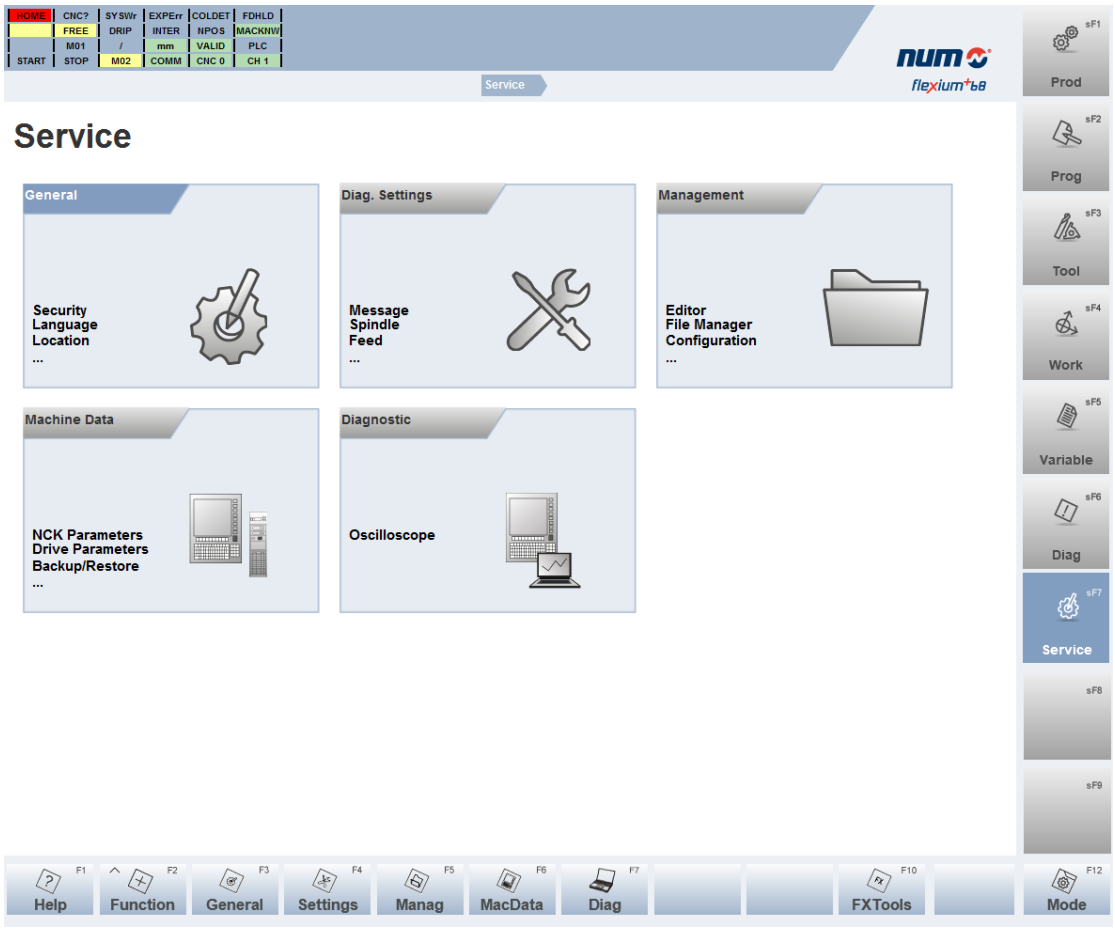
F3 > F7



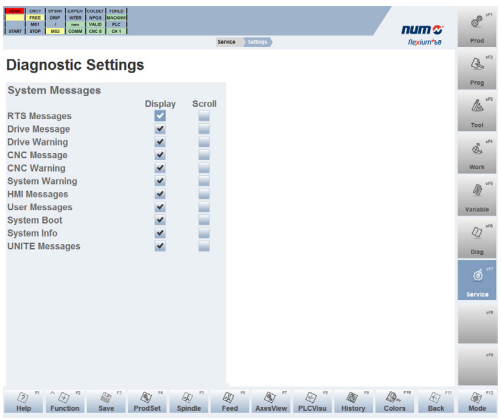
F3 > F9



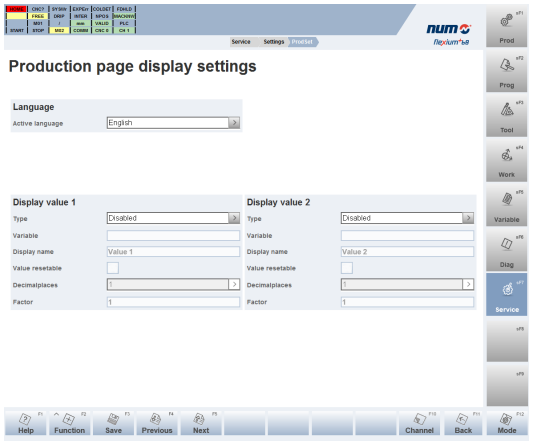
SERVICE Context - Settings F4



F4

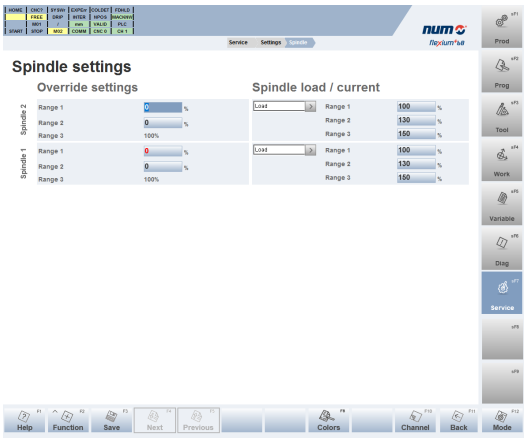


F4 > F4

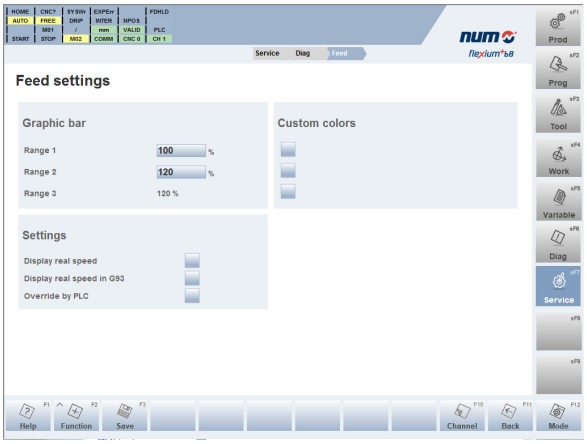


SERVICE Context - Settings F4

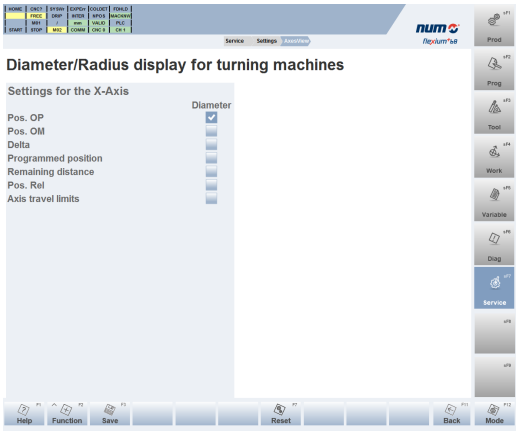
F4 > F5



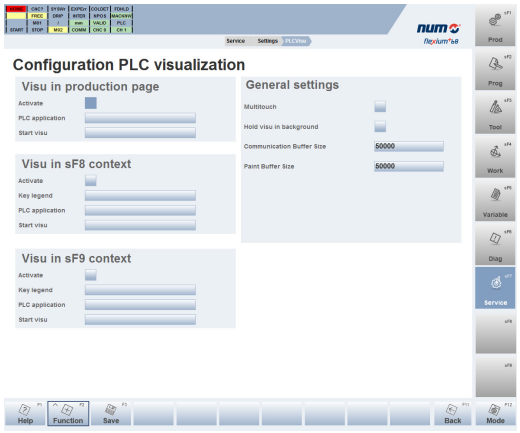
F4 > F6



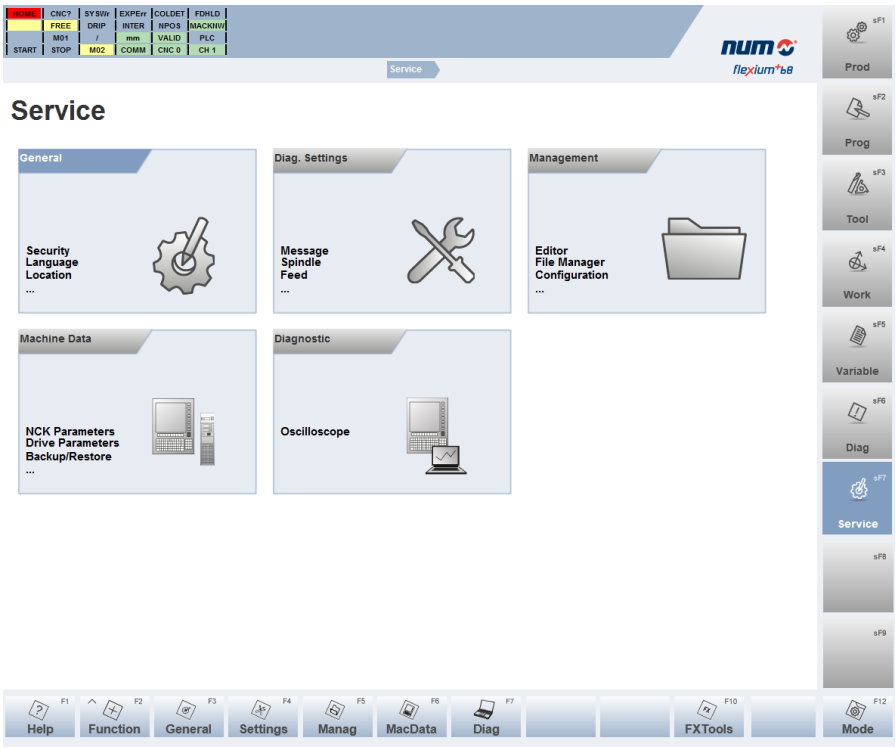
F4 > F7



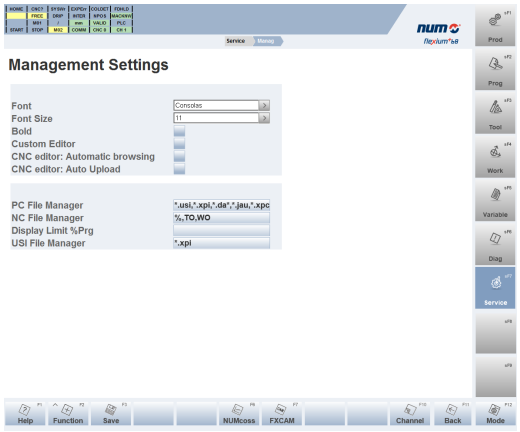
F4 > F8



SERVICE Context Management F5



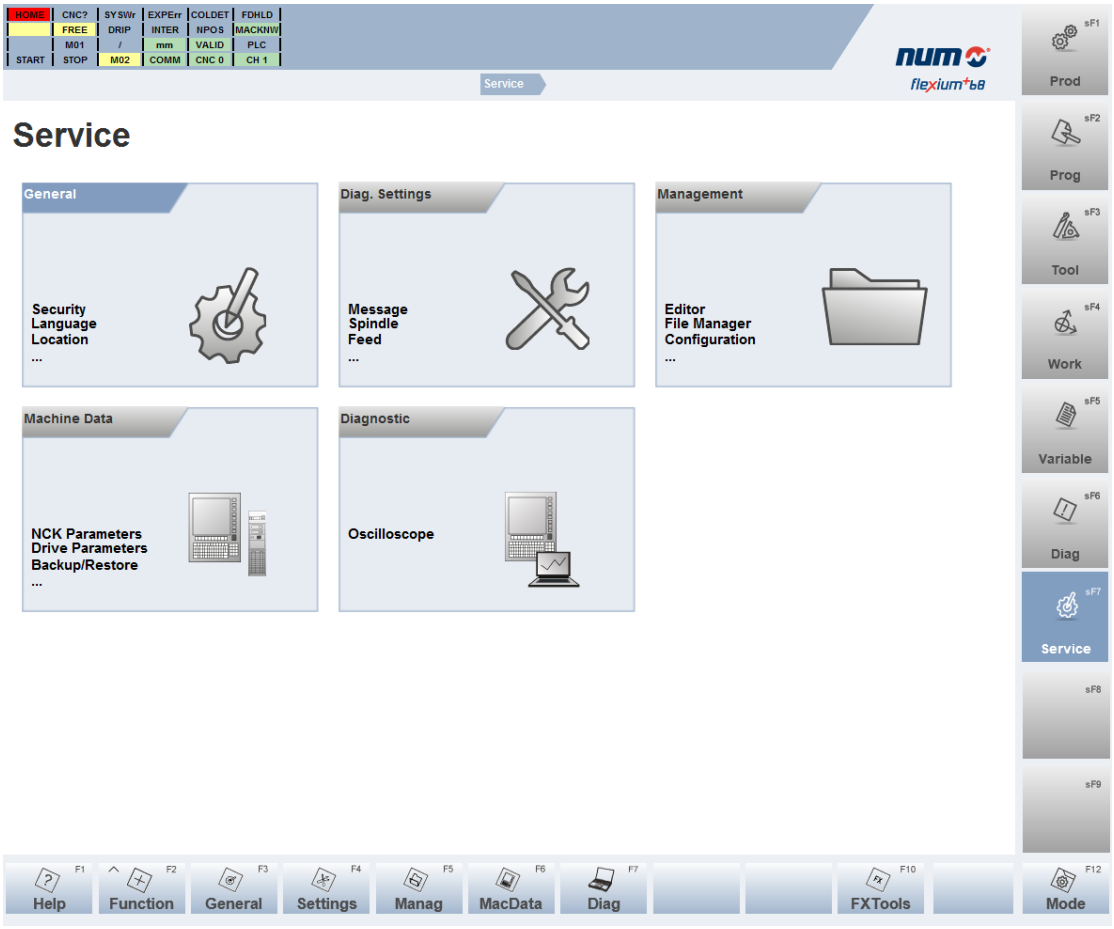
F5



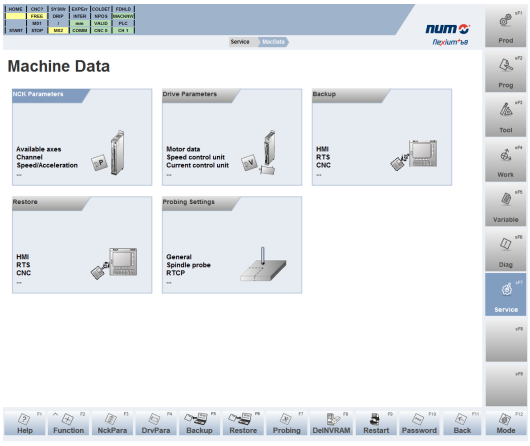
F5 > F6

F5 > F7

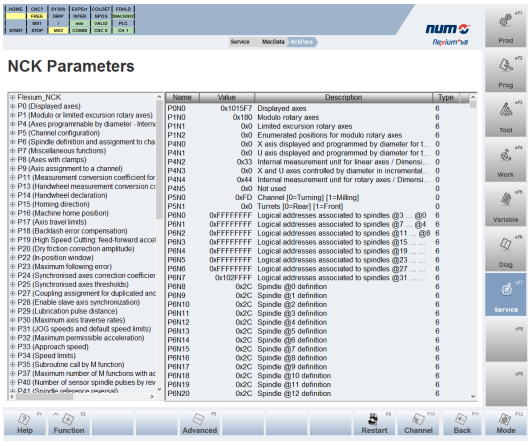
SERVICE Context Machine Data F6



F6



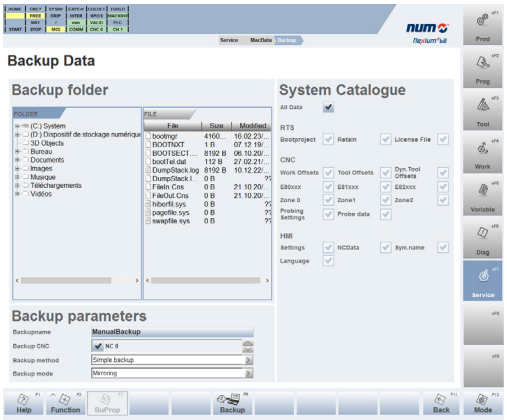
F6 > F3



SERVICE Context Machine Data F6

F6 > F4

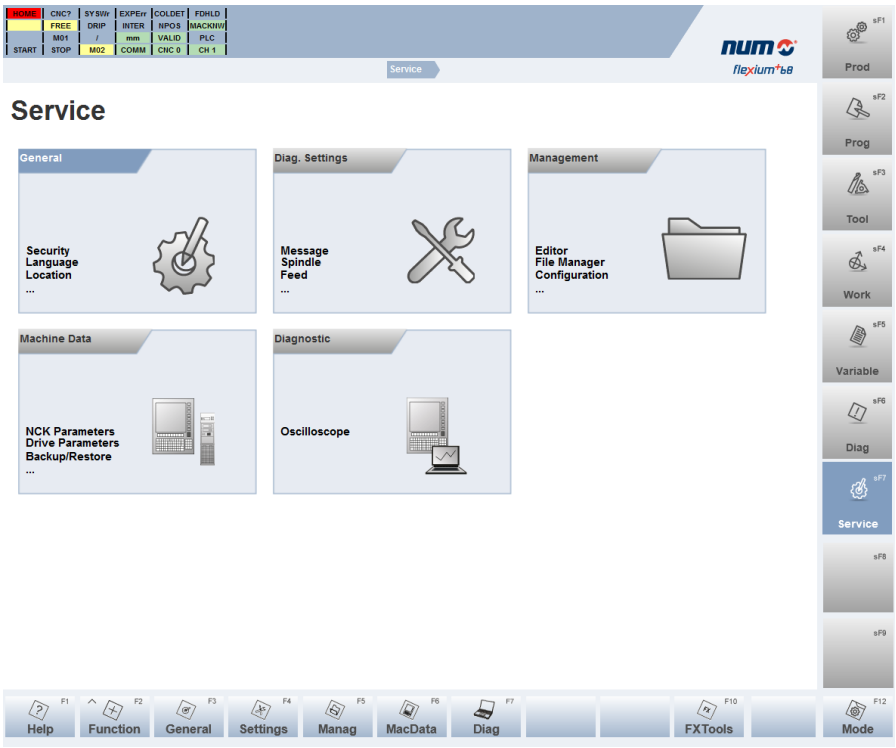
F6 > F5



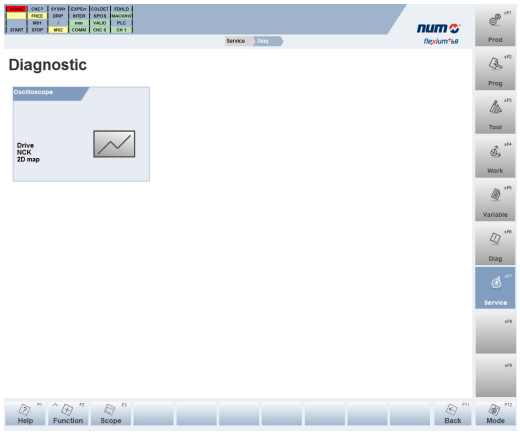
F6 > F6

F6 > F7

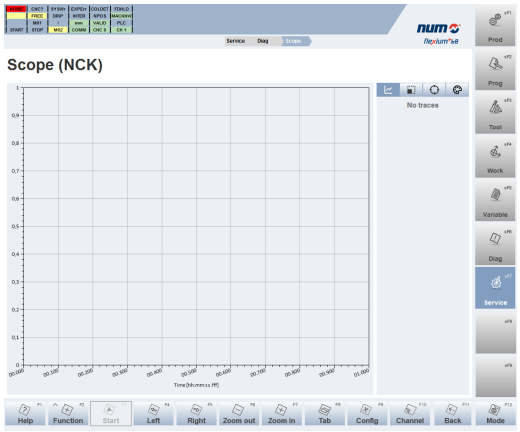
SERVICE Context Diagnostic F7



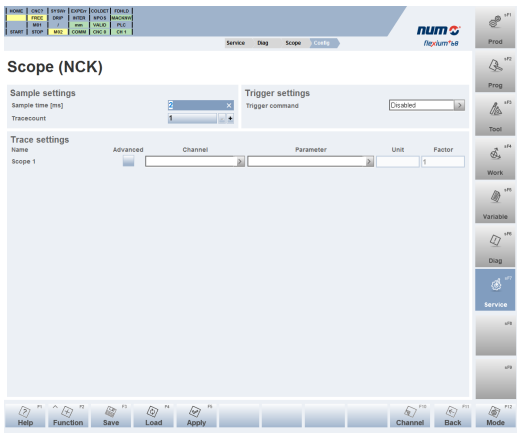
F7



F7 > F3



F7 > F9



ANNEX 2 - HMI functions

FUNCTION	CONTEXT	1st	2nd	3rd	FIELD
Select channel	ALL	F11			
Display machining time	PROD	F2			
Display part program time	PROD	F2			
Delete dialog list	PROD	F2			
Switch inch/metric	PROD	F2			
Switch OM/OP display	PROD	F3			
Display Lag/Delta/Distance to go	PROD	F4			
Display axes position	PROD	F6			
Display other types of axes position	PROD	F6	F2		
Display travel limits	PROD	F6	F2		
Display program in progress	PROD	F7			
Display second channel program in progress	PROD	F7	F5		
Display axes infos	PROD	F8			
Display tool orientation	PROD	F8	F7		
Display measure correction	PROD	F8	F8		
Display coupled axes	PROD	F9			
Display spindle monitor	PROD	F10			
Create new folder	PROG	F3			FOLDER
Acknowledge error 2007	PROG	F2			FOLDER
Remove USB hardware	PROG	F2			FOLDER
Create new file on drive	PROG	F2			FILE
Edit file property	PROG	F3			FILE
Delete file on disk	PROG	F4			FILE
Edit file on disk	PROG	F5			FILE
List recent drive disk	PROG	F6			FILE
Launch Flexium 3d	PROG	F7			FILE
Transfer from disk to NC	PROG	F8			FILE
Drip feed activate	PROG	F9			FILE
Disable drip feed	PROG	F10			FILE
Delete file in NC memory	PROG	F4			CNC
Edit file in NC memory	PROG	F5			CNC
List recent files from NC memory	PROG	F6			CNC
Transfer from NC to disk	PROG	F8			CNC
Activate part program	PROG	F9			CNC
Initialise tool offsets	TOOL	F2			
Clear a tool offset	TOOL	F3			
Manual measure of a tool	TOOL	F7			
Modify a tool type	TOOL	F8			
Increment tool wear	TOOL	F9			
Define absolute tool wear	TOOL	F10			
Define DAT3 on X-Y	WORK	F3			
Define DAT3 on Z-X	WORK	F4			

FUNCTION	CONTEXT	1st	2nd	3rd	FIELD
Define DAT3 on Y-Z	WORK	F5			
Define rotary axes offsets	WORK	F6			
Define 3-D workpiece compensation	WORK	F6			
DAT1 definition wizard	WORK	7			
DAT1 entry	WORK	F8			
DAT2 entry	WORK	F9			
Access to measuring cycles	WORK	F10			
Access to part measuring cycles	WORK	F10	F3		
Access to part calibration cycles	WORK	F10	F4		
Access to probe calibration cycles	WORK	F10	F5		
Display debug variables	VARIABLE				
Load set of debug variables	VARIABLE	F6			
Create new list of debug variables	VARIABLE	F7			
Select format of debug variables	VARIABLE	F8			
Access to copy explorer	VARIABLE	F9			
Create list of debug variables	VARIABLE	F10			
Display operation time	DIAG	F4			
Display messages	DIAG				
Display tool offsets	TOOL				
Display shifts	WORK				
Display safety diagnostic	DIAG	F5			
Display SAMX I/Os	DIAG	F5	F7		
Display CNC configuration	DIAG	F6			
Display CNC task timings	DIAG	F6	F3		
Display dynamic operator timings	DIAG	F6	F3	F6	
Display plc project	DIAG	F6	F5		
Display drive history	DIAG	F6	F6		
Display system functions	DIAG	F6	F7		
Display device list	DIAG	F6	F8		
Display drives test points	DIAG	F7			
Display exception file	DIAG	F8			
Display message history	DIAG	F9			
Mode setting general	SERVICE	F3			
Set box pc	SERVICE	F3	F4		
Change service password	SERVICE	F3	F5		
Configure custom tools	SERVICE	F3	F6		
Change file location	SERVICE	F3	F7		
Change security settings	SERVICE	F3	F9		
Change system time and date	SERVICE	F3	F10		
Mode diagnostic settings	SERVICE	F4			
Define additional variables in prod	SERVICE	F4	F4		
Define spindle override ranges	SERVICE	F4	F5		
Define spindle override ranges colours	SERVICE	F4	F5	F8	
Define feed override ranges	SERVICE	F4	F6		

FUNCTION	CONTEXT	1st	2nd	3rd	FIELD
Define display of axes properties	SERVICE	F4	F7		
Define plc visualisations	SERVICE	F4	F8		
Define history settings	SERVICE	F4	F9		
Define messages colours	SERVICE	F4	F10		
Define editor and extensions properties	SERVICE	F5			
Define NUMcoss parameters	SERVICE	F5	F6		
Install FXCAM application	SERVICE	F5	F7		
Define machine data	SERVICE	F6			
List and edit CNC parameters	SERVICE	F6	F3		
List and edit drive parameters	SERVICE	F6	F4		
Backup system	SERVICE	F6	F5		
Restore system	SERVICE	F6	F6		
Define probe settings	SERVICE	F6	F7		
Delete non-volatile ram on NCK	SERVICE	F6	F8		
Restart the system	SERVICE	F6	F9		
Change advanced password	SERVICE	F6	F10		
Open oscilloscope	SERVICE	F7			

Page deliberately empty

Published by NUM Spa

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

Printed March 2023

CNC HighEnd Applications