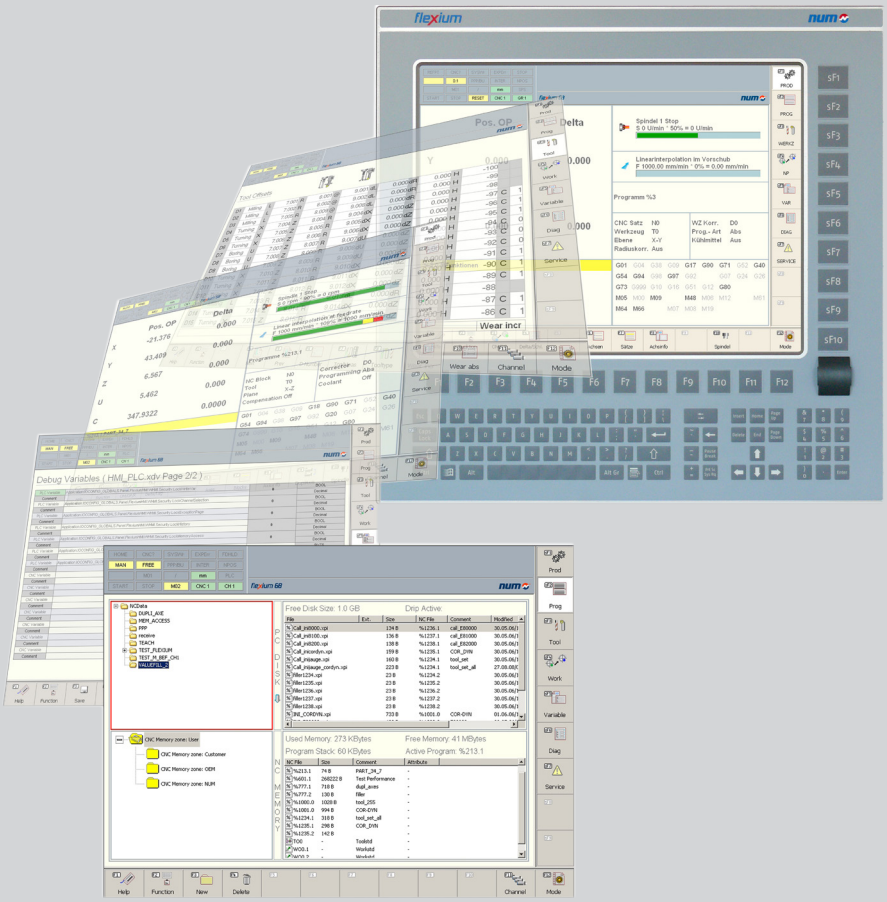
Flexium HMI

Operator Manual
Additional Functions

M00016EN-02

Edition 05.2013



WWW.NUM.COM

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.

Summary

Page deliberately empty.

Summary

Page deliberately empty.

1 Foreword

This manual is an addition to the Basic Operator Manual (Reference: M00017) of the Flexium Human Machine Interface (HMI) for Flexium systems.

It 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 CoDeSys visualization elements in the contexts sF1, sF8 or sF9

1.1 Document revisions

Date	Index	Reason for revision
09 - 2008	00	Document creation
05 - 2010	01	Revised to conform to software version 3.1.0.10
05 - 2013	02	Revised to conform to software version 3.4.0.0 New chapters: 2.3 : Teach-in directly in NC memory 6.0 : PLC visualization.

1.2 Use of the 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.

1.3 NUM Agency

The list of NUM agencies is given click here http://www.num.com

1.4 Index

The general index at the end of the manual facilitates access to infomation by keywords.
Please click here index

Page deliberately empty.

Page deliberately empty.

Summary

Page deliberately empty.

2 Teach-in

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

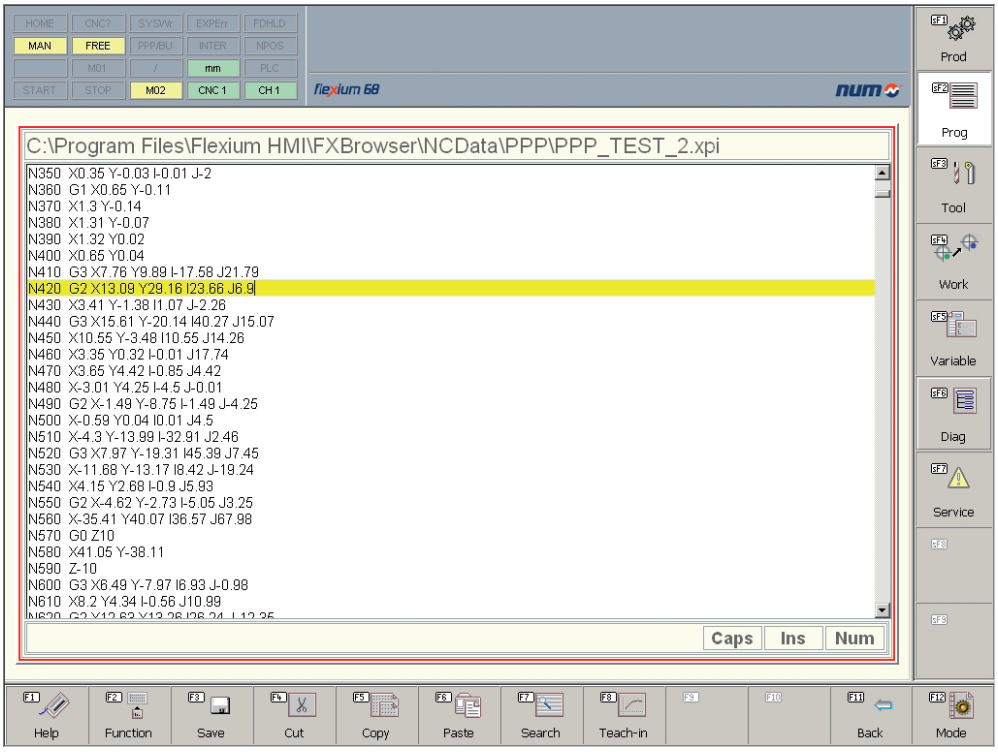
In addition to axes positions, CNC functions like G, M, F etc. may be entered in the programme. Teach-in is possible in part programmes (*.xpi) as well as in usinage files (*.usi).

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

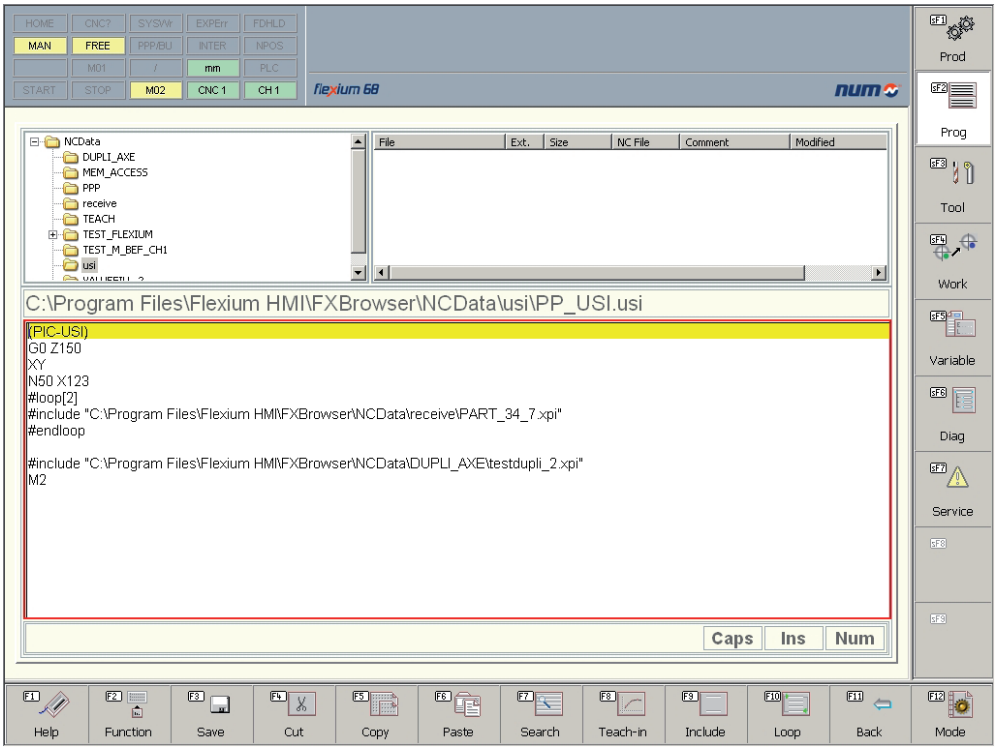
2.1 Access to the Teach-in function

Select the context “Programming	☞ Shift + F2
Select the right hand window of PC Disk	☞ 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:



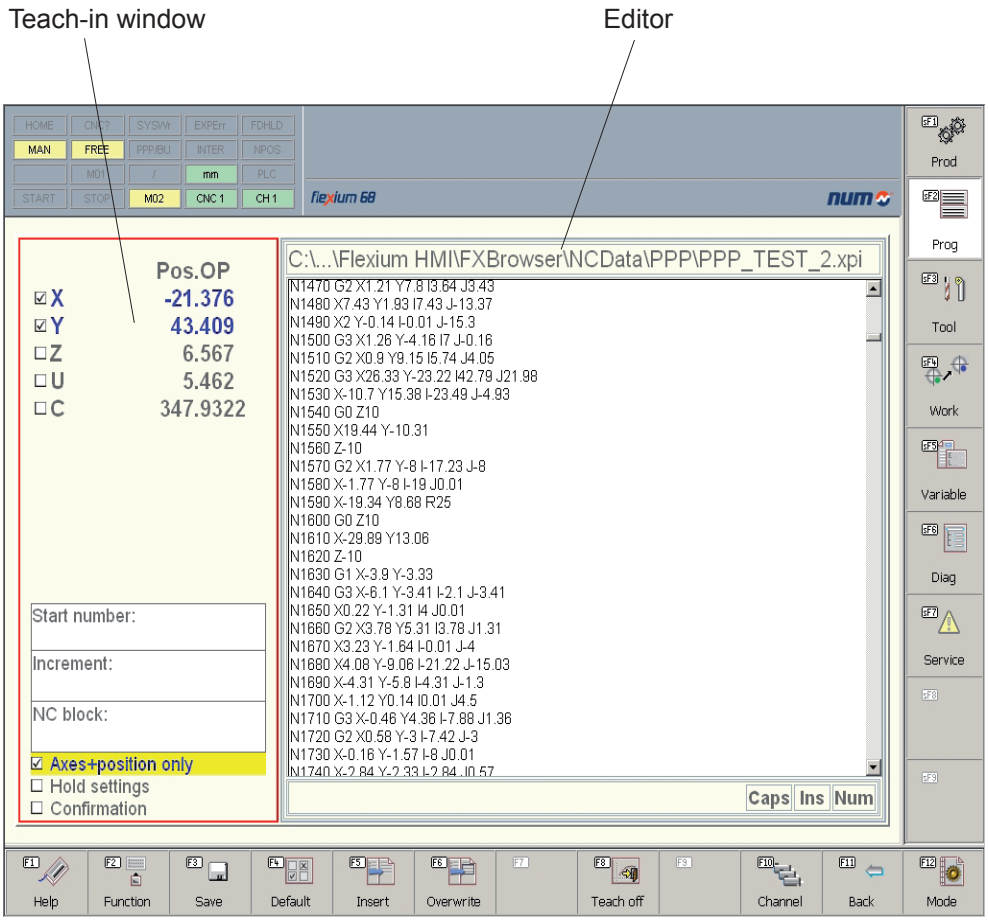
or for usinage files:



To activate the teach-in function	☞ F8

2.1.1 The 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 ↵

2.1.2 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.

2.2 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 .

2.3 Teach-in directly in NC Memory

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

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

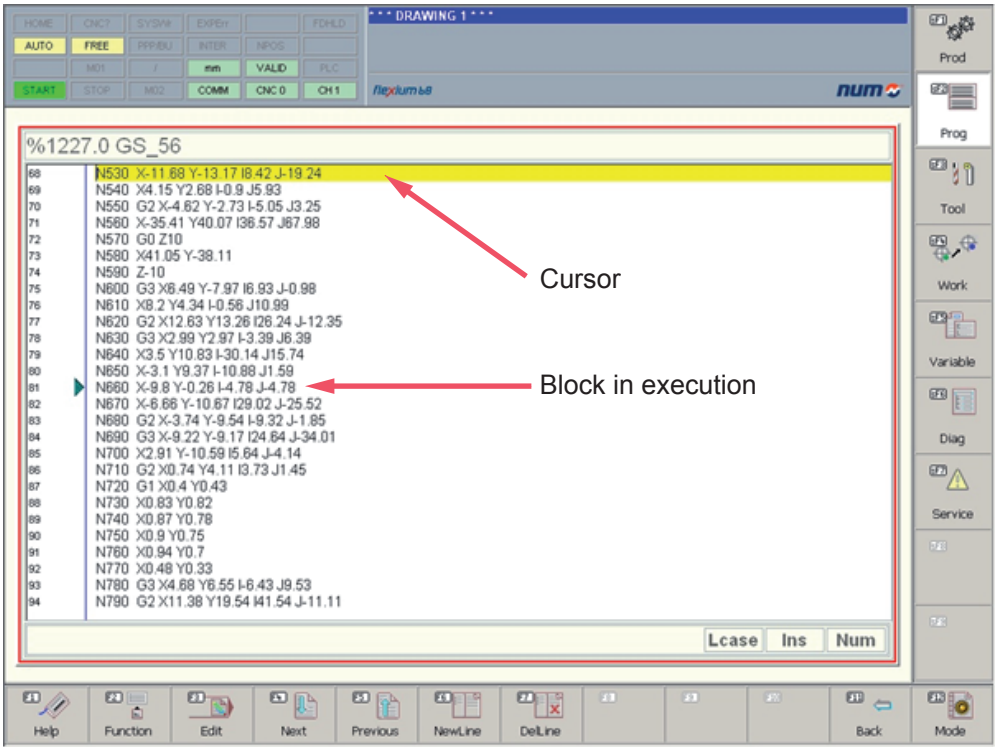
Procedure:

During part program execution, select the context “PROG” and memory area “User”.

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

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

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

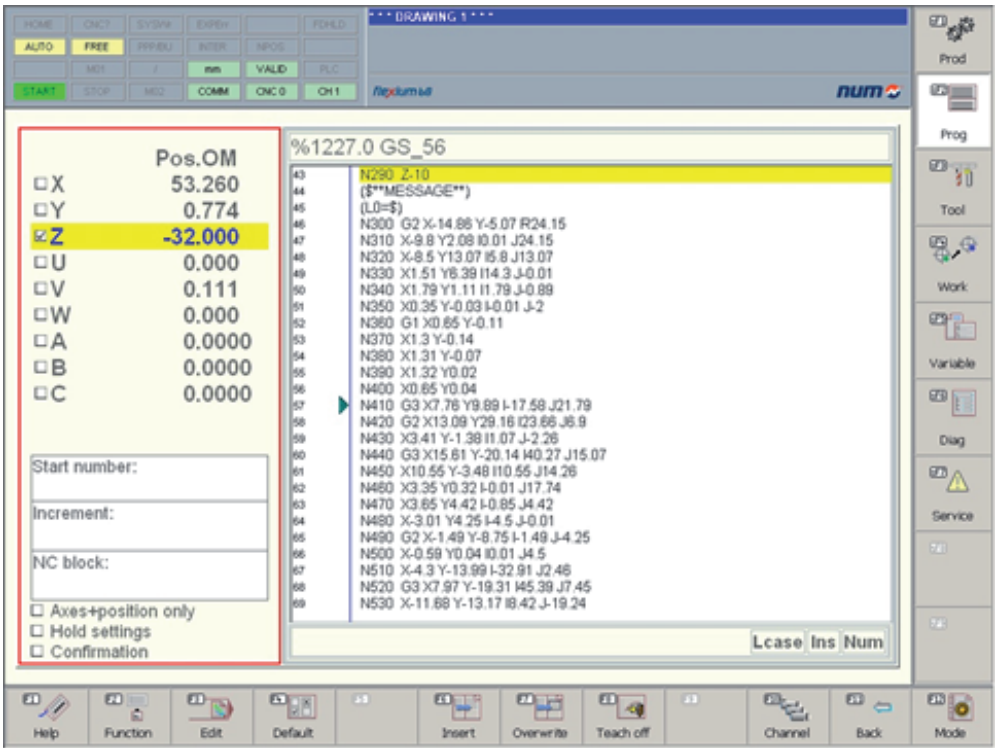


Possible actions

Press F11 to leave the editor
Press 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 Enter. Pressing Enter does not create a new line, rather this is performed by the function key F6.

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

Page deliberately empty.

Page deliberately empty.

Summary

Page deliberately empty.

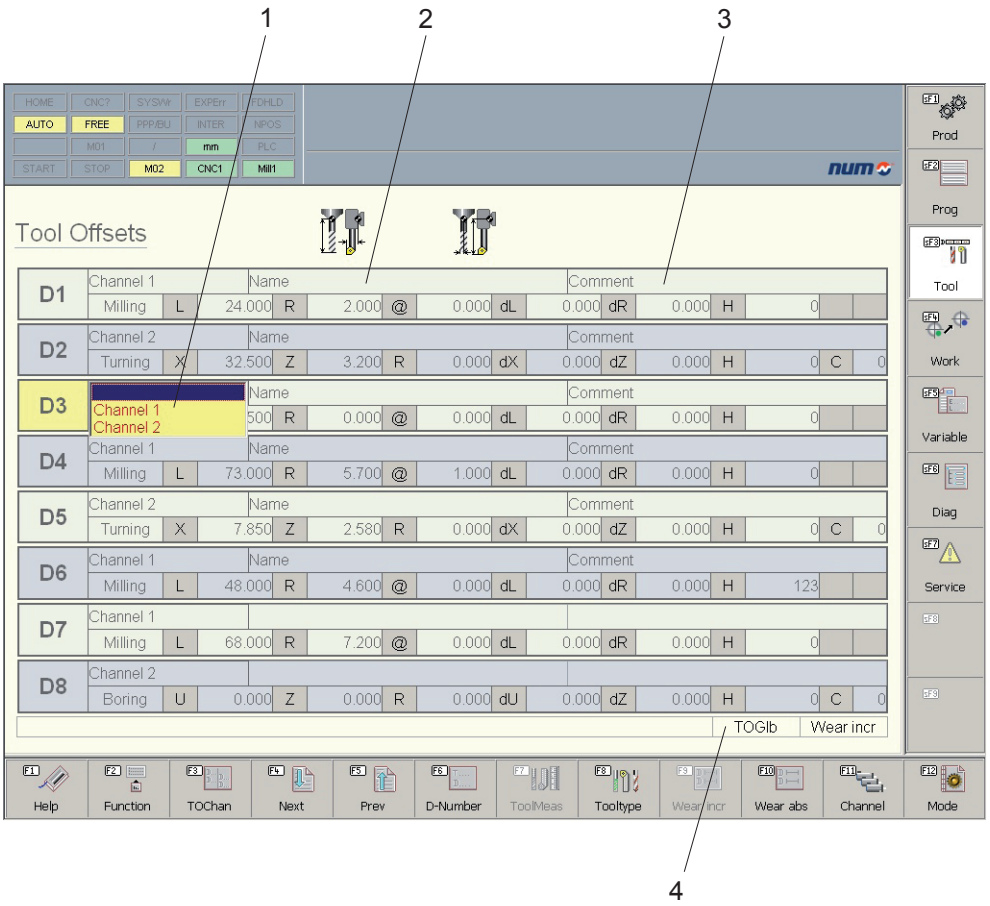
3 Extended tool table

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.

3.1 Displays

There are two kinds of tool table display.



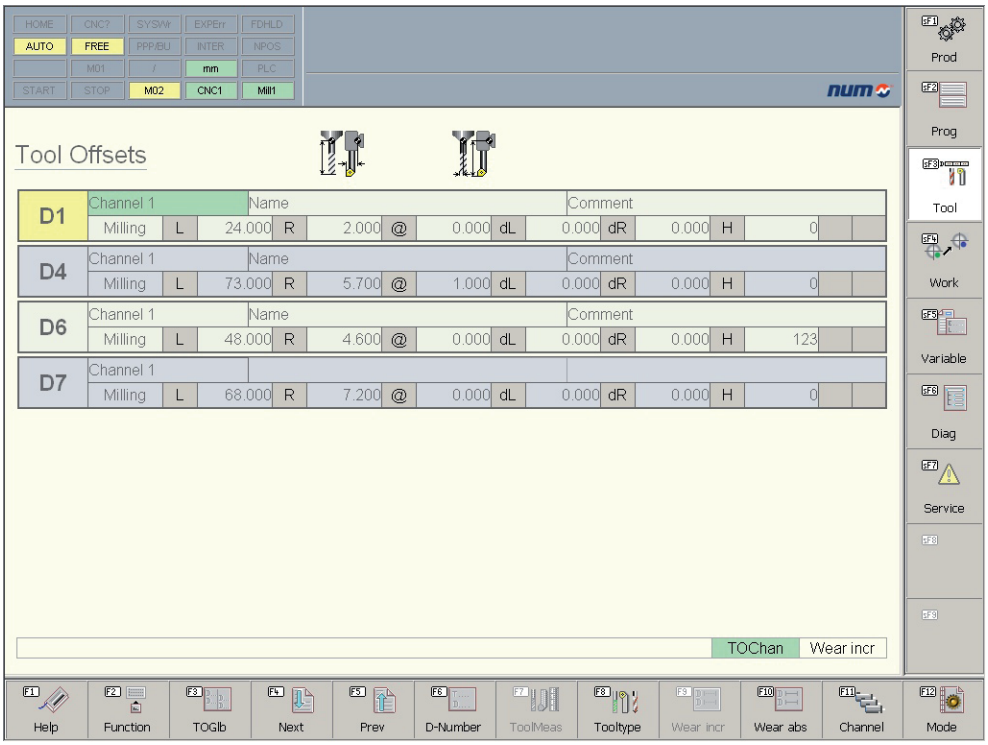
1 - Channel selection
2 - Name or comment field
3 - Name or comment field
4 - Kind of display

The global display shows all tools.

To toggle between displays	☞ F3

3.1.1 Channel display

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



3.2 Channel assignment and comments

For additional information on entering tool dimensions, refer to chapter “Context Tool” of the basic Flexium HMI operator manual.

3.2.1 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

3.2.2 Assigning a name and a comment

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

Page deliberately empty.

Summary

Page deliberately empty.

4 Symbolic names

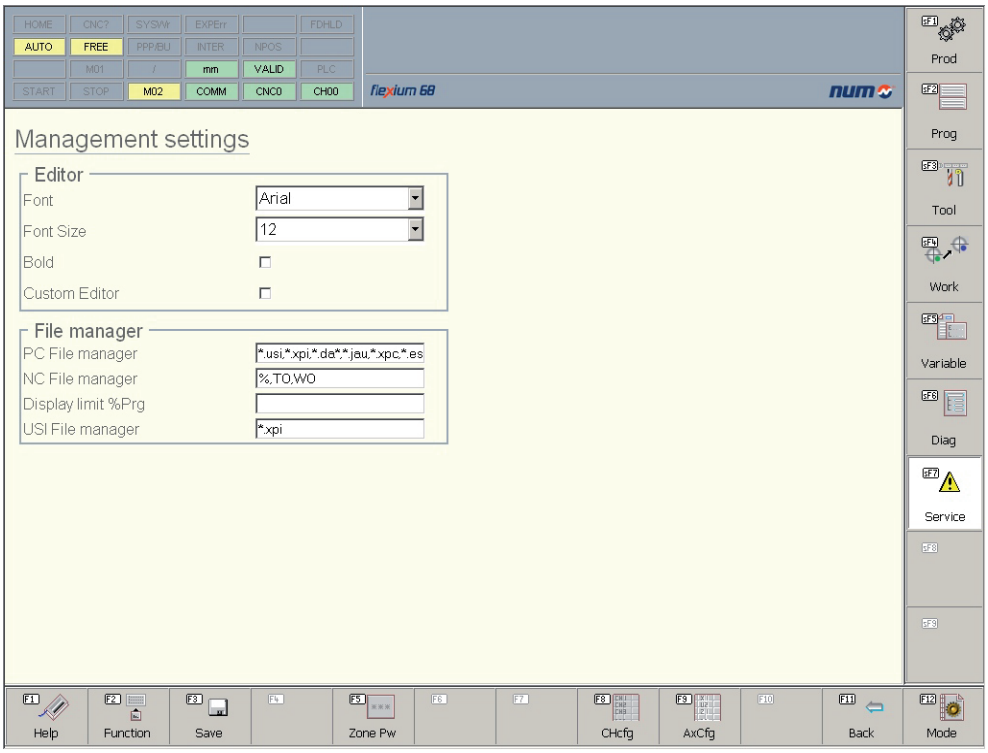
Gives the possibility to assign names to the axes, axes channels and CNC. These are displayed instead of usual address numbers

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

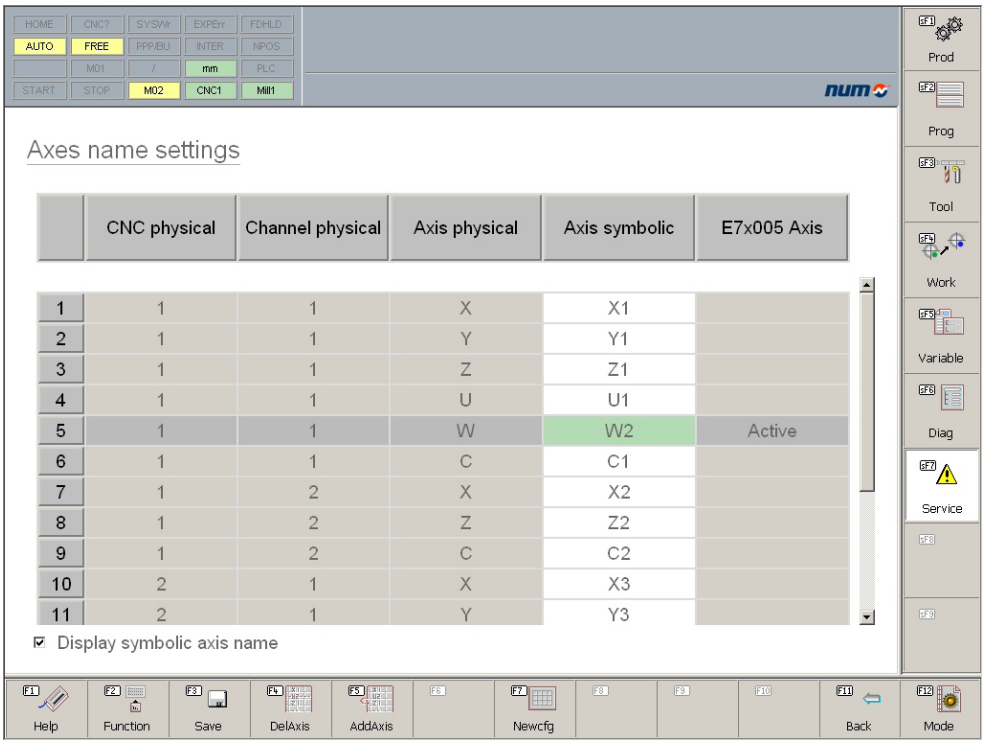
This opens the management settings page



Possible actions

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

4.2 Axis 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

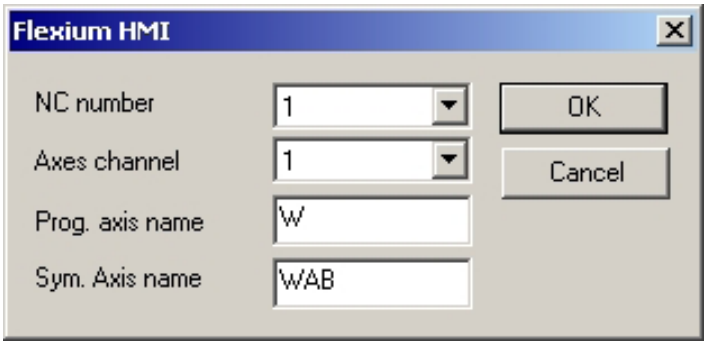
4.3 Entering axes names

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 ↵

4.4 Giving a name to an axis address assignment

For additional information on axis address assignment, refer to the Flexium CNC programming manual M00018.

Select the function “Add axis”	☞ F5



Move the cursor to the field “CNC 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

4.5 Deleting the name given to an axis address assignment

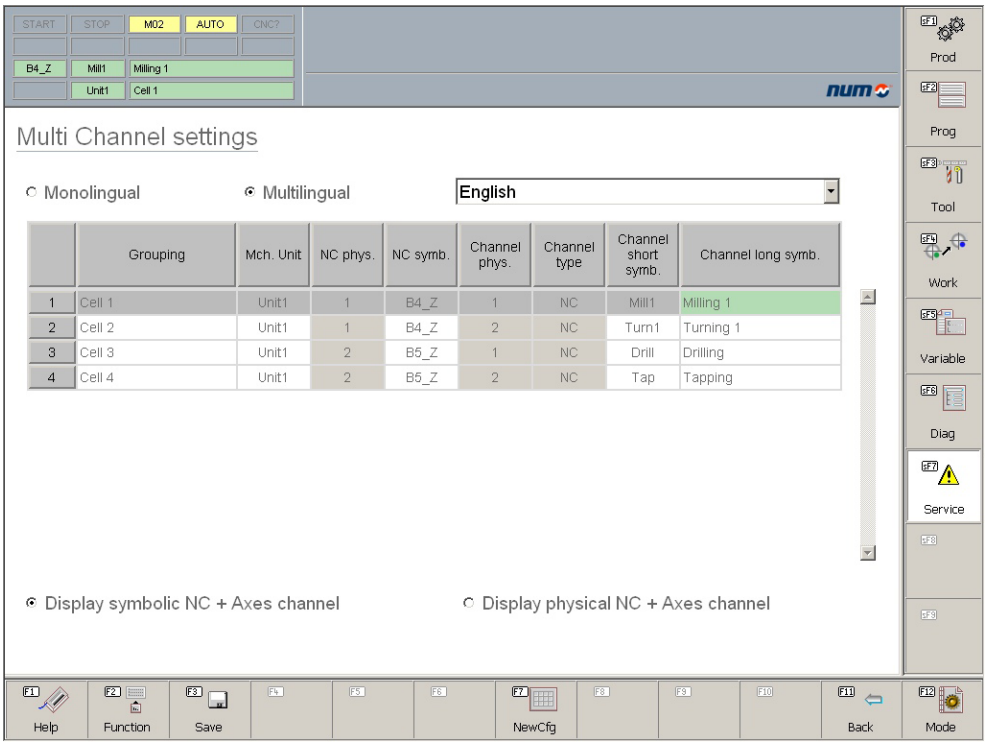
Move the cursor to the axis to be deleted	☞ ↑ or ↓
Delete the assignment	☞ F4

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

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

4.7.1 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.

4.7.2 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

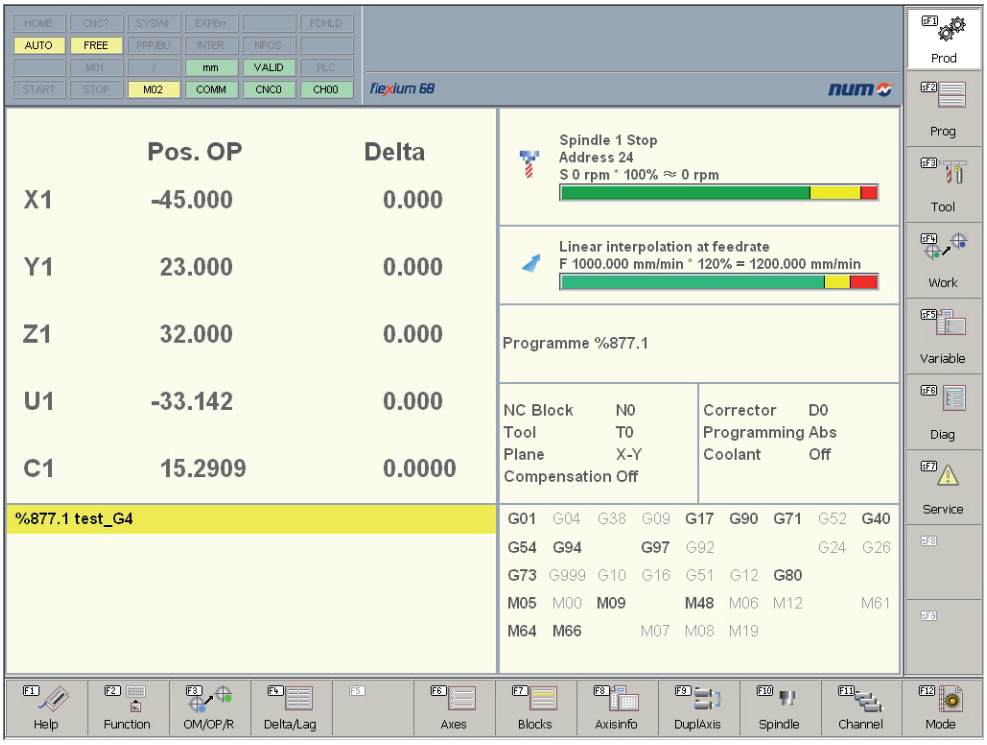
4.7.3 Entering names

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

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”:

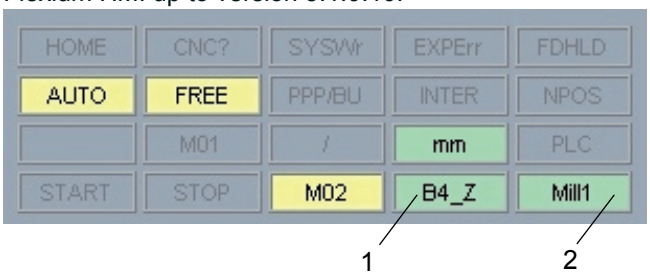


4.9 Display in the status window

The status window can be displayed in two different ways.

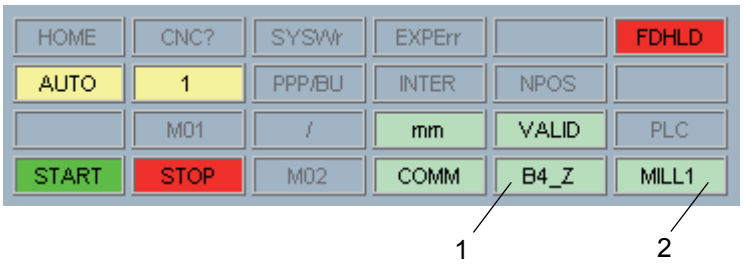
4.9.1 Status window with short symbolic names

Flexium HMI up to version 3.1.0.10:



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

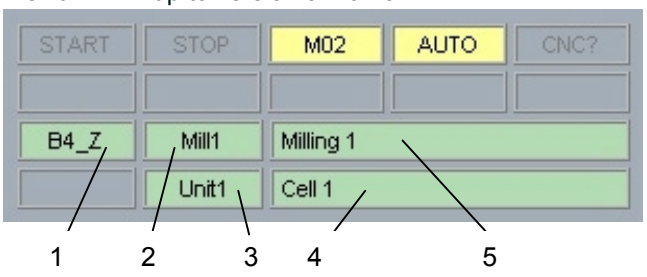
Flexium HMI from version 3.2.0.0 on:



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

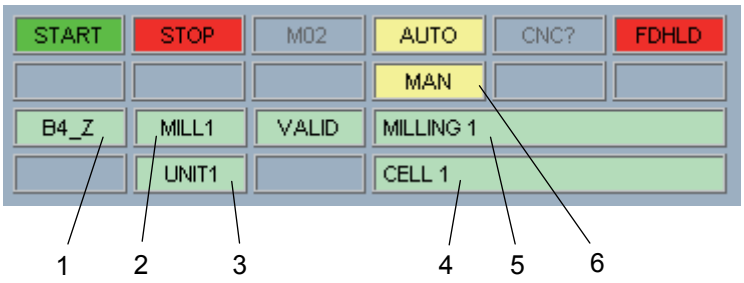
4.9.2 Status window with long symbolic names

Flexium HMI up to version 3.1.0.10:



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.

Flexium HMI from version 3.2.0.0 on:



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)

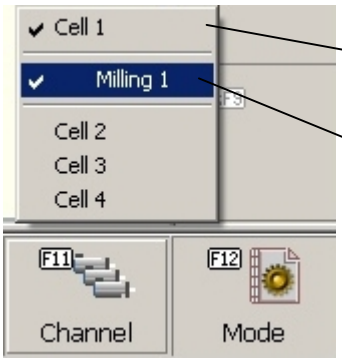
The other field have the same meaning as described in the chapter ‘Status window’ of the basic Flexium HMI operator manual.

4.9.3 Selecting the status window display

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

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.



Name of the grouping

Long symbolic name of the channel

4.10.1 Channel selection procedure

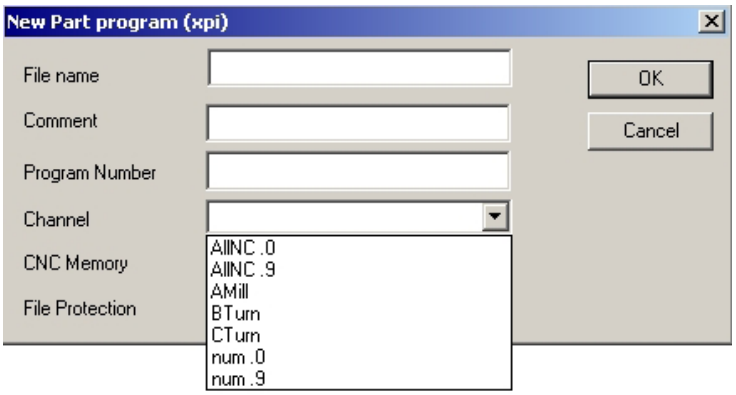
Open the channel selection	☞ F11
Move the cursor to the desired grouping	☞ ↑ or ↓ and Enter ↵
Move the cursor to the desired channel	☞ ↑ or ↓ and Enter ↵

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.

4.11.1 Example of creation of part programme



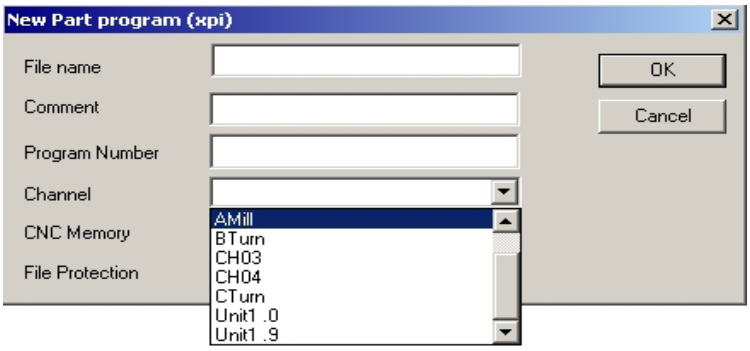
Channel assignment: Assigns the file to the settings of the axes channel configuration. The file gets automatically the number of the corresponding channel.

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.

4.11.2 Particularities with the function Multi-CNC

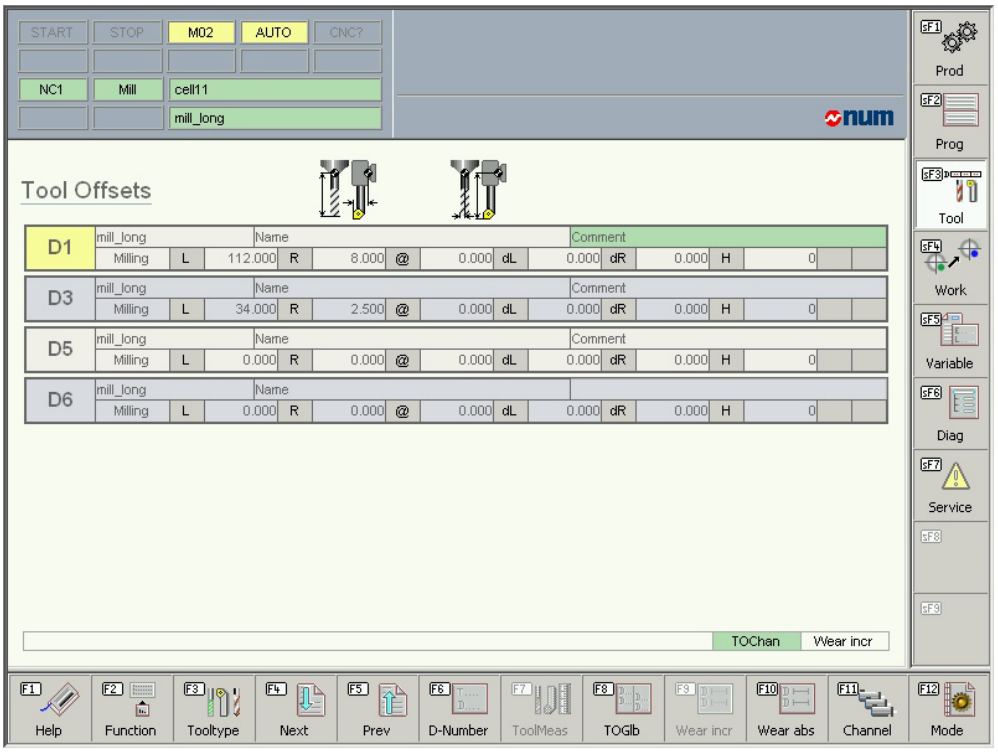
If the function Multi-CNC is activated in the system, an additional assignment to the Machine Units can be carried out. It is thus possible to load part programmes (*.xpi) as well as compact part programmes (*.xpc) automatically into all CNC belonging to a defined Machine Unit.



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.

4.12 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.



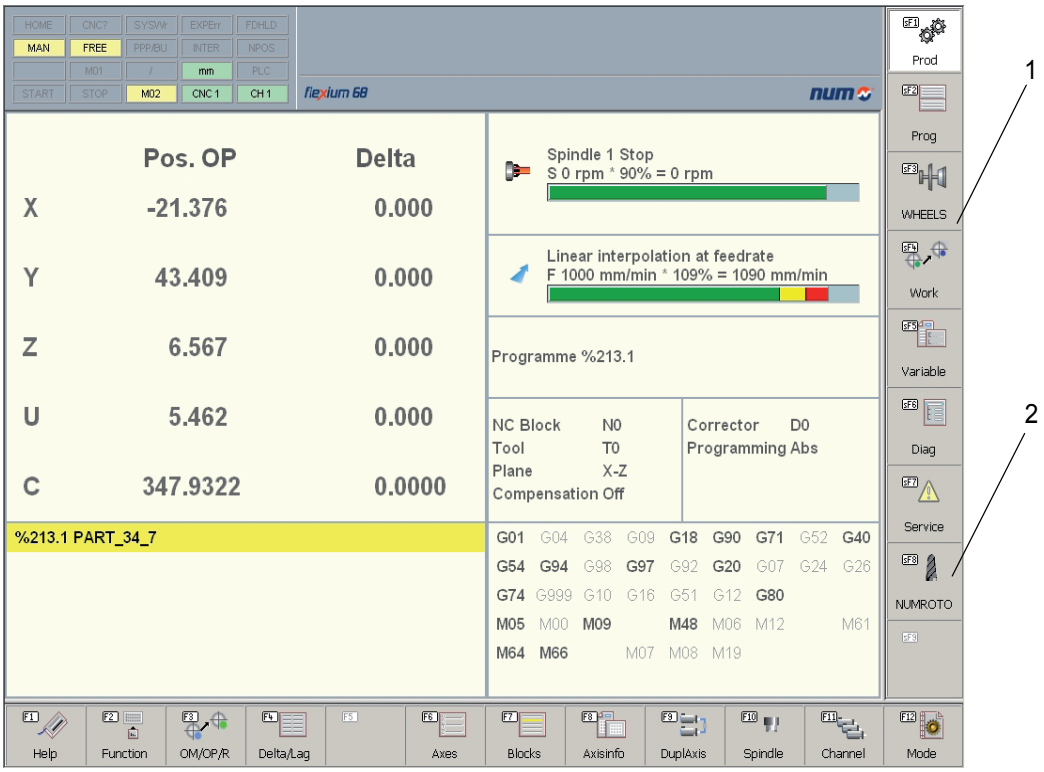
Page deliberately empty.

Summary

Page deliberately empty.

5 Extensions for NUMROTOplus®

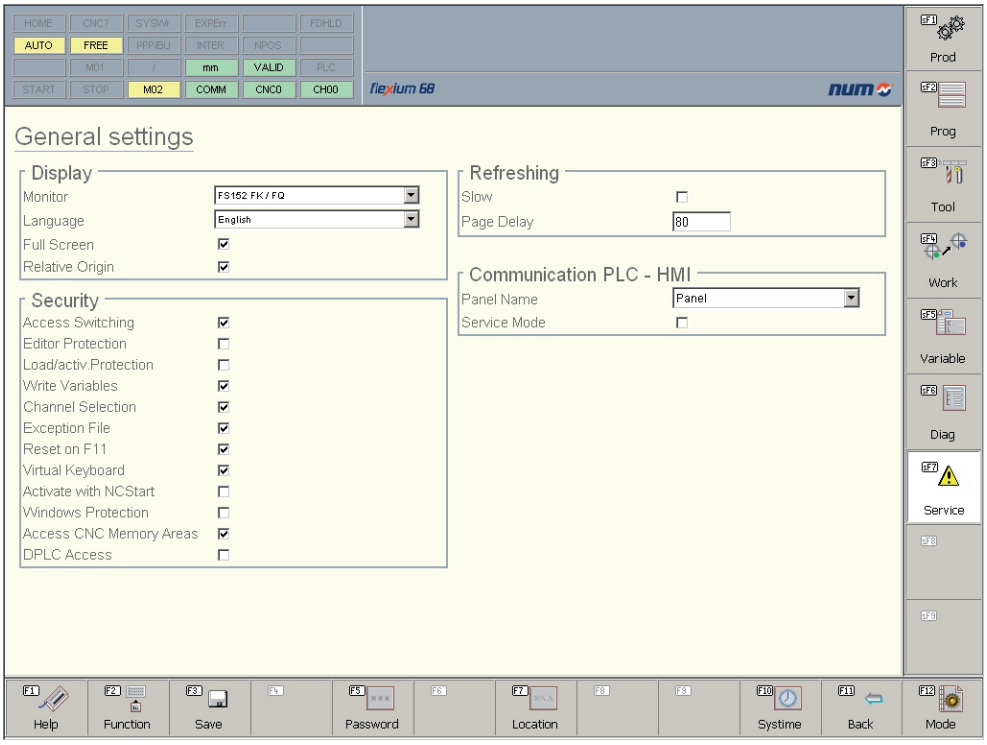
5.1 Particularities in the context bar



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

5.2 Setting in the context “Service”

The page “General settings” in the context “Service”



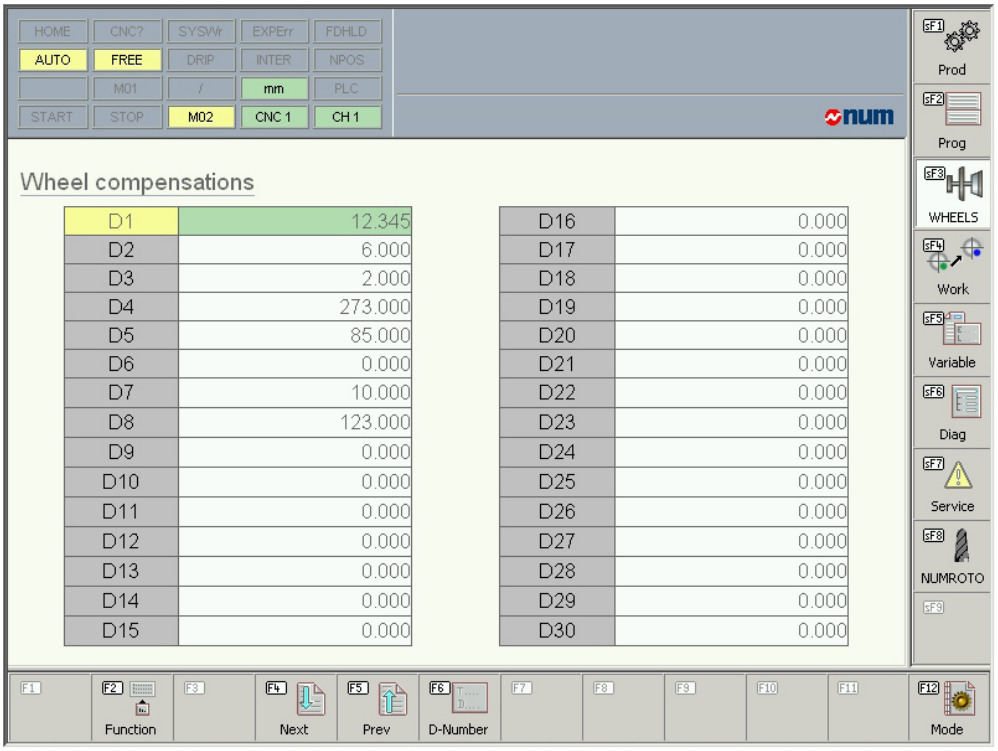
5.2.1 Explanation of the setting options in the field “Display”

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.

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↵

5.3.1 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↵

Page deliberately empty.

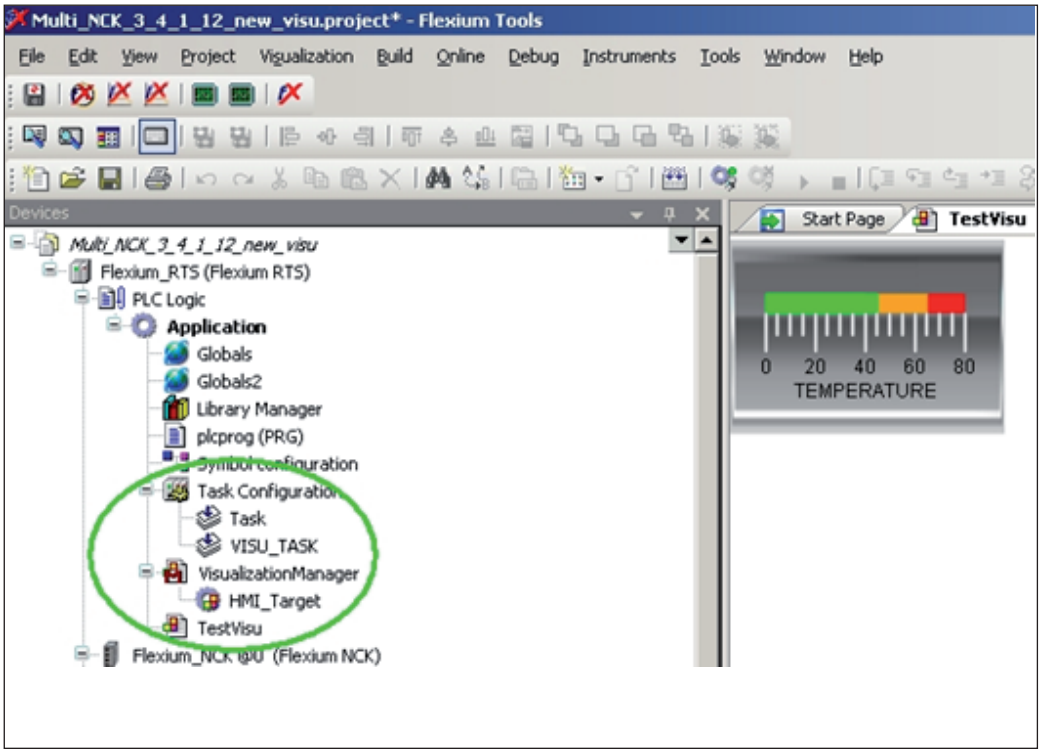
Summary

Page deliberately empty.

6 PLC visualization

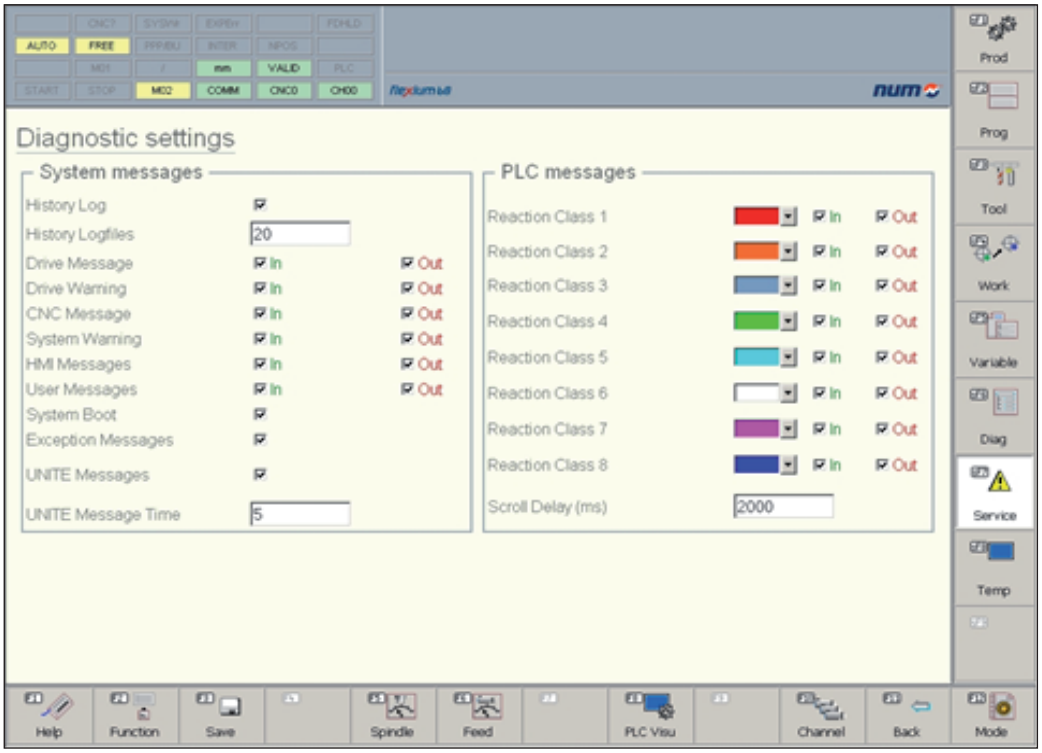
6.1 Integrated PLC visualization

From the Flexium HMI 3.4.0.0 on, it is possible to display CoDeSys visualization elements in the contexts sF1 “Production”, sF8 or sF9.

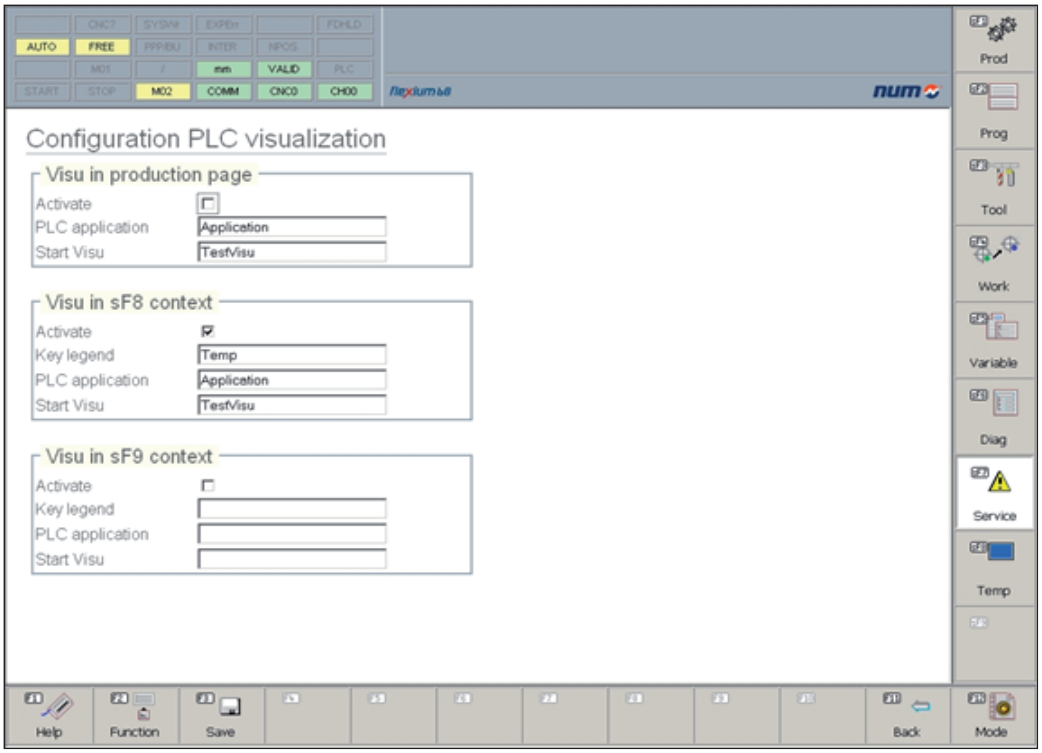


Settings

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



☞ F8 to access the visualization settings page

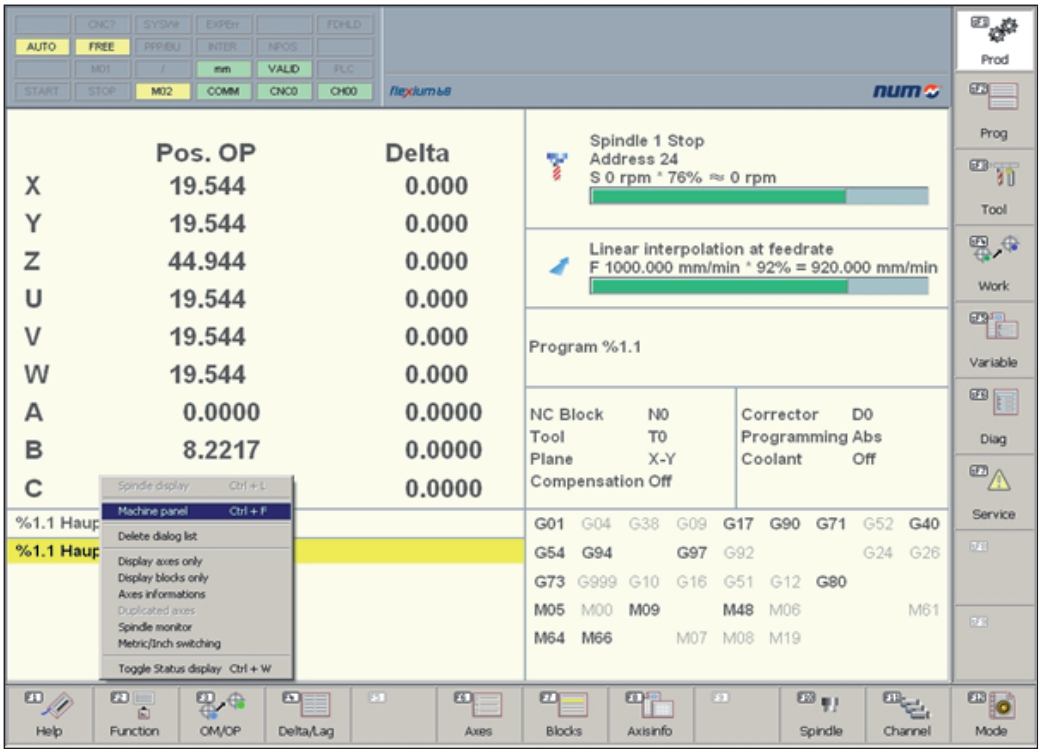


You may now select the desired location for your visualization display.
Enter a key legend, the PLC application and the visualization names. To activate the display, check the appropriate box.

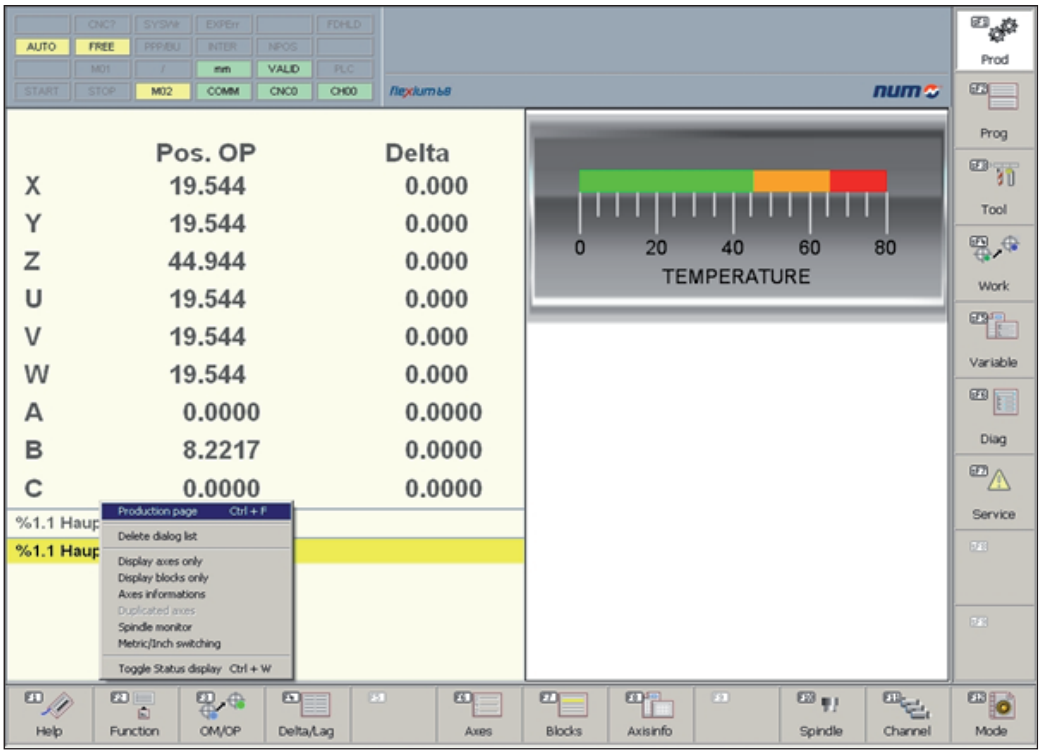
You have to close and restart Flexium HMI for the changes to take effect.

The visualization displays are now ready to be used.

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

Index

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 May 2013

CNC HighEnd Applications