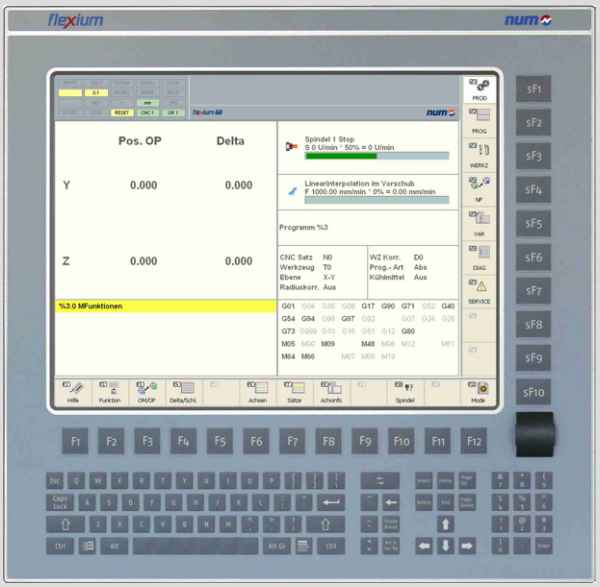
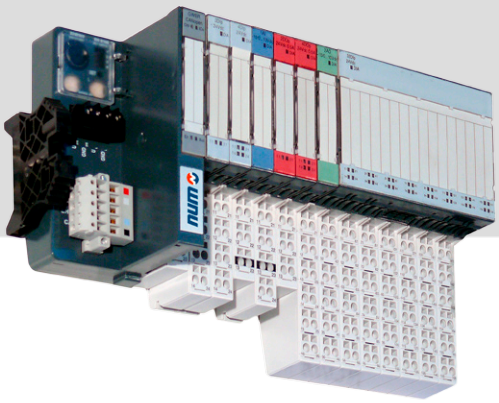
Flexium

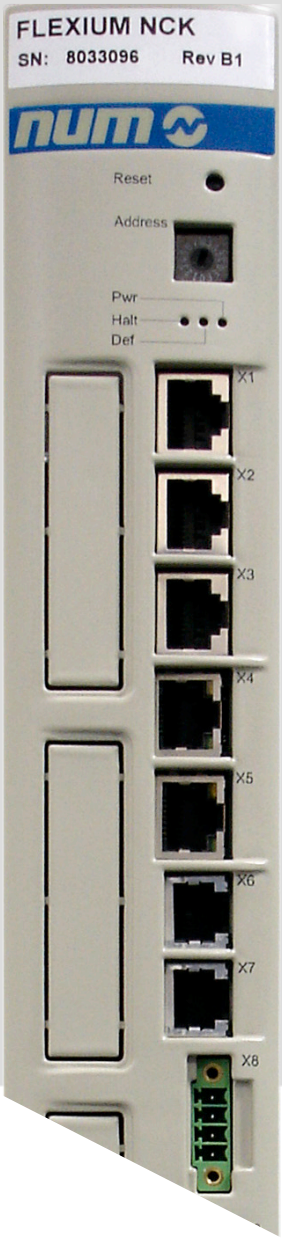
Parameters manual

M00021EN-00

Edition 11.2008







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.

Summary

1 Foreword 9
1.1 Use of the manual 9
1.2 Use of symbols in the manual 9
1.3 Document revisions 9
1.4 How to read the manual 10
1.5 NUM Agency 10
1.6 Index 10

2 Parameter Definition 15
2.1 Introduction 15
2.2 Rules for Writing the Parameters 15
2.2.1 8-bit and 16-bit Hexadecimal Words 15
2.2.2 32-bit Hexadecimal Words 16
2.2.3 Decimal Words 16
2.3 PLC Variable Mnemonics 17

3 List of Machine Parameters 21

4 Axis Declaration 27
4.1 Number of Channels in the System ▶ P97 27
4.2 Measured Axes ▶ P2 28
4.3 Displayed Axes ▶ P0 29
4.4 Modulo or Limited Excursion Rotary Axes ▶ P1 30
4.5 Servo-Controlled and Interpolated Axes ▶ P3 32
4.6 Channel Machine Configuration ▶ P5 33
4.7 Axes Programmable by Diameter and Internal Measurement and Dimension Acquisition and Display Format ▶ P4 34
4.8 Axes with Clamps ▶ P8 40
4.9 Axis Assignment to a Channel ▶ P9 42
4.10 Handwheel Declaration ▶ P14 45
4.11 Coupling Assignment for Duplicated and Synchronised Axes ▶ P27 48
4.12 Synchronised Axis Coupling Enable ▶ P28 50
4.13 Carrier or Carried Axes ▶ P64 51
4.14 Axes with Quantified Movements ▶ P65 52
4.15 Axis and Spindle Switching with Homing Preserved ▶ P70 53
4.16 DISC NT Axis presence ▶ P71 56
4.17 Tandem Function: master/slave coupled axes ▶ P85 57
4.18 Tandem Function: defines master axes with torque synchronisation ▶ P86 60

5 Measurement 65
5.1 Axis Measurement Direction ▶ P10 65
5.2 Axis Measurement Conversion Coefficient ▶ P11 66
5.3 Handwheel Measurement Direction ▶ P12 67
5.4 Handwheel Measurement Conversion Coefficent ▶ P13 68
5.5 Poor Signal and Encoder Channel Complementarity Check Declaration ▶ P25 70
5.6 Poor Signal and encoder Channel Complementarity Check ▶ P26 72
5.7 Minimum Time Between Two Consecutive Encoder Faults ▶ P77 73

6 Servo-Control 77
6.1 Maximum Axis Traverse Rates ► P30 77
6.2 JOG Speeds and Reduced Speeds ► P31 78
6.3 Maximum Permissible Acceleration ► P32 79
6.4 Direction of Axis Speed Reference ► P20 82
6.5 Servo-System Loop Gain Coefficient ► P21 83
6.6 Servo-Loop Time Constant ► P56 87
6.7 In-Position Window ► P22 88
6.8 Maximum Following Error ► P23 89
6.9 Dynamic Movement Control ► P57 91
6.10 Speed Anticipation Coefficient and Number of Filtered Terms for High Speed Cutting Machining ► P55 93
6.11 Characteristics of Synchronised Axis Pairs ► P24 97
6.12 Servo-Loop Error Tolerated on Circles ► P52 99
6.13 High Speed Cutting Machining ► P19 103
6.14 Approach Speed ► P33 105
6.15 Setting of the Speed Buildup in the Path ► P53 106
6.16 Time Constant per Axis ► P66 110
6.17 JOG Speed Setting 111
6.18 General Block Diagram of Servo-Control 112

7 Axis Travels 117
7.1 Reminders 117
7.2 Homing Direction ► P15 120
7.3 Datum Switch Position in Machine Dimensions ► P16 122
7.4 Axis Travel Limits ► P17 124
7.5 Backlash Error Compensation ► P18 126

8 Spindles 131
8.1 Principle of Bidirectional Spindle Indexing 131
8.1.1 Spindle Indexing 131
8.1.2 Indexed Stop Position 131
8.1.3 Indexed Stopping Condition 131
8.1.4 Spindle Speed Control by Bidirectional Servo-Loop 132
8.1.5 Spindle Indexing at Constant Acceleration 133
8.2 Rigid Tapping 134
8.2.1 Procedure 134
8.2.2 Feed Stop (Stop) 134
8.3 Spindle Declaration and assign spindle to the channels ► P6 135
8.4 Spindle Measurement Conversion ► P40 137
8.5 Spindle Reference Reversal ► P41 139
8.6 Spindle 1 Speed Ranges ► P46 140
8.7 Spindle 2 Speed Ranges ► P47 143
8.8 Spindle 3 Speed Range ► P48 146
8.9 Spindle Speed Ranges ► P49 149
8.10 Spindle Origins ► P42 152
8.11 Maximum Speed During Spindle Indexing and Maximum Speed of Spindles by PLC Programme ► P43 153
8.12 Spindle Indexing In-Position Window ► P44 155
8.13 Spindle Indexing Servo-Loop Gain ► P45 157
8.14 Spindle Acceleration and Minimum Spindle Reference in Hole Bottom ► P62 159
8.15 Cycle Axis Integration Time Constant, Cycle Axis Position Anticipation Time Constant and Spindle Zero Crossing Anticipation Time Constant ► P63 161

9 Miscellaneous Functions 167
9.1 Miscellaneous Function Parameter Setting ► P7 167
9.2 Subroutine Call by M Function ► P35 170
9.3 Sampling Period ► P50 171
9.4 Minimum Block Execution Time ► P51 172
9.5 Number of NC blocks used per channel for look ahead analysis ► P54 173
9.6 Interaxis Calibration Table and Programme Stack Reservation ► P58 174
9.7 Potentiometer Min-Max Range ► P67 175
9.8 Memory Area Allocation ► P95 176
9.9 Communication period – Address of master CNC – Timeout for the monitoring of PLC application counter ► P99 177
9.10 Backup in Path, Auto Recall after INTER ► P114 178

Page deliberately empty

Page deliberately empty

Page deliberately empty

Summary

1 Foreword 9
1.1 Use of the manual 9
1.2 Use of symbols in the manual 9
1.3 Document revisions 9
1.4 How to read the manual 10
1.5 NUM Agency 10
1.6 Index 10

Page deliberately empty

1 Foreword

1.1 Use of the manual

This manual is for use by qualified personnel only.

To reduce the risk due to an improper use of the product, do not perform any servicing other than that contained in this manual unless you are qualified to do so.

To ensure proper use of these products, please always read this User's manual carefully and retain for future reference. Should the unit require maintenance, please contact a NUM authorized service center only.

1.2 Use of symbols 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 Document revisions

Date	Index	Reason for revision
11 - 2008	00	Document creation

1.4 How to read the manual

The machine parameter manual gives information on how to set the parameters to customise the Flexium for a particular machine-tool.

Chapter 1 - Foreword:
- Use of the manual.

Chapter 2 - Parameter definition:
- Writing rules and PLC Variable Mnemonics.

Chapter 3 - List of Machine Parameters.

Chapter 4 - Axis declaration parameters:
- Machine setup data
- Special information.

Chapter 5 - Measurement setting parameters:
- Machine setup data and Axis Measurement
- Special information.

Chapter 6 - Axis servo-control parameters:
- Machine Servo-Control setup data
- Special information.

Chapter 7 - Axis travel setting parameters:
- Machine setup data
- Special information.

Chapter 8 - Spindle setting parameters:
- Machine setup data
- Special information.

Chapter 9 - Miscellaneous parameters:
- Machine setup data
- Special information.

1.5 NUM Agency

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

1.6 Index

The index at the end of the manual facilitates access to infomation by keywords.

Page deliberately empty

Page deliberately empty

Summary

2 Parameter Definition 15
2.1 Introduction 15
2.2 Rules for Writing the Parameters 15
2.2.1 8-bit and 16-bit Hexadecimal Words 15
2.2.2 32-bit Hexadecimal Words 16
2.2.3 Decimal Words. 16
2.3 PLC Variable Mnemonics 17

Page deliberately empty

2 Parameter Definition

2.1 Introduction

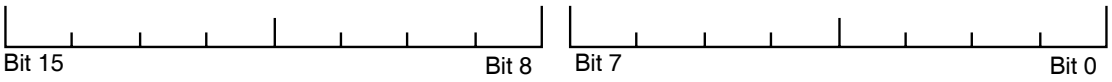
The machine parameters are used to dedicate the CNC system to a particular machine-tool.

2.2 Rules for Writing the Parameters

2.2.1 8-bit and 16-bit Hexadecimal Words

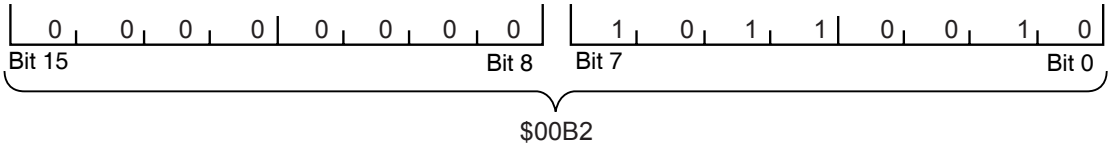
Each bit represents a particular function or system configuration (e.g. channel number). It is enabled when the bit is set and inhibited when the bit is zero.

Word Format



Example

Bits 1, 4, 5, 7 of word 1 are set.



Value of the word:	00B2

2.2.2 32-bit Hexadecimal Words

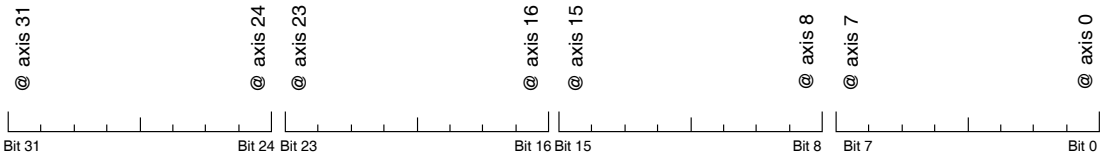
Each bit corresponds to the physical address of an axis.

The assignment principle is as follows:

- the axis with physical address 0 is assigned to bit 0,
- the axis with physical address 1 is assigned to bit 1,
- the axis with physical address 2 is assigned to bit 2,
- and so forth up to the axis with physical address 31.

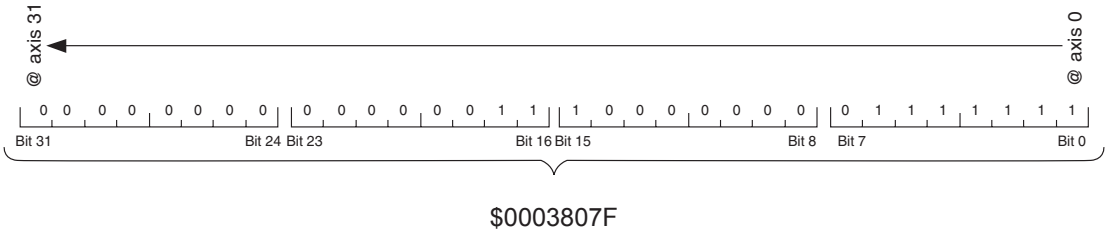
The physical address of an axis is assigned by the switch on the drive.

Word Format



Example

Bits 0, 1, 2, 3, 4, 5, 6, 15, 16 and 17 are set.



Value of the word:	0003807F

2.2.3 Decimal Words

Each word contains a signed or unsigned decimal value.

When entering the values, comply with the units specified for each type of word.

2.3 PLC Variable Mnemonics

The table below gives the addresses of the PLC variables corresponding to the mnemonics mentioned in this manual.

Mnemonic	Identification
WCNC.Channel[n].AckMFunction	Channel n M function Acknowledge
Spindle[n].PositionReached	Spindle n in position
RCNC.CNCReady	CNC ready
W_ALLCNC.ReducedSpeed	Low speed
Spindle[1].ReducedSpeed	Enabling of spindle 1 speed limiting
W_ALLCNC.ManualSpeed.SlowSpeed	JOG 1 selection
W_ALLCNC.ManualSpeed.FastSpeed	JOG 2 selection

Page deliberately empty

Summary

3 List of Machine Parameters 21

Page deliberately empty

3 List of Machine Parameters

Parameter	Category	Description	Page
P0	Axis declaration	Displayed axes	29
P1	Axis declaration	Modulo and limited excursion rotary axes	30
P2	Axis declaration	Measured axes	28
P3	Axis declaration	Servo-controlled and interpolated axes	32
P4	Axis declaration	Axes programmed by diameter Internal system measurement and dimension acquisition and display format	34
P5	Axis declaration	Channel machine configuration, Lathe or milling machine channels, Front/Rear turrets	33
P6	Spindles	Number of controlled spindles, Spindle type, Spindle assignments to the channels	135
P7	Miscellaneous	- Initialisation in inches or metric units, - Direction of automatic spindle search. - Transmission of T and M functions to the machine processor in SEARCH and TEST modes. - Forcing of T functions, - Interaxis calibration table enable, - Initialisation in G94 or G95, - Forcing of manual mode, - Homing selection Subroutine call on reset. - Feed stop with rigid tapping. - Speed variation with gradual acceleration.	167
P8	Axis declaration	Axes with clamps	40
P9	Axis declaration	Axis assignment to channels	42
P10	Measurements	Axis measurement direction	65
P11	Measurements	Axis measurement conversion coefficient	66
P12	Measurements	Direction of handwheel measurement	67
P13	Measurements	Handwheel measurement conversion coefficient	68
P14	Axis declaration	Handwheels	45
P15	Axis travel	Direction of homing Switch status test	120
P16	Axis travel	Reference switch position in machine dimensions	122
P17	Axis travel	Axis travel limit	124
P18	Axis travel	Backlash error compensation	126
P19	Servo-control	High Speed Cutting Machining	103
P20	Servo-control	Direction of axis speed reference	82
P21	Servo-control	Servo-system loop gain coefficient	83
P22	Servo-control	In-position window	88
P23	Servo-control	Maximum following error	89
P24	Servo-control	Synchronised axis control	97
P25	Measurements Measurements and axis declaration	Poor signal and encoder channel complementarity check declaration Poor signal and encoder channel complementarity check declaration. Declaration of connected measur. encoders	70 **
P26	Measurements	Poor signal and encoder channel complementarity check	72

Parameter	Category	Description	Page
P27	Axis declaration	Duplicated axis coupling definition	48
P28	Axis declaration	Synchronised axis coupling enable	50
P30	Servo-control	Maximum axis traverse rates	77
P31	Servo-control	JOG speed and reduced speed	78
P32	Servo-control	Maximum permissible acceleration	79
P33	Servo-control	Approach speed	105
P35	Miscellaneous	Subroutine call by M function	170
P40	Spindles	Spindle measurement conversion	137
P41	Spindles	Spindle reference reversal	139
P42	Spindles	Spindle origins	152
P43	Spindles	Maximum spindle speed Maximum spindle speed by PLC programme	153
P44	Spindles	Indexing in-position window	155
P45	Spindles	Spindle servo-system gain	157
P46	Spindles	Spindle 1 speed range	140
P47	Spindles	Spindle 2 speed range	143
P48	Spindles	Spindle 3 speed range	146
P49	Spindles	Spindle 4 speed range	149
P50	Miscellaneous	Sampling period (CNC)	171
P51	Miscellaneous	Minimum block execution time	172
P52	Servo-control	Servo-system error tolerated on circles	99
P53	Servo-control	Setting of speed buildup in the path	106
P54	Servo-control	Number of NC blocks used per channel	173
P55	Servo-control	Speed anticipation coefficient, number of terms used to calculate the filtered reference for very high speed machining	93
P56	Servo-control	Servo-loop time constant	87
P57	Servo-control	Dynamic movement control	91
P58	Miscellaneous	Interaxis calibration table reservation Programme stack size	174
P62	Spindles	Spindle acceleration Minimum spindle reference in hole bottom	159
P63	Spindles	-Cycle Axis Integration Time Constant Cycle Axis Position Anticipation time Constant - Spindle Zero Crossing Anticipation Time Constant	161
P64	Axis declaration	Carried or carrier axes	51
P65	Axis declaration	Axes with quantified movements	52
P66	Servo-control	Time constant per axis	110
P67	Miscellaneous	Potentiometer Min-Max range	175
P70	Axis declaration	List of 32-bit DISC NT axis measurem. Switchable digital axes or spindles	53
P71	Axis declaration	DISC NT Axis presence	56
P77	Measurements	Minimum time between two consecutive encoder faults so as not to trigger a CNC error	73

Parameter	Category	Description	Page
P85	Axis declaration	Tandem Function: master/slave coupled axes	57
P86	Axis declaration	Tandem function: defines master axes with torque synchronisation	60
P95	Miscellaneous	Part programme memory segment size	176
P97	Axis declaration	Number of channels in the system	27
P99	Miscellaneous	Communication period Address of master CNC Timeout for the monitoring of PLC application counter	177
P114	Miscellaneous	Backup in path, auto recall after INTER	178

Page deliberately empty

Summary

4 Axis Declaration 27

4.1 Number of Channels in the System ► P97 27
4.2 Measured Axes ► P2 28
4.3 Displayed Axes ► P0 29
4.4 Modulo or Limited Excursion Rotary Axes ► P1 30
4.5 Servo-Controlled and Interpolated Axes ► P3 32
4.6 Channel Machine Configuration ► P5 33
4.7 Axes Programmable by Diameter and Internal Measurement and Dimension Acquisition and Display Format ► P4 34
4.8 Axes with Clamps ► P8 40
4.9 Axis Assignment to a Channel ► P9 42
4.10 Handwheel Declaration ► P14 45
4.11 Coupling Assignment for Duplicated and Synchronised Axes ► P27 48
4.12 Synchronised Axis Coupling Enable ► P28 50
4.13 Carrier or Carried Axes ► P64 51
4.14 Axes with Quantified Movements ► P65 52
4.15 Axis and Spindle Switching with Homing Preserved ► P70 53
4.16 DISC NT Axis presence ► P71 56
4.17 Tandem Function: master/slave coupled axes ► P85 57
4.18 Tandem Function: defines master axes with torque synchronisation ► P86 60

Page deliberately empty

4 Axis Declaration

4.1 Number of Channels in the System ► P97

Description

Defines the maximum number of channels controlled by the system.

Principle

The system can control a maximum of eight channels. There two types of channels:
- CNC channels,
- Auxiliary channels.

Word N0 : specifies the number of CNC channels (min 1, max 8).

Word N1 : specifies the number of Auxiliary channels (min 0, max 7).

The system assigns the CNC channels in numerical order, starting with the first channel (channel 1) then the Auxiliary channels after the last CNC channel.

Maximum number of channels		
Product	CNC channels	Auxiliary channels
Flexium 6	1	0
Flexium 68	8	7

Programme Parameter (see "Flexium Programming Manual M or T).

Programme parameter E41102 is used to read the number of CNC channels.

Example

The system controls three CNC channels and two Auxiliary channels.

Word N0	03
Word N1	02

The channels are assigned as follows:

Channel number	CNC	Auxiliary
Channel 1	X	
Channel 2	X	
Channel 3	X	
Channel 4		X
Channel 5		X
Channel 6	Unassigned	
Channel 7	Unassigned	
Channel 8	Unassigned	

4.2 Measured Axes ► P2

Description

Used to declare the measured axes of the machine and check for the presence of axis interfaces.

Principle

The bit position gives the physical axis address.

The bit is set 1 to declare the axis as measured.

If the axes declared in P2 are not detected as present by the system on the bus, the message "ERROR 30x with x : number of the higher missing axis" is displayed.

If an interface is assigned both to a spindle (by P6) and to an axis (by P2), it is assigned by priority to the spindle.

If an interface is assigned both to a handwheel (by P14) and to an axis (by P2), it is assigned by priority to the handwheel.

If an interface is detected by the system but is not declared in parameter P2, it can be used by the dynamic operators (write of speed reference and read of measurement).

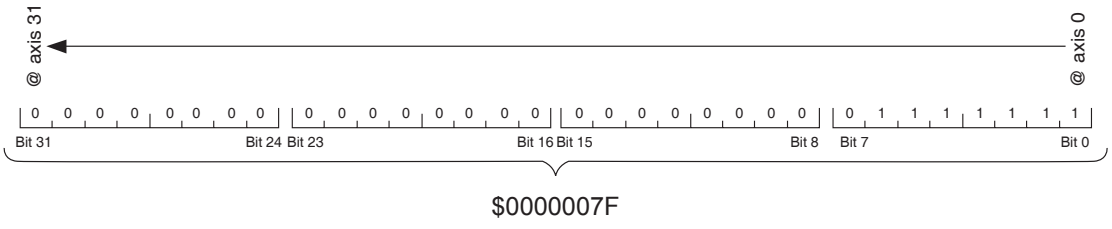
Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameters E931xx are used to read the value of P2.

Example

The seven axes of this configuration are measured.

Axis @	**0**	**1**	**2**	**3**	**4**	**5**	**6**
Axis name	X	Y	Z	U	W	B	C



Word N0	0000007F

4.3 Displayed Axes ► P0

Description

Defines the axes displayed.

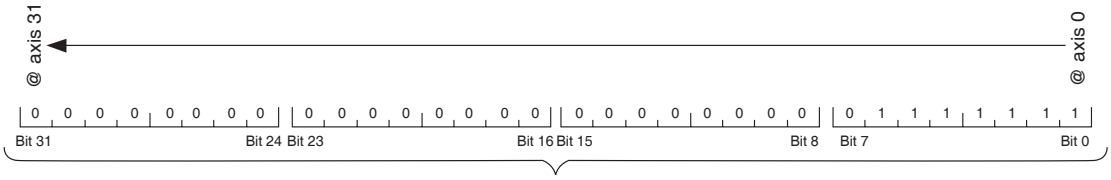
Principle

The bit position gives the physical axis address.
The bit is a 1 to declare the axis as displayable if it is assigned to a channel, i.e. declared in P9.

If the axis is declared in P2 or P3, it is displayed on the HMI pages.

Example

The axes at addresses 0 to 6 are displayable.



$0000007F

Word N0	0000007F

4.4 Modulo or Limited Excursion Rotary Axes ► P1

Description

Declares rotary axes as modulo 360 degrees or limited excursion type.

Principle

The bit position gives the physical axis address.

Word N0: specifies the modulo 360 degrees rotary axes.

A rotary axis is modulo if the value of the current position returns to zero after rotating by 360 degrees. The travel of a modulo rotary axis is not limited by software stops.

The bit is a 1 to declare the rotary axis as modulo.

Word N1: specifies the limited excursion rotary axes.

A rotary axis with limited excursion is an axis controlled like a linear axis.

Axes with limited excursion are axes:

- with an excursion less than 360 degrees with respect to a reference position (twist head),
- that it is desired to control over several revolutions.

The bit is a 1 to declare the rotary axis with limited excursion.

Axes declared in either lists cannot be programmed in inches and cannot be assigned a scaling factor.

If an axis is declared modulo and with limited excursion, it is considered modulo by the system. After Reset, each rotating axis is forced to the original state (modulo or limited excursion) set by this parameter on power-up.

Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameters E932xx are used to read the value of word N0 of P1.

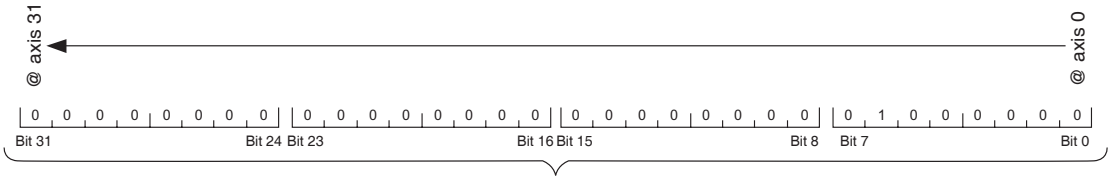
Examples

The axis with address 2 is modulo 360 degrees.

$00000004

Word N0	00000004

The axis with address 6 has limited excursion.



$00000040

Word N1	00000040

4.5 Servo-Controlled and Interpolated Axes ► P3

Description

Used to declare servo-controlled and interpolated axes.

Principle

The bit position gives the physical axis address.

The system manages servo-control of the axes if they are declared both in P3 and P2.

Programme Parameters (see "Flexium Programming Manual M or T).

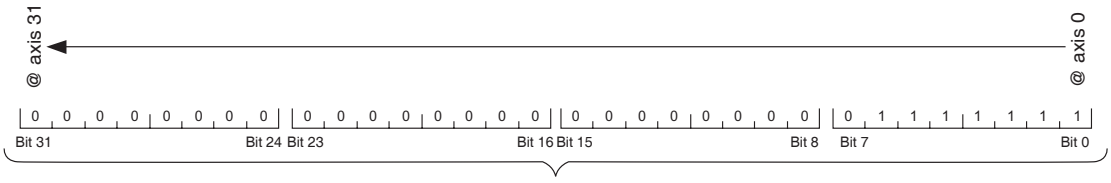
Parameter E910xx is used to read or change the state of these axes. A reset returns the CNC to the state specified by P3.

Extended NCK access (see PLC library "Extended NCK access")

The state of the axes can be modified by a Extended NCK request. A reset (CNC reset) resets the CNC to the state specified by P3.

Example

The axes with addresses 0 to 6 are servo-controlled and interpolated.



$0000007F

Word N0	0000007F

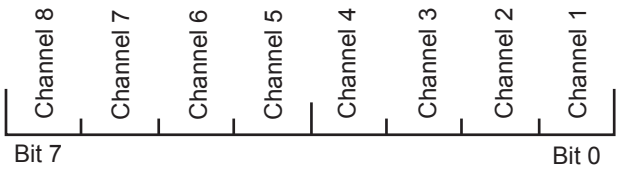
4.6 Channel Machine Configuration ► P5

Description

Used to define the machine configuration:
- lathe and milling machine channels,
- front and rear turrets.

Principle

Each bit represents the address of a channel.



Word N0: specifies the basic programming functions and defines the graphic display types.

A bit set to 0 defines a lathe channel or set to 1 defines a milling machine channel.

Word N1: further defines a lathe channel assignment.

A bit is set to 0 for a rear turret or set to 1 for a front turret.

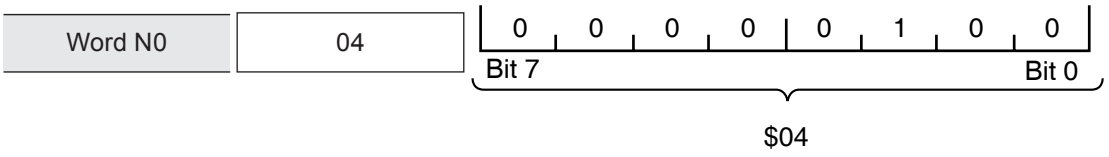
Programme Parameters (see "Flexium Programming Manual M or T).

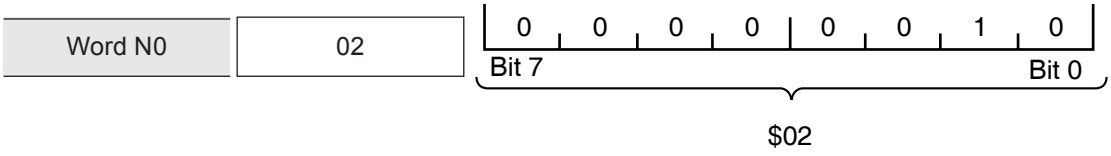
Programme parameter E11016 is used to enable the front turret or rear turret by programming.
A reset (CNC INIT) sets this parameter to the state of word N1 of P5.

Example

Setting the parameters for the following machine configuration.

Channel	1	2	3
N0	Lathe	Lathe	Milling machine
N1	Rear turret	Front turret	





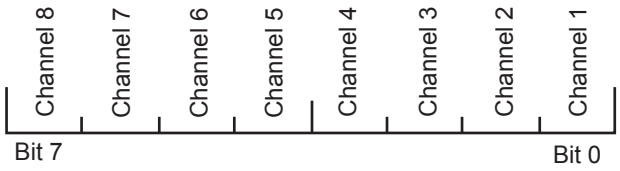
4.7 Axes Programmable by Diameter and Internal Measurement and Dimension Acquisition and Display Format ► P4

Description

Used to:
- declare the X and U axes of each channel as programmable by diameter,
- define the internal system measurement unit and the format used for acquisition and display of dimensions,
- specify that control of the X and/or U axes of each channel is referenced to the diameter in incremental JOG mode and handwheel mode (including NMauto),
- specify the divider for the rotary axes.

Principle

Each bit of words N0 and N1 represents the address of a channel.



Word N0: characterises the X axes of the different channels.

If the bit is set to 1, the X axis for the channel specified is always displayed by diameter and is programmed by diameter for turning (G20 state).

Word N1: characterises the U axes of the different channels.

If the bit is set to 1, the U axis for the channel specified is always displayed by diameter and is programmed by diameter for turning (G20 state).

Word N2: defines the internal system measurement units and the format used for linear axis dimension acquisition and display.

To reduce sampling effects and improve acceleration setting possibilities, the nano-interpolation (or Nano IPO) function makes it possible to have an internal interpolation unit that is smaller than the format used for dimension acquisition and display.

Word N2 specifies :
- the internal measurement unit by the units digit
- the dimension acquisition and display format by the tens digit.

The format used for dimension acquisition and display and the internal system measurement is defined by the following significant values:

Value of digit	Format used for dimension acquisition and display	Internal measurement unit	Maximum machine travel
1	One decimal digit	0.10 mm	10000 m
2	Two decimal digits	0.01 mm	1000 m
3	Three decimal digits	µm	100 m
4	Four decimal digits	0.10 µm	10 m
5	Five decimal digits	0.01 µm	1 m

For other values of N2 (0, 6, etc.), the system requires the micrometre as internal unit.

The basic unit is mm.
The internal measurement unit directly affects machine travels.

A digit is a number from 0 to 9.

Example:

Word N2	34

The tens digit = 3: The dimension acquisition and display format is micrometres.

The units digit = 4: The internal measurement unit is tenths of a micrometre.

The ratio between the internal system measurement unit and the dimension acquisition and display format is limited to 10.

Word N3: characterises the X and/or U axes controlled with reference to diameter in incremental JOG and handwheel mode.

If X or U are required, then set their corresponding channel bit in this word (X axes then appear in words 0 and 3 while U axes appear in words 1 and 3).

Since the measurement unit is really with respect to radius, if the jogs are set to radius while the display is set for diameter, then only every second demand increment will be taken into account.

Word N4 : specifies the internal system measurement unit and the dimension acquisition and display format for rotary axes .

This word is common to all the rotary axes of the machine, whether or not they are modulo.

It specifies the internal unit for the rotary axes and therefore for the machine parameters expressing dimensions (travel limit, maximum following error, in-position window, etc.) and the acquisition and display format for the dimensions of these axes.

To reduce sampling effects and improve acceleration setting possibilities.

The nano-interpolation (or Nano IPO) function makes it possible to have an internal interpolation unit that is smaller than the unit used for dimension programming and display.

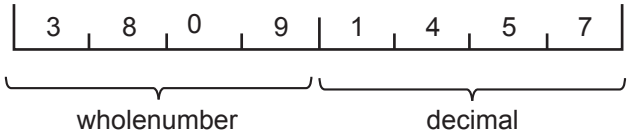
Word N4 specifies:
- the internal measurement unit by the units digit
- the dimension acquisition and display format by the tens digit.

The programming format for measurement of rotating axes (with limited excursion or modulo 360°) is extended so that it is possible to use an extra digit for the whole number part expressed in degrees.

Depending on the definition of the increment for rotating axes set by P4N4, the format changes as follows:

Value of N4	Definition of the increment	Old programming format	New programming format	Internal unit of measurement
0	10-4°	4.4	4.4	1/10000
1	10-1°	4.1	5.1	1/10
2	10-2°	4.2	5.2	1/100
3	10-3°	4.3	5.3	1/1000
4	10-4°	4.4	4.4	1/10000

Example of position of digits for programming formats (= 3809,1457):



This example shows a programming format with a whole number of 4 digits and a decimal fraction of 4 digits.

In the case of rotary axes, certain machine configurations associate a small diameter axis with a large diameter axis.
The small diameter axis requires less resolution than the large diameter axis.
A programme parameter is created to divide the following error by 10 (see programme parameter E922xx)

The ratio between the internal system measurement unit and the dimension acquisition and display format is limited to 10.

Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameter E11005 is used to temporarily inhibit programming by diameter in a turning phase.

A reset forces E11005 high.

Example

E11005 = 0: Programming by diameter inhibited.

E11005 = 1: Programming by diameter enabled by G20 on the X and U axes if they were declared in P4.

Programme parameter E922xx (xx = 0 to 31) is used to divide display of the following error for each axis.

Example

E92201 = 1, the following error on axis 1 is divided by 10.

E92200 = 0, the following error display is not modified.

At power up, programme parameters E922xx are set to 0.

The action of parameter E922xx can be combined with the action of the nano-interpolation function.
In this case, it is possible to display the following error on an axis with a definition 100 times smaller than the internal measurement unit.

Example

Word N0: The X axes of the three channels are programmed by diameter.

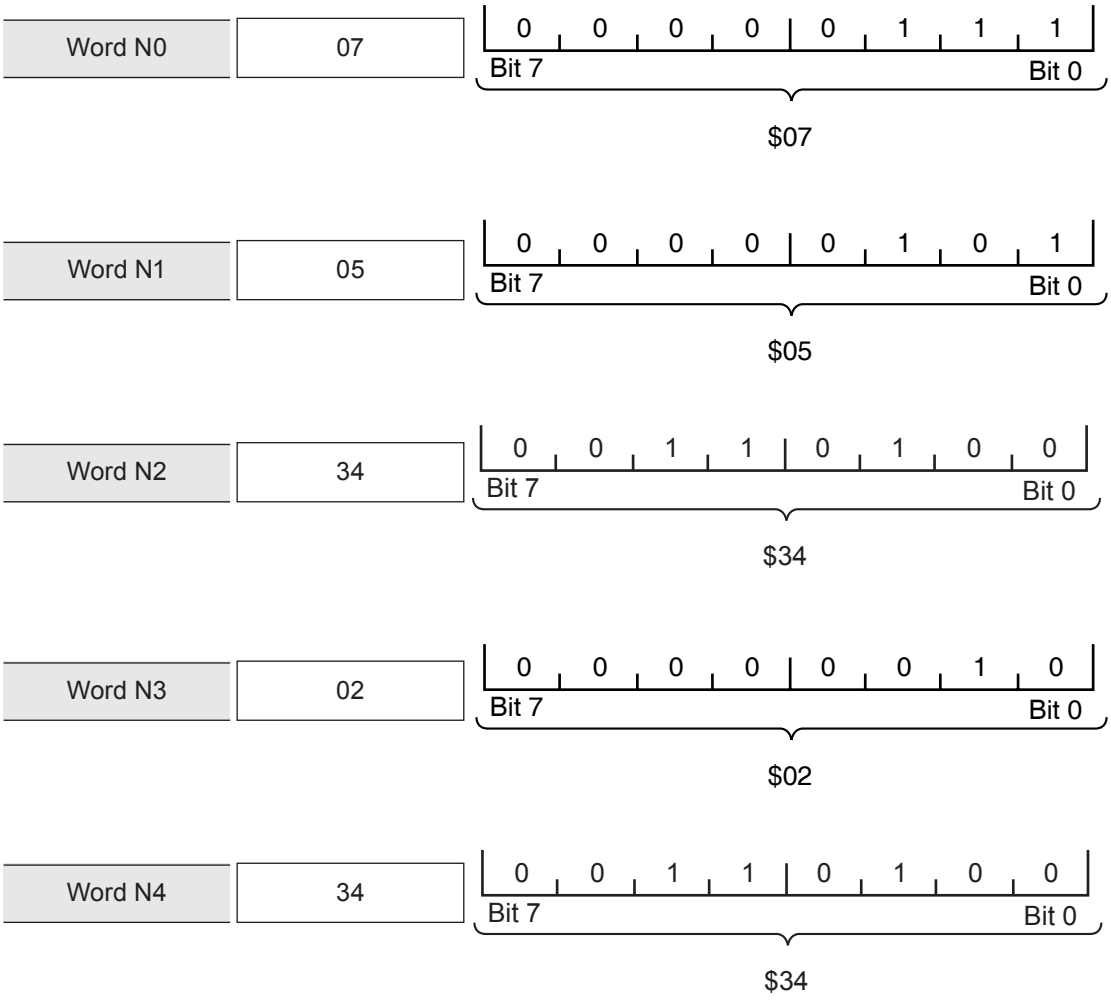
Word N1: The U axes of channels 1 and 3 are programmed by diameter.

Word N2: The internal measurement unit for linear axes is in tenths of a micrometre and the dimension acquisition and display format is in micrometres.

Word N3: The X axes of channels 2 is programmed by diameter in incremental JOG.

Word N4: The internal measurement unit of the rotary C axis is in ten thousandths of a degree and the dimension acquisition and display format is in thousandths of a degree.

	Channel 1			Channel 2			Channel 3		
Axis name	X	Z	U	X	Z	C	X	Z	U



4.8 Axes with Clamps ► P8

Description

Used to declare the machine axes with clamps.

The axes are clamped by miscellaneous function M10 and unclamped by miscellaneous function M11.

Principle

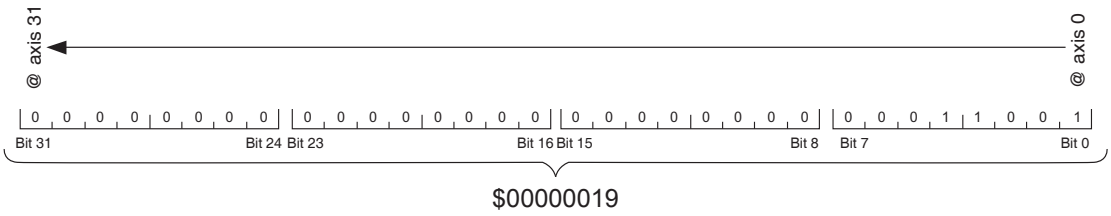
The bit position gives the physical axis address.

The bit is set to 1 declare the axis with clamps.

Example

Of the seven axes of this configuration, the axes at addresses 0, 3 and 4 have clamps.

Axis @	**0**	**1**	**2**	**3**	**4**	**5**	**6**
Axis name	X	Y	Z	U	W	B	C



Word N0	00000019

Programme Parameters (see "Flexium Programming Manual M or T).

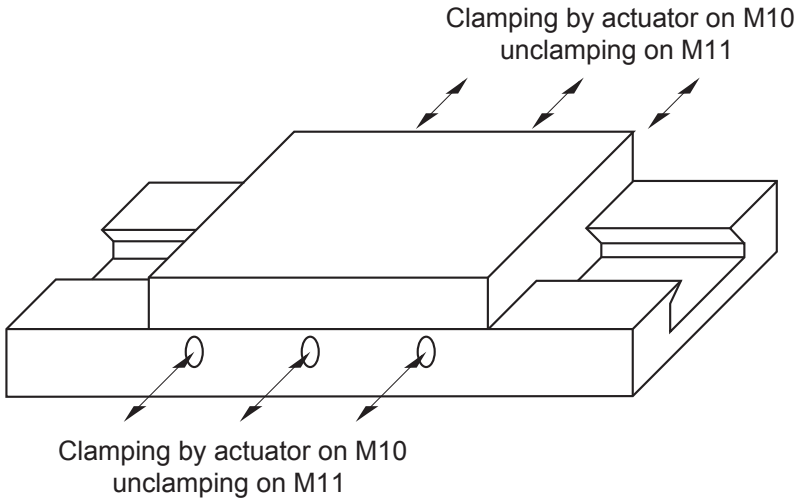
Programme parameter E9130x (x = physical axis number) is used to declare that an axis can be clamped or not. When this parameter is reset, the axes revert to the setting of machine parameter P8.

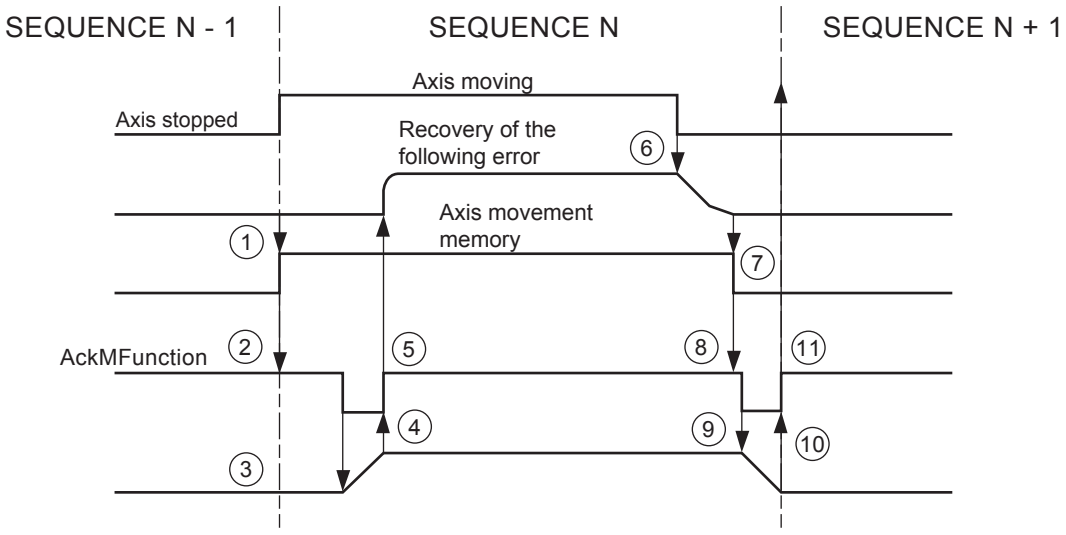
E9130x = 1 axis x with clamp

E9130x = 0 axis x according to P8.

Functional Diagram

The CNC processor requires this information since the CNC reference is not provided until the PLC has acknowledged an end of unclamp report (Channel[g] AckMFunction where g is the channel number from 1 to 8).





1. Internal timeout
2. Wait (for AckMFunction)
3. Axis unclamping (AckMFunction = 0)
4. AckMFunction = 1
5. Movement
6. Deceleration
7. End of movement
8. Wait (for AckMFunction)
9. Axis clamping
10. AckMFunction = 1
11. Programme continued (unprogrammed clamped axis)

4.9 Axis Assignment to a Channel ► P9

Description

Used to associate the symbolic axis names with the physical addresses and channels.

Principle

The system can control up to 32 axes. These axes are assigned to eight channels with a maximum of nine axes in a channel.

If the total number of axes and spindles defined in this parameter is higher than that permitted by system customisation, the message "TOO MANY AXES OR SPINDLES" is displayed at initialisation. After acknowledgement, the "PRSOV" indicator is displayed in the CNC screen status window. The AUTO, SINGLE, DRYRUN, and MDI modes are inhibited.

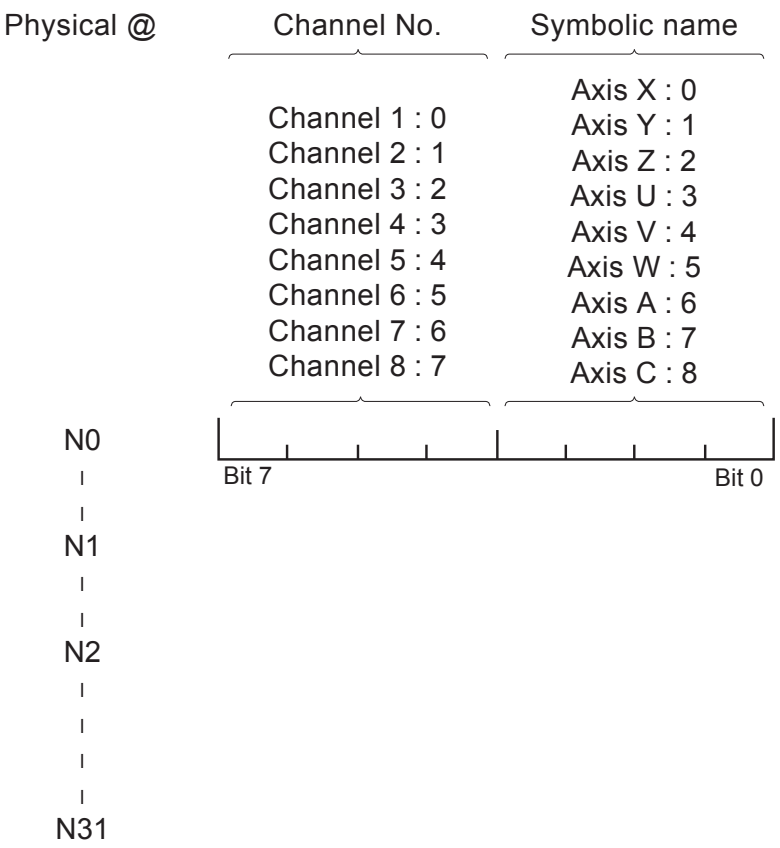
The word number gives the physical axis address. The values defines channels and symbolic addresses.

Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameter E7x005 is used to assign a CNC axis to a Auxiliary channel by programming. A reset reconfigures the system in the state of P9.

Word Format

The low half-byte specifies the symbolic name of the axis and the high half-byte specifies the channel to which the axis belongs.



If two axes have the same assignment by mistake, only the first is recognised except with the high-speed machining option, for which two synchronised axes must always have the same assignment.
The axis at address 31 cannot exist unless all the preceding axes have been declared.

The value FF means that the specified axis address has no symbolic name and does not belong to any channel.

The value Fx indicates that the specified axis address has a symbolic name but does not belong to any channel.

Example

Machine configuration corresponding to the table below:

- Channel 1 including X and Z axes
- Channel 2 including X and W axes
- Channel 3 including U,W and C axes.

Axis @	**0**	**1**	**2**	**3**	**4**	**5**	**6**
Axis name	X	Z	X	W	U	W	C
Channel concerned	1	1	2	2	3	3	3

Word N0	00

The axis at address 0 is the X axis of channel 1.

Word N1	02

The axis at address 1 is the Z axis of channel 1.

Word N2	10

The axis at address 2 is the X axis of channel 2.

Word N3	15

The axis at address 3 is the W axis of channel 2.

Word N4	23

The axis at address 4 is the U axis of channel 3.

Word N5	25

The axis at address 5 is the W axis of channel 3.

Word N6	28

The axis at address 6 is the C axis of channel 3.

Word N7 and others	FF

The axes from address 7 until 31 do not have a symbolic name and do not belong to any channel.

4.10 Handwheel Declaration ► P14

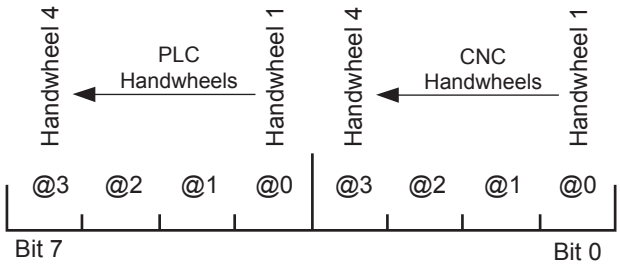
Description

Used to declare handwheels and filter the handwheel measurement.
One handwheel can be assigned to CNC or PLC but not both.

Principle

Word N0: specifies handwheel.

The bit position gives the physical handwheel address.



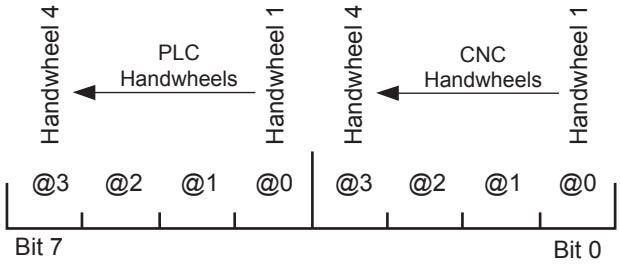
A bit is set to declare the handwheel (presence of an axis encoder).

If an encoder is assigned both to a handwheel (by P14) and to an axis (by P2), it is assigned by priority to the handwheel.

Word N1: specifies the filtering of the handwheel measurement.

Filtering consists of forcing the two LSBs of the measurement to zero, i.e. the bits resulting from multiplication by 4 of the increments, performed by the axis card.

The bit position corresponds to the physical address of the handwheel to which filtering of the measurement is applied. When the bit is a 1, filtering is enabled on the handwheel addressed.



Word N2: Enables the monitoring of errors (equivalent to P25 N0)

Word N3: Specifies the physical connection (set bit if present):

Bit 0	handwheel 1@0
Bit 1	handwheel 2@1
Bit 2	handwheel 3@2
Bit 3	handwheel 4@3

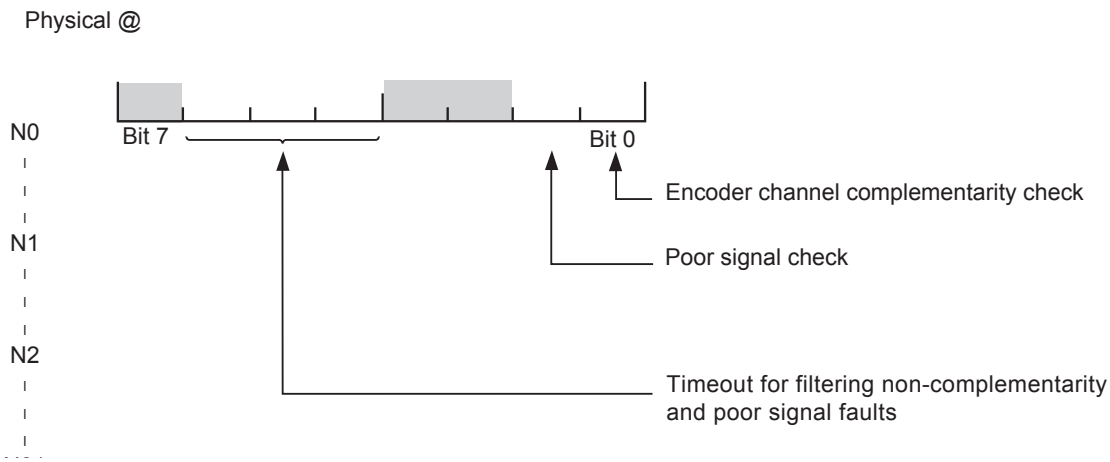
Word N4: Poor signal and encoder channel complementary settings for handwheel 1@0

Word N5: Poor signal and encoder channel complementary settings for handwheel 2@1

Word N6: Poor signal and encoder channel complementary settings for handwheel 3@2

Word N7: Poor signal and encoder channel complementary settings for handwheel 4@3

Word format



Filtering Timeout

Bits	6	5	4	
	0	0	0	32 µs
	0	0	1	16 µs
	0	1	0	8 µs
	0	1	1	4 µs
	1	0	0	2 µs
	1	0	1	1 µs
	1	1	0	500 ns
	1	1	1	250 ns

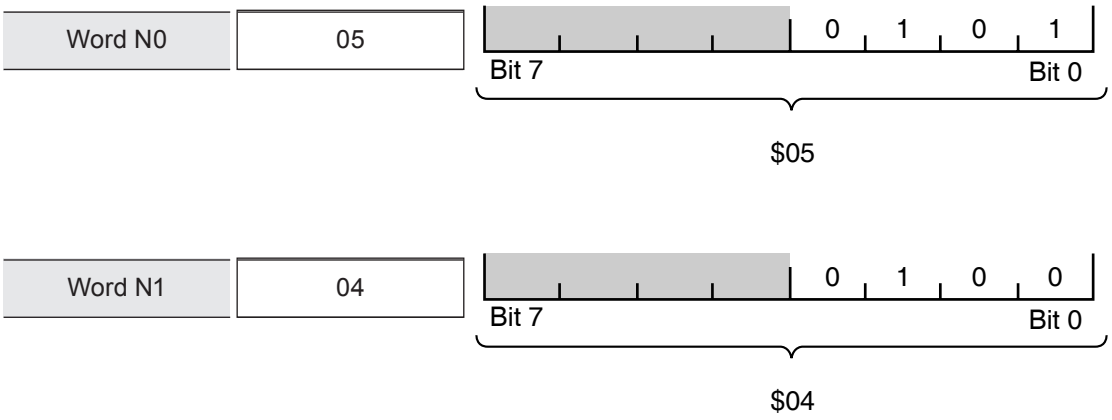
Default value: $0

Extended NCK access (see PLC library "Extended NCK access")

The handwheel measurement filters can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state declared in P14, word N1.

Example

Two handwheels are assigned to addresses 0 and 2 and the measurement of the handwheel at address 2 is filtered.



4.11 Coupling Assignment for Duplicated and Synchronised Axes ► P27

Description

Used to assign a slave axis to a master axis.

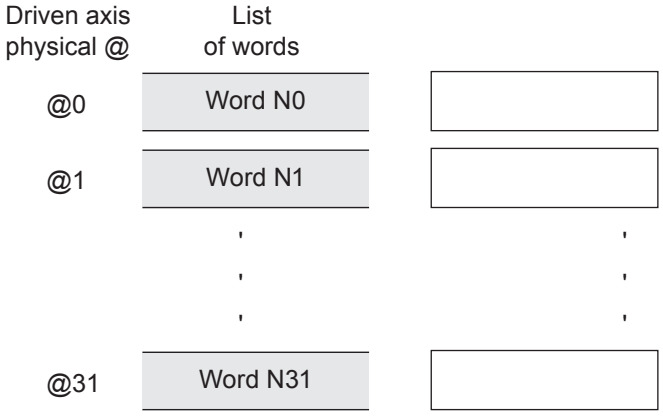
Concerns the different types of coupling:

- simple coupling (drive axis reference copied to driven axis),
- coupling with symmetry (reverse of the drive axis reference copied to the driven axis),
- coupling with synchronisation (the driven axis reference is drive axis reference + a servoed correction).

Principle

The word number gives the physical address of the driven axis of a pair. By convention, it is also the coupling number.

The word contains the physical address of the drive axis. The value FF specifies the absence of a drive axis.



Programme Parameters (see "Flexium Programming Manual M or T).

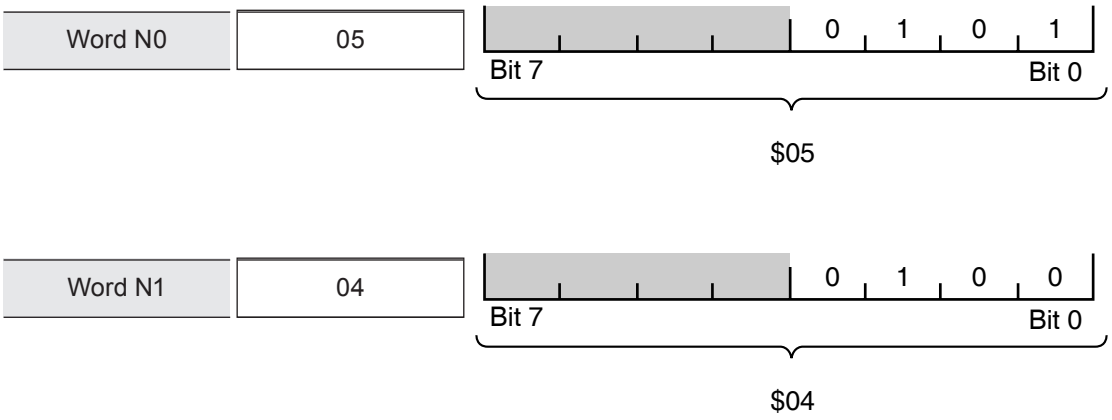
Parameters E94100 to E94131 are used to assign a driven axis to a drive axis by programming. A reset (CNC init) sets these parameters to the state of P27.

Extended NCK access (see PLC library "Extended NCK access")

The handwheel measurement filters can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state declared in P14, word N1.

Example

Two handwheels are assigned to addresses 0 and 2 and the measurement of the handwheel at address 2 is filtered.



4.12 Synchronised Axis Coupling Enable ► P28

Description

Enables synchronisation of a driven axis with a drive axis.

Principle

The synchronised axes are duplicated axes where the driven axis is synchronised to the drive axis.

The bit position gives the physical address of the driven axis of a pair. By convention, it is also the coupling number.

If the bit is set to 1, the driven axis is synchronised with its drive axis declared in P27.

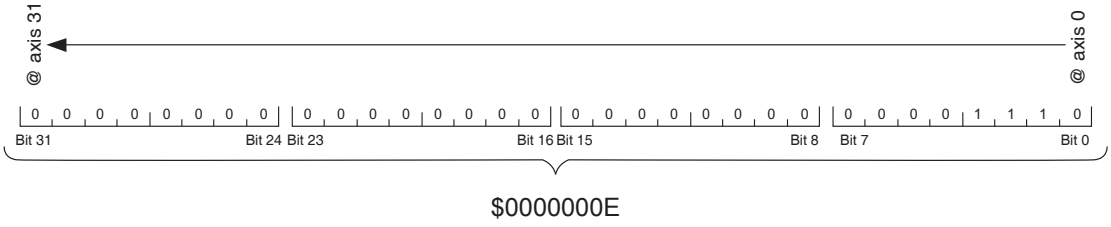
STOP A coupling cannot be both synchronised and symmetric.

Programme Parameters (see "Flexium Programming Manual M or T).

Parameters E96200 to E96231 are used to enable duplication and synchronisation by programming. A reset (CNC init) sets these parameters to the state of P28.

Example

The axes at physical addresses 1, 2 and 3 are synchronised with their drive axes.



$0000000E

Word N0	0000000E

4.13 Carrier or Carried Axes ► P64

Description

Creates carrier/carried linear axis pairs by associating a primary axis with a secondary axis: X/U, Y/V or Z/W.

Principle

The bit position gives the physical address of the axis.

If the bits of the axis pair are set, the axis is declared as carrier or carried axis according to its declaration in P9 as primary (X, Y, Z) or secondary (U, V, W).

On a lathe, if one of the lathe axes is called U or W, it must be declared as carried. A secondary axis cannot accept tool dimensions unless it is declared as carried.

Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameter E7x006 is also used to create a carrier/carried axis pair.
A reset (CNC INIT) reconfigures the CNC in the state specified by P64.

Example

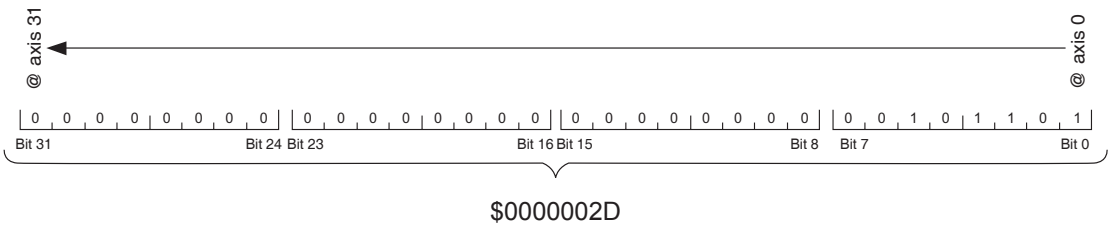
E70006 = 0 The X and U axes are independent.

E72006 = 1 Carrier/carried Z and W axis pair.

Example

Creation of X/U and Z/W carrier/carried axis pairs. The Y and V axes are independent.

Axis @	**0**	**1**	**2**	**3**	**4**	**5**
Axis name	X	Y	Z	U	V	W



$0000002D

Word N0	0000002D

4.14 Axes with Quantified Movements ► P65

Description

Used to define the axis movement increment (step).

Principle

This parameter applies only to the axes declared as modulo in P1 and clamped by function M10 in P8.

The values are expressed in internal unit. The internal unit is deg/10, deg/100, deg/1000 or deg/10000 for rotary axes (see P4 "Axes Programmable by Diameter and Internal Measurement" in the parameter manual). A value of zero corresponds to axis movement by increments of 1/10000 of a degree.

The word number gives the physical axis address.

For instance, this parameter applies to movements of a turntable controlled by 1/2 degree and measured to one ten-thousandths of a degree (HIRTH coupling).

Driven axis physical @	List of words	
@0	Word N0	
@1	Word N1	
'	'	'
@31	Word N31	

Example

The axis at address 3 has a movement increment of 0.5 degrees, the axis at address 4 of 2.5 degrees and the axis at address 5 of 1 degree.

Axis @	**0**	**1**	**2**	**3**	**4**	**5**	**6**
Axis name	X	Y	Z	A	B	C	
Modulo axes				Yes	Yes	Yes	
Clamped axes				Yes	Yes	Yes	

The value of words N0 to N2 and N6 to N31 is 0.

Word N3	5000
Word N4	25000
Word N5	10000

4.15 Axis and Spindle Switching with Homing Preserved ► P70

Description

Specifies the DISC-NT axes using 32-bit measurements.

Used to declare switchable digital axes or spindles.

Principle

Word N0: specifies the address of the sensor coupler to which the axis-sensor or the handwheel is connected:

Bit 0 to bit 7	Address of sensor or handwheel connected to a full axis/spindle coupler 0 (0 to 31 or FF if not used)
Bit 8 to bit 15	Address of handwheel connected to reduced handwheel coupler 0 (0 to 3 or FF if not used)
Bit 16 to bit 23	Address of sensor or handwheel connected to a full axis/spindle coupler 1 (0 to 31 or FF if not used)
Bit 24 to bit 31	Address of handwheel connected to reduced handwheel coupler 1 (0 to 3 or FF if not used)

Word N1 to N5: not significant.

Word N6: specifies the sensorless digital spindles. Set to 0 the bit at the position of the address (24-27) of the sensorless spindle.

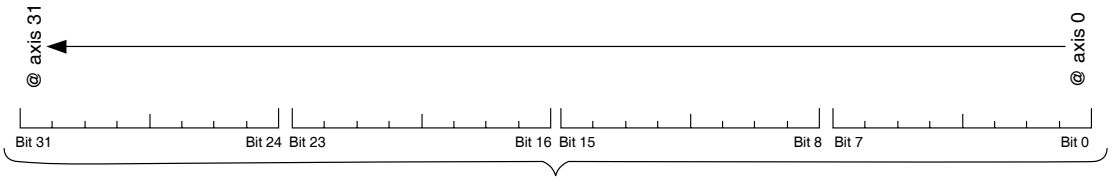
Word N7: specifies the DISC-NT axes using 32-bit measurements.

(This function is restricted to switchable axes and spindles with preservation of measurements and homing).

Measurements are normally processed on 16 bits.

Considering the usual definition of a C axis measurement (e.g. 3,600,000 pulses per revolution) and the usual maximum speed of a spindle (several thousand rpm), it may be necessary to read the drive measurement on 32 bits so as not to be limited by the size of the measurement increment.

This function is achieved by resetting the bit corresponding to the address of the switchable axis in word N7. The default value is $FFFFFFFF.



Conditions

Switching between a digital spindle and a digital axis is possible without losing the homing on the axis or spindle (the drive measures the spindle or axis with the motor encoder or an additional encoder) provided the two following conditions are satisfied:

- The drive MULTI and DIVI coefficients are the same for the axis configuration and the spindle configuration
- The measurement encoder is always mechanically interfaced with the spindle and the axis.

To determine the axes whose measurements must be processed on 32 bits, the following inequality giving the measurement increment per sample must be verified:

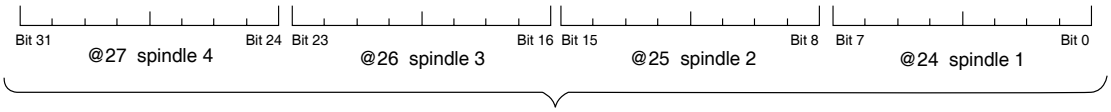
(Spindle speed $_{max}$. x Axis modulo x Sampling time) / 60 < 2^{15}

Spindle speed $_{max}$.	In rpm divided by 60 for conversion to seconds
Axis modulo	360 degrees x Parameter P4 Word N4 (in internal measurement unit)
Sampling time	Parameter P50 / 10^6 (division required to convert o seconds)
2^{15}	Size of the measurement register.

The measurement increment is calculated on 32 bits.
Only for update of the measurement.
The measurement increment is used by other software functions, but only on 16 bits, format unchanged to limit modifications.

Word N8 : specifies the digital spindles that are switchable with a digital axis.

This function is achieved by declaring the physical address of the axis to be switched with the spindle in the byte corresponding to the physical address of the spindle.

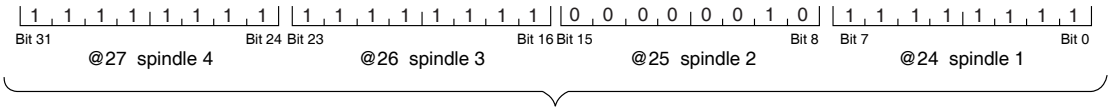


Conditions

The axis switchable with a spindle must be declared in parameter P2.

Example

The axis whose physical address is @2 is switchable with spindle 2 whose physical address is @25.



$ FFFF02FF

Word N8	FFFF02FF

Programme Parameters (see "Flexium Programming Manual M or T):

Programme parameter E942xx (where xx is the spindle drive address) is used to switch this spindle with an axis using its physical address:

- To switch from spindle to axis E94225 =2
- To switch from axis to spindle E94225 =25

For a configuration allowing switching from axis to spindle, the drive is identified by the spindle address and at CNC initialisation, the CNC is configured for spindle operation.

4.16 DISC NT Axis presence ► P71

Description

Specifies the present of a DISC NT axis.

Principle

Each word corresponds to the physical address of DISC NT axis.

The setting of the word is the physical address of DISC NT axis.

A setting of $FF indicates the absence of axis.

Driven axis physical @	List of words	
@0	Word N0	
@1	Word N1	
	'	'
	'	'
	'	'
@31	Word N31	

4.17 Tandem Function: master/slave coupled axes ► P85

Description

Specifies the master and slave axes to be associated by coupling.

This coupling can be processed in several ways by the Tandem Function:

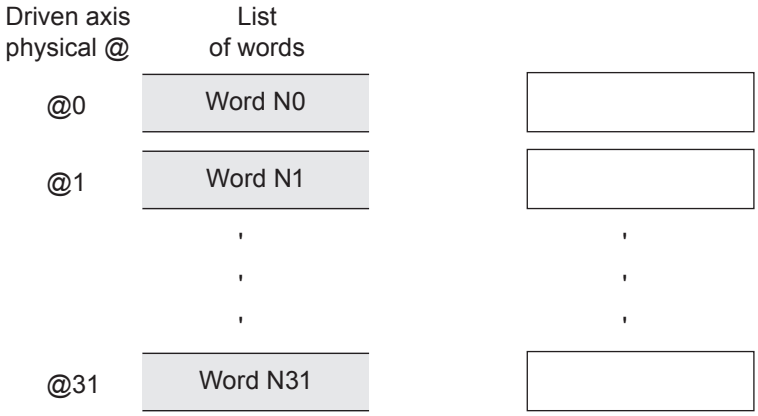
- By torque duplication
- By torque synchronisation
- By anti-backlash (torque synchronisation with a preload current value)

Principle

Each word corresponds to the physical address of the slave axis.

The value of the word is the physical address of the master axis.

A setting of $FF indicates the absence of coupling.



Example

Torque duplication

If an axis motor has insufficient torque for accelerating a high load, several motors can be mechanically coupled and declared as master/slave by the Tandem Function.
The master motor torque is communicated to the slave motors so that the drives of the motors coupled in tandem can develop the required resultant torque for accelerating the load.

Torque synchronisation

Certain applications using the Tandem Function require equalisation of the master and slave motor torques. A master/slave torque difference control function equalises the master and slave torques.

Torque synchronisation with anti-backlash

In the case of a rack and pinion drive with backlash, an opposing preload torque can be applied to the master and slave motors to increase the transmission stiffness.
The opposing torque reduces the adverse effects of backlash by holding the teeth of each drive pinion against the rack teeth.
In addition, control of the preload totally eliminates vibration during transient phases of movement.

Conditions

Declaration of parameters P85 and P86 determines the typical configuration of the MDLU3 drive by writing the drive parameters. (see the Drive Parameters manual AMOMAN007)

In the case of torque duplication, a maximum of three slave axes can be associated with a master axis and only one master axis can be associated with each slave axis.
With torque synchronisation, a single slave axis is associated with a single master axis.

All the couplings can be modified on the fly by a Extended NCK Access request (see the Tandem function in the Flexium Programming manual).
A slave servo-drive cannot become a master or independent axis unless it is declared in P2 (P2 cannot be modified on the fly).

In the case of reassignment of an axis servo-drive (declared by the Tandem Function) as a spindle servo-drive, the axis must first be configured as an independent axis (neither master nor slave) before being able to assign it to a spindle. THE TANDEM FUNCTION IS NOT AVAILABLE FOR SPINDLES.

Errors

One machine error message and eight system initialisation error messages are associated with the Tandem Function.

- Machine error 79: ***Illegal Tandem configuration***

Error 79 is generated by an incorrect on-the-fly Tandem configuration.
This occurs when the tandem configuration defined in machine parameters P85 and P86 is inconsistent with the tandem configuration defined in drive parameters.
(see the Drive Parameters manual AMOMAN007)

- The following messages can be displayed at system initialisation:

DRIVE S/W VERSION IS NOT COMPATIBLE WITH THE TANDEM FUNCTION	The drive software version is below 2.30.
BAD TANDEM CONFIGURATION: @SLAVE EQUALS @MASTER	The slave axis address is the same as the master axis address.
BAD TANDEM CONFIGURATION: MASTER AXIS ALREADY DECLARED AS SLAVE AXIS	The master axis is already declared as a slave axis.
BAD TANDEM CONFIGURATION: INCOMPATIBLE SLAVE AXIS	An axis declared as slave is incompatible with the Tandem Function.
BAD TANDEM CONFIGURATION: INCOMPATIBLE MASTER AXIS	The axis declared as master is incompatible with the Tandem Function.
BAD TANDEM CONFIGURATION: NUMBER OF SLAVE AXES > MAX IN TORQUE DUPLICATION	The maximum number of slave axes is limited to 3 with torque duplication.
BAD TANDEM CONFIGURATION: NUMBER OF SLAVE AXES > 1 IN ANTI-BACKLASH	Only one slave axis can be declared with backlash compensation.
TANDEM DRIVE PARAMETER WRITE FAILURE	An error occurred during drive parameter write by the CNC.

4.18 Tandem Function: defines master axes with torque synchronisation ► P86

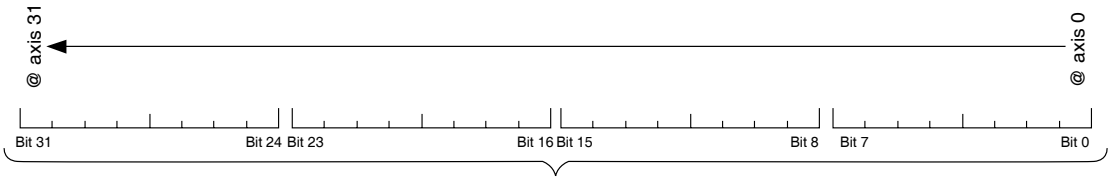
Description

Defines master axes with torque synchronisation.
Torque synchronisation may include backlash compensation (i.e. a preload current defined in the MDLU drive).

This parameter differentiates the master axes from those that use only torque duplication.

Principle

This function is activated by setting the bit corresponding to the physical address of the master axis.



Page deliberately empty

Page deliberately empty

Summary

5 Measurement 65
5.1 Axis Measurement Direction ► P10 65
5.2 Axis Measurement Conversion Coefficient ► P11 66
5.3 Handwheel Measurement Direction ► P12 67
5.4 Handwheel Measurement Conversion Coefficent ► P13 68
5.5 Poor Signal and Encoder Channel Complementarity Check Declaration ► P25 70
5.6 Poor Signal and encoder Channel Complementarity Check ► P26 72
5.7 Minimum Time Between Two Consecutive Encoder Faults ► P77 73

Page deliberately empty

5 Measurement

5.1 Axis Measurement Direction ► P10

Description

Used to define the direction of axis measurement.

Principle

The bit position gives the physical axis address.

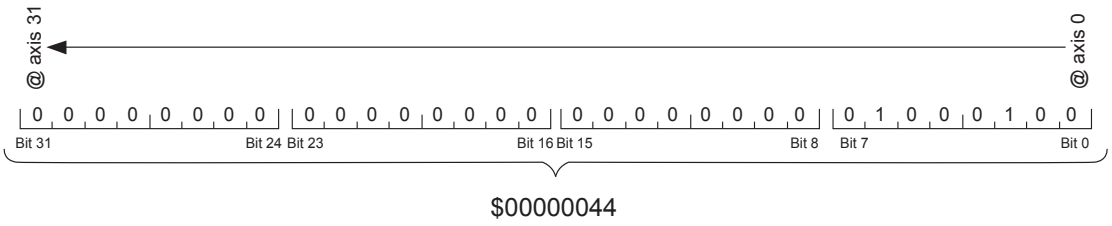
If the bit is set to 0, the encoder for the corresponding axis is mounted in the positive direction (the "measurement step" supplied by the encoder increases as the tool moves in the positive direction of the axis).

If the bit is set to 1, the axis encoder is reverse-mounted (the measurement decreases as the axis moves in the positive direction).

Example

The axes at addresses 2 and 6 are equipped with a reverse-mounted encoder.

Axis @	0	1	2	3	4	5	6
Name axis	X	Y	Z	U	W	B	C
Forward-mounted axis	Yes	Yes		Yes	Yes	Yes	
Reverse-mounted axis			Yes				Yes



Word N0	00000044

Extended NCK access (see PLC library "Extended NCK access")

The axis measurement direction can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P10.

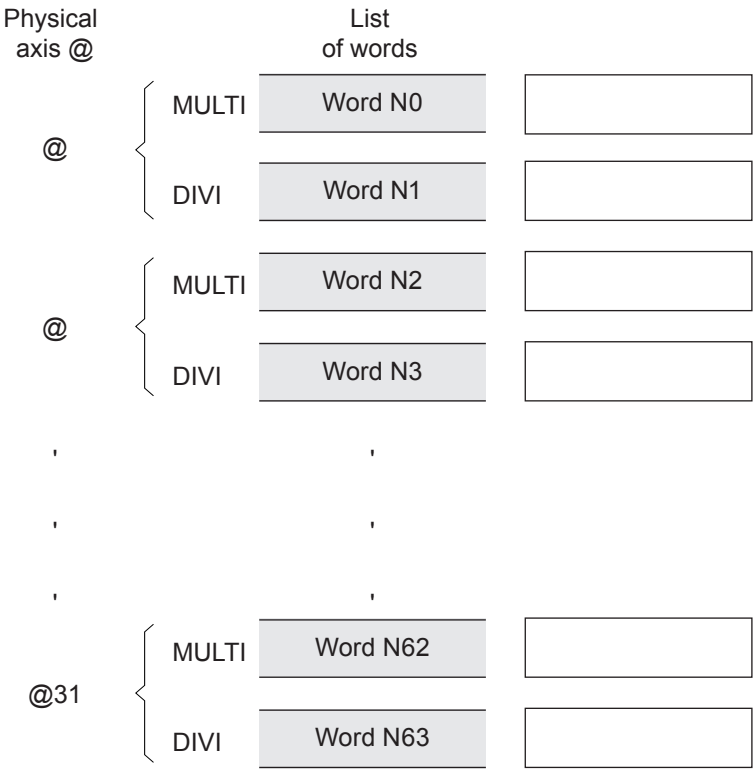
5.2 Axis Measurement Conversion Coefficient ► P11

Description

Used only for analog axis.
Adapts the measurement supplied by the position encoder to the internal CNC measurement.

Principle

Each pair of words corresponds to the values assigned to an axis. The first word of the pair indicates the multiplier coefficient (MULTI) and the second the divider coefficient (DIVI).



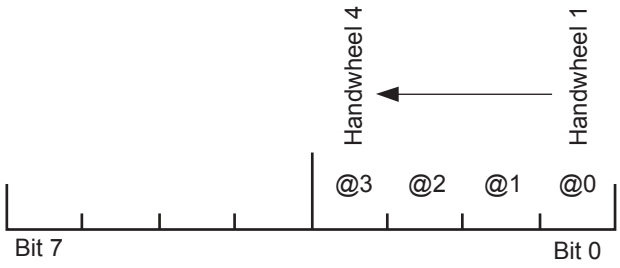
5.3 Handwheel Measurement Direction ► P12

Description

Used to define the handwheel measurement direction.

One handwheel can be assigned to CNC or PLC but not both.

Principle



If the bit is set to 0, the handwheel encoder is mounted in the positive direction (the measurement step output by the encoder increases as the dial moves in the positive direction of the graduations).

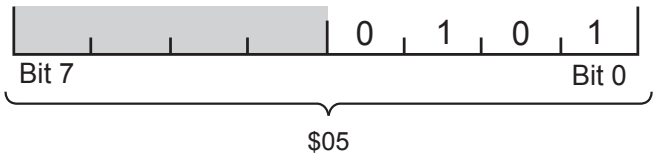
If the bit is set to 1, the encoder is reverse-mounted (the measurement decreases as the dial moves in the positive direction of the graduations).

Extended NCK access (see PLC library "Extended NCK access")

The parameter can be modified by a Extended NCK request.
A reset (CN reset) resets the CNC to the state specified by P12.

Example

The handwheels at addresses @0 and @2 are equipped with reverse-mounted encoders.



Word N5	0

5.4 Handwheel Measurement Conversion Coefficent ► P13

Description

Moves the handwheel by an increment equivalent to the internal measurement unit (in manual mode).

Principle

Words N0 to N7 : defines pairs of words corresponding to the values assigned to the handwheels.

The first word of the pair indicates the multiplier coefficient (MULTI).

The second word of the pair indicates the divider coefficient (DIVI).

The conversion formula is as follows:

Movement (internal measurement unit) = Encoder measurement (No. of lines) x MULTI/DIVI

Where Encoder measurement = Number of lines output by the encoder x 4.

MULTI/DIVI = Movement (internal measurement unit)/Number of encoder lines x 4

The internal measurement is expressed in mm/10, mm/100, µm, µm/10 or µm/100 for linear axes (see P4 parameter).

Words N8 and N9 : defines the measurement increment for the handwheel in automatic mode.

This pair of words allow that the increments related to rotation of the handwheel are added to the increments computed by the CNC when preparatory function G12 is enabled.
- N8 contains MULTI.
- N9 contains DIVI.

Words N10 to N17 : defines the measurement increment for the handwheel in automatic mode for rotary axes.

When using rotary axes (modulo 360 and/or with limited excursion) driven by a handwheel, a second MULTI/DIVI pair can be defined for each handwheel. This pair is automatically taken into account whenever the handwheel is used to drive a rotary axis.

If these words are set to 0, the MULTI/DIVI values from words N0 to N7 are used regardless of whether the driven axis is linear or rotary. N10 to N17 are initialised with default values of 0.

The conversion formula and other definitions remain the same.

Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameter E32001 is also used to adjust the overspeed coefficient on the path in G12. A reset (CNC INIT) reconfigures the CNC in the state specified by P13.

Extended NCK access (see PLC library "Extended NCK access")

The handwheel measurement conversion coefficients can be modified by a Extended NCK request. A reset (CNC reset) resets the CNC to the state specified by P13.

Example

On the axis at address 28 (N0 and N1), the encoder outputs 500 lines per revolution.
It is desired to move the axis by 2 mm per encoder revolution.

The internal measurement is expressed in µm/10.

20000 = 500 x 4 x MULTI/DIVI

20000 = 2000 x MULTI/DIVI

MULTI/DIVI = 10

Word N0	10
Word N1	1

5.5 Poor Signal and Encoder Channel Complementarity Check Declaration ► P25

Description

Used to declare the axes that can be affected by the poor signal and encoder channel complementarity checks regardless of the type of measurement used (incremental, encoded references, combined or SSI encoder).

Word N0 specifies for all the position encoders (axis, spindle) and all the handwheels connected to analogue axis cards.

This word has a default value of $FFFFFFFF (check of all the position encoders and all the handwheels).

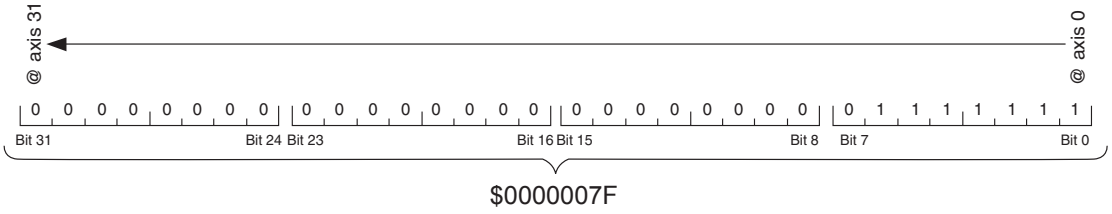
The bit position corresponds to the physical address of the axis.

If the bit is set to 0, the check is inhibited on the axis concerned.

If the bit is set to 1, the check is enabled on the axis concerned and gives access to the fault filtering values (see parameter P26).

Example

Check enabled on the axes at addresses 0 to 6.



Word N0	0000007F

Word N1: On a CNC reset, this word indicates whether an encoder (axis, spindle or handwheel) connected to an analogue interface is physically connected or disconnected.

This word has a default value of $FFFFFFFF (encoder connected).

When this word equals zero, the encoder is disconnected.

This function allows systems to be started with the encoders disconnected. In this case, the encoder type is incremental and homing requests are rejected.

For an SSI encoder, measurement is inhibited (no serial frame).

Physical disconnection of an encoder requires declaring the encoder as not checked and not connected.

Extended NCK access (see PLC library "Extended NCK access")

Declaration of the measurement encoder check and connection can be modified by a Extended NCK request. A reset (CNC reset) resets the CNC to the state declared in P14, word N1.

The word N1 of the parameter P25 is meaningless with DISC NT.

5.6 Poor Signal and encoder Channel Complementarity Check ► P26

Description

Used only for analog axis.
Activates the encoder channel complementarity and poor signal check and assigns the timeout for filtering faults.

Principle

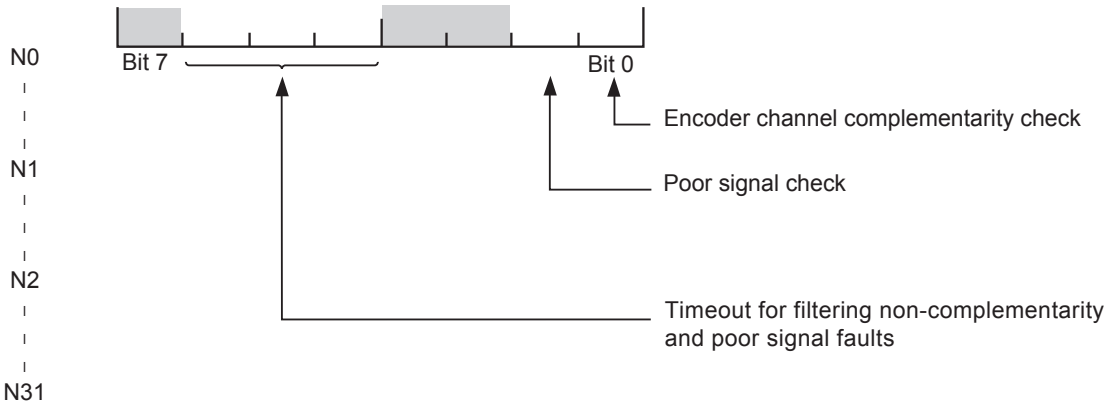
The word number gives the physical axis address.

If the bit is set to 1, the check is active. Default value: 00

STOP The check cannot be activated in P26 unless it is first declared in P25.

Word Format

Physical @



Filtering Timeout

Bits	6	5	4	
	0	0	0	32 µs
	0	0	1	16 µs
	0	1	0	8 µs
	0	1	1	4 µs
	1	0	0	2 µs
	1	0	1	1 µs
	1	1	0	500 ns
	1	1	1	250 ns

Default value: $73

Poor signal and encoder channel complementarity check enabled with a 250 ns filter timeout.

5.7 Minimum Time Between Two Consecutive Encoder Faults ► P77

Description

Specifies the minimum time between two consecutive encoder faults so as not to trigger a CNC error.

Principle

This parameter is used only for SSI measurement systems.

The settings are in minutes. A setting of zero triggers an error each time a fault is detected.

Default value: 00

The bit position corresponds to the axis address.

Page deliberately empty

Summary

6 Servo-Control 77

6.1 Maximum Axis Traverse Rates ► P30 77
6.2 JOG Speeds and Reduced Speeds ► P31 78
6.3 Maximum Permissible Acceleration ► P32 79
6.4 Direction of Axis Speed Reference ► P20 82
6.5 Servo-System Loop Gain Coefficient ► P21 83
6.6 Servo-Loop Time Constant ► P56 87
6.7 In-Position Window ► P22 88
6.8 Maximum Following Error ► P23 89
6.9 Dynamic Movement Control ► P57 91
6.10 Speed Anticipation Coefficient and Number of Filtered Terms for High Speed Cutting Machining ► P55 93
6.11 Characteristics of Synchronised Axis Pairs ► P24 97
6.12 Servo-Loop Error Tolerated on Circles ► P52 99
6.13 High Speed Cutting Machining ► P19 103
6.14 Approach Speed ► P33 105
6.15 Setting of the Speed Buildup in the Path ► P53 106
6.16 Time Constant per Axis ► P66 110
6.17 JOG Speed Setting 111
6.18 General Block Diagram of Servo-Control 112

Page deliberately empty

6 Servo-Control

6.1 Maximum Axis Traverse Rates ► P30

Description

Used to define the maximum axis traverse rates.

Principle

The values are expressed in mm/min for linear axes and deg/min for rotary axes.

The maximum traverse rate is 30000 increments per sample after multiplication by 4 and after MULTI/DIVI.

The word number gives the physical axis address.

Physical axis @	List of words	
@0	Word N0	
@1	Word N1	
'	'	
'	'	
'	'	
@31	Word N31	

Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameter E970xx is used to read the maximum axis traverse rate.

Extended NCK access (see PLC library "Extended NCK access")

The maximum axis speed can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P30.

6.2 JOG Speeds and Reduced Speeds ► P31

Description

Used to define the speeds corresponding to manual mode and the reduced speeds requested by the PLC.

Principle

Words N0 to N2 : define the speeds of movement on the axes:

- slow JOG for N0,
- normal JOG for N1,
- fast JOG for N2.

The speeds are expressed in mm/min.

They can be overridden by the feed rate potentiometer.

The normal and fast speeds are accessible from the machine panel.

Words N3 and N4 : used to set the parameters of the reduced speeds requested by the PLC function:

(VREDUCT = 1).

Word N3 gives the reduced speed on linear axes in mm/min.

Word N4 gives the reduced speed on rotary axes in deg/min.

6.3 Maximum Permissible Acceleration ► P32

Description

Used to define the maximum permissible accelerations for each of the axes or each of the spindles.

Principle

Each pair of words defines the accelerations on an axis or on a spindle

The first word gives the acceleration for the work rates.

The second gives the accelerations for the fast speeds: "DRYRUN, FAST JOG and G0 modes".

The values are expressed in mm/s^2 for linear axes.

The values are expressed in deg/s^2 for rotary axes and spindles.

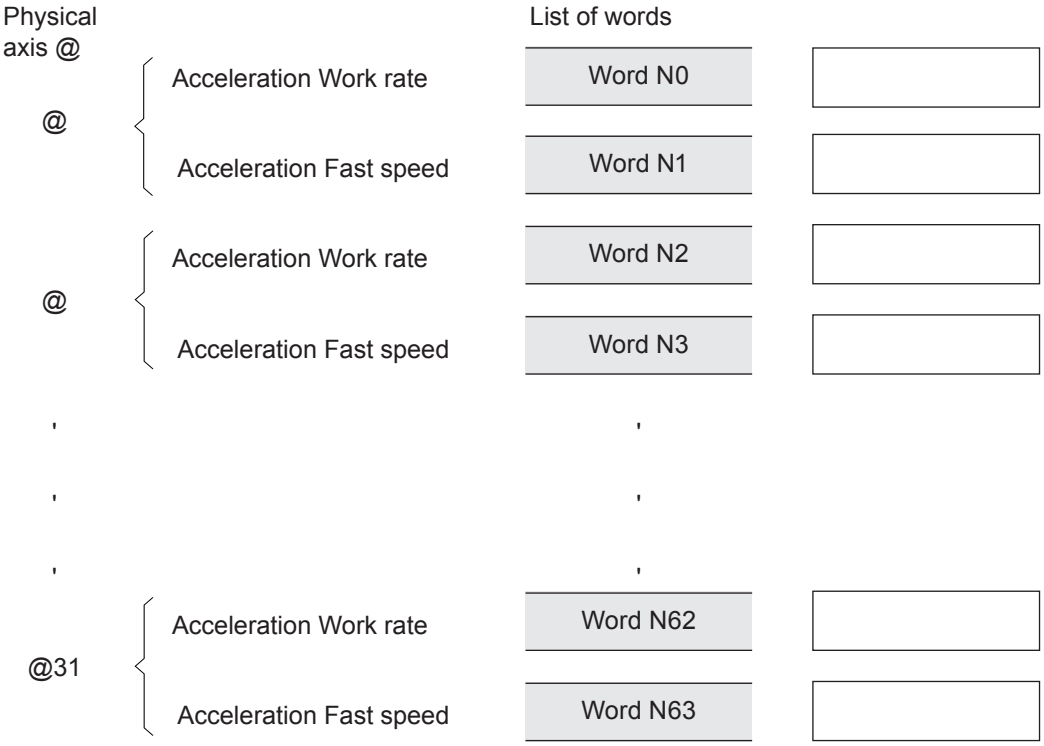
The values are expressed according to the settings of parameter P6 as shown in the table below for spindles:

Default value in deg/s^2

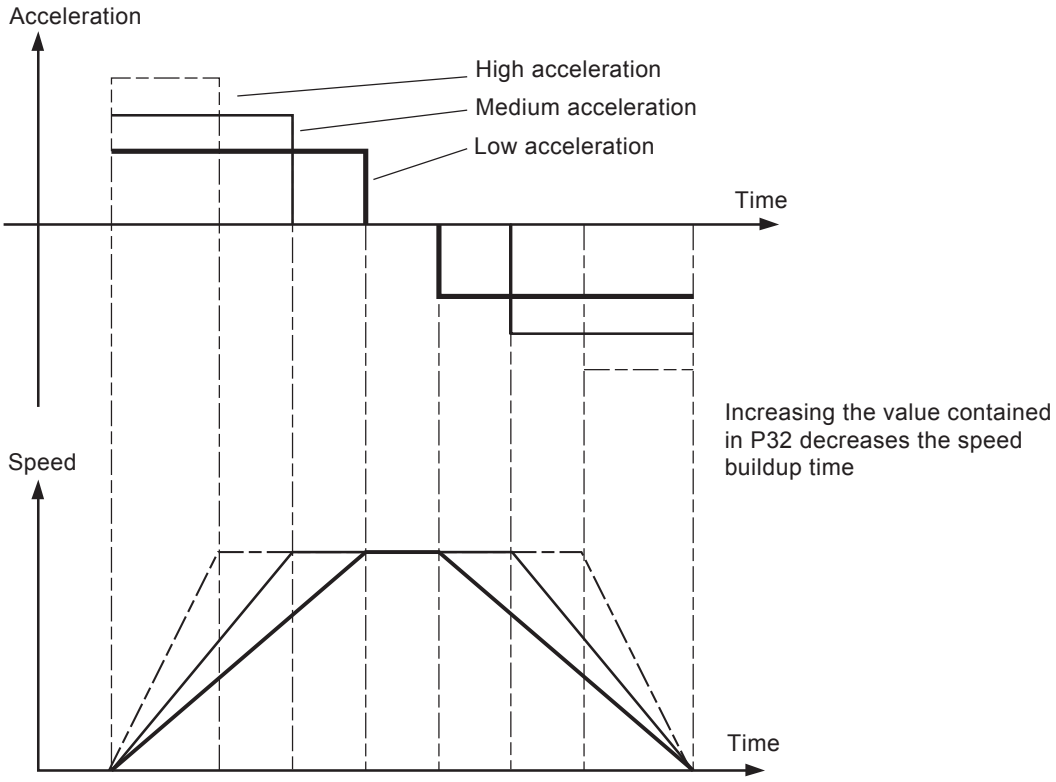
Parameter P6 Word 1 Bit 4	Programming format	Parameter P6 Word1 Bit6	X10	Acceleration unit
0	S05	0	NO	deg/s^2
0	S05	1	YES	10 deg/s^2
1	S32	0	NO	10^{-2} deg/s^2

The two words of the pair can have the same values.
For each servo-controlled axis, the accelerations must be defined and must be greater than 1 increment/sample^2 (sample = RTC in P50).

Physical addresses @24, @25, @26 and @27 are reserved for the acceleration settings of spindles 1, 2, 3 and 4 when present (words 48 and 49 for @24, words 50 and 51 for @25, words 52 and 53 for @26, words 54 and 55 for @27).


Physical axis @
List of words
@
Acceleration Work rate
Word N0
Acceleration Fast speed
Word N1
@
Acceleration Work rate
Word N2
Acceleration Fast speed
Word N3
@31
Acceleration Work rate
Word N62
Acceleration Fast speed
Word N63

Influence of the Acceleration on the Speed Buildup Diagram



When words 48, 50, 52 and 54 are equal to 0 and these words are the spindle accelerations, the spindles are started and stopped instantaneously without managing the accelerations. The same is true when programme parameter E9033b is set to 0.
The spindles on which the acceleration is set to zero must not be declared with constant acceleration (parameter P6, words 1 to 4, bit 5 equal to 0).

Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameter E971xx is used to read the acceleration at the work rate.

Programme parameter E972xx is used to read the acceleration at fast speed.

Programme parameter E9033b (b from 0 to 3) is used to read the indexed spindle acceleration expressed in revolutions per deg/s^2.

Extended NCK access (see PLC library "Extended NCK access")

The axis accelerations can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P32.

6.4 Direction of Axis Speed Reference ► P20

Description

Used to reverse the direction of the axis speed reference sent to the axis servo-drive.

Principle

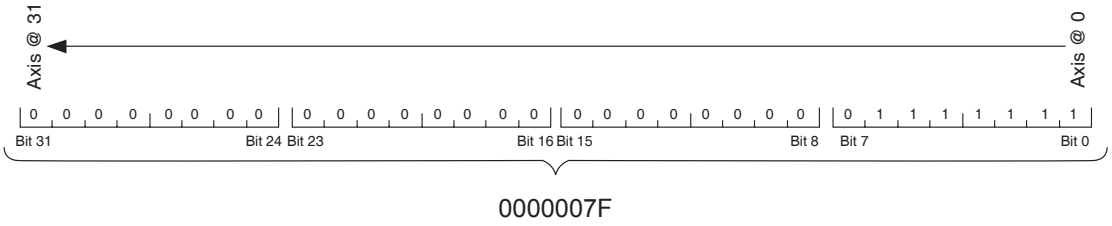
For DISC NT, this parameter is located in the servo-drive.

The bit position gives the physical axis address.

If the bit is set to 1, a positive speed reference applied to the servo-drive results in a negative movement of the axis.

Example

The axes at addresses 0 to 6 move in the negative direction for a positive speed reference.



0000007F

Word N0	0000007F

Extended NCK access (see PLC library "Extended NCK access")

The speed reference direction can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P20.

6.5 Servo-System Loop Gain Coefficient ► P21

Description

Defines the coefficient used by the CNC to calculate the reference supplied to the servo-drive according to the following error.

A Smith corrector is inserted in the position loop. It is automatically calculated by the CNC at initialisation.

It depends on the following parameters:
- P50 (sampling period)
- P56 (servo-loop time constant) or P66 (time constant per axis).

Before parameter P21 can be set, it is first necessary to set parameter P56 or P66.

Principle

Each word corresponds to the physical address of an axis:
- the value in word N0 is assigned to the axis with physical address 0,
- the value in word N1 is assigned to the axis with physical address 1,
- the value in word N2 is assigned to the axis with physical address 2,
- etc.

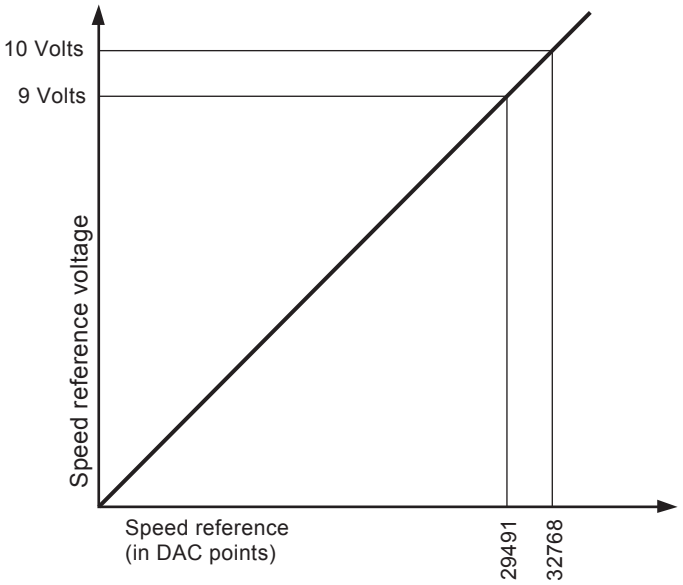
The value of each word gives the proportional action coefficient applied to the following error on the axis concerned to obtain the speed reference.

Used for analog axis only.

KVAR Calculation

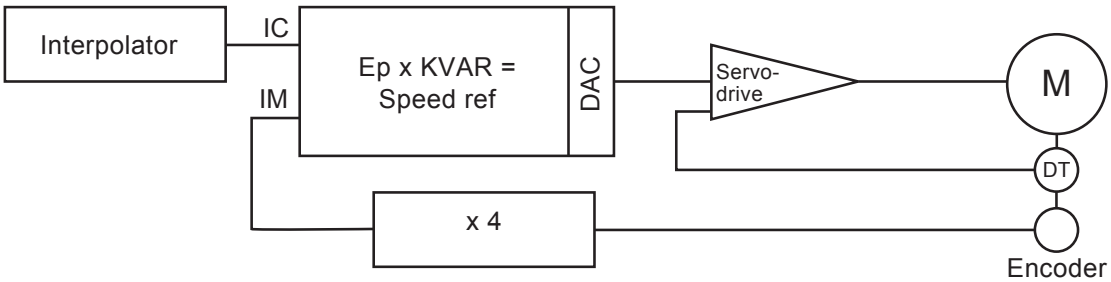
The CNC function generates a theoretical speed reference voltage of 10 V maximum.
This voltage is provided by a DAC on an axis encoder card. Generally, the servo-drives are set so that the axis speed is obtained for a servo-drive input voltage of 9 V.

The 15-bit DAC of the axis encoder card provides an output voltage of 10 V for 32768 DAC points. The output is 9 V for 29491 DAC points.



For VMAX, we have:

Max speed reference (in DAC points) = KVAR x max Ep



i.e.:

KVAR = Max speed reference (in DAC points)/Ep max x k

Where Ep (in mm or degrees) = Required position - Measured position

Coefficient k is used to take into account the internal system unit.

Coefficient k can take on the following values:

Internal measurement - Linear axes	Internal measurement - Rotary axes	Coefficient k
1/10 mm	1/10°	100
1/100 mm	1/100°	10
µm	1/1000°	1
1/10 µm	1/10000°	0.1
1/100 µm	-	0.01

The speed and following error are related by the servo-loop gain.

The equation is:

Gain (s^{-1})= Speed (in mm/s or deg/s)/Ep (in mm or deg)

hence :

Ep max = VMAX/Gain

Hence the equation for KVAR versus the gain

KVAR = 29491 x Gain/Max. speed x k

29491 DAC points correspond to a servo-drive input voltage of 9 V for maximum speed.

Equation for KVAR as a function of the time constant (T (in s) = 1/Gain):

KVAR = (29491 / (VMAX x T)) x k

Where:
- T : Position servo-loop time constant (s),
- VMAX : Maximum axis speed (mm/s or deg/s),
- k: Internal measurement coefficient.

The following error displayed on the CNC is in internal units (deg/10 to deg/10000 for rotary axes or mm/10 mm to µm/100 for linear axes)
For setting and monitoring P21, parameter P55, i.e. the speed anticipation coefficient, must be set to zero.

Equation for the Gain

In practice, the gain is often expressed in m/min/min of following error.

A gain of 1 m/min/min corresponds to a gain:

$G = 1000/60 \times 1 = 16.66\ s^{-1}$

and a time constant of:

$T = 1/16.66 = 0.060$ s, i.e. 60 ms.

Example

Calculation of KVAR for the axis at physical address 0 (VMAX = 100 mm/s, T = 60 ms, i.e. 0.06 s, internal unit μm).

KVAR = 29491 / (100 x 0.06) x 1 = 4915.16

Word N0	4915

Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameters E980xx (x: axis number from 0 to 31) are used to read or write servo-system loop gain coefficients whose value is expressed in thousandths.

Extended NCK access (see PLC library "Extended NCK access")

The servo-system loop gain coefficients can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P21.

6.6 Servo-Loop Time Constant ► P56

Description

Defines the position servo-loop time constant.

Principle

Each word corresponds to a channel:
- word N0 for channel 1,
- word N1 for channel 2,
- etc.

The value is in ms with a basic value of 60 ms.

This parameter is used for dynamic movement control and speed control at the end of a block.

The time constant is defined by the following equation:

$$T = 1/\text{Gain} = Ep/VIT$$

Where:
- T is in s,
- Ep is in mm or degrees,
- VIT is in mm/s or deg/s.

Example

For channel 1, the axis position servo-loop gain is 1 for a speed of 1 m/min (16.66 mm/s) and the following error is 1 mm.

T = 1/16.66 = 0.06 s, i.e. 60 ms

Word N0	60

The time constant defined this way is used for axes whose time constant was not specified in machine parameter P66, i.e. for which the corresponding word in P66 is set to 0.

Extended NCK access (see PLC library "Extended NCK access")

Programme parameter E41006 is used to read the servo-system loop gain time constant in ms

The time constants can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P56.

6.7 In-Position Window ► P22

Description

A distinction is made between two cases:

For stopping an axis at the end of movement, defines the following error threshold below which the CNC considers the movement to be finished.

At the end of a block including function G9, defines the following error threshold below which the CNC considers the movement in the block to be finished.

Principle

Each word corresponds to the physical address of an axis.
- The value in word N0 is assigned to the axis with physical address 0,
- The value in word N1 is assigned to the axis with physical address 1,
- The value in word N2 is assigned to the axis with physical address 2,
- etc.

Physical axis @	List of words	
@0	Word N0	
@1	Word N1	
'	'	
'	'	
'	'	
@31	Word N31	

This parameter is used for movements:
- in MANUAL, MDI and SINGLE modes
- at the end of a programme executed in AUTO mode or in a block including function G09 (accurate stop at the end of the block).

The CNC considers a movement to be finished when the residual following error on the axis falls below the value set in P22.

This value contributes to determining the stopping accuracy of an axis passively.

The values are expressed in the internal measurement unit: mm/10, mm/100, µm, µm/10 or µm/100 for linear axes or deg/10, deg/100, deg/1000 or deg/10000 for rotary axes (see P4 parameter "Axes Programmable by Diameter and Internal Measurement").

6.8 Maximum Following Error ► P23

Description

Defines the maximum permissible following error for each axis.

Principle

Each word corresponds to the physical address of an axis.

- The contents of word N0 are assigned to the axis with physical address 0,
- The contents of word N1 are assigned to the axis with physical address 1,
- The contents of word N2 are assigned to the axis with physical address 2,
- etc.

Physical axis @	List of words	
@0	Word N0	
@1	Word N1	
'	'	
'	'	
'	'	
@31	Word N31	

The values are expressed in the internal measurement unit: mm/10, mm/100, µm, µm/10 or µm/100 for linear axes or deg/10, deg/100, deg/1000 or deg/10000 for rotary axes (see P4 parameter "Axes Programmable by Diameter and Internal Measurement").

If the maximum following error is exceeded, the CNC goes into fault status and displays one of errors 40 to 71.

This value is called EMAX.

The practical equation for EMAX is:

EMAX = VMAX x T x 11/10/k

Where:

- T is the time constant of the position servo-loop (in ms),
- VMAX is the maximum speed on the axis (in mm/s or deg/s).

Coefficient k can take on the following values:

Internal measurement - Linear axes	Internal measurement - Rotary axes	Coefficient k
1/10 mm	1/10°	100
1/100 mm	1/100°	10
µm	1/1000°	1
1/10 µm	1/10000°	0.1
1/100 µm	-	0.01

Example

Calculation of EMAX for the axis at physical address 0 (VMAX = 100 mm/s, T = 60 ms, internal unit µm).

EMAX = 100 x 60 x 1 x 11/10 = 6600

Word N0	6600

Extended NCK access (see PLC library "Extended NCK access")

The maximum following errors can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to te state specified by P23.

6.9 Dynamic Movement Control ► P57

Description

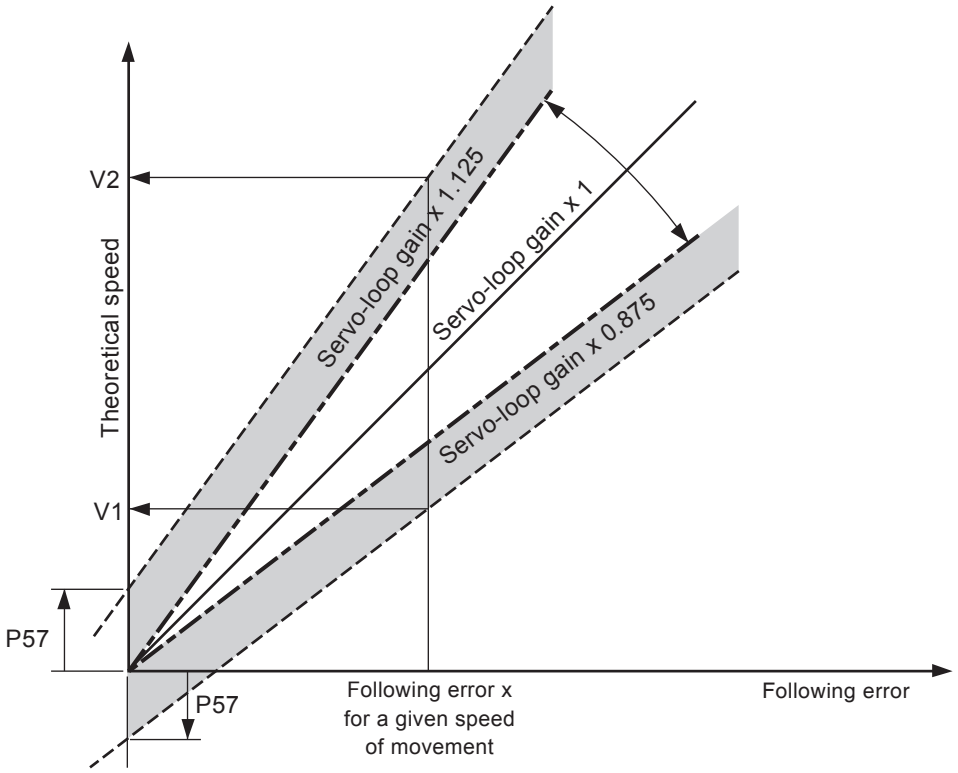
Used each sample (RTC in P50) to compare the real speed of an axis with its theoretical speed deduced from the following error.

Principle

Each word corresponds to the physical address of an axis.
- The contents of word N0 are assigned to the axis with physical address 0,
- The contents of word N1 are assigned to the axis with physical address 1,
- The contents of word N2 are assigned to the axis with physical address 2,
- etc.

Physical axis @	List of words	
@0	Word N0	
@1	Word N1	
'	'	
'	'	
'	'	
@31	Word N31	

The values are expressd in micrometres per sample.



The servo-loop gain multipliers (1.125 and 0.875) are defined by the system.

A following error gives two theoretical speeds, V1 and V2. The real axis speed must be between these two values for correct operation of the position servo-loop.

If the real speed is outside the theoretical speed band, the system generates one of errors 40 to 71.

For correct use of the settings of this parameter, the position servo-loop time constants must be specified in parameter P56.
The value of zero in a word disables dynamic control on the corresponding axis.

Extended NCK access (see PLC library "Extended NCK access")

Dynamic movement control can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P57.

6.10 Speed Anticipation Coefficient and Number of Filtered Terms for High Speed Cutting Machining ► P55

Description

Defines:

- The percentage of the speed reference reinjected in the position servo-loop so that it operates with a small following error, for words N0 to N7,
- The "2j+1" terms used to calculate the filtered reference for very high speed machining, in words N8 to N15.

Parameter P55 can be cancelled by a programme parameter to be able to use the dynamic axis alignment function (see the note at the end of this section)

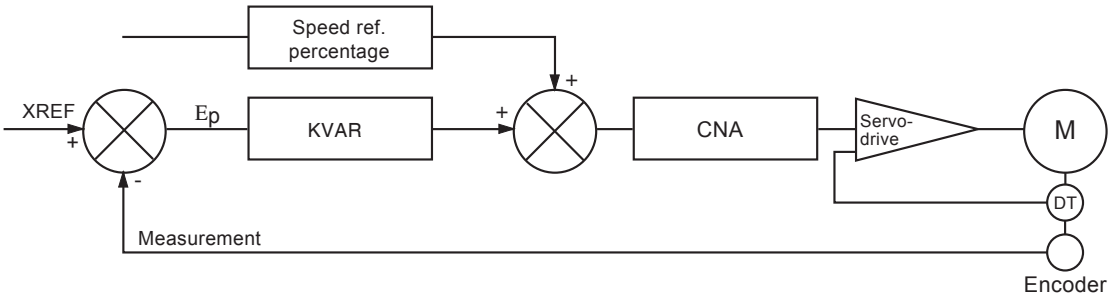
Principle

Each word represents a channel:

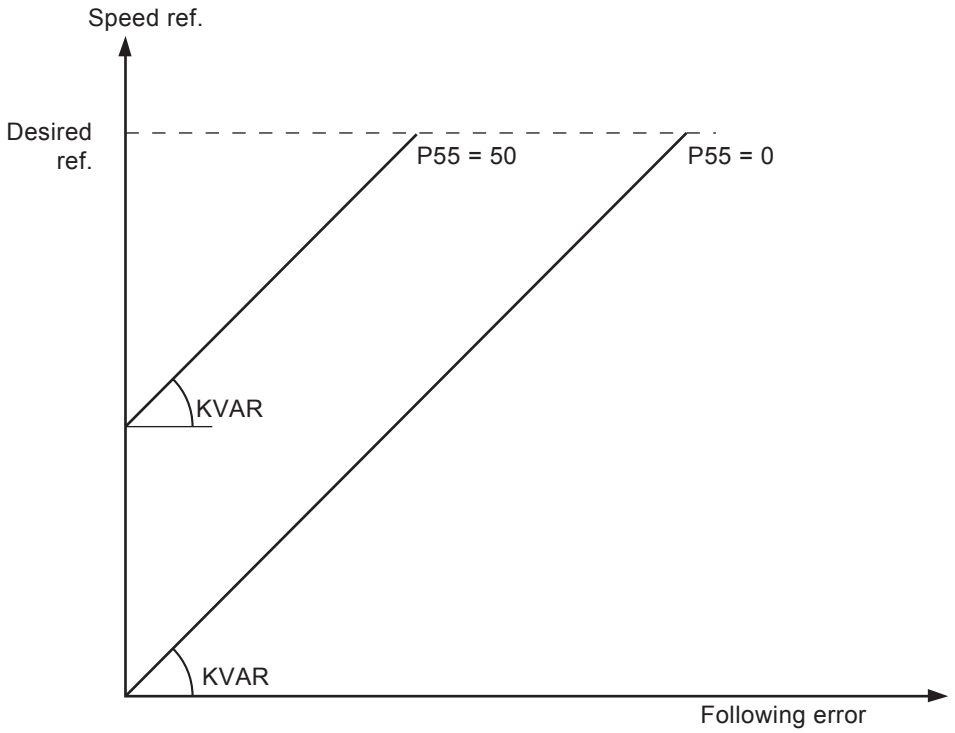
- Words N0 and N8 for channel 1,
- Words N1 and N9 for channel 2,
- etc.

Words N0 to N7 define a speed reference expressed as a percentage between 0 and 100.

If a value higher than 100 is set in P55, a value of 100 percent is used by the system.



A value of zero has no effect on the system.



Anticipation does not change the gain of the position loop.

STOP This parameter must be set to zero to determine parameter P21 (KVAR).

Words N8 to N15 : defines the number of references used to determine the value applied to the system.

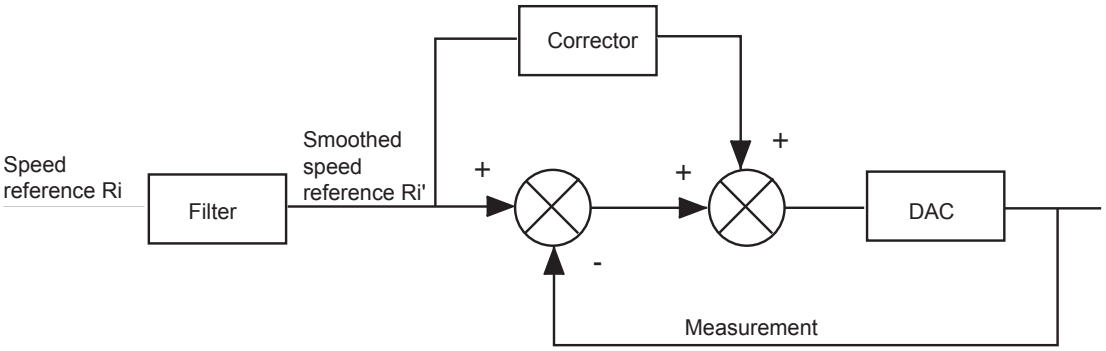
For very high speed machining, a filter smoothes the speed reference. This filter averages several consecutive settings.

The number of terms used for the filter specifies the number of calculated references averaged to calculate the smoothed value applied to the system.

Filtered Reference Calculation Method

$$R'i = \frac{\sum Ri'_{-1} + Ri - Ri_{-2j-1}}{2j + 1}$$

Each word specifies the number of terms "2j+1".
The order of magnitude of "2j+1" is typically around 5.



Antipitch detection requires the setting of this HSC filter to be above 0.

Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameter E11012 inhibits filter correction if the number of terms is not zero.
Setting the parameter enables the correction. A reset has no action on the value of E11012.

Programme parameter E32005 is the image of machine parameter P55 words 8-15. This parameter can be read and written and retains the last value programmed after a reset.

It can be written if:
- the high-speed machining is present; otherwise error 4 is declared
- anticipation is disabled (E11012 = 0), otherwise error 95 is returned
- its value is less than 14; otherwise error 94 is declared.

Extended NCK access (see PLC library "Extended NCK access")

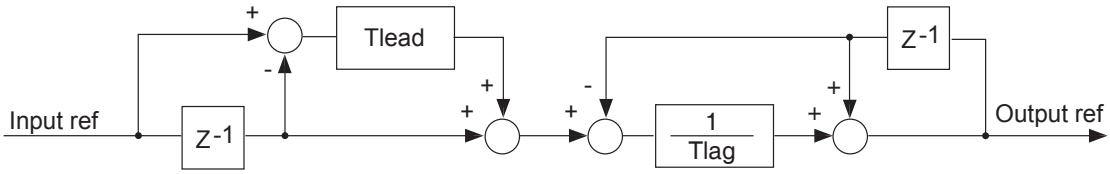
The speed anticipation coefficient can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P55.

Note on Use of Parameter P55

This parameter can be cancelled by programme parameter E923xx to be able to use the dynamic axis alignment function (see the note at the end of this section)

Description

When interpolating several axes with different dynamic characteristics without degrading individual performance on any axis, it is possible to apply a dynamic alignment function to each axis to ensure that the path is correct (or phase lead and lag compensation on the position reference)



Programme Parameters (see "Flexium Programming Manual M or T):

Programme parameter E923xx (xx = 0 to 31) enables the following programme parameters:
- E986xx (xx = 0 to 31) : Anticipation time constant in hundredths of ms
- E986yy (yy= 32 to 63) : Delay filter time constant in hundredths of ms

Programme parameter E11012 has precedence over programme parameter E923xx.
When a channel is declared for HSC, setting programme parameter E923xx generates error 95.
Enabling HSC via E11012 automatically resets E923xx for the programme's channel.

6.11 Characteristics of Synchronised Axis Pairs ► P24

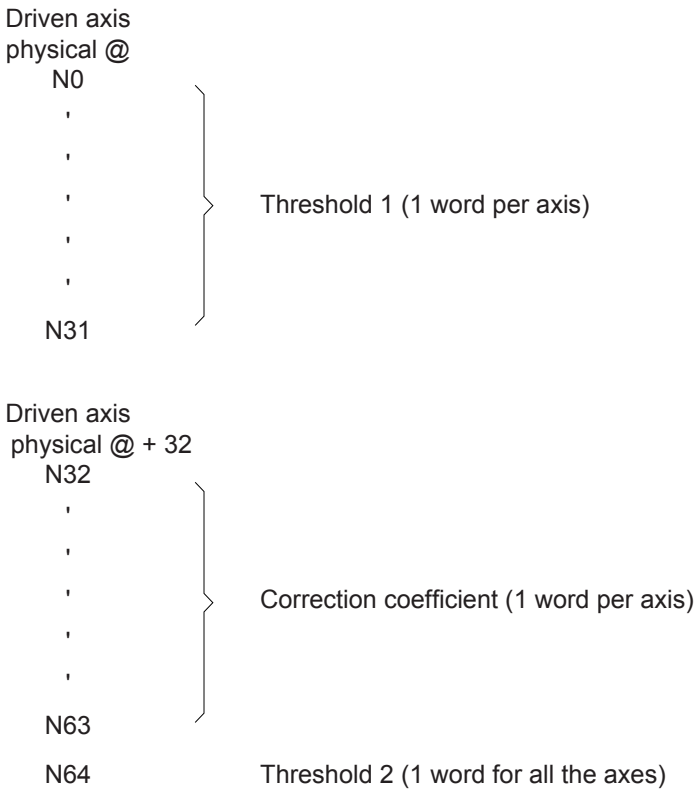
Description

Defines the synchronisation error tolerated by an axis, the correction coefficient for each axis and the maximum synchronisation error.

Principle

The word number gives the physical address of the driven axis of a pair. By convention, it is also the number of the axis pair.

Each word corresponds to the physical address of an axis.



Words N0 to N31 define threshold 1, which is the maximum permissible error (in internal measurement units) after synchronisation. Above this threshold, the system generates an error but allows the driven axis to be resynchronised on its drive axis. The default value is 160 µm.

Words N32 to N63 define the correction coefficient (in 1/1000) for each axis. It is a fraction of the error reinjected into the driven axis servo-loop to achieve synchronisation. The default value is 250.

Word N64 defines threshold 2, which is the maximum synchronisation error (in internal measurement units) tolerated for all the axis pairs. Above this value, the system generates an error and inhibits synchronisation. The default value is 600 µm.

The values must be strictly limited to the range 0-32767.
Entry of an incorrect value generates error 39.

The internal unit can be mm/10, mm/100, µm, µm/10 or µm/100 for linear axes or deg/10, deg/100, deg/1000 or deg/10000 for rotary axes (see P4 parameter "Axes Programmable by Diameter and Internal Measurement").

The error generated when thresholds 1 and 2 are exceeded causes reset of CN_PRET and display of error 39.

6.12 Servo-Loop Error Tolerated on Circles ► P52

This chapter covers the management of errors in tracing circles:

Word N0: Defines the maximum servo error allowed for circles.

Description

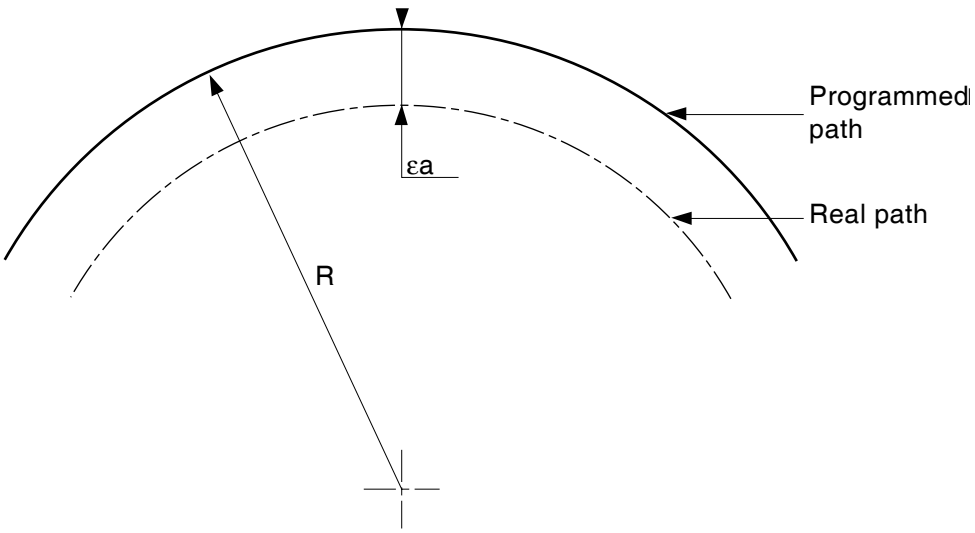
This parameter is used to limit the feed rate based on the path curvature so as never to exceed the maximum specified error.

It is always applied, both when HSC is enabled by programme parameter E11012 = 1, and when it is disabled by E11012 = 0.

However, it must be set differently for the two above cases.

Principle

εa (servo-loop error) is the error in mm between a programmed path and the real path.



For conventional machining (E11012 = 0)

The basic value is 50 µm.

εa is approximated by the following equation:

$$\varepsilon a = Vmax^2 \times T^2 / 2R$$

Where:

- εa Servo-loop error (in m),
- Vmax Max speed (in m/s),
- T Time constant (in s),
- R Radius (in m).

This equation can be reformulated to give the maximum tolerated speed:

$$Vmax = \frac{\sqrt{2R \times \varepsilon a}}{T}$$

Example:

Calculation of the maximum tolerated speed for εa = 50 µm T= 60 ms and R = 50 mm.

We have:

$$Vmax = \frac{\sqrt{2 \times 50.10^{-3} \times 50.10^{-6}}}{60.10^{-3}} = \frac{\sqrt{5.10^{-6}}}{60.10^{-3}} = \frac{2.236}{60} = 0.037 \text{ m/s, i.e. } 2.236 \text{ m/min.}$$

For HSC machines (E11012 = 1)

For HSC, this parameter must be set to a low value, especially for finishing (a few micrometres).

This parameter is applied not only to circles but also to curves programmed by small segments (TABCYLS).

For the same setting of P52, speed limiting is less penalising for HSC than for normal machining, because the CNC uses a higher feed rate, allowed by HSC. This speed advantage justifies the use of HSC.

Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameter E32002 is the image in micrometres of machine parameter P52.
This parameter can be read and written and retains the last value programmed after a reset.

Word N1: Defines the path checks in RTCP.

Description

This parameter defines the maximum angular value between 2 check points.

If one or more rotating axes are programmed in RTCP, it is left to the user to choose the number of check points on the trajectory.

Principle

To verify the trajectory that has just been calculated, the numerical control will test the conditions to be met over a number of check points.

These points are specified so that they are uniformly distributed along the trajectory with the angular distance between two consecutive points less than or equal to the angular distance defined in P52N1.

For each point thus specified, checks are carried out for the axes of the pivot point.

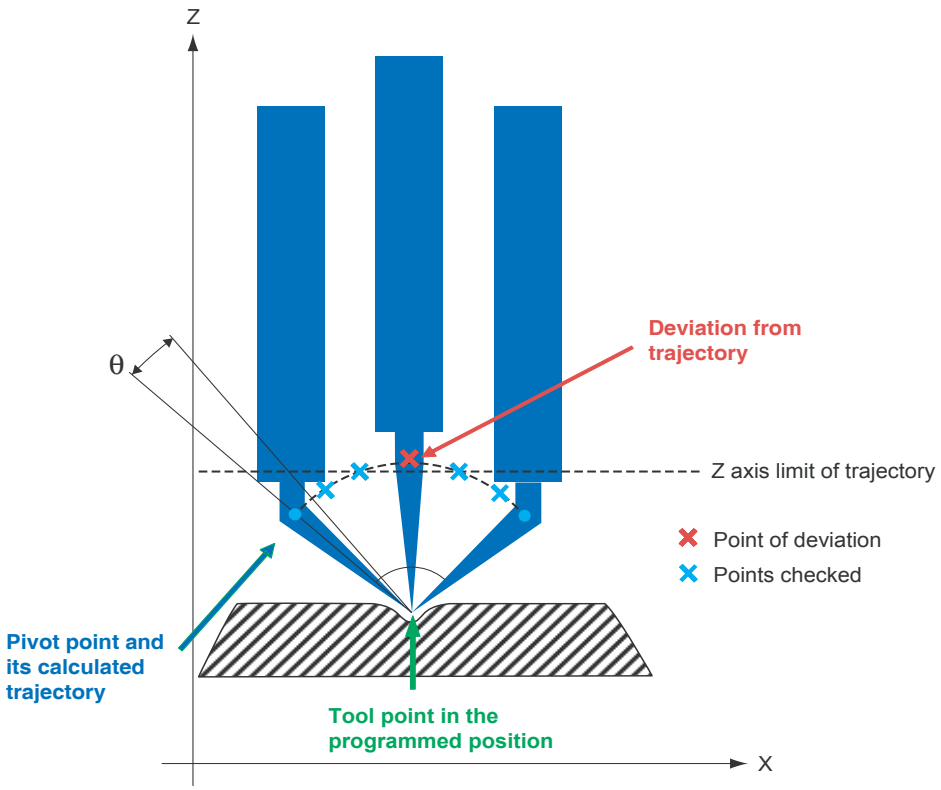
The checks are valid for the following types of interpolated trajectories:
- Linear interpolation,
- Circular interpolation,
- Segmented polynomial interpolation.

Only smoothed polynomial interpolation is not covered.

Parameter P52N1 is set in degrees. The system automatically takes account of the definition of rotating axes given by P4N4. The system allows values from 5° to 45°. Outside this range, it sets the value to 0, which leads to invalidation of the function.

Determination of the value of parameter P52N1 has to be a compromise between the precision required in checking the trajectories and the additional time required for preparation of the block.

This diagram shows how the check is carried out:



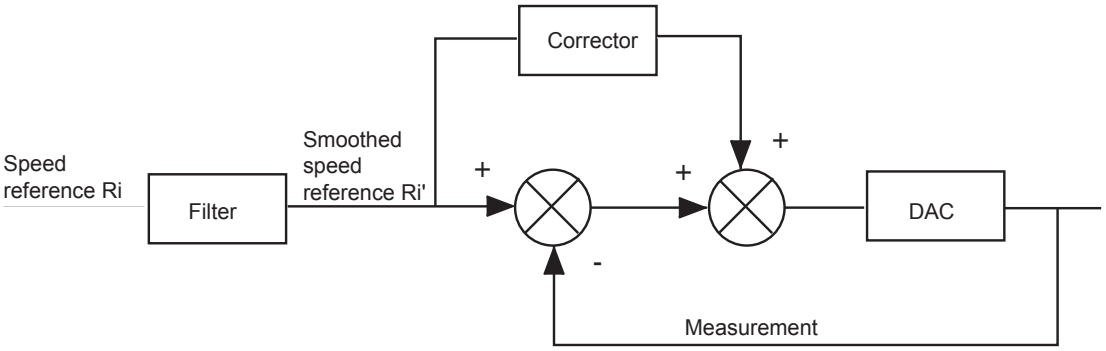
θ: The angle θ is calculated by the CNC so that the check points are uniformly distributed along the circular trajectory and the angular distance between two consecutive points is less than that given by P52N1.

6.13 High Speed Cutting Machining ► P19

Description

This parameter is used to set:
- the acceleration anticipation for each axis,
- the dry friction correction amplitude for each axis and activate the antipitch function in the MDLU drive.
- the dry friction correction time constant for each axis.

Principle



Words N0 to N31: define the acceleration anticipation

Each word corresponds to the physical address of an axis. The acceleration anticipation is expressed in µs. The order of magnitude is 400 µs.

The acceleration anticipation Ka' is applied in the following formula:

$$Cor = \frac{Kv}{2j+1}(Ri-Ri_{-2j-1}) + \frac{Ka}{2j+1}(Ri-Ri_{-2j-1}+Ri_{-2j-2})$$

The speed anticipation corrector "Kv" is set to 100 percent and cannot be modified.

Words N32 to N63: define the dry friction correction amplitude

Each word corresponds to the physical address of an axis.
The dry friction correction amplitude is expressed in internal measurement units.
The order of magnitude is 10 µs.

Words N64 to N95: define the dry friction correction time constant

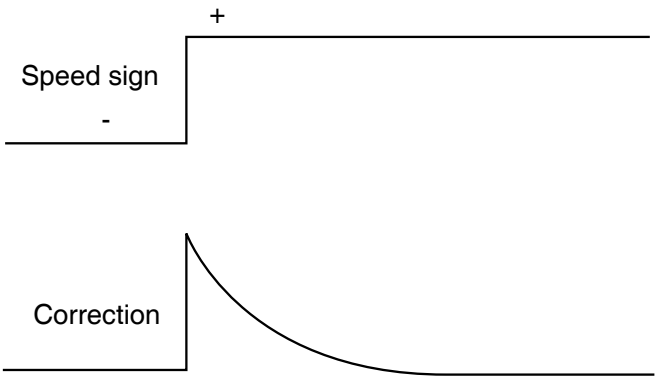
Each word corresponds to the physical address of an axis. The dry friction correction time constant is expressed in hundredths of ms. The order of magnitude is 5000 hundredths of ms.

Dry friction elimination.

The dry friction correction amplitude Fi and dry friction correction time constant Rf are applied in the equation:

$$Cor\_Fsi = Fi\text{-}Fi_{-1} + Cor\_Fsi_{-1}\left(1 - \frac{Te}{Tf}\right)$$

where:
- Cor_Fsi = dry friction correction
- T = sampling period



Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameters E982xx are the image in micrometres of machine parameter P19 words 32-63. They can be read and written and retain the last value programmed after a reset.

Programme parameters E983xx are the image in hundredths of a ms of machine parameter P19 words 64-95. They can be read and written and retain the last value programmed after a reset.

Programme parameters E981xx are the image in µs of machine parameter P19 words 0 - 31. They can be read and written and retain the last value programme after a reset.

6.14 Approach Speed ► P33

Description

Specifies the maximum speed when passing from one block to another.

Principle

Each word corresponds to the physical address of an axis. The value is expressed in mm/min or in deg/min. The default value is 0.

The approach speed is defined by the equation:

$$Va = \gamma \times T$$

where:
- γ is the maximum permissible acceleration in mm/s^2 or deg/s^2 (P32)
- T is the servo-loop time constant in ms (P56).

If the very high speed machining option is present, the approach speed should be divided by a value of around 9.

Example

Where $\gamma = 1000\ mm/s^2$ and T = 60 ms, we have:

$Va = \gamma \times T = 1000 \times 60 \times 10^{-3} = 60\ mm/s$, i.e. 3600 mm/min

Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameter E9730xx is used to read and write this speed in m/min.

Extended NCK access (see PLC library "Extended NCK access")

This parameter can be modified by a Extended NCK request.

6.15 Setting of the Speed Buildup in the Path ► P53

Description

This parameter limits the acceleration derivative (jerk) during a speed buildup with gradual acceleration. The type of jerk is specified by the setting of parameter E11013.

The jerk is expressed in m/s^3.

To facilitate understanding of this physical phenomenon, it is preferable to express it as an acceleration buildup time.

Depending on the setting of E11013, parameter P53 defines the minimum time of the acceleration and/or speed buildup phase of the so-called sine square speed law.

This time is in ms and is common to all the channels.

For a trapezoidal jerk profile (E11013 = -1), the setting of P53 is limited to 200 sampling periods (200*P50).

The setting of P53 can be read and written by programme parameter E32006.

Principle

The speed buildup profile is specified by the setting of E11013.

Several types of acceleration profile can be selected.

E11013	=	-1	Trapezoidal jerk (parabolic acceleration profile) recommended for HSC
E11013	=	+1	Square jerk (sawtooth acceleration profile)
E11013	=	0	Constant acceleration.

The setting of this parameter is saved after a reset.

At power on, sawtooth acceleration is enabled (i.e. E11013 = +1) if the $\sin^2$ speed profile is selected in parameter P7, word 1, bit 5 (=1).

Trapezoidal Jerk : E11013 = -1

Description

This type of speed buildup profile is defined by parameter P53 (or E32006) in conjunction with parameter E32008.

This jerk profile allows faster speed buildups and stopping, especially for high speeds of movement (e.g. positioning blocks).

It allows a speed buildup profile with gradual acceleration and deceleration especially well suited to high speed cutting (HSC) by:
- Decreasing mechanical jerks and stresses on the machine
- Decreasing profile errors and synchronisation errors.

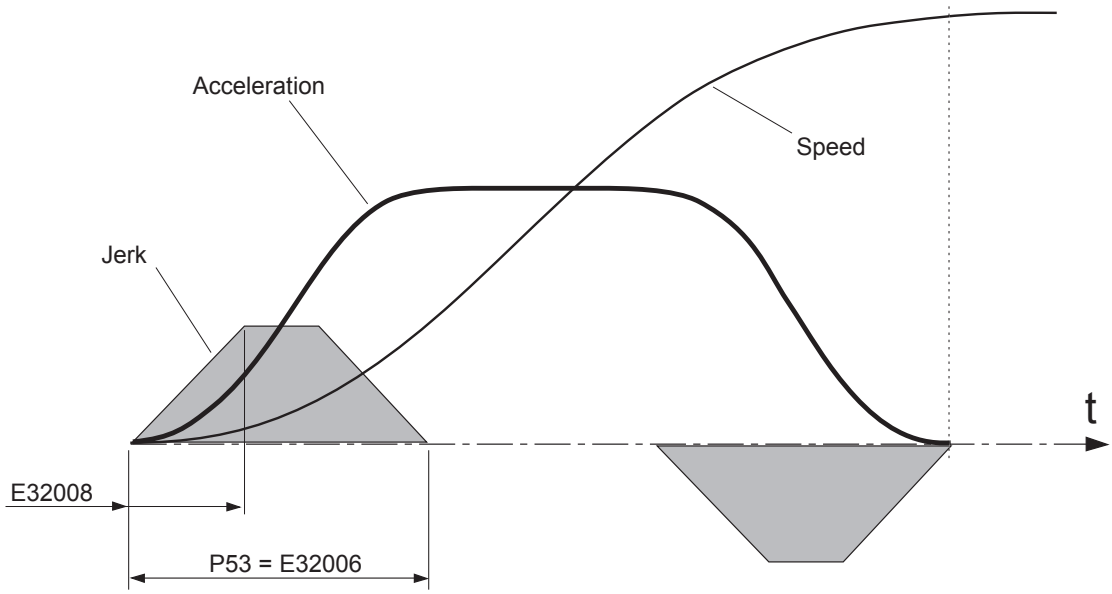
This jerk profile allows a higher maximum acceleration (P32) to be set since the machine is less sensitive to acceleration effects because acceleration is gradual.

Parameter P53 sets the time in ms of increase to the maximum acceleration.

Parameter E32008 sets the time in ms of increase to the maximum jerk.

These two parameters define a trapezoidal jerk profile.

The maximum jerk applied to each machine axis depends on parameters P53 and E32008 setting the speed buildup and maximum acceleration applied to the axis (P32): $J_{MAX} = g_{MAX}/(P53-E32008)$.



Recommendation

E32008 is adjustable between 0 and P53/2:
- If E32008 = 0, the jerk profile is constant by segments, which produces a trapezoidal acceleration
- If E32008 = P53/2, the jerk profile is a sawtooth, which produces a parabolic acceleration by segments

It is recommended to set:
- Parameter P53 to two or three times the position loop time constant (P56), and
- Parameter E32008 to between P53/3 and P53/2.

Smoothing the acceleration buildup profile (increasing P53 and E32008) increases the speed buildup time.

To reduce the machining time, it is often preferable to increase the maximum acceleration (P32) within the limit of the machine performance rather than increasing the jerk.

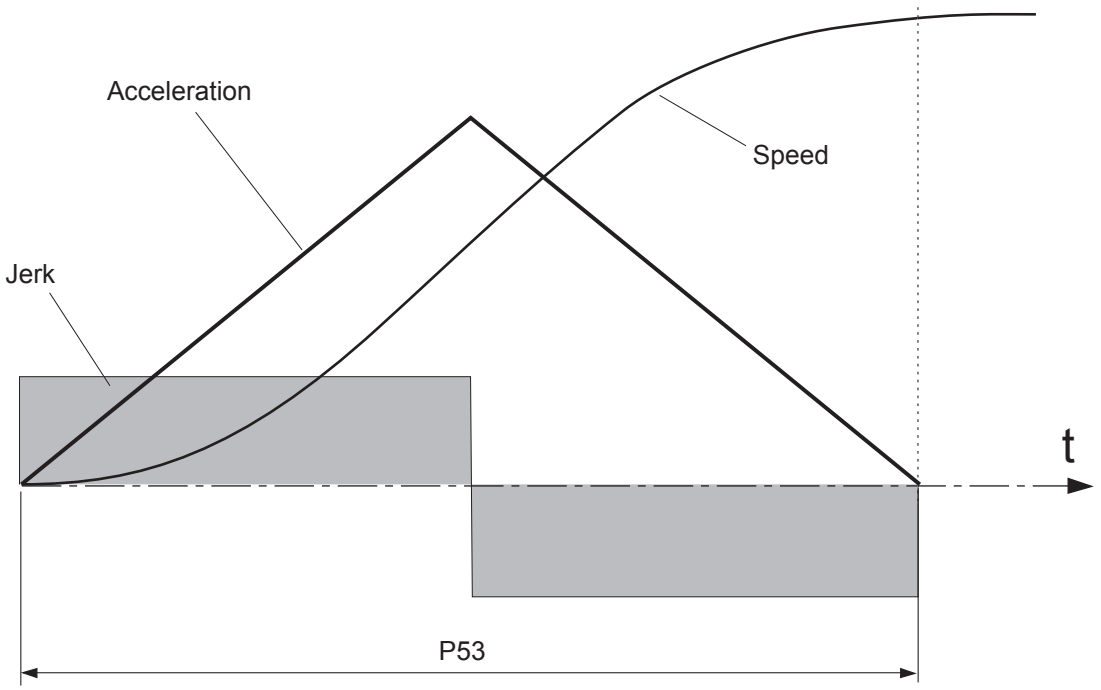
Trapezoidal acceleration cannot be used with sequenced thread cutting. In G38, E11013 is forced to 0. It is necessary to reset E11013 to -1 after thread cutting. In G33, trapezoidal acceleration is cancelled during the cycle, but is resumed automatically at the end.

Note:

Sawtooth acceleration : E11013 = 1.

Description

Sawtooth acceleration allows more fluid speed variations when the geometric profile includes a close sequence of curve radius changes.



With this type of speed buildup, acceleration is gradual but the jerk level is variable as it depends on the speed level.

The jerk is a maximum for a speed level DV = (P32/2)*P53 in which case it is $J_{MAX\ AXIS} = 2*P32_{AXIS}/P53$.

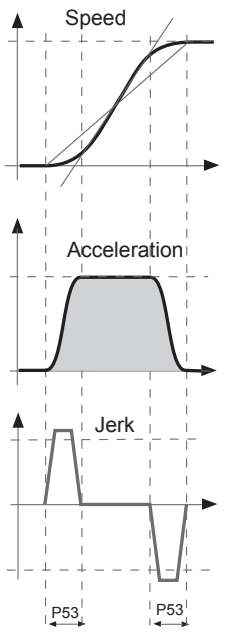
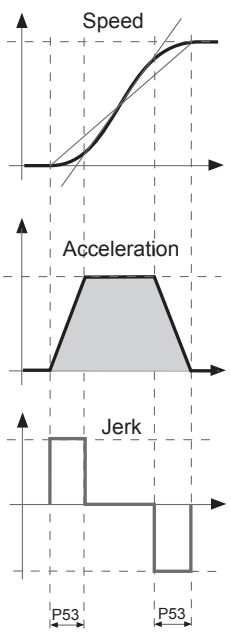
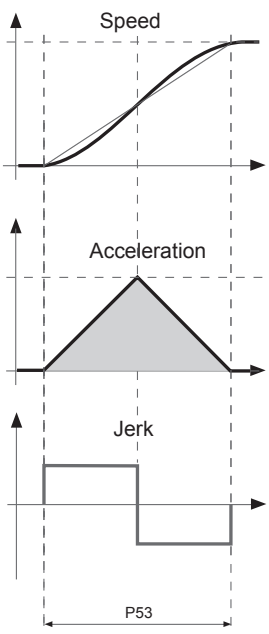
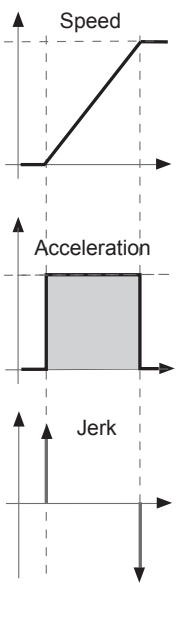
Recommendation

It is recommended to set parameter P53 to three or four times the position loop time constant (P56).

Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameter E32006 is used to read and write this time in ms.

Comparison of Speed Builup Profile

	Conditions			
	E11013= -1	**E11013= -1**	**E11013= 1**	**E11013= 0**
	P53	**P53**	**P53**	**-**
	E32008=P53/3	E32008=0	-	-
Speed curve **Acceleration curve** **Jerk Curve**				
Role of P53	P53 sets the acceleration buildup time		P53 sets the minimum speed buildup time	P53 is not significant
Acceleration	The maximum acceleration P32 is reached for a long time, giving a short speed buildup time.		The maximum acceleration P32 is reached only for a very short time	The speed buildup time is at minimum level
Calculation of $J_{MAX\ AXE}$	$J_{MAX\ AXIS} = \frac{P32_{AXIS}}{P53 - E32008}$	$J_{MAX\ AXIS} = \frac{P32_{AXIS}}{P53}$	$J_{MAX\ AXIS\ maximum} = \frac{2 \times P32_{AXIS}}{P53}$	–
Type of Jerk	The shape and level of the trapezoidal jerk can be set	The square jerk has a constant level which can be set	The jerk is square but the jerk level varies according to the geometric profile and the speed variations required	The jerk level is very high (theoretically infinite)
Type of machine	HSC	HSC	HSC	Conventional

6.16 Time Constant per Axis ► P66

Description

This parameter defines a position loop time constant for each axis.

This time constant is expressed in ms, with a basic time of 60 ms.

Default value forced to zero.

Principle

In earlier versions, machine parameter P56 was used to specify a common time constant for all the axes of a channel. With total speed anticipation, different time constants can be used on the axes in a channel, corresponding to the optimum settings.

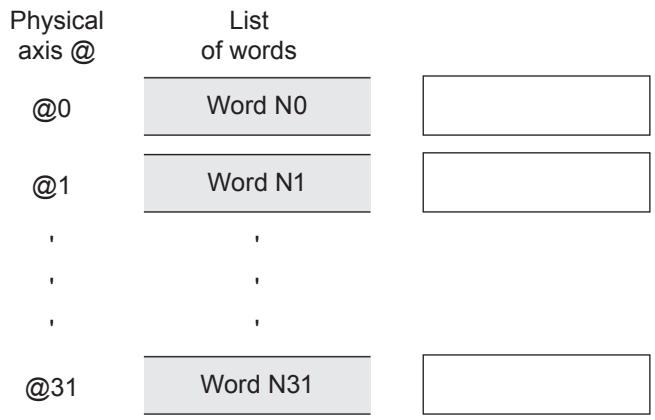
The time constant per axis is used with total and partial speed anticipation, dynamic movement control and analysis of angle crossing speeds for axes whose parameter P33 is set to 0.

The time constant per channel P56 continues to be used for speed limiting based on path curve radii to control the servo-control error.

P56 is also used for axes whose word in P66 is set to 0.

Each word corresponds to the physical address of an axis:

- The contents of word N0 are assigned to the axis with physical address 0
- The contents of word N1 are assigned to the axis with physical address 1
- etc.



6.17 JOG Speed Setting

The JOG speeds are set by parameter P31

The speeds are expressed in mm/min or deg/min.
They can be overridden by the feed rate potentiometer.

Slow JOG

This speed is accessible only by programming the PLC function.
It can be used as approach speed.

Word N0	

Normal JOG

This is the nomal speed for manual movement. It is set at 1 m/min.

Word N1	

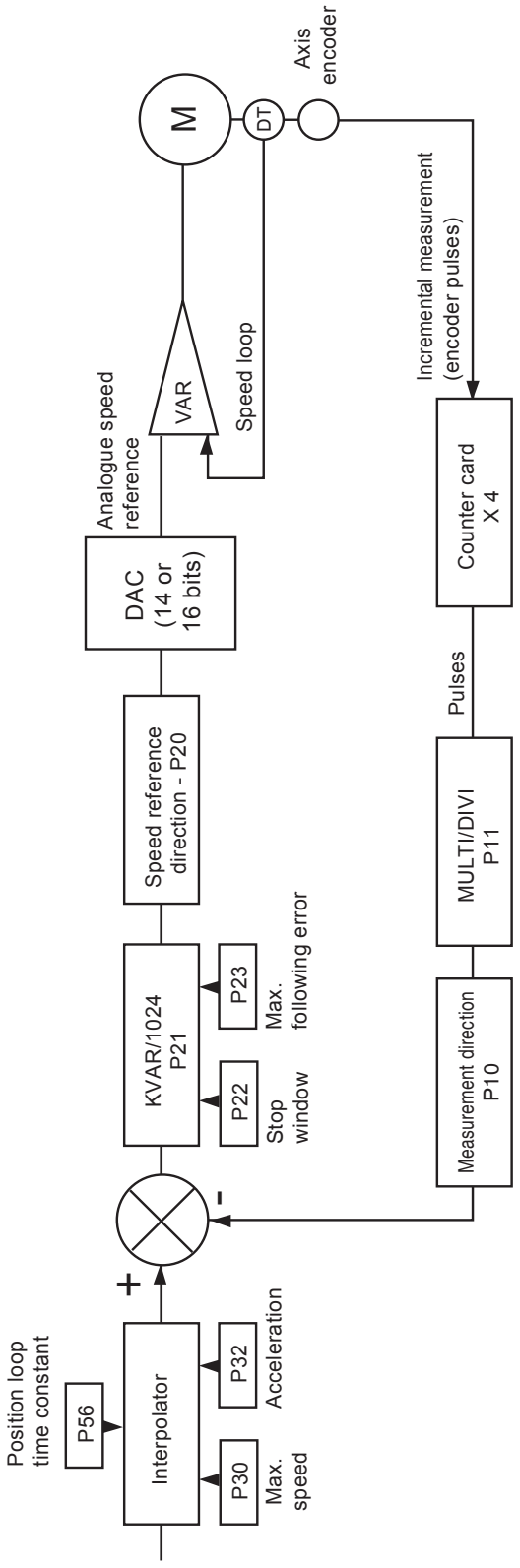
Fast JOG

Enables the axes to be moved at maximum speed.
The value set in word N2 is the maximum speed of the slowest axis.

Word N2	

The JOG type is selected by the PLC for signals W_ALLCNC.ManualSpeed.SlowSpeed and fastSpeed.

6.18 General Block Diagram of Servo-Control



Page deliberately empty

Page deliberately empty

Summary

7 Axis Travels 117
7.1 Reminders 117
7.2 Homing Direction ► P15 120
7.3 Datum Switch Position in Machine Dimensions ► P16 122
7.4 Axis Travel Limits ► P17 124
7.5 Backlash Error Compensation ► P18 126

Page deliberately empty

7 Axis Travels

7.1 Reminders

The measurement origin " OM " and machine zero point " Om " coincide.

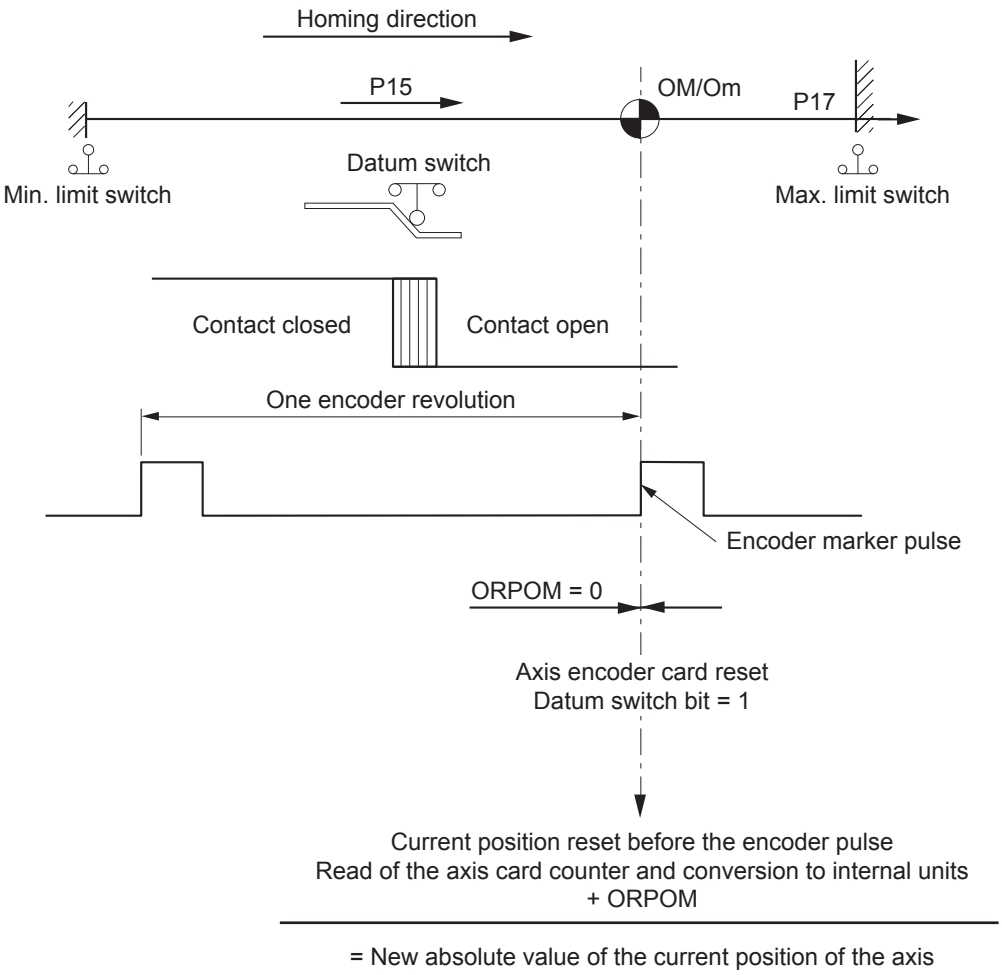
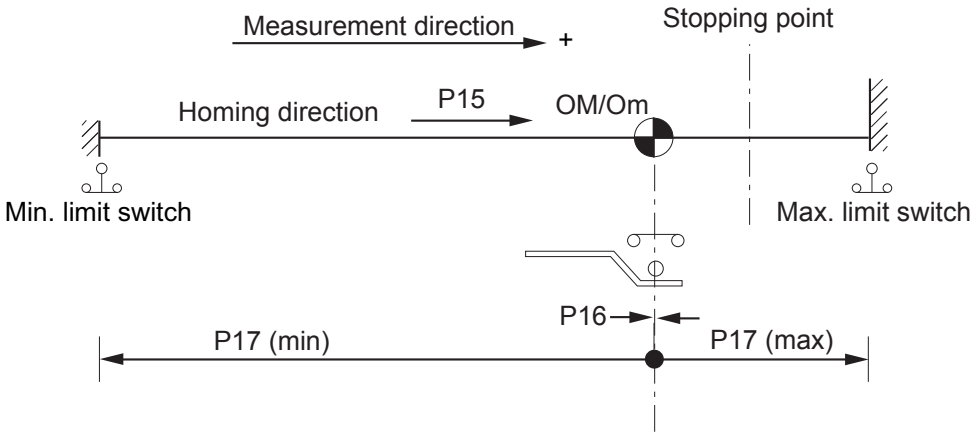


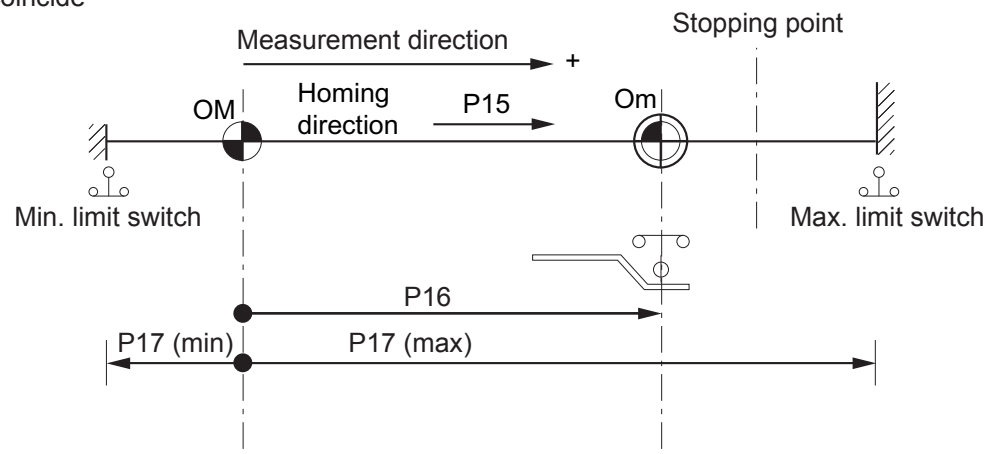
Figure 7.1 - Detail of homing

Summary of the Different Cases of Homing on the Datum switch

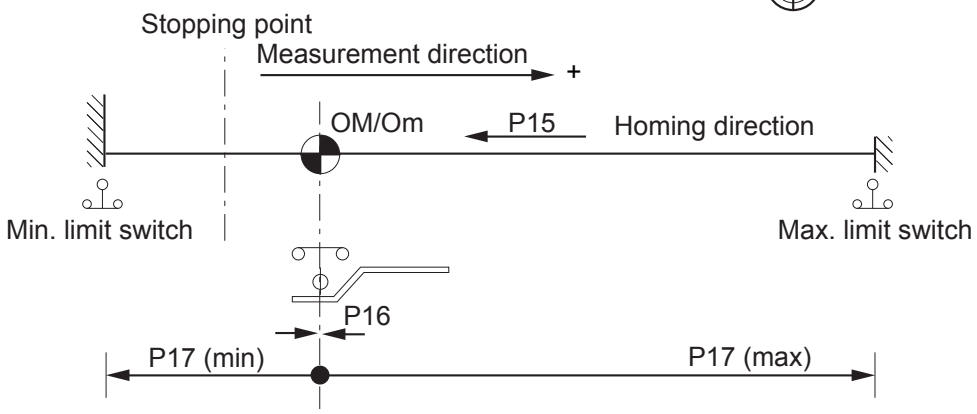
Case 1: Positive homing direction
The measurement origin "OM" and machine zero point "Om" coincide



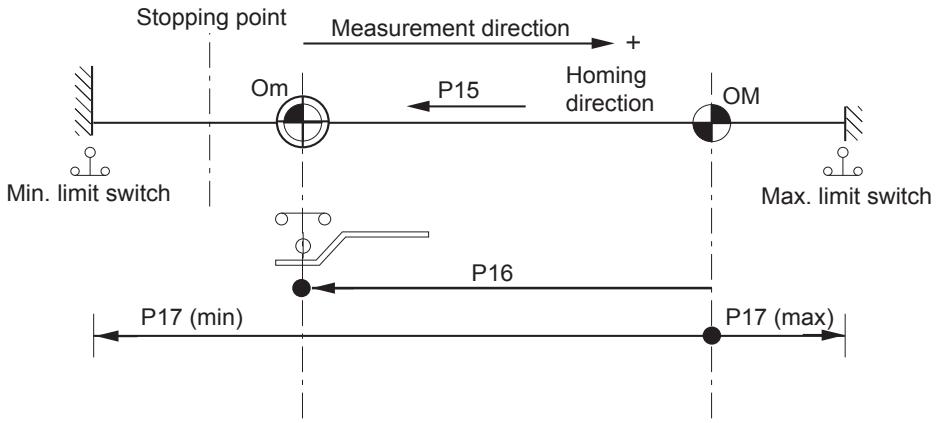
Case 2: Positive homing direction
The measurement origin "OM" and machine zero point "Om" do not coincide



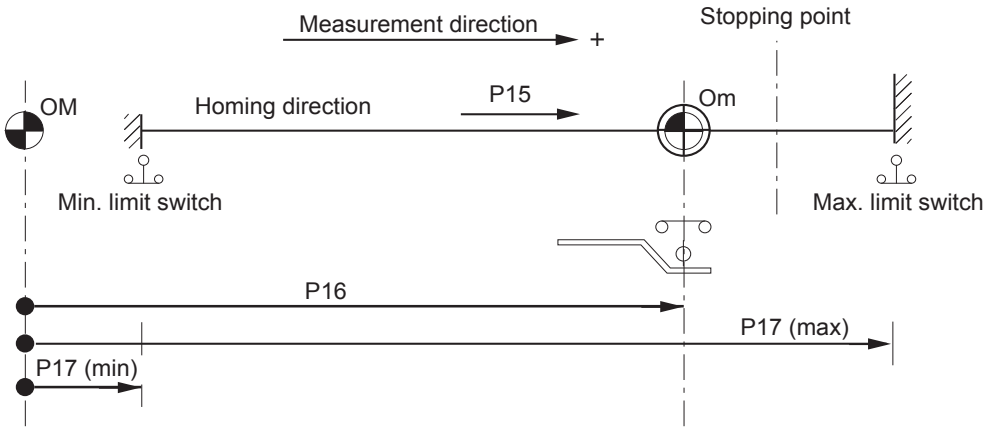
Case 3: Negative homing direction
The measurement origin "OM" and machine zero point "Om" coincide



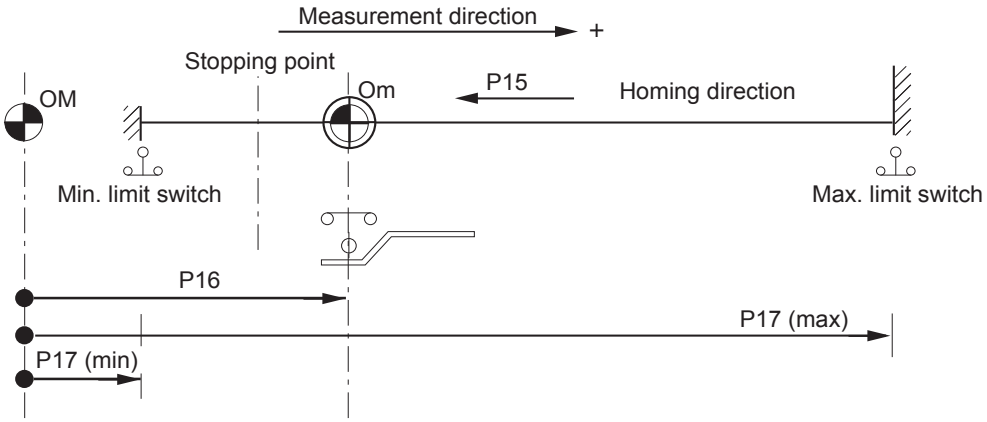
Case 4: Negative homing direction
The measurement origin "OM ◓" and machine zero point "Om ⊕" do not coincide



Case 5: Positive homing direction
The measurement origin "OM ◓" is not accessible



Case 6: Negative homing direction
The measurement origin "OM ◓ " is not accessible



7.2 Homing Direction ► P15

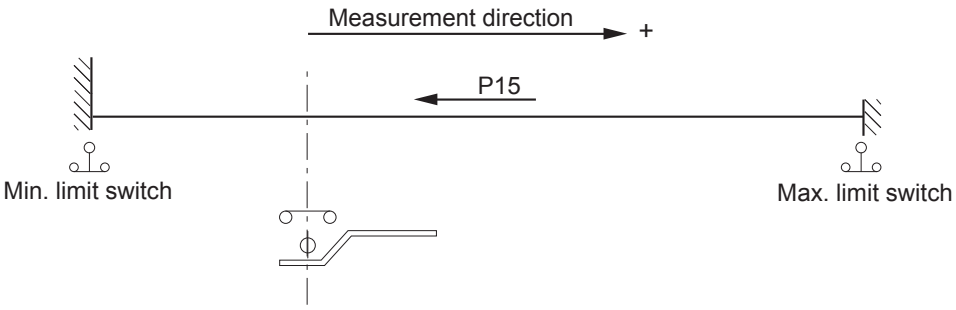
Description

Defines the direction from which the datum switch is approached in homing mode.
Inhibits the switch status test on axes which do not have a datum switch.

Principle

The bit position gives the physical axis address.

Word N0 : specifies the datum switch approach direction in homing mode.



If the bit is set to 1, homing is in the negative direction on the axis.

Word N1: inhibits the switch status test for axes which do not have a datum switch.
If the bit is set to 1, the datum switch is not tested on the corresponding axis.
In homing mode, homing occurs on the first marker pulse encountered.

Programme Parameters (see "Flexium Programming Manual M or T):

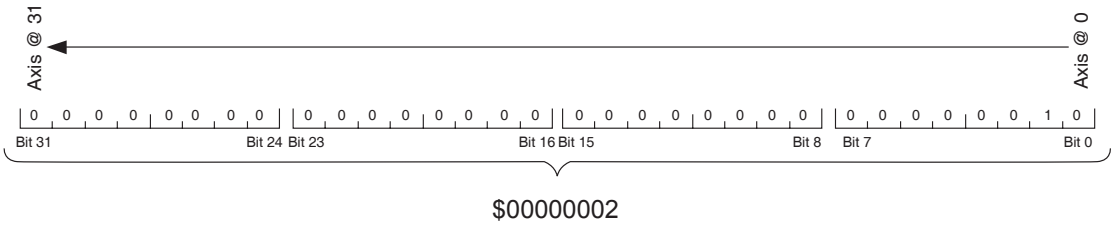
Programme parameter E933xx is used to read the value of the homing direction.

Extended NCK access (see PLC library "Extended NCK access")

The direction of homing on the machine zero point can be modified by a Extended NCK request. A reset (CNC reset) resets the CNC to the state specified by P15.

Example

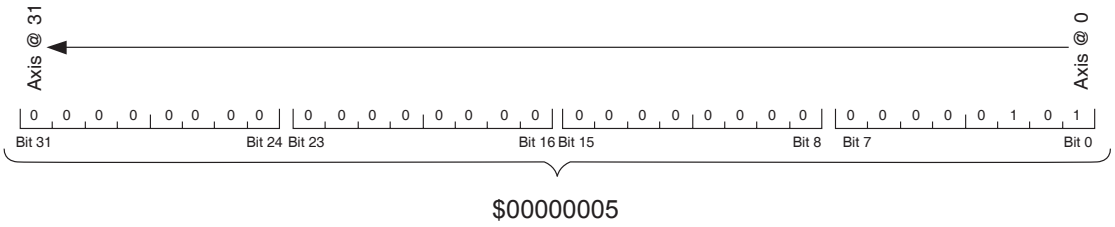
For the axis at physical address 1, homing is in the negative direction.



$00000002

Word N0	00000002

The axes at physical addresses 0 and 2 do not have a datum switch.



$00000005

Word N1	00000005

7.3 Datum Switch Position in Machine Dimensions ► P16

Description

Used to set the measurement origin (ORPOM) of an axis within or outside the travel limits.

Principle

The word number gives the physical address of the axis.

Physical axis @	List of words	
@0	Word N0	
@1	Word N1	
'	'	
'	'	
'	'	
@31	Word N31	

The values are expressed in the internal measurement unit: mm/10, mm/100, µm, µm/10 or µm/100 for linear axes and deg/10, deg/100, deg/1000 or deg/10000 for rotary axes (see P4 parameter "Axes Programmable by Diameter and Internal Measurement" elsewhere in this manual).

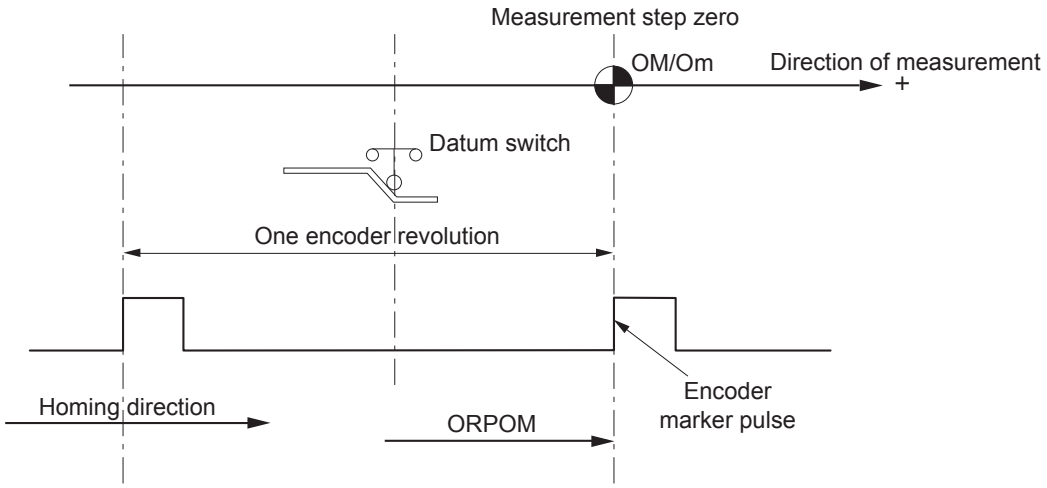
Each word contains the position of the datum switch on the corresponding axis in machine dimensions. The sign of ORPOM is as viewed from the measurement origin OM to the datum switch, taking the axis direction into account. (See examples earlier in this section.).

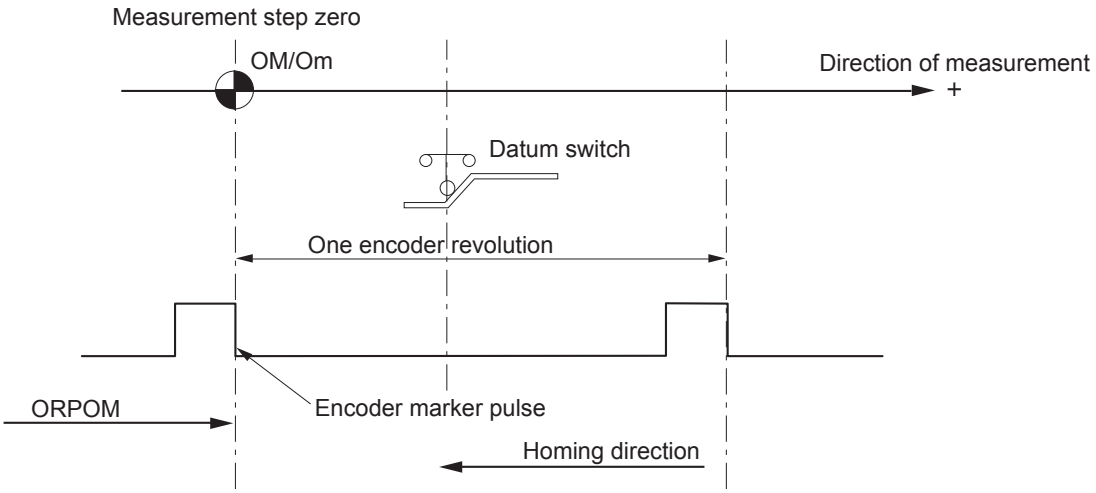
Extended NCK access (see PLC library "Extended NCK access")

The measurement origin on an axis can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P16.

Case of Encoder

ORPOM gives the encoder step ZERO encountered after the datum switch during a homing opeation.





7.4 Axis Travel Limits ► P17

Description

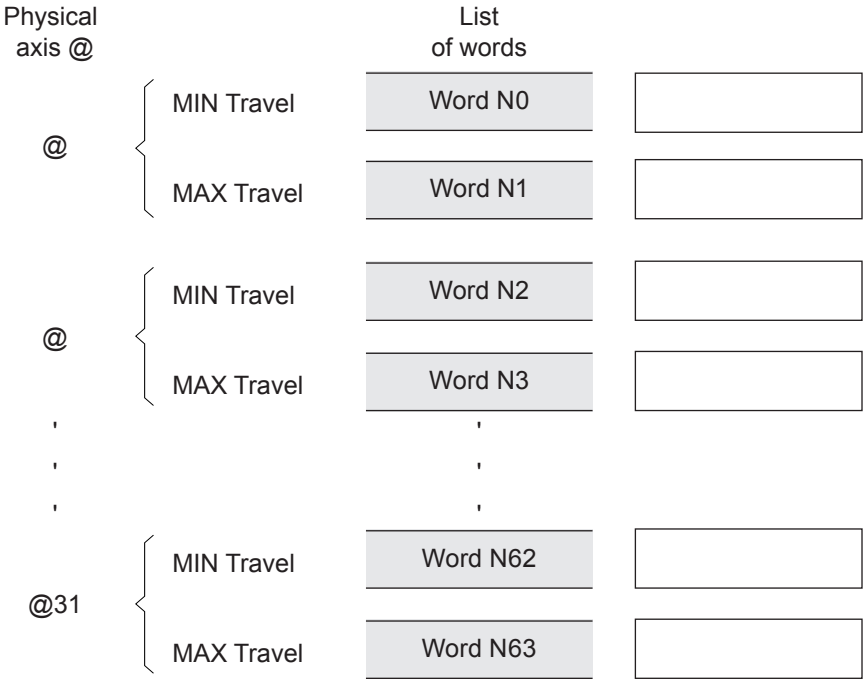
Defines the axis travel limits. Used to limit software travel in the measurement space.

Principle

Each pair of words defines the upper and lower travel limits for an axis.

The first word in the pair gives the lower limit (minimum or negative) of MIN travel.

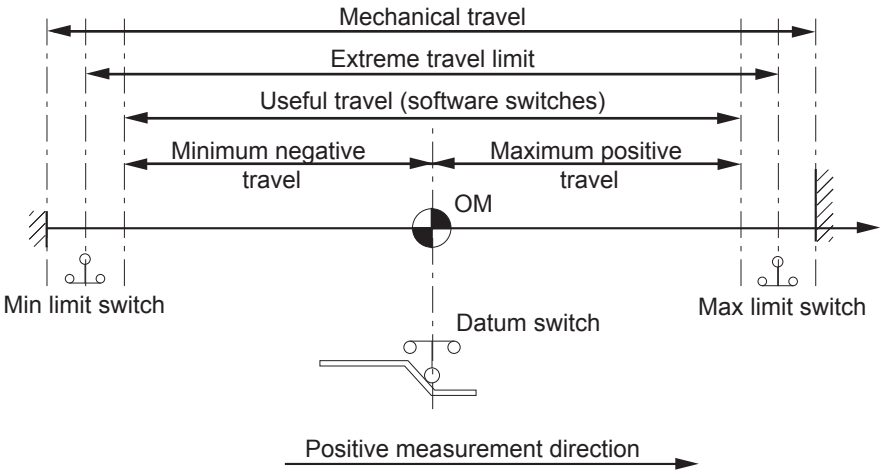
The second word of the pair gives the upper limit (maximum or positive) of MAX travel.



The values are expressed in the internal measurement unit: mm/10, mm/100, µm, µm/10 or µm/100 for linear axes and deg/10, deg/100, deg/1000 or deg/10000 for rotary axes (see P4 parameter "Axes Programmable by Diameter and Internal Measurement" in the parameter manual).

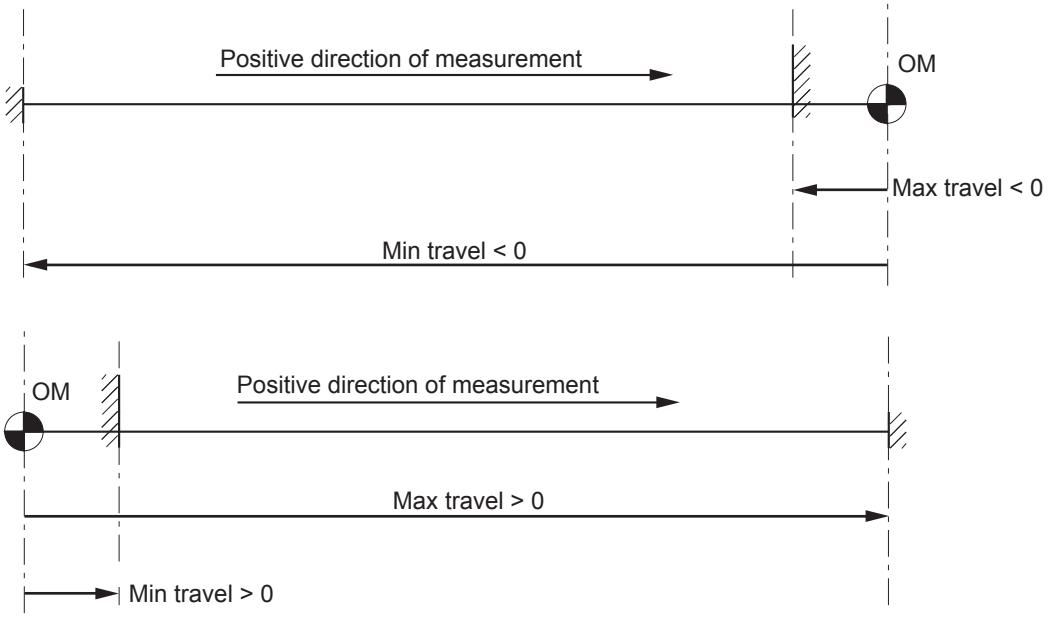
Extended NCK access (see PLC library "Extended NCK access")

The travels on an axis can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P17.



The signed MIN/MAX travel limits are as viewed from the measurement origin OM.

If the measurement origin is outside the travel limits, the minimum travel may be positive or the maximum travel may be negative.



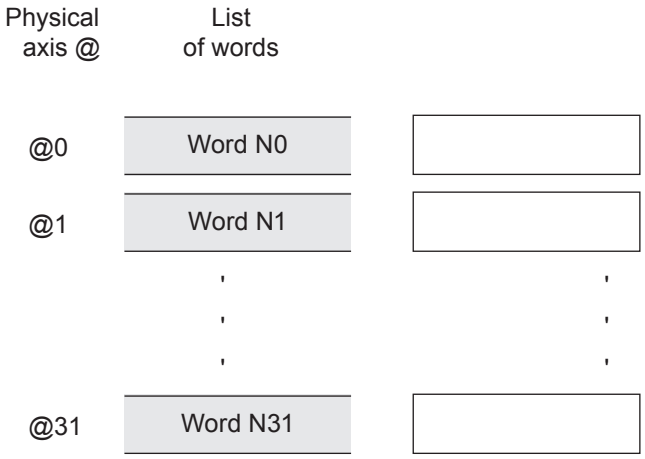
7.5 Backlash Error Compensation ► P18

Description

Used to correct positioning errors due to backlash.

Principle

The word number gives the physical axis address.



The values are expressed in the internal measurement unit: mm/10, mm/100, µm, µm/10 or µm/100 for linear axes and deg/10, deg/100, deg/1000 or deg/10000 for rotary axes (see P4 parameter "Axes Programmable by Diameter and Internal Measurement").

The absolute value contained in a word gives the maximum backlash on the corresponding axis. The sign determines the direction of backlash correction.

Value > 0: Positive correction applied when the axis moves in the positive direction.

Value < 0: Negative correction applied when the axis moves in the positive direction.

Extended NCK access (see PLC library "Extended NCK access")

The backlash error compensation can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P18.

Page deliberately empty

Page deliberately empty

Summary

8 Spindles 131
8.1 Principle of Bidirectional Spindle Indexing 131
8.1.1 Spindle Indexing 131
8.1.2 Indexed Stop Position 131
8.1.3 Indexed Stopping Condition 131
8.1.4 Spindle Speed Control by Bidirectional Servo-Loop 132
8.1.5 Spindle Indexing at Constant Acceleration 133
8.2 Rigid Tapping 134
8.2.1 Procedure 134
8.2.2 Feed Stop (Stop) 134
8.3 Spindle Declaration and assign spindle to the channels ► P6 135
8.4 Spindle Measurement Conversion ► P40 137
8.5 Spindle Reference Reversal ► P41 139
8.6 Spindle 1 Speed Ranges ► P46 140
8.7 Spindle 2 Speed Ranges ► P47 143
8.8 Spindle 3 Speed Range ► P48 146
8.9 Spindle Speed Ranges ► P49 149
8.10 Spindle Origins ► P42 152
8.11 Maximum Speed During Spindle Indexing and Maximum Speed of Spindles by PLC Programme ► P43 153
8.12 Spindle Indexing In-Position Window ► P44 155
8.13 Spindle Indexing Servo-Loop Gain ► P45 157
8.14 Spindle Acceleration and Minimum Spindle Reference in Hole Bottom ► P62 159
8.15 Cycle Axis Integration Time Constant, Cycle Axis Position Anticipation Time Constant and Spindle Zero Crossing Anticipation Time Constant ► P63 161

Page deliberately empty

8 Spindles

8.1 Principle of Bidirectional Spindle Indexing

Parameters P42 to P45 are used to set the indexing for one to four spindles.

8.1.1 Spindle Indexing

Spindle indexing is activated by miscellaneous function M19 (see "Flexium Programming Manual M or T"). It should be noted that if the spindle is stopped, it is indexed by the shortest path.

Servo-control is bidirectional on the end point.

Function M19 is transmitted to the PLC at the beginning of the indexing cycle and Spindle[n].PositionReached (spindle n), is set when the spindle is in position.
Servo-control continues as long as function M19 remains active.

Function M19 is cancelled by functions M00, M01, M02, M03, M04 or M05.

8.1.2 Indexed Stop Position

The indexed stop position can be chosen by programming using a value in deg/10000.

Address C must be declared in P9 and P1 (modulo).

If no value is programmed, the value zero is taken by default.

A value for the indexed stop position is programmed:

- with the C address for a lathe channel,
- with the EC address for a milling machine channel.

The spindle origin offset is defined by parameter P42. This parameter contains an angular offset (expressed in the internal measurement units).

It is taken into account in an indexing.

P42 = Offset (in degrees) x 4096/360

Stopping in position is achieved by a symmetric servo-loop allowing the use of reversible servo-drives.

Example of Indexed Stop

For a lathe M19C (stop on spindle zero pulse)

M19C45 (stop at 45 degrees)

For a milling machine M19EC (stop on spindle zero pulse).

8.1.3 Indexed Stopping Condition

The stop is declared complete when:

- the absolute value of the position error is less than the in-position window defined by parameter P44,
- the spindle speed is considered sufficiently low (less than 3/4096 rev/sample).

The in-position window is defined by parameter P44 (expressed in internal measurement units).

8.1.4 Spindle Speed Control by Bidirectional Servo-Loop

Control is by a reversible servo-drive only.

The current position approaches the indexed position.

The speed setting is calculated by:

Spindle reference = Position error x P45

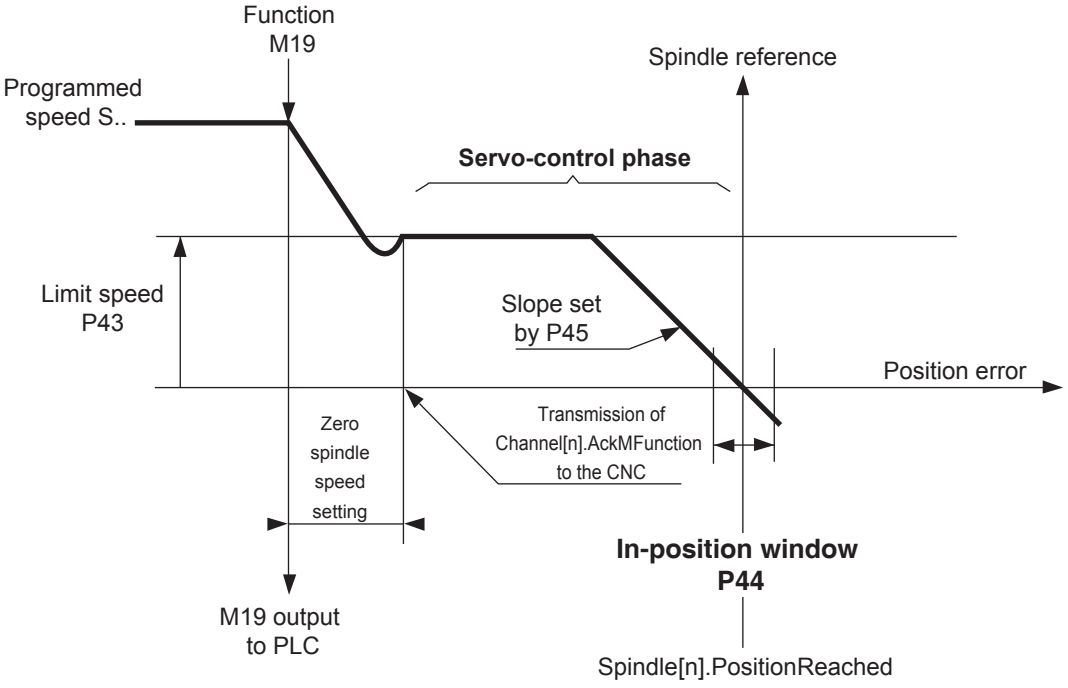
P45 is expressed in rpm/rev, i.e. 1/min.

The speed reference computed is then clipped to a limit speed defined by parameter P43 or specified by the programmed value (G92 S.. "Spindle speed limit in rpm/rev"), whichever is smaller.

The servo-control phase starts when the M19 report is set to 1, after detection of a measured spindle speed less than P43.

The current position is beyond the limit position.

The system returns the spindle to the limit position. The system is continuously servo-controlled.



8.1.5 Spindle Indexing at Constant Acceleration

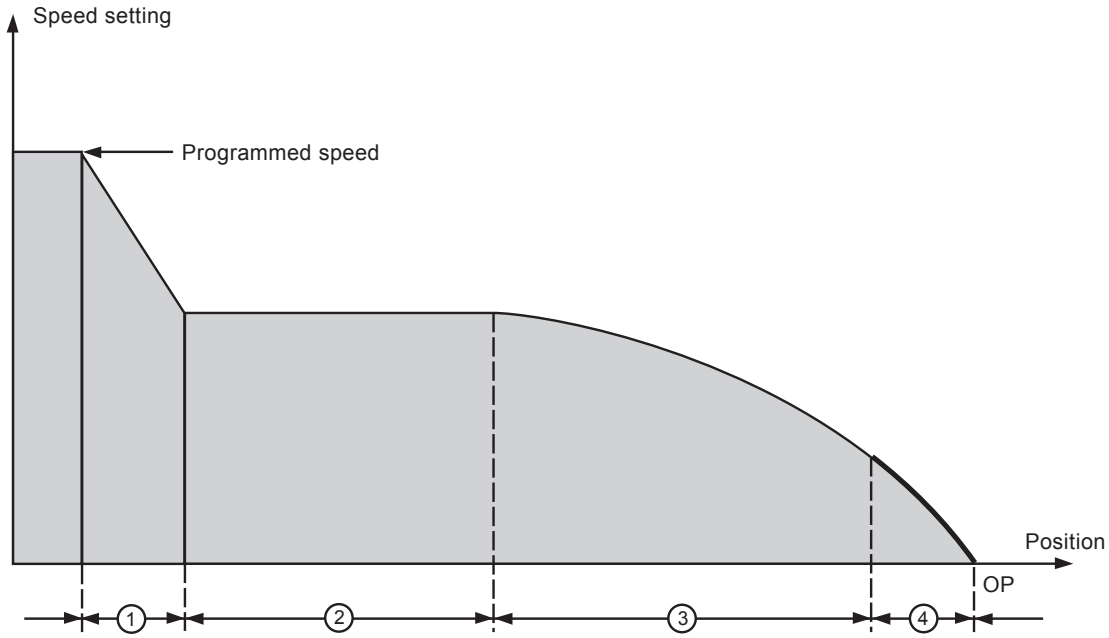
With spindle indexing, when bit 5 of words N1 to N4 of P6 is a 1, the positioning speed is at constant deceleration.

The deceleration is expressed in deg/s^2 in parameter P32 (see 6.3).

The limit speed P43 is expressed in rpm.

The in-position window P44 is expressed in internal measurement units.

The servo-loop gain P45 is expressed in rpm/rev. It is only used at the end of positioning when deceleration at constant "gamma" is requested.



1) The setting decreases along the deceleration slope (parameter P32) to the limit speed

2) The setting remains equal to the limit speed:
- until arriving in the deceleration region (if the AckMFunction = 1), or
- as long as AckMFunction = 0.

3) The setting decreases to the indexing point along a deceleration slope with constant "gamma" if the position is 180 degrees from the end point and if the AckMFunction = 1

4) The servo-control gain is constant up to the indexing point

3) and 4) are carried out on one spindle revolution

If the programmed speed is less than the limit speed, the programmed speed is used as indexing speed.

8.2 Rigid Tapping

This function allows tapping to be performed without using a floating tool-holder in order to improve the quality of machining. The rigid tapping characteristics are set in parameters P62 and P63.

Rigid tapping requires the use of a responsive spindle servo-drive and an angular spindle position measurement. All during machining, the cycle axis position is strictly servoed to the angular spindle position.

This option can be used on spindles 1 and 2 controlled by channels 1 to 5.

Rigid tapping can also be used in conjunction with the inclined plane option.

8.2.1 Procedure

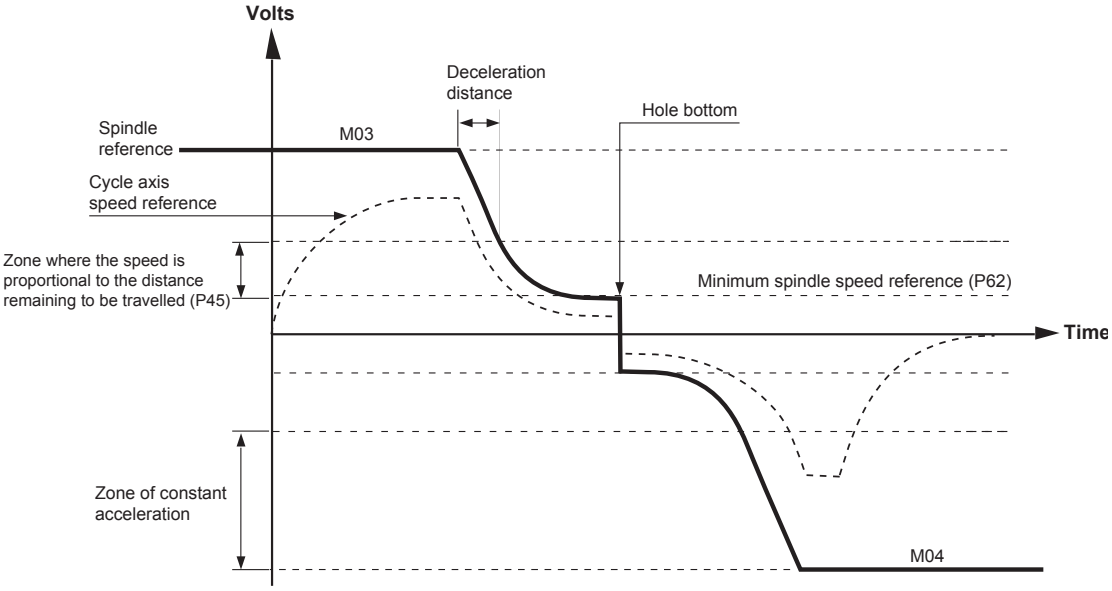
At the beginning of infeed, the cycle axis is synchronised on the spindle zero pulse.
This synchronisation is repeated each cycle.

The start of movement is anticipated with respect to the spindle zero pulse in order to obtain a gradual acceleration (parameter P63, spindle crossing anticipation time constant).

When approaching the bottom of the tapped hole, the spindle slows down according to the preset deceleration value (parameter P62, spindle deceleration).

Near the hole bottom, a minimum spindle speed is applied to ensure reaching the hole bottom (parameter P62, minimum speed reference in the hole bottom).

The spindle then reverses and accelerates up to the programmed speed. The infeed and outfeed speeds may be different. At the end of outfeed, the spindle reverses and the axis/spindle coupling is cancelled.



8.2.2 Feed Stop (Stop)

The action of Stop for rigid tapping depends on machine parameter P7 word 1 bit 4:

- Bit = 0: Stop causes detapping before arriving in the hole bottom
- Bit = 1: Stop inhibited during tapping; at the end of the cycle, the system goes into Stop state.

8.3 Spindle Declaration and assign spindle to the channels ► P6

Description

Used to declare the measured spindles and define the spindle types.

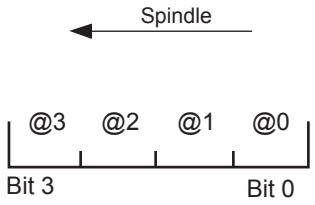
Principle

Word N0 : defines measured spindles

Word N0 is a list of 4 bits (bit 0-3) that address spindles 1 to 4.

The bit position gives the physical spindle address.

The physical addresses of the encoders are as follows:



The bit is set to 1 to indicate that the spindle is controlled by NC.

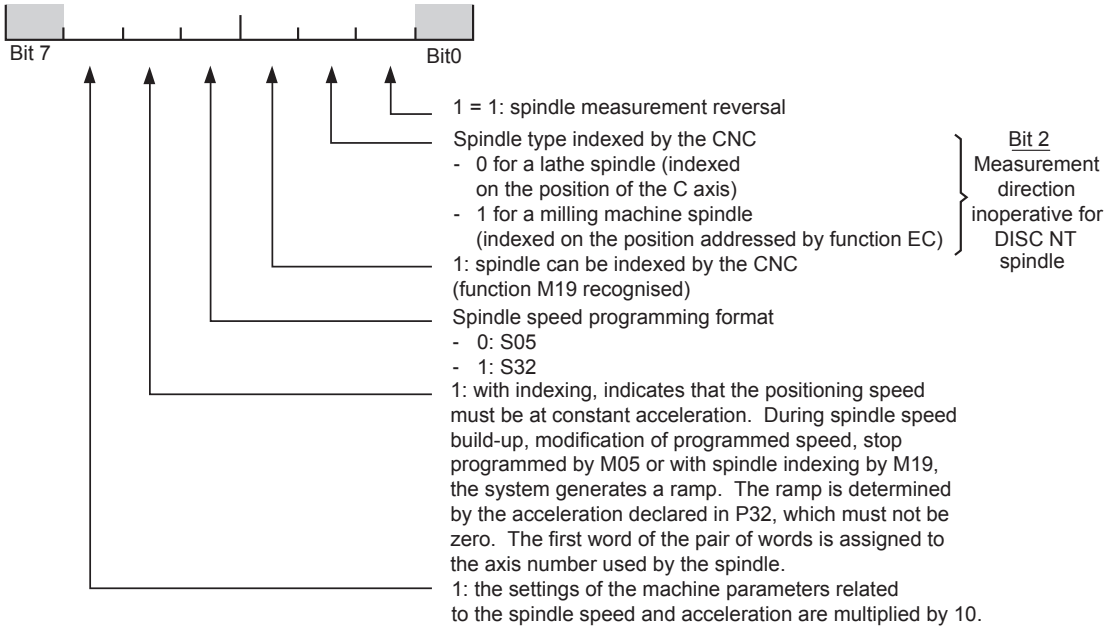
If an encoder is assigned both to a spindle (by P6) and to an axis (by P2), it is assigned by priority to the spindle.
If a spindle is declared in P6 but does not have an axis encoder, the measurement of this spindle is simulated by the CNC.

Words N1-N4 : defines spindle type.

A list of four words:

- word N1 for spindle 1,
- word N2 for spindle 2,
- word N3 for spindle 3,
- word N4 for spindle 4.

Only bits 1 to 6 are significant.



When bit 6 of these words is set, speeds from 10 to 600000 rpm can be programmed without changing the format of the parameters.

If bit 5 of these words is set to 0, this indicates constant gain indexation (see parameter P45).

The parameters concerned by this function are spindle speed ranges P46 to P49, gains P45 (in rpm/rev), maximum speed during spindle indexing P43 and spindle accelerations P32.

The maximum measured speed remains equal to 32768 measurement increments per sample.
For instance, if P50 = 5000 and P40 < 4096, Vmax = (32768/4096)*60000/5 = 96000 rpm.

Words N5 to N8: assign spindle to the channels.

- Word N5 corresponds to spindle 1
- Word N6 corresponds to spindle 2
- Word N7 corresponds to spindle 3
- Word N8 corresponds to spindle 4

The value specified in each word corresponds to the channel assigned:

- 0 for channel 1
- 1 for channel 2
- ...
- 7 for channel 8

Extended NCK access (see PLC library "Extended NCK access")

The spindle declarations can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P6.

8.4 Spindle Measurement Conversion ► P40

Description

Used to define the number of measurement points output by the spindle encoder.

Principle

This parameter includes four decimal values that correspond to the number of measurement points output by the spindle encoder for one revolution.

The values are expressed as a number of points per revolution.

Spindle 1	Word N0	
Spindle 2	Word N1	
Spindle 3	Word N2	
Spindle 4	Word N3	

The use of an encoder requires applying a multiplier of 4 on the input of the axis card.

The internal units depend on the entered values.

The system automatically determines the spindle modulo (number of measurement units per revolution). The spindle modulo is the power of two just above or equal to the number of increments per encoder revolution.

The spindle modulo cannot be less than 4096.

Extended NCK access (see PLC library "Extended NCK access")

The spindle measurement conversion can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P40.

Example

				Automatically determined Internal measurement unit
If P40 =	2000 pts/rev The spindle modulo is equal to:	2^{12} (4096)	⟶	360/4096 deg
If P40 =	5000 pts/rev The spindle modulo is equal to:	2^{13} (8192)	⟶	360/8192 deg
If P40 =	20000 pts/rev The spindle modulo is equal to:	2^{15} (32768)	⟶	360/32768 deg
If P40 =	65536 pts/rev The spindle modulo is equal to:	2^{16} (65536)	⟶	360/65536 deg

8.5 Spindle Reference Reversal ► P41

Description

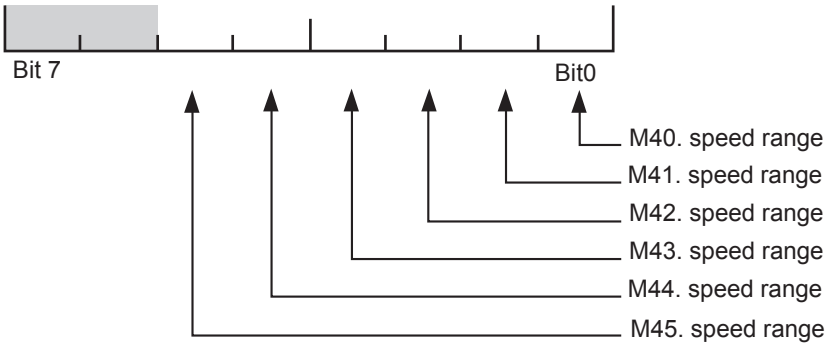
Used to define the spindle reference direction.

Spindle 1	Word N0	
Spindle 2	Word N1	
Spindle 3	Word N2	
Spindle 4	Word N3	

Principle

Each word corresponds to a spindle from 1 to 4.

Only bit 0 to 5 of each word are significant. The bits of each word are addressable by spindle speed ranges M40 to M45. The bit is set to 1 to indicate that the spindle reference is reversed.



Extended NCK access (see PLC library "Extended NCK access")

The spindle measurement conversion can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P41.

8.6 Spindle 1 Speed Ranges ► P46

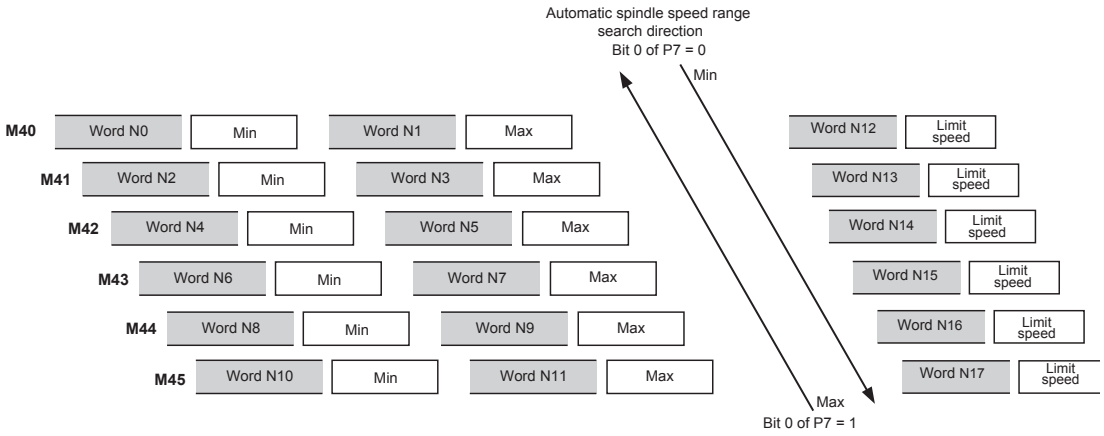
Description

Used to define the speed ranges for spindle 1 as well as the limit speed for each range.

Principle

This parameter includes six pairs of words defining the min and max speeds of ranges M40 to M45 and six words (N12-N17) giving the limit speed for each range.

The automatic range search is not enabled unless P7, word 0, bit 7 equals 0.



The values are expressed as a function of parameter P6 as shown in the table below:
Default value in rpm

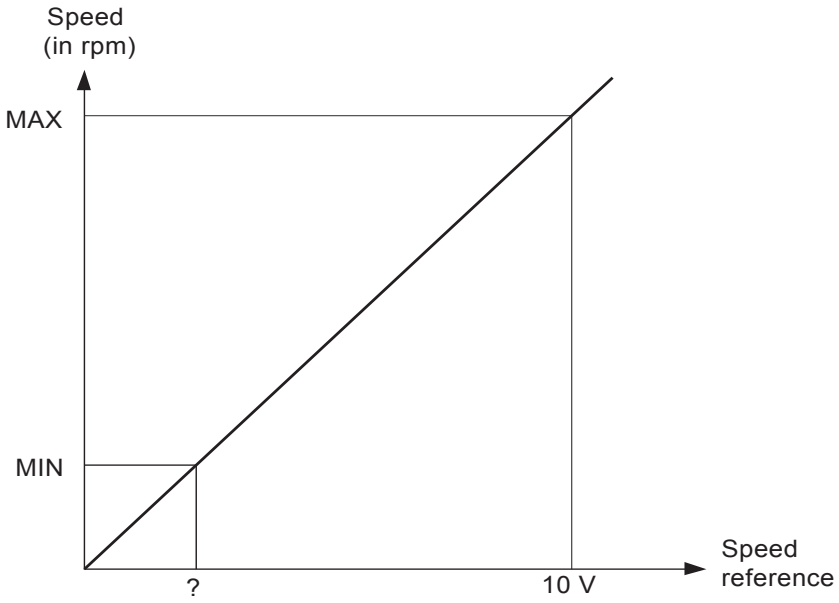
Parameter P6 Word 1 Bit 5	Programming format	Parameter P6 Word1 Bit6	X10	Speed unit	Maximum spindle rotation speed
0	S05	0	NO	rpm	65535 rpm
0	S06	1	YES	10 rpm	655350 rpm
1	S32	0	NO	rpm/100	655 rpm

The limit speed must be less than or equal to the maximum speed for the range. If the value is zero (default value), the maximum speed of the range is considered as limit speed.

If a programmed spindle speed is above the limit speed but below the maximum speed of the range and the range is programmed, it is refused with error 29. If the range is not programmed, a search for a range compatible with the speed is made (provided automatic range search is enabled: P7, word 0, bit 7 = 0).

If a speed range does not exist on a machine, the pair of values defining this range must be initialised with zero.

The maximum speed in a range from M40 to M45 must correspond to a servo-drive speed reference voltage of 10 V.

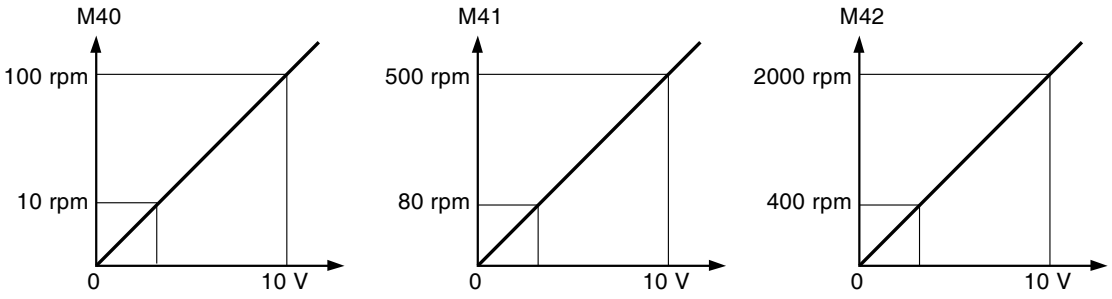


Extended NCK access (see PLC library "Extended NCK access")

The speed ranges of spindle 1 can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P46.

Example

	Min	Max
M40	10	100
M41	80	500
M42	400	2000



8.7 Spindle 2 Speed Ranges ► P47

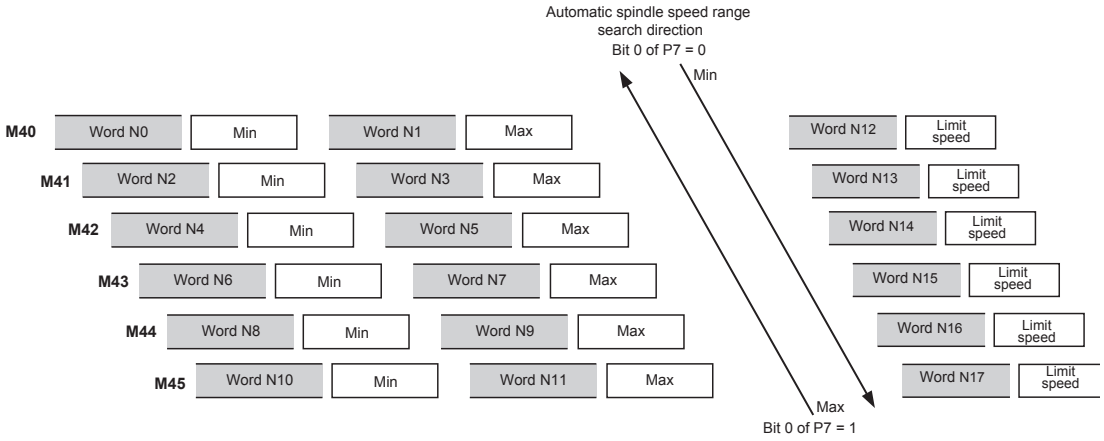
Description

Used to define the speed ranges for spindle 2 as well as the limit speed for each range.

Principle

This parameter includes six pairs of words defining the min and max speeds of ranges M40 to M45 and six words (N12-N17) giving the limit speed for each range.

The automatic range search is not enabled unless P7, word 0, bit 7 equals 0.



The values are expressed as a function of parameter P6 as shown in the table below:
Default value in rpm

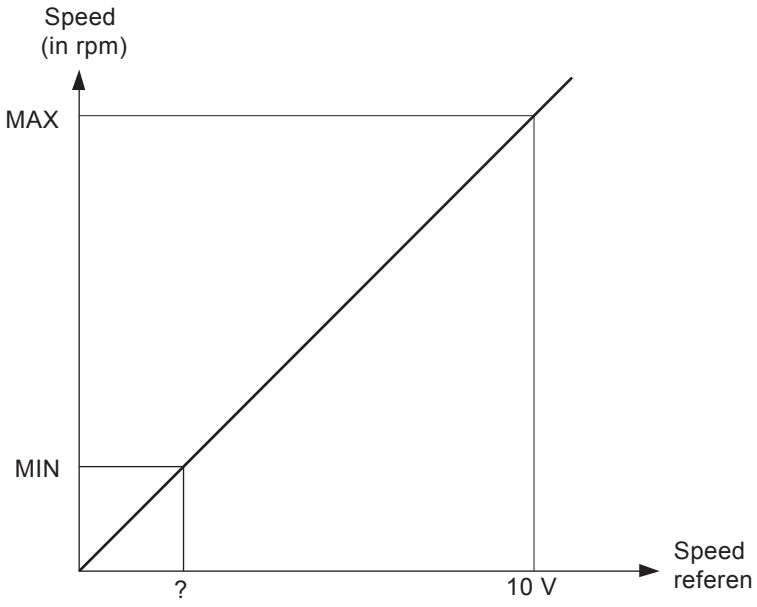
Parameter P6 Word 1 Bit 5	Programming format	Parameter P6 Word1 Bit6	X10	Speed unit	Maximum spindle rotation speed
0	S05	0	NO	rpm	65535 rpm
0	S06	1	YES	10 rpm	655350 rpm
1	S32	0	NO	rpm/100	655 rpm

The limit speed must be less than or equal to the maximum speed for the range. If the value is zero (default value), the maximum speed of the range is considered as limit speed.

If a programmed spindle speed is above the limit speed but below the maximum speed of the range and the range is programmed, it is refused with error 29. If the range is not programmed, a search for a range compatible with the speed is made (provided automatic range search is enabled: P7, word 0, bit 7 = 0).

If a speed range does not exist on a machine, the pair of values defining this range must be initialised with zero.

The maximum speed in a range from M40 to M45 must correspond to a servo-drive speed reference voltage of 10 V.

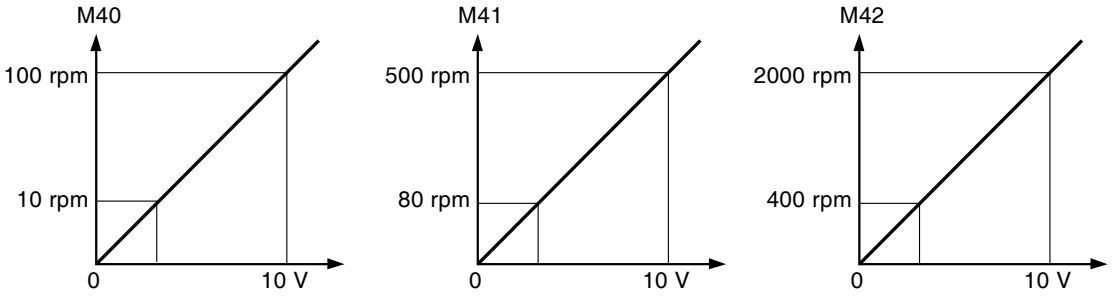


Extended NCK access (see PLC library "Extended NCK access")

The speed ranges of spindle 2 can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P47.

Example

	Min	Max
M40	10	100
M41	80	50
M42	400	2000



8.8 Spindle 3 Speed Range ► P48

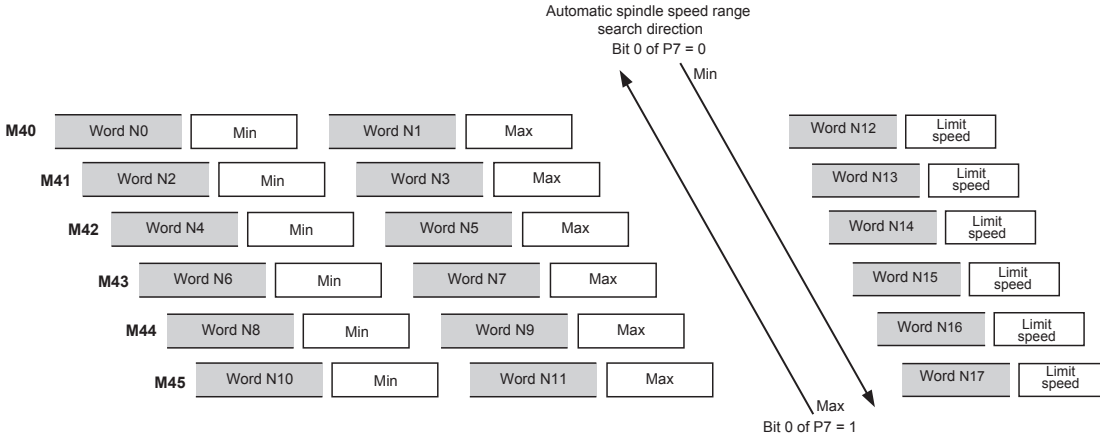
Description

Used to define the speed ranges for spindle 3 as well as the limit speed for each range.

Principle

This parameter includes six pairs of words defining the min and max speeds of ranges M40 to M45 and six words (N12-N17) giving the limit speed for each range.

The automatic range search is not enabled unless P7, word 0, bit 7 equals 0.



The values are expressed as a function of parameter P6 as shown in the table below:
Default value in rpm

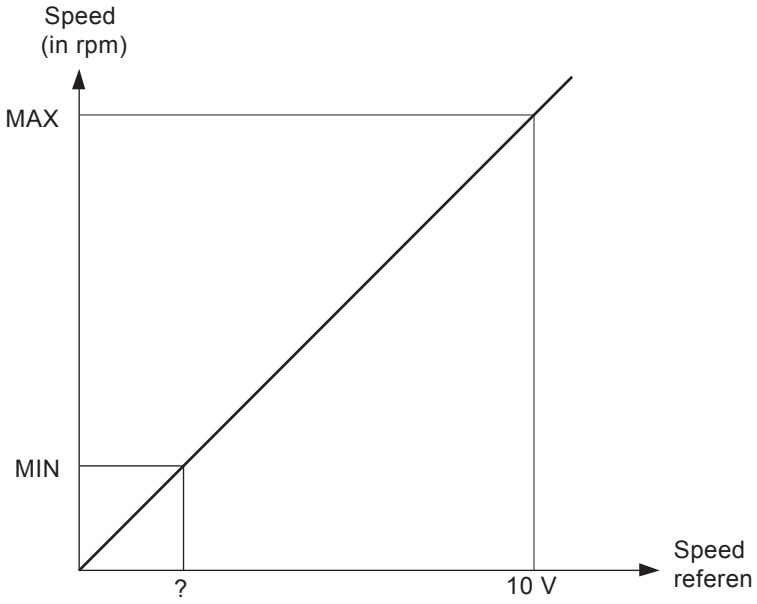
Parameter P6 Word 1 Bit 5	Programming format	Parameter P6 Word1 Bit6	X10	Speed unit	Maximum spindle rotation speed
0	S05	0	NO	rpm	65535 rpm
0	S06	1	YES	10 rpm	655350 rpm
1	S32	0	NO	rpm/100	655 rpm

The limit speed must be less than or equal to the maximum speed for the range. If the value is zero (default value), the maximum speed of the range is considered as limit speed.

If a programmed spindle speed is above the limit speed but below the maximum speed of the range and the range is programmed, it is refused with error 29. If the range is not programmed, a search for a range compatible with the speed is made (provided automatic range search is enabled: P7, word 0, bit 7 = 0).

If a speed range does not exist on a machine, the pair of values defining this range must be initialised with zero.

The maximum speed in a range from M40 to M45 must correspond to a servo-drive speed reference voltage of 10 V.

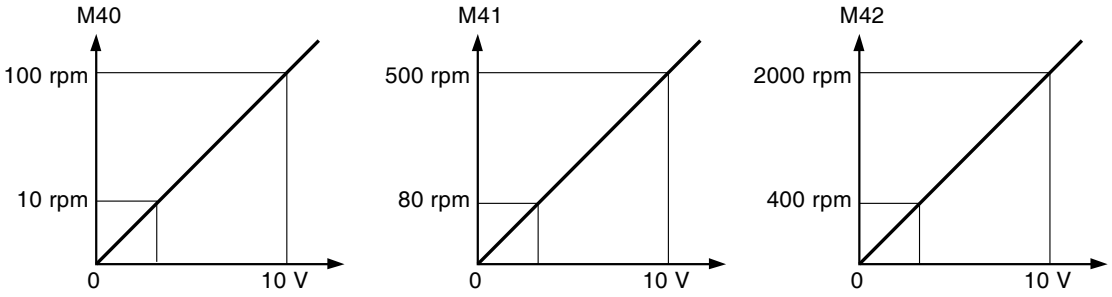


Extended NCK access (see PLC library "Extended NCK access")

The speed ranges of spindle 3 can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P48.

Example

	Min	Max
M40	10	100
M41	80	00
M42	400	2000



8.9 Spindle Speed Ranges ► P49

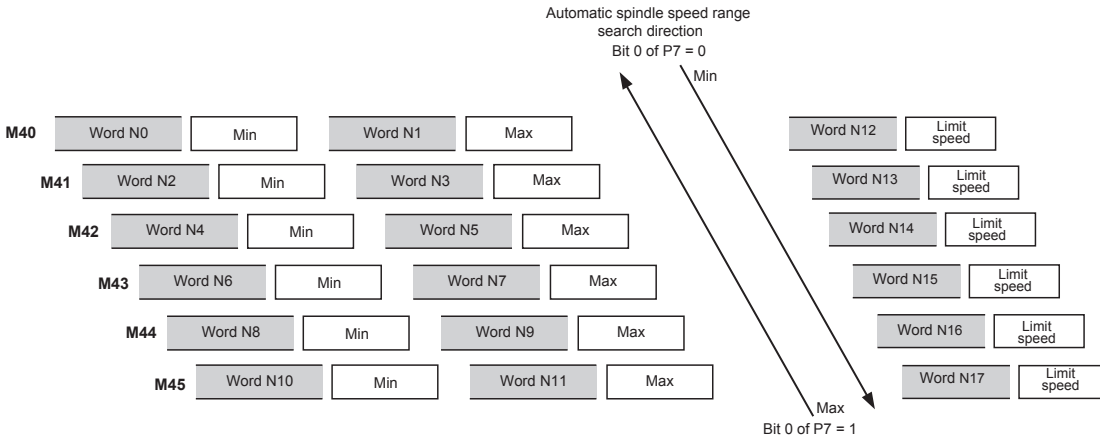
Description

Used to define the speed ranges for spindle 4 as well as the limit speed for each range.

Principle

This parameter includes six pairs of words defining the min and max speeds of ranges M40 to M45 and six words (N12-N17) giving the limit speed for each range.

The automatic range search is not enabled unless P7, word 0, bit 7 equals 0.



The values are expressed as a function of parameter P6 as shown in the table below:
Default value in rpm

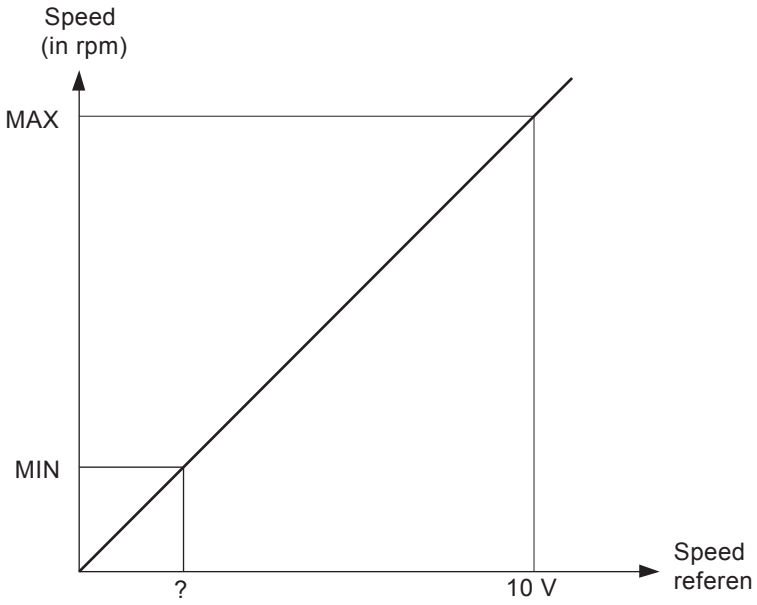
Parameter P6 Word 1 Bit 5	Programming format	Parameter P6 Word1 Bit6	X10	Speed unit	Maximum spindle rotation speed
0	S05	0	NO	rpm	65535 rpm
0	S06	1	YES	10 rpm	655350 rpm
1	S32	0	NO	rpm/100	655 rpm

The limit speed must be less than or equal to the maximum speed for the range. If the value is zero (default value), the maximum speed of the range is considered as limit speed.

If a programmed spindle speed is above the limit speed but below the maximum speed of the range and the range is programmed, it is refused with error 29. If the range is not programmed, a search for a range compatible with the speed is made (provided automatic range search is enabled: P7, word 0, bit 7 = 0).

If a speed range does not exist on a machine, the pair of values defining this range must be initialised with zero.

The maximum speed in a range from M40 to M45 must correspond to a servo-drive speed reference voltage of 10 V.

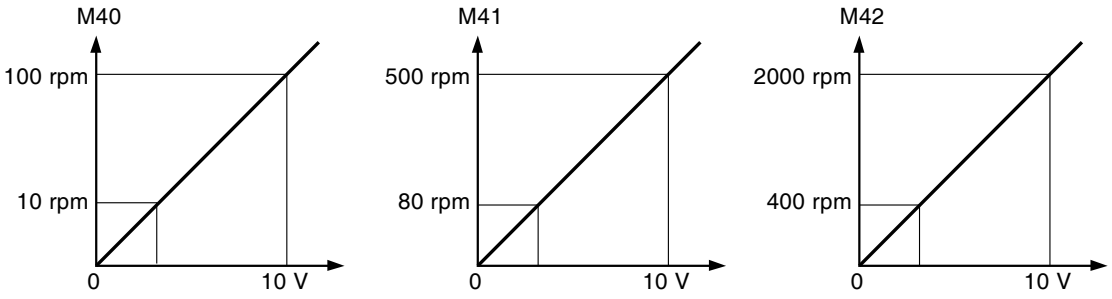


Extended NCK access (see PLC library "Extended NCK access")

The speed ranges of spindle 4 can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P49.

Example

	Min	Max
M40	10	100
M41	80	500
M42	400	2000



8.10 Spindle Origins ► P42

Description

Used to define the spindle origin positions.

Principle

This parameter includes four decimal values defining the origin positions of spindles 1 to 4.

Spindle 1	Word N0	
Spindle 2	Word N1	
Spindle 3	Word N2	
Spindle 4	Word N3	

The values are expressed in internal measurement units (see 8.5).

The spindle origin setting is effective on the first encoder marker pulse encountered during the first revolution:

- if P42 = 0 —> Spindle measurement = 0 on first marker pulse,
- if P42 = x —> Spindle measurement = x on first marker pulse.

Extended NCK access (see PLC library "Extended NCK access")

The spindle origins can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P42.

8.11 Maximum Speed During Spindle Indexing and Maximum Speed of Spindles by PLC Programme ► P43

Description

Used to define the maximum spindle speed during indexing.

Used to define the maximum spindle speed by PLC programme.

Principle

Word N0 to N3 : defines the maximum spindle speed generated by the indexing function for spindles 1 to 4.

Spindle 1	Word N0	
Spindle 2	Word N1	
Spindle 3	Word N2	
Spindle 4	Word N3	

The values are expressed as a function of parameter P6 as shown in the table below:
Default value in rpm

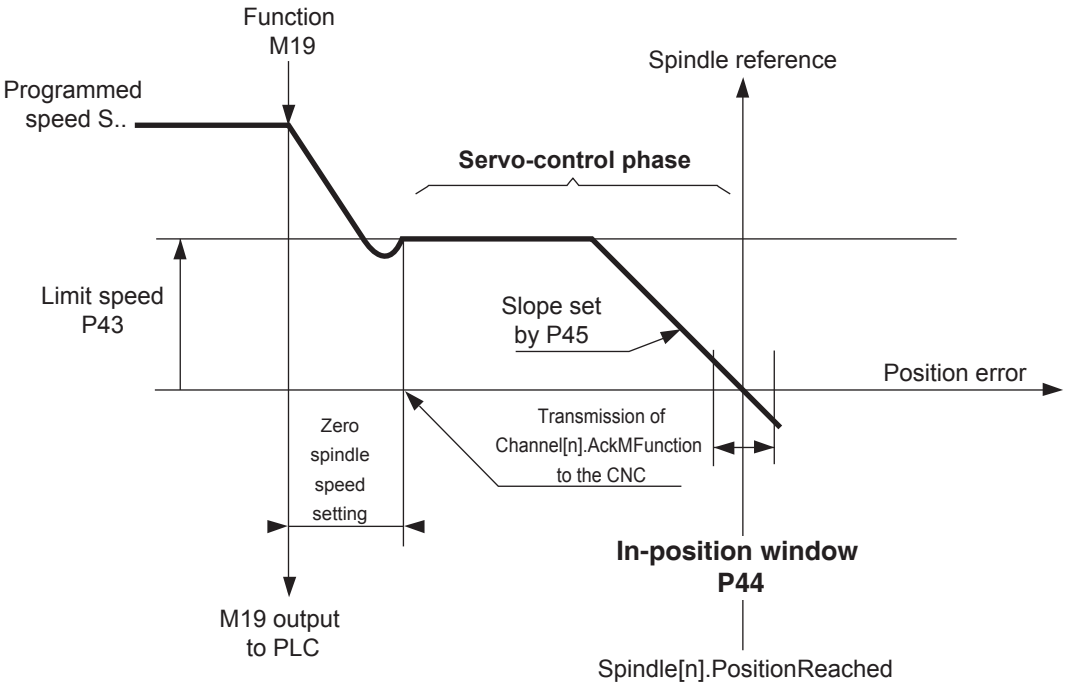
Parameter P6 Word 1 Bit 5	Programming format	Parameter P6 Word1 Bit6	X10	Speed unit	Maximum spindle rotation speed
0	S05	0	NO	rpm	65535 rpm
0	S06	1	YES	10 rpm	655350 rpm
1	S32	0	NO	rpm/100	655 rpm

Words N4 to N7: define the maximum speed of spindles 1 to 4

For safety reasons, the PLC can enable a maximum spindle speed via words Spindle[s].ReducedSpeed.

Spindle 1	Word N0		V_Reduit_Br1
Spindle 2	Word N1		V_Reduit_Br2
Spindle 3	Word N2		V_Reduit_Br3
Spindle 4	Word N3		V_Reduit_Br4

A value other than 0 set in a word specifies a reduced spindle speed. This value is in rpm.



Programme Parameters (see "Flexium Programming Manual M or T):

Programme parameter E9030b (b = 0 to 3) is used to read and write the maximum speed. A reset (CNC reset) sets this parameter to the state of P43.

Extended NCK access (see PLC library "Extended NCK access")

The maximum speed during spindle indexing can be modified by a Extended NCK request. A reset (CNC reset) resets the CNC to the state specified by P43.

8.12 Spindle Indexing In-Position Window ► P44

Description

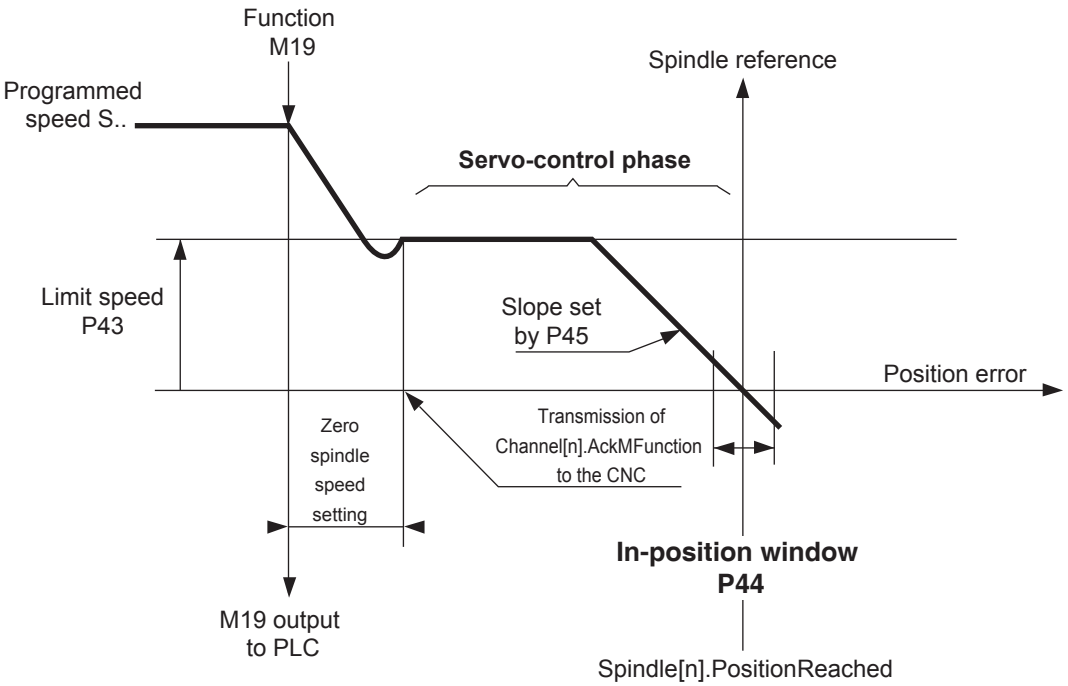
Used to define the spindle servo-loop accuracy.

Principle

Defines the spindle servo-loop accuracy for spindles 1 to 4.

Spindle 1	Word N0	
Spindle 2	Word N1	
Spindle 3	Word N2	
Spindle 4	Word N3	

The values are expressed in the internal measurement units.



Programme Parameters (see "Flexium Programming Manual M or T):

Programme parameter E9031b (b = 0 to 3) is used to read and write the in-position window. A reset (CNC reset) sets this parameter to the state of P44.

Extended NCK access (see PLC library "Extended NCK access")

The spindle indexing in-position window can be modified by a Extended NCK request. A reset (CNC reset) resets the CNC to the state specified by P44.

8.13 Spindle Indexing Servo-Loop Gain ► P45

Description

Used to define the spindle servo-loop ramp.

Principle

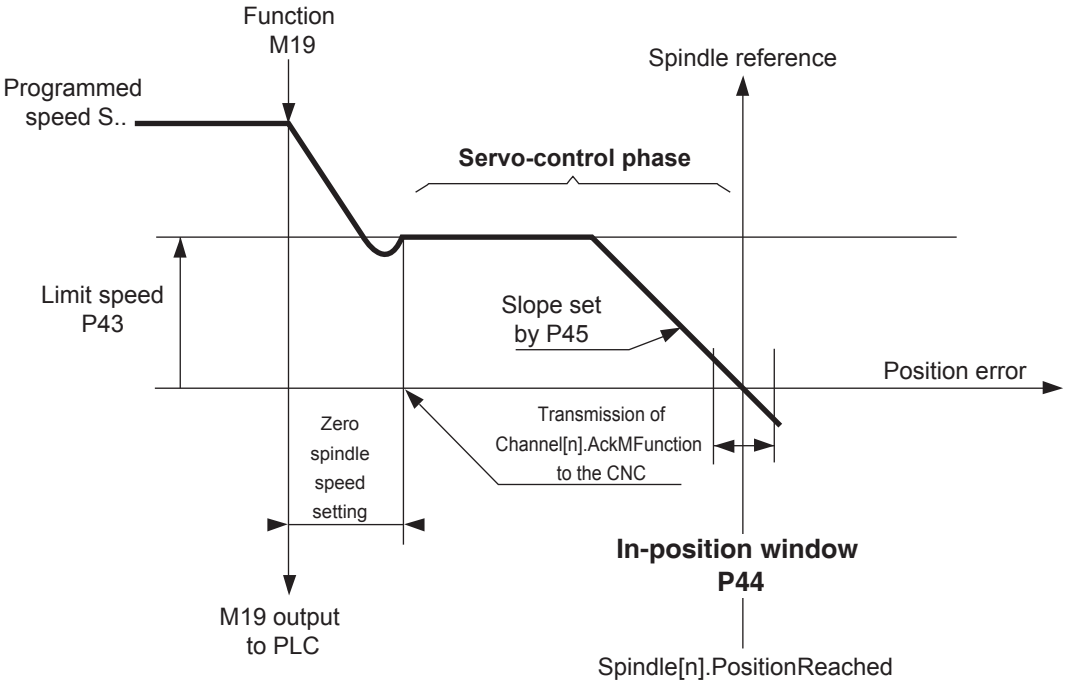
Defines the value of the servo-loop ramp for spindles 1 to 4.

Spindle 1	Word N0	
Spindle 2	Word N1	
Spindle 3	Word N2	
Spindle 4	Word N3	

The values are expressed as a function of parameter P6 as shown in the table below:

The values are expressed in rpm/rev.

Parameter P6 Word 1 Bit 5	Programming format	Parameter P6 Word1 Bit6	X10	Speed unit	Maximum spindle rotation speed
0	S05	0	NO	rpm/rev	65535 rpm/rev
0	S06	1	YES	10 rpm/rev	655350 rpm/rev
1	S32	0	NO	rpm/100/rev	655 rpm/rev



Programme Parameters (see "Flexium Programming Manual M or T):

Programme parameter E9032b (b = 0 to 3) is used to read and write the spindle gain.
A reset (CNC reset) sets this parameter to the state of P45.

Extended NCK access (see PLC library "Extended NCK access")

The spindle indexing servo-loop gain can be modified by a Extended NCK request.
A reset (CNC reset) resets the CNC to the state specified by P45.

8.14 Spindle Acceleration and Minimum Spindle Reference in Hole Bottom ► P62

Concerns only rigid tapping.

Description

Used to define the spindle acceleration and minimum spindle speed reference in the hole bottom for the spindles.

Spindle 1	Word N0	
Spindle 1	Word N1	
Spindle 2	Word N2	
Spindle 2	Word N3	
Spindle 3	Word N4	
Spindle 3	Word N5	
Spindle 4	Word N6	
Spindle 4	Word N7	

Word N0, Word N2, Word N4, Word N6 : the value A set in these four words defines the deceleration distance at the hole bottom for spindles 1, 2, 3 and 4 respectively.

A is in revolutions per s^2.

The deceleration distance (D) is given by the equation:

$$D = Pitch \times S^2/2 \times A \times 60^2$$

where:

- Pitch = tap pitch
- S = spindle speed

Word N1, Word N3, Word N5, Word N7 : defines the minimum speed reference for spindles 1, 2, 3 and 4 respectively.

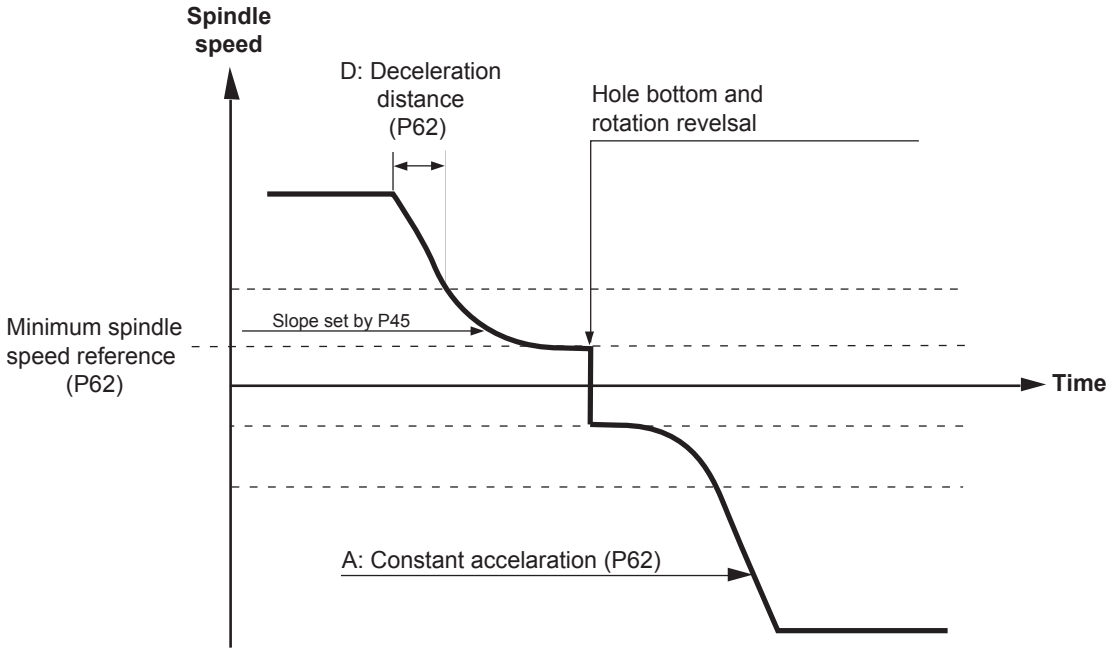
When the cycle axis enters the deceleration zone, there is first a constant deceleration, then the spindle speed reference becomes a function of the distance remaining to be travelled to reach the hole bottom.

The spindle speed reference would normally become zero on reaching the hole bottom, which would have the effect of immobilising it there. Therefore, a minimum speed reference is substituted as soon as the spindle speed reaches a certain level.

Word N1 defines the minimum speed reference for spindle 1, word N3 for spindle 2, word N5 for spindle 3 and word N7 for spindle 4.

The value is expressed in millivolts. The recommended value is from 50 to 100 mV.

Refer to 8.15 (parameter P63) for the method used to set this parameter.



8.15 Cycle Axis Integration Time Constant, Cycle Axis Position Anticipation Time Constant and Spindle Zero Crossing Anticipation Time Constant ► P63

Concerns only rigid tapping.

Description

This parameter includes eight channels of three words, i.e. one per channel.

Channels		List of wors	
Channel 1	Acceleration Work rate	Word N0	
	Acceleration Fast speed	Word N1	
	Acceleration Fast speed	Word N2	
'		'	
'		'	
Channel 8	Acceleration Work rate	Word N21	
	Acceleration Fast speed	Word N22	
	Acceleration Fast speed	Word N23	

Word N0 : defines the integration time constant for the cycle axis. It completely cancels the cycle axis position error with respect to the spindle.

Integration time constant = 1000/Ti

Ti is the number of samples and Ti > = 2.

Word N1: defines the cycle axis position anticipation time constant. It is used to cancel the axis servo-control error rapidly and approximately.

Its value is 10^3 x Σp/V. Σp is in micrometres and V is in micrometres per smp, i.e. 10^3 x T/smp where T = servo-loop time constant and smp = sample period.

In practice, the equation for the time constant is:

Cycle axis position anticipation time constant = 10^6 x Value P56/Value P50

Word N2 : defines the spindle zero crossing anticipation time constant. It is used to define the advance of the axis with respect to the spindle zero crossing in order to gradually accelerate the axis.

The value is expressed as a number of samples. It is obtained by the equation:

Spindle zero crossing anticipation time constant = (Max speed/γ) x 10^6/Value P50

where:

- Max speed = maximum authorised tapping speed in mm/s
- γ = acceleration authorised on the axis in mm/s².

The faster the accelerations allowed on the cycle axis, the lower the value programmed in this constant.

The values entered are constant for all the axes of a channel.
So, if multiple axes are required to perform rigid tapping, it is also necessary to set the same gain for these axes (KVAR set in P21).

Adjusting and checking the values set in parameters P62 and P63

To make the checks, write a part programme including a rigid tapping cycle.

The spindle servo-drive acceleration ramp affects operation of the cycle.
The steepest ramp possible should be chosen.

Below, the adjustments are made for one channel (parameter P63, words 0, 1 and 2) and spindle 1 (parameter P62, words 0 and 1).

Setting P63

- Set word 0 to 0,
- Enter the theoretical values calculated by the above equations in words 1 and 2.

Setting P62

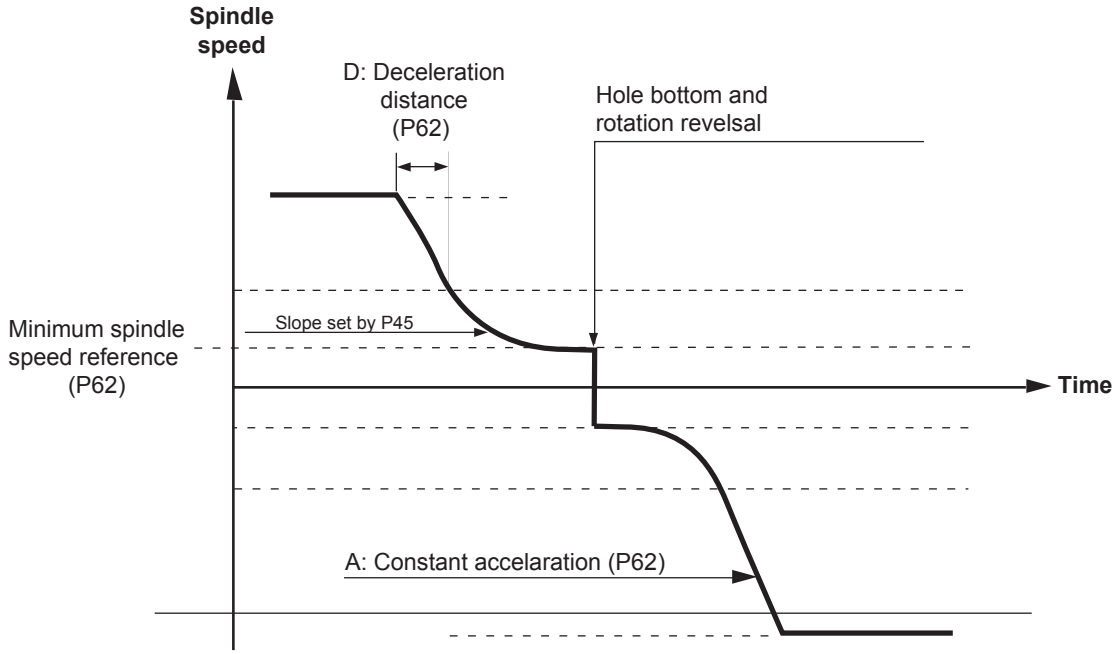
- Word 0: keep the default value of 10,
- Word 1: set the value supplied by the servo-drive manufacturer or keep the default value of 100.

Checks

Use an oscilloscope to check the spindle speed on the cycle axis (see drawing).

- Concerning the deceleration distance, if A is too high, the deceleration is too long and the spindle goes beyond the hole bottom dimension.
- Check the spindle acceleration.
- Check the acceleration induced on the axis when the spindle rotation reverses.
- Check that the following error is cancelled when the speed has settled and during reversal.

STOP The deceleration cannot be greater than allowed by the servo-drive ramp.



Improving the settings

Search for the optimum value of word 0 in P63 by gradually increasing the setting without exceeding 1000 until an instability appears. Enter the maximum value obtained divided by 2. The result should be less than 500.

Check that the following error is small on reversal. If not, decrease the value of word 0 of P62.

Page deliberately empty

Summary

9 Miscellaneous Functions 167

9.1 Miscellaneous Function Parameter Setting ► P7 167
9.2 Subroutine Call by M Function ► P35 170
9.3 Sampling Period ► P50 171
9.4 Minimum Block Execution Time ► P51 172
9.5 Number of NC blocks used per channel for look ahead analysis ► P54 173
9.6 Interaxis Calibration Table and Programme Stack Reservation ► P58 174
9.7 Potentiometer Min-Max Range ► P67 175
9.8 Memory Area Allocation ► P95 176
9.9 Communication period – Address of master CNC – Timeout for the monitoring of PLC application counter ► P99 177
9.10 Backup in Path, Auto Recall after INTER ► P114 178

Page deliberately empty

9 Miscellaneous Functions

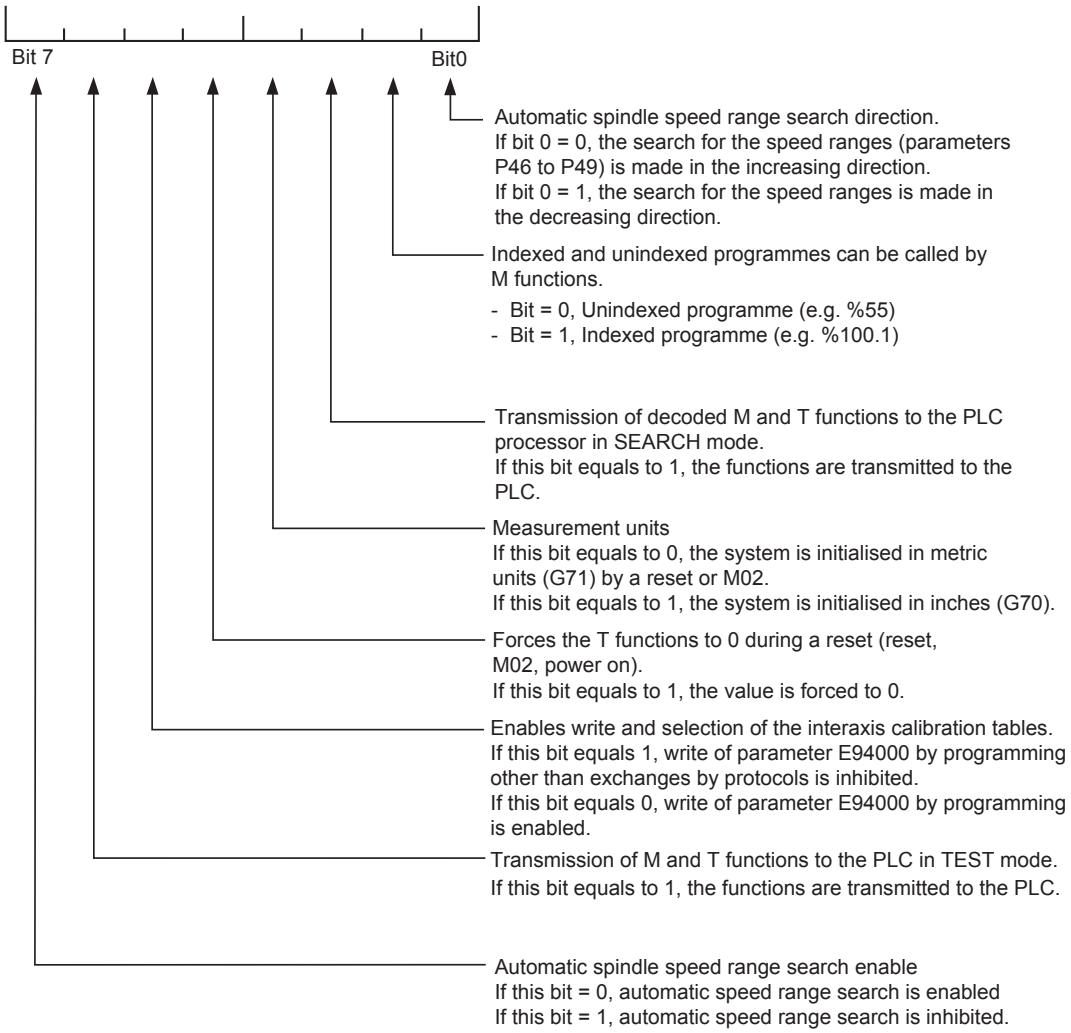
9.1 Miscellaneous Function Parameter Setting ► P7

Description

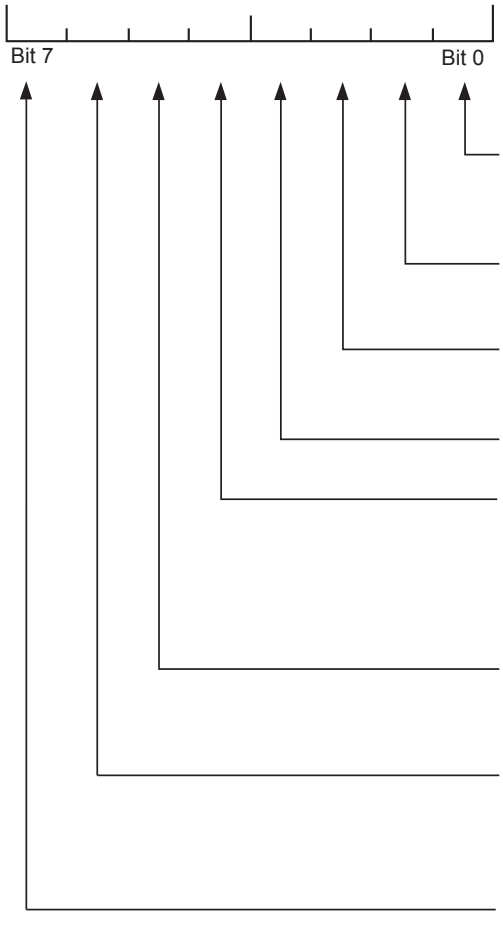
Definition of miscellaneous functions.

Principle

Word N0



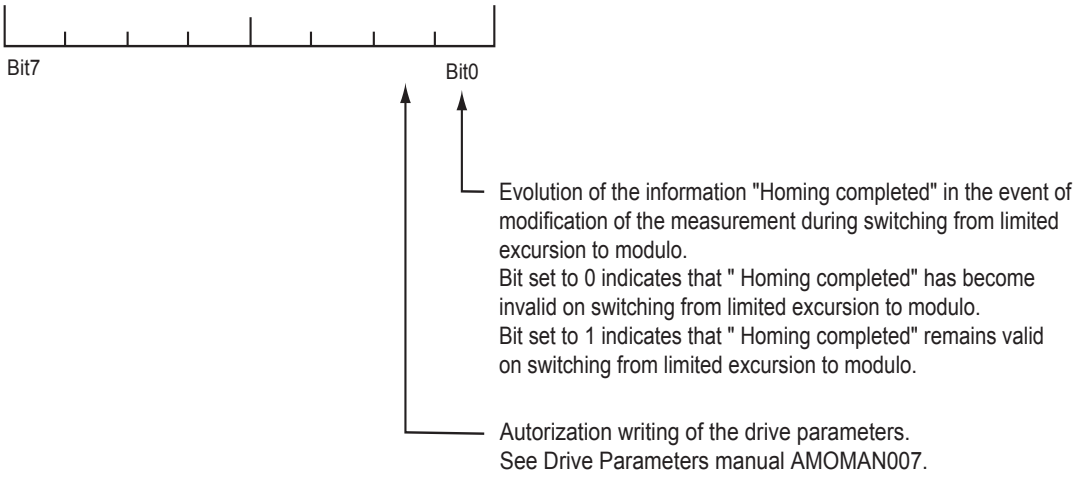
Word N1


Bit 7
Bit 0
Choice of feed rate programming unit.
If this bit equals to 1, the system is initialised in mm/rev (G95).
If this bit equals to 0, the system is initialised in mm/min (G94).
Forcing of MANUAL mode.
If this bit is set to 1, miscellaneous function M12
forcing the MANUAL mode is processed.
Homing selection
If this bit is set to 0, homing from the keyboard is enabled.
If this bit is set to 1, homing from the keyboard is inhibited.
Subroutine call on a reset
If this bit is set to 1, subroutine call on a reset is enabled.
Feed stop for rigid tapping
If this bit is set to 0, detapping is commanded before
reaching the hole bottom if a feed stop should occur.
If this bit is set to 1, feed stop is inhibited during the
tapping phase; where applicable, the system is
in Feed Stop state at the end of the cycle.
S ramp Speed variation
If this bit is set to 0, linear speed variation is enabled.
If this bit is set to 1, gradual acceleration is enabled.
G52 prohibited when an inclined plane is valid.
If this bit is set to 0, programming an inclined plane axis
in G52 returns error 24.
If this bit is set to 1, programming an inclined plane axis
in G52 is enabled.
This bit enables or inhibits detection of error 149
By default, the bit = 0 and detection is enabled.

Programme Parameters (see "Flexium Programming Manual M or T):

Programme parameter E11013 enables S ramp speed variation.
At power on, E11013 is set to the value of bit 5 of word N1. However, this programme parameter is not changed by a reset.

Word N2



Programme Parameters (see "Flexium Programming Manual M or T):

Switching from limited excursion to modulo 360° returns current measurement to modulo 360° if necessary.

The measurement is therefore modified if the axis lies outside the range [0°, 360°].

If the measurement is modified the value of parameter P7N2 bit 0 determines the status of the information "Reading of Origin completed" (E911xx).

If the measurement was not modified, the status of the information "Reading of Origin completed" remains unchanged.

9.2 Subroutine Call by M Function ► P35

Description

Used to assign a subroutine number to a miscellaneous M function.

Principle

Sixteen miscellaneous M functions can call 16 different subroutines.

The words of this parameter are associated in pairs from word 0 to word 31. The first word in the pair gives the M function number and the second gives the subroutine number.

Set unused words to zero.
Miscellaneous function M0 cannot be used to call a subroutine.

Example

Word N0	6
Word N1	9206
Word N2	85
Word N3	9285

Function M6 calls subroutine %9206.

Function M85 calls subroutine %9285.

9.3 Sampling Period ► P50

Description

Defines the CNC sampling period.

Word N0 : defines the CNC sampling period (T) in µs.

The value must be a multiple of 200 µs.

The minimum sampling period accepted by the system is 2000 µs and the maximum is period is 20.000 µs. If the value of P50 is outside these limits, the system forces the limit values.

Programme Parameters (see "Flexium Programming Manual M or T).

Programme parameter E41005 is used to read the value of the sampling period.

9.4 Minimum Block Execution Time ► P51

Description

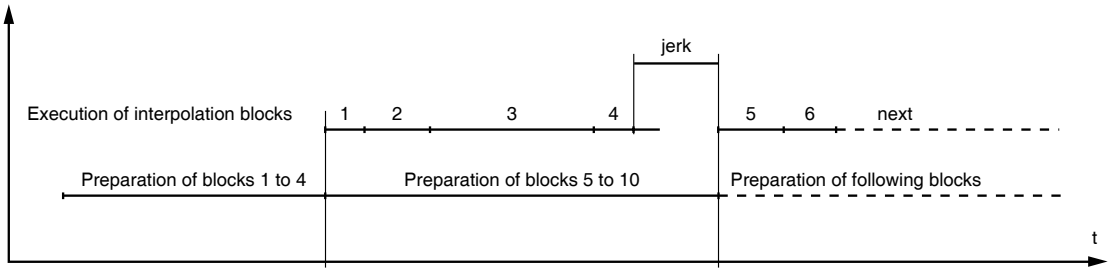
Word N0 : defines the minimum time for executing a block by interpolation.

The block execution time is displayed in milliseconds. The default value is 0 ms.

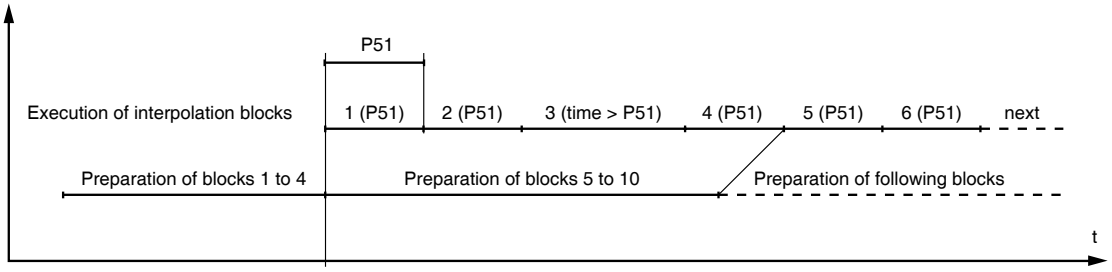
Principle

Before execution, the system prepares a variable number of blocks.

During execution of short interpolation blocks, the movements may be completed before the system has time to prepare the following blocks (see diagram), causing jerky movements.



The jerks can be avoided by setting a minimum interpolation block execution time in parameter P51 to allow sufficient time for preparing the following blocks (see diagram).



The larger the number of channels and axes, the longer the time required to prepare the blocks. The minimum block execution time must be set according to the system configuration.

9.5 Number of NC blocks used per channel for look ahead analysis ► P54

Description

Defines the number of NC blocks in the look-ahead buffer per channel.

The minimum number of blocks per channel is 10 and the maximum is 1000.
If the value of P54 is outside these limits, the system forces the limit value.

The value 0 tells the system to automatically determine the most appropriate value.

9.6 Interaxis Calibration Table and Programme Stack Reservation ► P58

Description

Used to allocate the interaxis calibration table and specify the programme stack size.

Principle

Word N0 : used to allocate the interaxis calibration table or internal parameters.

Defines the number of parameters from E81000 to E81999 "Master axis reference positions" and E82000 to E82999 "Slave axis corrections".

The parameters are reserved in ascending order from E81000 to E81xxx and E82000 to E82xxx.

The value entered must be less than or equal to 1000.

If a value higher than 1000 is entered, the system automatically limits the number of parameters to 1000.

Word N1 : used to reserve the programme stack size for the channels.

The value entered corresponds to the size in words (2 bytes) per channel. The minimum stack size must be above 10 kbytes.

The total programme stack size is displayed on the "Programming context".

Example

Word 0	200

Parameters E81000 to E81199 and E82000 to E82199 can e used.

Word 1	20480

The programme stack has a size of 20 kbytes per channel.

9.7 Potentiometer Min-Max Range ► P67

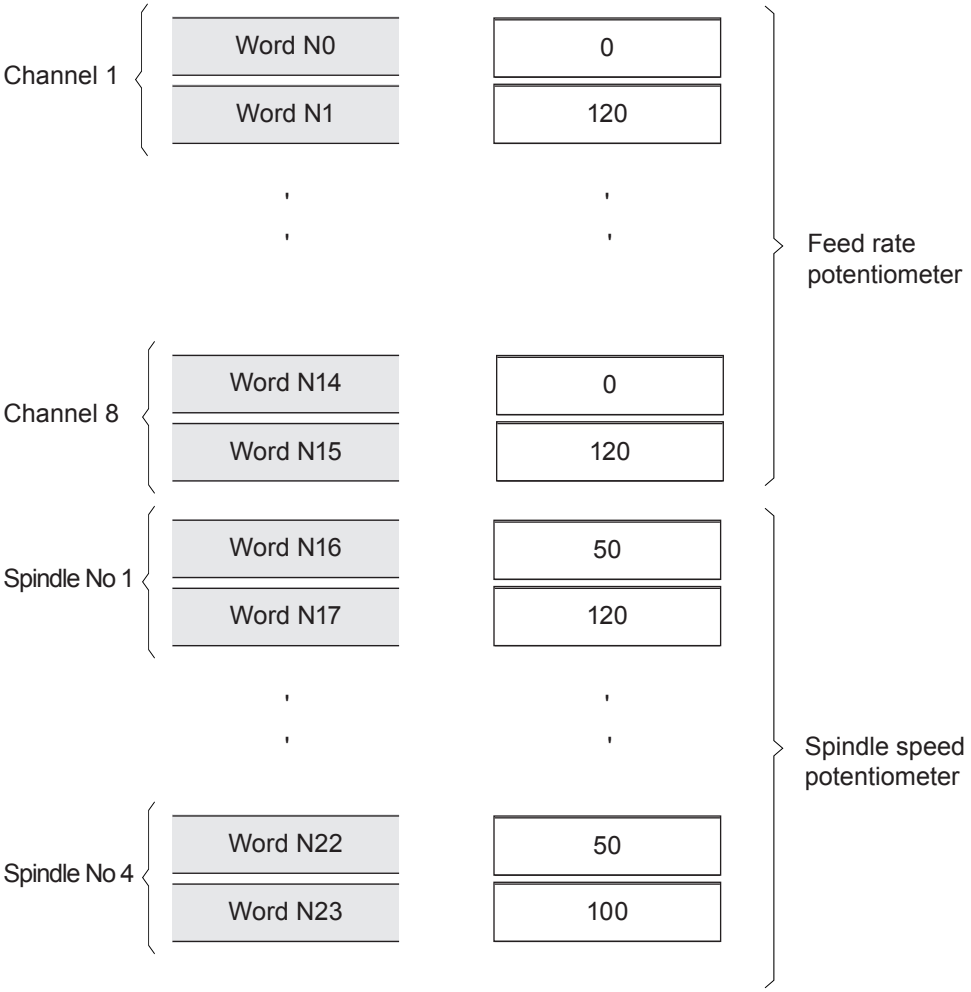
Description

The values are specified as percentages in 16-bit words.

All the values must be positive or zero.

255% is the highest possible value
0% is the lowest possible value.

- 100% must be within the min-max range.
- The max values must always be higher than the min values
- If these conditions are not satisfied, the default value is set at power on:
  - 0-120% for feed rate potentiometers
  - 50-100% for spindle speed potentiometers.



9.8 Memory Area Allocation ► P95

Description

Used to allocate memory areas for part programmes.

Principle

The RAM dedicated to part programmes is divided into four areas:
- free access area 0,
- protected areas 1 to 3.

The protected areas are designed to protect certain programmes (resident macros) which may be proprietary and to ensure integrity of machine operation (the programmes in protected areas cannot be edited).
The total memory size is defined when customising the system and the size of areas 1 to 3 can be modified in words N0 to N2.

The three protected areas include:
- Customer area 1,
- OEM area 2,
- NUM area 3.

The memory area is expressed in kbytes (1024 bytes).

Any remaining area not allocated by this parameter is assigned to area 0 "Free access user area".

Word N0 : used to define the size allocated to area 1 (Customer area).

Word N1 : used to define the size allocated to area 2 (OEM area).

Word N2 : used to define the size allocated to area 3 (NUM area).

Part program memory area

NUM area 3
OEM area 2
Customer area 1
Free access area 0

9.9 Communication period – Address of master CNC – Timeout for the monitoring of PLC application counter ► P99

Description

Defines the communication period between NC and PLC.

Specifies the address of master NC.

Defines the timeout used to monitor communications with the PLC application.

Principle

Word N0 : Defines the communication period between the NC and PLC in numbers of NC sampling periods (P50). Value between 3 and 6.

In a multi-NC system, this value must be greater than the number of NCs.

Word N1 : Specifies the address of the master NC.

The address on the physical NCK unit is selected by means of the rotary switch on the front.

In a multi-NC system, this value must be different for all NCKs.

Word N2 : Defines the timeout in number of NC sampling periods (P50) used to monitor communications with the PLC application.

If PLC application period is less than N0, then set N2=N0+1.

If PLC application period is bigger than N0, then set N2=(Tplc/P50)+1.

If PLC is in free-run mode, find out the maximum length of its period and apply the above formula.

9.10 Backup in Path, Auto Recall after INTER ► P114

Description

Defines the number of stored blocks that can be backed up in the path, the number of points that can be stored during axis retraction in INTER mode and the distance after which approach to the recall point is carried out at the work rate.

Principle

Word N0 : defines the number of stored blocks used for backup in the path after a feed stop.

Backup and return are carried out at the speed programmed in the stored blocks (see Flexium HMI Operator Manual).

A maximum of 100 blocks can be stored.

Word N1 : defines the number of points to be stored during axis retraction in INTER mode.

This to allow axis recall on the same path (see Flexium HMI Operator Manual).

A maximum of 10 points can be stored.

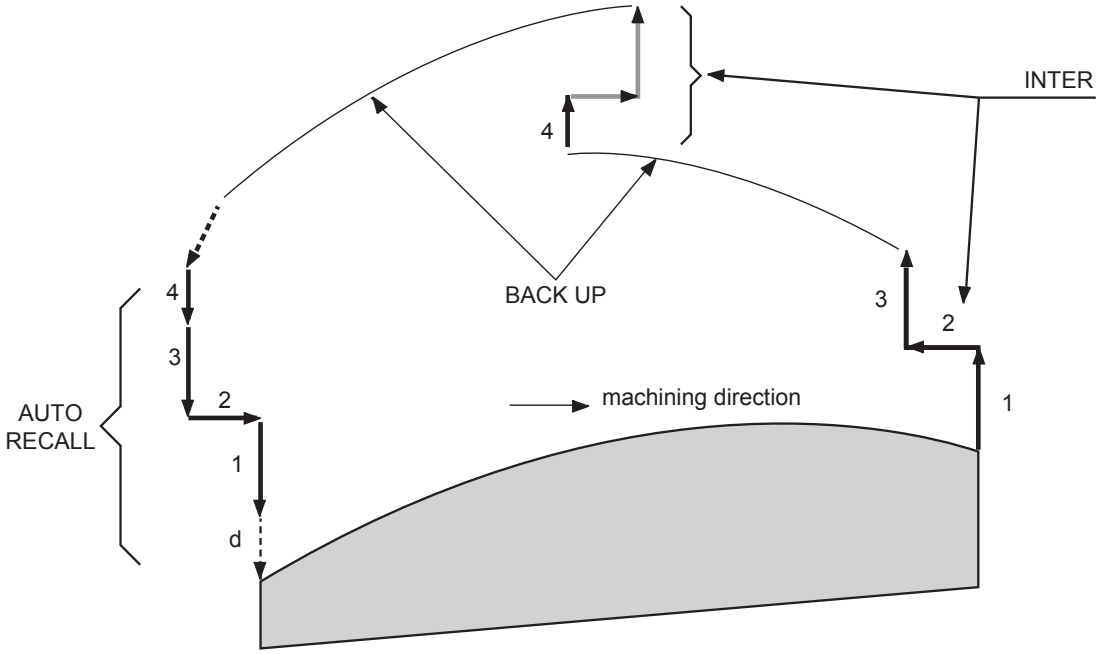
Auto recall can be combined with backup and return in the path.

Word N2 : specifies the distance to be covered at the work rate.

Auto recall is carried out at the JOG rate, but the final approach to the recall point can be carried at the work rate.

Example

Four stored points (1 to 4). The thin line is the rest of INTER that was not stored.
d = recall distance at the work rate



Page deliberately empty

A

Anti-backlash 57
Anticipation Time Constant 161
Antipitch 95
Approach Speed 105
Axes 27
  CNC 27
  Displayed 28
  Interpolated 32
  Measured 52
  Servo-controlled 32
Axes 2
  Measurement Direction 65
  Programmable by Diameter 34
  Traverse Rates 77
  with Clamps 40
  with Quantified Movements 52
Axis
  Drive 48
  Driven 48
Axis and Spindle Switching 53, 56

B

Backlash 126
Backlash Error Compensation 126
Backup in Path 178
Bidirectional Spindle Indexing 131

C

Carrier or Carried Axes 51
Checking
  Homing 120
Communication period 177
Consecutive Encoder Faults 73
Coupling Assignment for
  Duplicated Axes 50
  Synchronised Axes 43
Coupling with
  Symmetry 48
  Synchronisation 48

D

Data Tables 65, 131
Datum Switch Position 122
Decimal Words 16
Dimension Acquisition and Display Format 34
Direction of Axis Speed Reference 82
Displayed Axes 29
DIVI 66
Drive Axis 48
Driven Axis 48
Dry Friction
  Correction Time Constant 103
  Elimination 104
Dynamic Alignment Function 96
Dynamic Movement Control 87

E

EMAX 90
Encoder Channel Complementarity 70

F

Following Error 83
Functions
  Miscellaneous Functions 167

G

Gain 83, 157
Graphic Element 175

H

Handwheel Measurement Direction 67
High Speed Cutting Machining 93, 103
Homing 117, 118
Homing Direction 120
HSC machines 100

I

Indexed Stop 131
In-Position Window 88
Integration Time Constant 161
Interaxis Calibration Table Reservation 174
Internal Measurement 34
Internal Units 137
Interpolated Axes 32

J

Jerk
  Square 106
JOG Speeds 78
JOG Speed Setting 111
  Fast JOG 111
  Normal JOG 111
  Slow JOG 111

K

KVAR 84

L

Limited Excursion Rotary Axes 30

M

Master axes with Torque Synchronisation 60
Master/Slave Coupled Axes 57
Maximum Permissible Acceleration 79
Maximum Speed During Spindle Indexing 153
Measured Axes 28
Measurement Conversion Coefficient 66
Memory Area 176
Memory Area Allocation 176
milling machine channels 33
Minimum Spindle Reference in Hole Bottom 159
Miscellaneous Function 167
Miscellaneous Times 177
Modulo Axes 52
MULTI 66

N

Nano-Interpolation Function 38

P

P0 29
P1 30
P2 28
P3 32
P4 34
P5 33
P6 135
P7 167
P8 40
P9 42
P10 65
P11 66
P12 67
P13 68
P14 45
P16 122
P17 124
P18 126
P19 103
P20 82
P21 83
P22 88
P23 89
P24 97
P25 70
P26 72
P27 48
P28 50
P30 77
P31 78
P32 79
P33 105
P35 170
P40 137
P41 139
P42 152
P43 153
P44 155
P45 157
P46 140
P47 143
P48 146
P49 149
P50 171
P51 172
P52 99
P53 106
P54 173
P55 93
P56 87
P57 91
P58 173, 174
P62 159
P63 161
P64 51
P65 52
P66 110
P67 175
P70 53, 56
P71 56
P77 73
P85 57
P86 60
P95 176
P97 27
P99 177
P114 178
PLC
Variable Mnemonics 17
Poor Signal 70, 72
Position Anticipation Time Constant 162
Potentiometer Min-Max Range 175
Programme Stack Reservation 174

R

Rear Turrets 33
Reduced Speeds 78
Rigid Tapping 134
Rules for Writing the Parameters 15

S

Sawtooth Acceleration 108
Servo-Controlled 32
Servo-Loop Error 99
Servo-System Loop Gain Coefficient 83
Setting
of the Speed Buildup in the Path 106
Simple Coupling 48
Smith corrector 83
Speed
Anticipation Coefficient 93
Buildup Diagram 81
Speed Buildup Diagram 81
Spindle
Spindle Speed Ranges 136, 139, 149
Spindle 2
Indexing 131
Indexing In-Position Window 155
Measurement Conversion 137
Origins 152
Reference 139
Zero Crossing 161
Spindle[n].PositionReached 131
Square jerk 106
Subroutine Call 170
Synchronisation error 97
Synchronised Axis Coupling 50

T

Tandem Function 57, 58
Time Constant 83, 87, 110
Time Constant per Axis 110
Timeout 177
Torque Duplication 58
Torque Synchronisation 58
Trapezoidal Jerk 106

Page deliberately empty

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 November 2008

CNC HighEnd Applications