NUM

PARAMETER MANUAL

0101938818/9

Table of Contents

Record of Revisions

Date	Revision	Revised pages	Added pages	Deleted pages
12 - 98	9	1, 5, 7 to 16 2-2 to 2-4 4-6, 4-8, 4-25 5-21, 5-24 6-40 8-11, 8-13, 8-14, 8-32 to 8-34 9-1, 9-3, 9-5 to 9-7, 9-9 to 9-11 9-13 to 9-30, 10-20 I-1, I-2 Questionnaire	17, 18 9-31, 9-32	

Date	Revision	Reason for Revisions
10 - 91	0	Conforming to NUM 1060 software at index A. Document creation.
01 - 92	1	Conforming to NUM 1060 software at index A. Miscellaneous corrections.
01 - 93	2	Conforming to NUM 1060 software at index D. Miscellaneous corrections. P4 - Word N2, internal system measurement. P27 - Definition of coupling for duplicated axes. P28 - Confirmation of coupling for synchronised axes. P31 - Words N3 and N4. P58 - Word N2, programme stack size. P59 - Graphic, display and print configurations. P84 - Timeout for DNC1000. P95 - Part programme memory segment sizes. P98 - Choice of machine processor programming language.
06 - 94	3	Conforming to NUM 1060 software at index E. Miscellaneous corrections. P100 - MAPWAY/ETHWAY network and station number. P110 - UNI-TELWAY master settings. P111 - UNI-TELWAY slave settings. P112 - Settings of the line assigned to the PLCTOOL link.
11 - 94	4	Conforming to NUM 1060 software at index F Miscellaneous corrections UNI-TE requests added for parameters P3, P10, P11, P20, P15-P18, P21, P23, P30, P32, P55-P57, P40-P49, P6. P7 - Homing selection - Subroutine call on reset - Feed stop for rigid tapping - Speed variation with gradual acceleration P21 - Setting for DISC axes P62 - Rigid tapping P63 - Rigid tapping

Date	Revision	Reason for Revisions
		P70 - Mapping by card P71 - Axes controlled by QDD P72 - Direction of motor rotation P73 - Maximum motor speed P74 - Proportional action coefficient of the speed servo-loop corrector P75 - Integral action coefficient of the speed servo-loop corrector P99 - Maximum time allocated to the PLC application
01 - 95	5	Conforming to NUM 1060 software at index G Miscellaneous corrections P6 - Words N5 to N8: Spindle assignment to axis groups P19 - Very High Speed Machining P33 - Approach speed P55 - Words N8 to N15 - Number of terms used to calculate the filtered reference for Very High Speed Machining P99 - Word N2: Fast execution on PLC
04 - 96	6	Conforming to NUM software at index J Miscellaneous corrections Parameters modified: - P11: Words DIVI - P26: Added bits 4-6 - P46-P49: Added words 12-17 (limit speed) New parameters: - P34: Definition of measurement sensor type and parameters - P36: Measurement sensor graduation and number of graduations - P76: Speed sensor measurement increment average - P77: Speed measurement filter - P78: Torque reference filter - P114: Backup in path, auto recall after intervention
09 - 97	7	Conforming to NUM software at Index K Miscellaneous corrections Parameters modified: - P4: Added word 4 - P13: Added words N10 to N17 (MULTI/DIVI for rotary axes) - P14: Added word N1 (handwheel measurement filtering) - P25: Added word N1 (enable/inhibit QVN motor encoder control) - P53: Adjustment of maximum permissible acceleration on the axes - P66: Time constant per axis - P79: Static current limiting - P85: Declaration of a torque slave QVN application - P86: Slave rotation direction same as or different from master - P87: Specification of preload current - P110: Added word N7 (number of polling cycles) Deleted Chapter 12 (see DISC Integration Manual 938907)
06 - 98	8	Conforming to NUM software at Index M Miscellaneous corrections Parameters modified: - P25 : Added text for word N1 = 0 - P36 : Combined sensor - P50 : Changes concerning word N0 - P84 : Added file download - P99 : Added word N3 - P100, P110, P111: Added new information

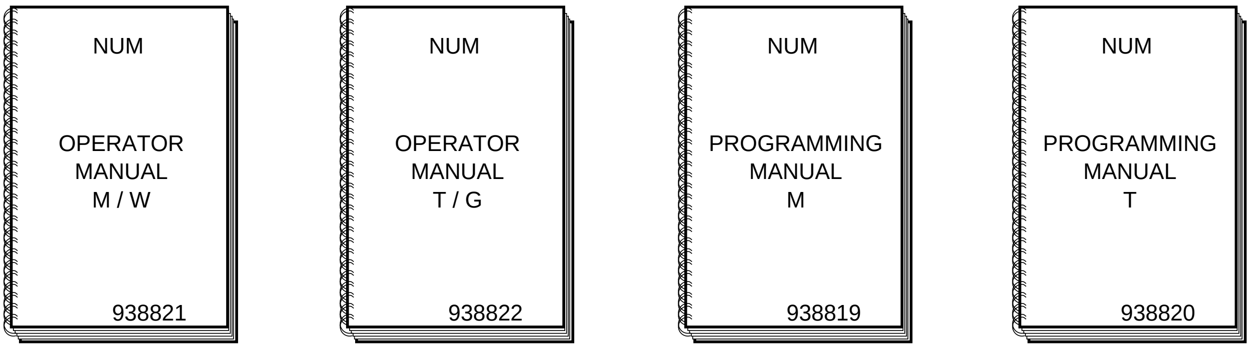
Date	Revision	Reason for Revisions
12 - 98	9	Conforming to NUM software at index M Miscellaneous corrections - List of parameters modified -Parameters added: - P67: Potentiometer min/max range - P88, P89, P90, P91, P92, P93, P94, P116, P117, P118, P119, P120 (QVN parameters, see remark, p. 2-3) - P115: Parameter invalid (MODICON) Parameters modified: - P2: Added remarks - P6: Added information for DISC NT on bit 2 - P9: Recommendations concerning T*plus* and M*plus* - P36: Added CAUTION (DISC NT) - P40: Deleted remark - P50: Additions in word N0 - P58: Created word N2 (1050) - P59: Documented word N3 - P62: Number of words changed to 8 - P66: Added “for CNC software at index LS and above” - P96: Added an additional language FADISUX - P97: Deleted ”1020” (total number of groups) - P99: Added “Reminder” - changed designation - P112: Added sketch for 1020/1040

Foreword

NUM 1020/1040/1060 Documentation Structure

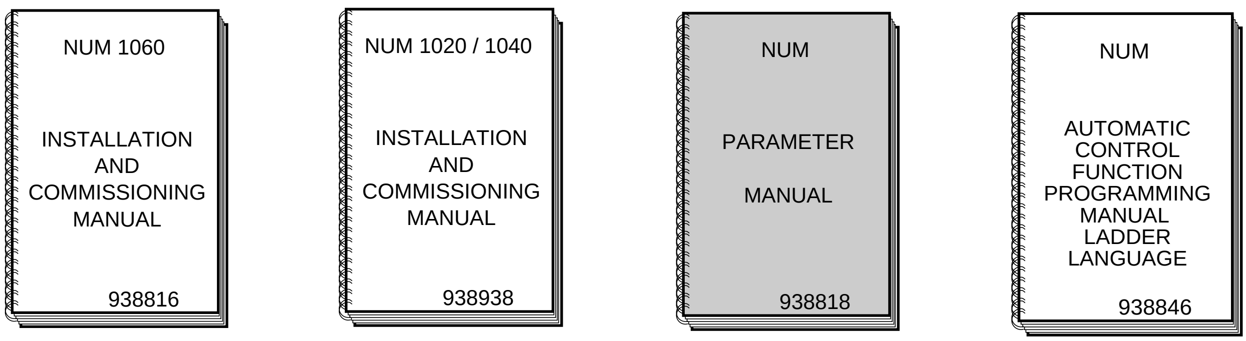
User Documents

These documents are designed for use of the CNC.



Integrator Documents

These documents are designed for setting up the CNC on a machine.



List of NUM 1020/1040/1060 Utilities

A series of utilities are available for the NUM for integration and use of the systems.

These utilities may be included in the basic version or available as options.

Depending on the function performed by each utility, its use is described in the integration manual or operator manual, as appropriate.

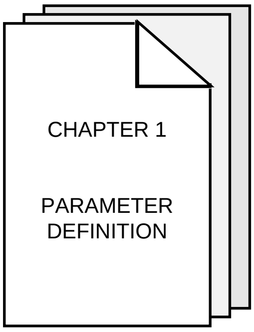
The table below lists the utilities and gives the references of the document describing them:

Utility	Name	Manual	Chapter	Application
UT0	utility management	operator manuals	8	NUM 1020/1040/1060
UT2	axis calibration	installation and commissioning manuals (1020/1040 or 1060)	10 11	NUM 1020/1040 NUM 1060
UT3	resident macros	operator manuals	8	NUM 1020/1040/1060
UT5	parameter integration	parameter manual	12	NUM 1020/1040/1060
UT7	programme debugging	automatic control function programming manual ladder language	16	NUM 1020/1040/1060 programming in ladder language
UT12	option locking	operator manuals	8	NUM 1020/1040/1060
UT20	interaxis calibration	installation and commissioning manual (1020/1040 or 1060)	11 12	NUM 1020/1040 NUM 1060
UT22	axis parameter integration	SET_TOOL manual	8	NUM 1060

REMARK: *Utility 22 is no longer used starting with CNC software at index K and the SET_TOOL software at index E.*

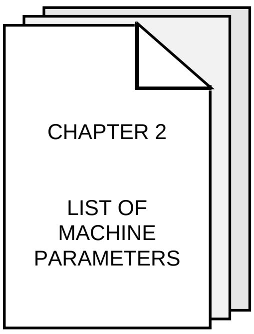
Parameter Manual

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

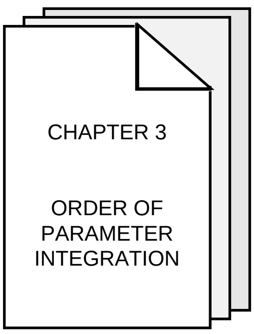


Parameter definition and structure.

Writing rules.

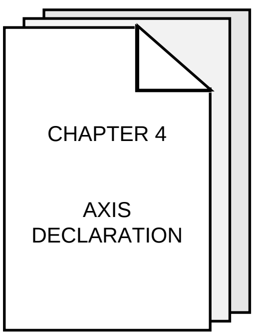


Alphanumeric list of the parameters.

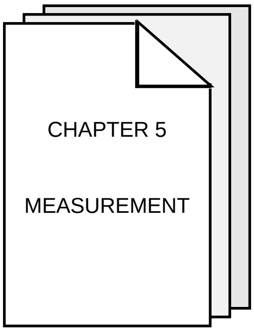


Order of parameter integration by subject.

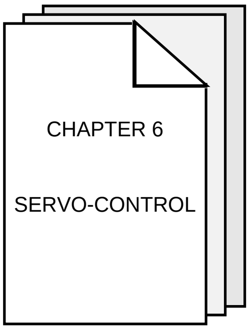
Tables for recording the values assigned to each word of each parameter.



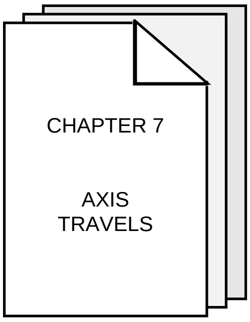
Axis declaration parameters.
- Machine setup data tables
- Special information.



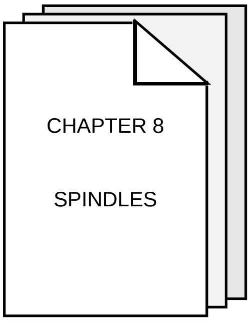
Measurement setting parameters.
- Machine setup data tables
- Special information.



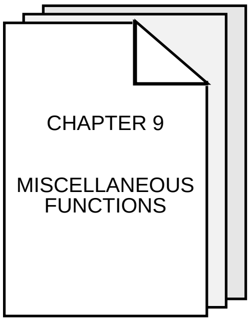
Axis servo-control parameters.
- Machine setup data tables
- Special information.



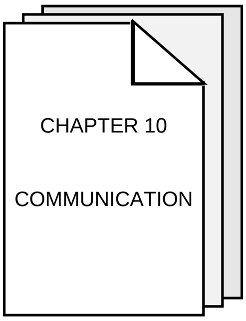
Axis travel setting parameters.
- Machine setup data tables
- Special information.



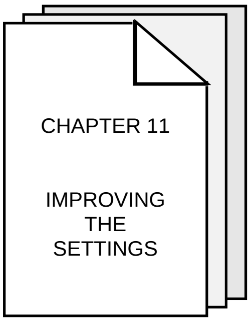
Spindle setting parameters.
- Machine setup data tables
- Special information.



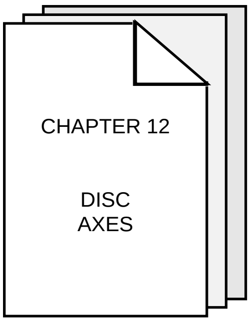
Miscellaneous parameters.
- Machine setup data tables
- Special information.



Parameters dedicated to communication.
- Special information.



Setting improvement procedure.



QVN axis setting parameters.

(see DISC Integration Manual 938907)

CHAPTER 13

PARAMETER INTEGRATION TOOL

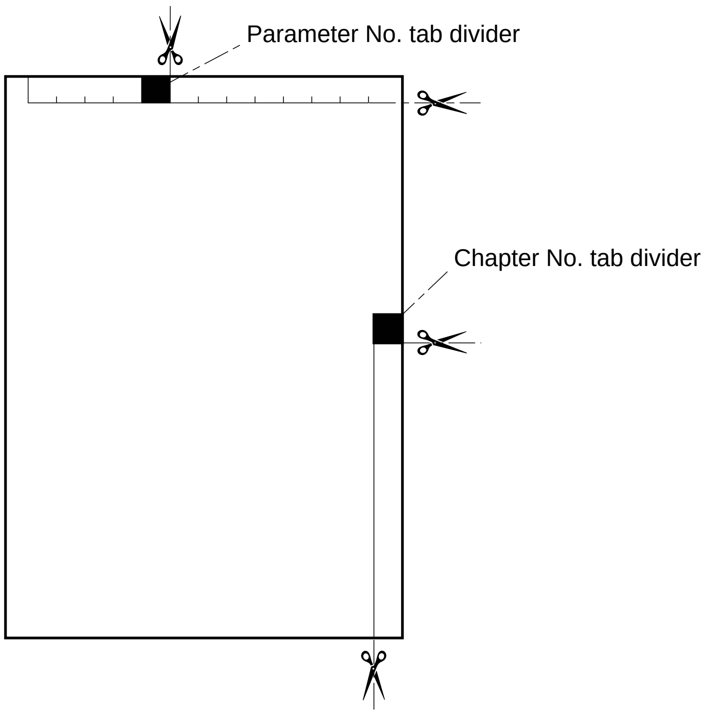
Procedure for use of the parameter integration tool on the CNC.

Overview of the CNC and its role with respect to the machine tool.

Use of the Parameter Manual

Tab Dividers

The chapters of this document include greyed tab dividers at the top of the page and along the edge. Cutting out these tab dividers facilitates access to the information concerning the parameters.



Chapter 3

The tables in this chapter are used for recording the settings assigned to each word of each parameter. The parameters are classified in the order of integration.

These tables facilitate CNC parameter entry.

Data Tables

The data tables at the beginning of each chapter are used for recording the data and settings specific to the machine. These data and settings can also be recorded in the tables of chapter 3 after analysing and formatting the parameters.

Procedures

The manual includes procedures (in particular in Chapter 13).

The actions to be carried out are illustrated as follows:

Reset the system. ☞ Y —

On the right are displayed the keys to be pressed, which can have two forms:

— Square keys: operator panel keys.

UTIL Rectangular keys: soft keys displayed in the bar at the bottom of the screen, actuated by pressing function keys F2 to F11 located under the screen.

Index

The index at the end of the volume gives access to information by keywords.

Agencies

The list of NUM agencies is given at the end of the volume.

Questionnaire

To help us improve the quality of our documentation, we kindly ask you to return the questionnaire at the end of this volume.

1 Parameter Definition

1.1 Introduction

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

1.2 Rules for Writing the Parameters

The machine parameters are different types of words. The word type and comment are displayed on the pages of utility 5 (see Chapter 12).

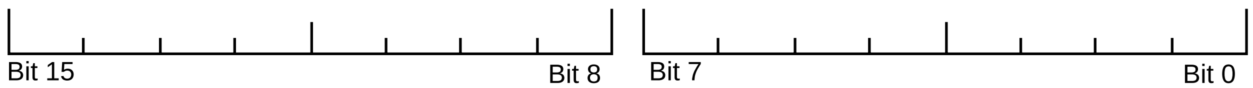
Word type	Comment	Expression	Values
0	BIN.BYTE	8-bit hexadecimal	00-FF
1	DECI. 16 bits	Signed decimal	-32768 to +32767
2	DECI. 32 bits	Signed decimal	-99,999,999 to +99,999,999
5	DEC. 16 bits UNSIGN.	Unsigned decimal	00 to 65535
4	DEC. 32 bits UNSIGN.	Unsigned decimal	00 to 99,999,999
6	BIN. 32 bits	32-bit hexadecimal	00 00 00 00 to FF FF FF FF
7	BIN. 16 bits	16-bit Hexadecimal	00 00 to FF FF
8	CHARACTER STRING	ASCII character string	All alphanumeric characters
9	FLOATING POINT		

1.2.1 8-bit and 16-bit Hexadecimal Words

Parameter types 0 and 7.

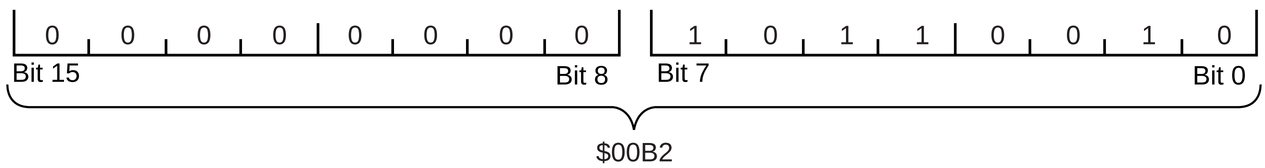
Each bit represents a particular function or system configuration (e.g. axis group number). It is enabled when the bit is set and inhibited when the bit is zero.

Word Format



Example

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



Value of the word:	00B2

1.2.2 32-bit Hexadecimal Words

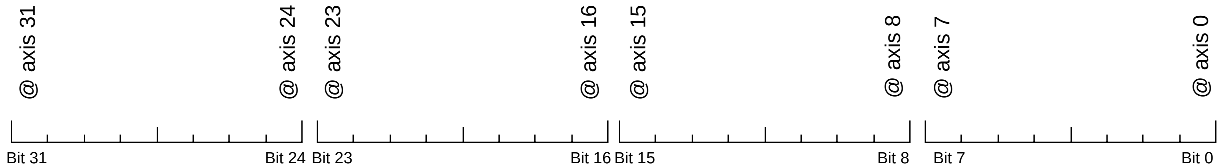
Parameter type 6.

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.

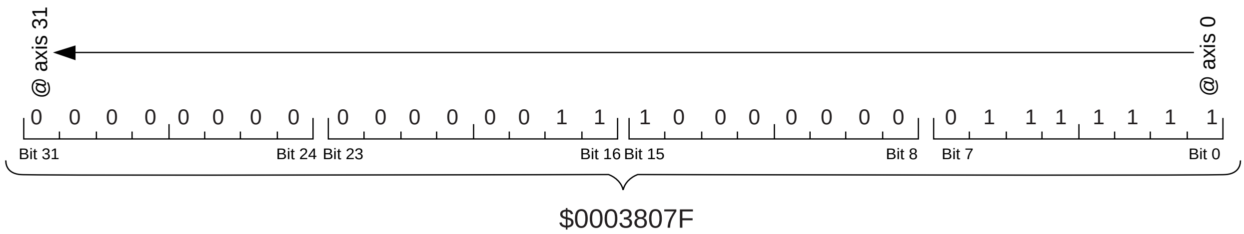
REMARK: The physical address of an axis is assigned by the wiring of pins 11, 12, 13, 23, 24 and 25 on the axis encoder card connector (see «Installation and Commissioning Manual»).

Word Format



Example

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



Value of the word:	0003807F

1.2.3 Decimal Words

Parameter types 1, 2, 4 and 5.

Each word contains a signed or unsigned decimal value.

> ⚠ **CAUTION**
>
> When entering the values, comply with the units specified for each type of word.

1.2.4 ASCII Character String

Parameter type 8.

Each word contains an ASCII character string.

> ⚠ **CAUTION**
>
> When entering the values, comply with the maximum number of characters.

1.3 PLC Variable Mnemonics

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

Mnemonic	Identification	Assembler	Ladder
CRM	M function report	A.10E	
CRM1	Group 1 M function report		%W100.5
CRM2	Group 2 M function report		%W200.5
CRM3	Group 3 M function report		%W300.5
CRM4	Group 4 M function report		%W400.5
CRM5	Group 5 M function report		%W500.5
CRM6	Group 6 M function report		%W600.5
CRM7	Group 7 M function report		%W700.5
CRM8	Group 8 M function report		%W800.5
POSBR1	Spindle 1 in position	E.11E	%R13.0
POSBR2	Spindle 2 in position	E.11F	%R13.1
POSBR3	Spindle 3 in position		%R13.2
POSBR4	Spindle 4 in position		%R13.3
CN_PRET	CNC ready	E.108	%R5.0
V_REDUIT	Low speed	A.111	%W4.7
VITMAN1	JOG 1 selection	A.109	%W4.1
VITMAN2	JOG 2 selection	A.10A	%W4.2

2 List of Machine Parameters

Parameter	Category	Type	Description	See page
P0	Axis declaration	6	Displayed axes	4-10
P1	Axis declaration	6	Modulo and limited excursion rotary axes	4-12
P2	Axis declaration	6	Measured axes	4-8
P3	Axis declaration	6	Servo-controlled and interpolated axes	4-14
P4	Axis declaration	0	Axes programmed by diameter Internal system measurement	4-18
P5	Axis declaration	0	Axis group machine configuration Lathe or milling machine groups Front/Rear turrets	4-16
P6	Spindles	0	Number of controlled spindles Spindle type Spindle assignments to axis groups	8-10
P7	Miscellaneous	0	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	9-6
P8	Axis declaration	6	Axes with clamps	4-22
P9	Axis declaration	0	Axis assignment to groups	4-24
P10	Measurements	6	Axis measurement direction	5-4
P11	Measurements	5	Axis measurement conversion coefficient	5-6
P12	Measurements	0	Direction of handwheel measurement	5-10
P13	Measurements	5	Handwheel measurement conversion coefficient	5-12
P14	Axis declaration	0	Handwheels	4-28
P15	Axis travel	6	Direction of homing Switch status test	7-8
P16	Axis travel	2	Reference switch position in machine dimensions	7-10
P17	Axis travel	2	Axis travel limit	7-12
P18	Axis travel	1	Reversal error compensation	7-14
P19	Servo-control	1	Very high speed machining	6-34
P20	Servo-control	6	Direction of axis speed reference	6-12
P21	Servo-control	4	Servo-system loop gain coefficient	6-14
P22	Servo-control	5	In-position window	6-20
P23	Servo-control	4	Maximum following error	6-22
P24	Servo-control	5	Synchronised axis control	6-30
P25	Measurements	6	Poor signal and encoder channel complementarity check declaration	5-14
P26	Measurements	0	Poor signal and encoder channel complementarity check	5-16
P27	Axis declaration	0	Duplicated axis coupling definition	4-30
P28	Axis declaration	6	Synchronised axis coupling enable	4-32
P29	Gear cutting	1	Reserved for NUM 1060 HG gear cutting	Not documented

Parameter	Category	Type	Description	See page
P30	Servo-control	4	Maximum axis traverse rates	6-6
P31	Servo-control	4	JOG speed and reduced speed	6-8
P32	Servo-control	5	Maximum permissible acceleration	6-10
P33	Servo-control	5	Approach speed	6-36
P34	Measurement	6	Definition of measurement sensor type and parameters	5-18
P35	Miscellaneous	5	Subroutine call by M function	9-8
P36	Measurement	5	Measurement sensor graduation and number of graduations	5-22
P37	Communication	0	Character format in DNC1	10-2
P38	Communication	6	Translation of special DNC1 characters	10-4
P39	Communication	4	DNC1 timeouts	10-6
P40	Spindles	4	Spindle measurement conversion	8-12
P41	Spindles	0	Spindle reference reversal	8-14
P42	Spindles	4	Spindle origins	8-24
P43	Spindles	5	Maximum spindle speed	8-26
P44	Spindles	5	Indexing in-position window	8-28
P45	Spindles	5	Spindle servo-system gain	8-30
P46	Spindles	5	Spindle 1 speed range	8-16
P47	Spindles	5	Spindle 2 speed range	8-18
P48	Spindles	5	Spindle 3 speed range	8-20
P49	Spindles	5	Spindle 4 speed range	8-22
P50	Miscellaneous	5	Sampling period (CNC and QVN)	9-10
P51	Miscellaneous	5	Minimum block execution time	9-12
P52	Servo-control	5	Servo-system error tolerated on circles	6-32
P53	Servo-control	5	Setting of the maximum permissible acceleration on the axes	6-38
P55	Servo-control	5	Speed anticipation coefficient, number of terms used to calculate the filtered reference for very high speed machining	6-26
P56	Servo-control	5	Servo-loop time constant	6-18
P57	Servo-control	5	Dynamic movement control	6-24
P58	Miscellaneous	5	Interaxis calibration table reservation Programme stack size	9-14
P59	Miscellaneous	0	Graphic, display and print configurations	9-16
P62	Spindles	1	Spindle acceleration Minimum spindle reference in hole bottom	8-32
P63	Spindles	1	Cycle axis integration time constant Cycle axis position anticipation time constant Spindle zero crossing anticipation time constant	8-34
P64	Axis declaration	6	Carried or carrier axes	4-34
P65	Axis declaration	4	Axes with quantified movements	4-36
P66	Servo-control	5	Time constant per axis	6-40
P67	Miscellaneous	5	Potentiometer Min-Max range	9-18
P70	QVN	6	Mapping by cards	*
P71	QVN	0	Axes controlled by QVN	*
P72	QVN	6	Direction of motor rotation	*

Parameter	Category	Type	Description	See page
P73	QVN	5	Maximum motor speed	*
P74	QVN	4	Proportional action coefficient of the speed servo-loop corrector	*
P75	QVN	4	Integral action coefficient of the speed servo-loop corrector	*
P76	QVN	0	Speed sensor measurement increment average (motor sensor)	*
P77	QVN	5	Speed measurement filter	*
P78	QVN	5	Torque reference filter	*
P79	QVN	1	Static current limiting	*
P80	Miscellaneous	0	Xoff character choice	9-20
P84	Communication	5	File upload and download timeout	10-8
P85	QVN	0	Declaration of a torque slave QVN application	*
P86	QVN	6	Slave rotation direction same as or different from master	*
P87	QVN	1	Specification of preload current	*
P88	QVN	5	Interpolated speed reference low-pass filter	*
P89	QVN	5	Interpolated speed reference rejection filter N2j : fn N2j + 1 : Zn	*
P90	QVN	5	Interpolated speed reference rejection filter N2j : fd N2j + 1 : Zd	*
P91	QVN	5	Undocumented parameter	
P92	QVN	5	Interpolated speed reference rejection filter N2j : fn N2j + 1 : Zn	*
P93	QVN	5	Interpolated speed reference rejection filter N2j : fd N2j + 1 : Zd	*
P94	QVN	5	Undocumented parameter	
P95	Miscellaneous	5	Part programme memory segment size	9-22
P96	Miscellaneous	8	Type of machine for PROCAM Language selection	9-24
P97	Axis declaration	0	Number of axis groups in the system	4-6
P98	Miscellaneous	0	Integrated machine processor programming language selection	9-26
P99	Miscellaneous	5	Miscellaneous times	9-28
P100	Communication	0	MAPWAY/ETHWAY network and station number	10-10
P101		7	Invalid parameter	Not documented
P102		5	Invalid parameter	Not documented
P103		0	Invalid parameter	Not documented
P104		0	Invalid parameter	Not documented
P105		0	Invalid parameter	Not documented
P106		0	Invalid parameter	Not documented

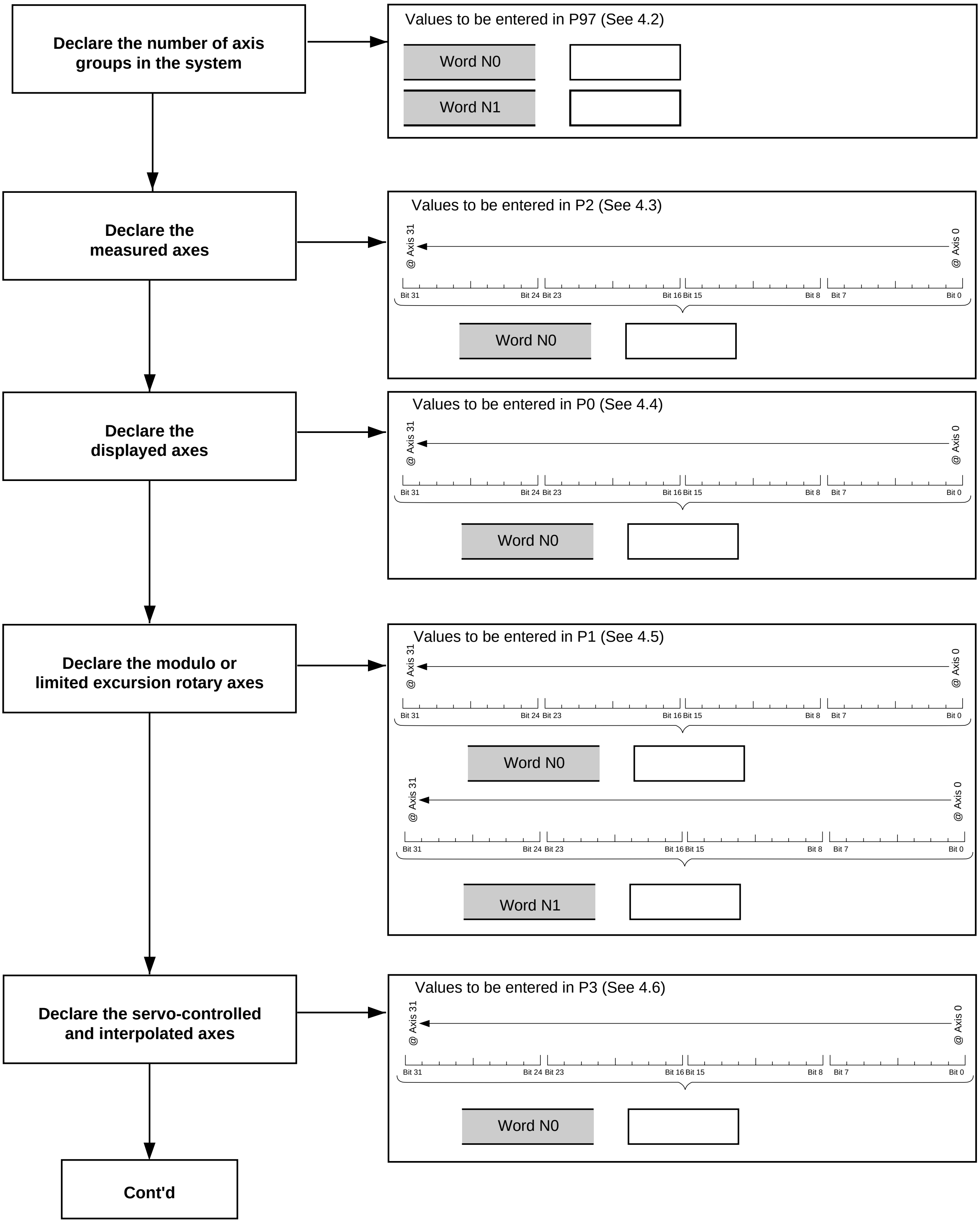
Parameter	Category	Type	Description	See page
P107		8	Invalid parameter	Not documented
P110	Communication	0	UNI-TELWAY master settings	10-12
P111	Communication	0	UNI-TELWAY slave settings	10-16
P112	Communication	0	Settings of the line assigned to PLCTOOL	10-20
P113		9	Invalid parameter	Not documented
P114	Miscellaneous	5	Backup in path, auto recall after INTERV	9-30
P115			Invalid parameter (Modicom)	Not documented
P116	QVN	4	Kalman filter: Ratio Kc/j	
P117	QVN	4	Kalman filter: Kalman gain for position estimation	*
P118	QVN	4	Kalman filter: Kalman gain for speed estimation	*
P119	QVN	4	Kalman filter: Kalman gain for interference estimation	*
P120	QVN	1	Kalman filter: Monitoring device	*

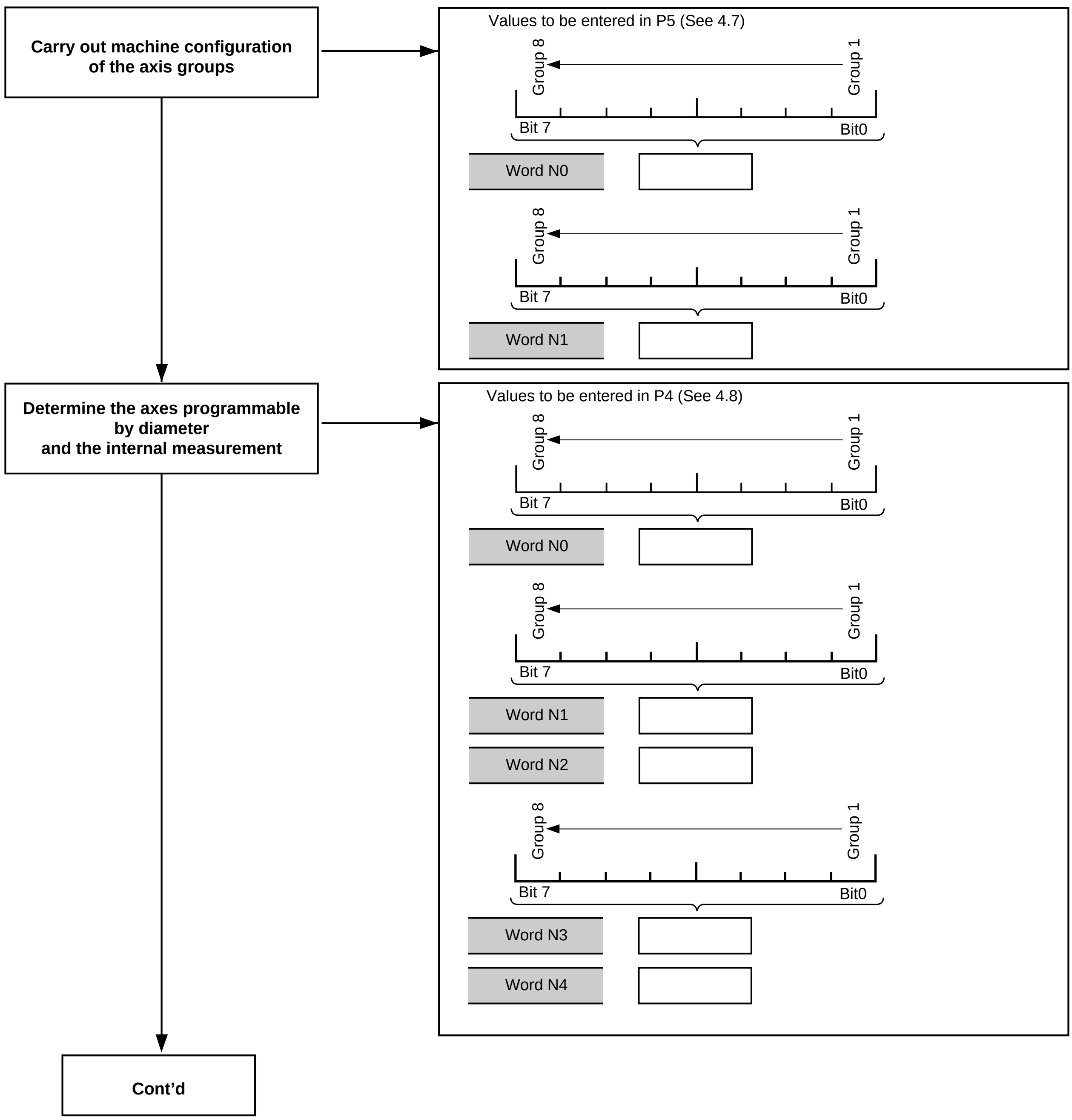
REMARK: ** The QVN parameters (P85 to P94 and P116 to P120) are not covered in this manual. For programming them, see DISC Integration Manual 938907.*

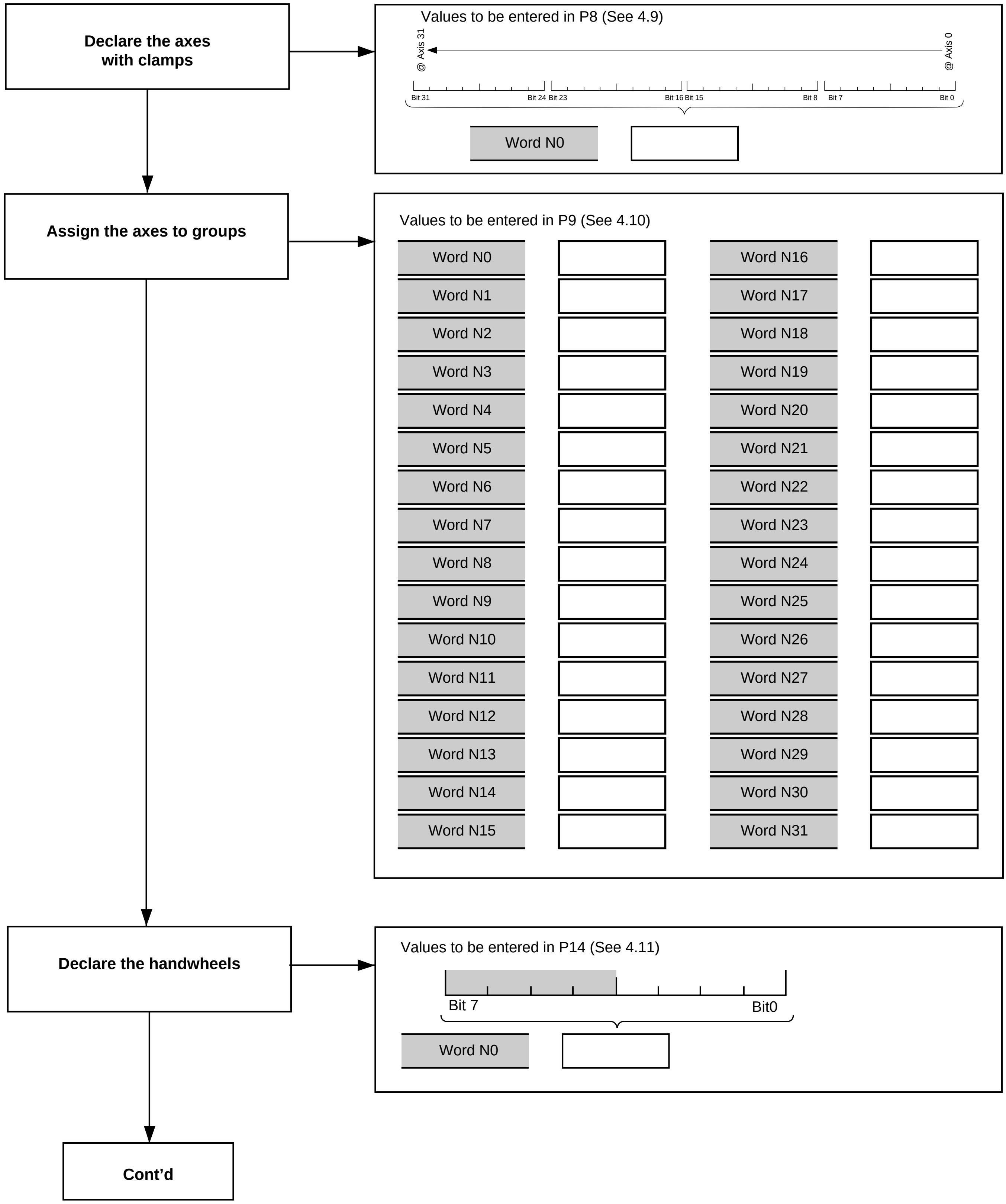
3 Order of Parameter Integration

3

3.1 Axis Declaration




Carry out machine configuration of the axis groups
Values to be entered in P5 (See 4.7)
Group 8
Group 1
Bit 7
Bit0
Word N0
Group 8
Group 1
Bit 7
Bit0
Word N1
Determine the axes programmable by diameter and the internal measurement
Values to be entered in P4 (See 4.8)
Group 8
Group 1
Bit 7
Bit0
Word N0
Group 8
Group 1
Bit 7
Bit0
Word N1
Word N2
Group 8
Group 1
Bit 7
Bit0
Word N3
Word N4
Cont'd


Declare the axes with clamps
Values to be entered in P8 (See 4.9)
@ Axis 31
@ Axis 0
Bit 31
Bit 24
Bit 23
Bit 16
Bit 15
Bit 8
Bit 7
Bit 0
Word N0
Assign the axes to groups
Values to be entered in P9 (See 4.10)
Word N0
Word N1
Word N2
Word N3
Word N4
Word N5
Word N6
Word N7
Word N8
Word N9
Word N10
Word N11
Word N12
Word N13
Word N14
Word N15
Word N16
Word N17
Word N18
Word N19
Word N20
Word N21
Word N22
Word N23
Word N24
Word N25
Word N26
Word N27
Word N28
Word N29
Word N30
Word N31
Declare the handwheels
Values to be entered in P14 (See 4.11)
Bit 7
Bit0
Word N0
Cont'd

Assign the couplings for the duplicated axes

Values to be entered in P27 (See 4.12)

Word N0		Word N16	
Word N1		Word N17	
Word N2		Word N18	
Word N3		Word N19	
Word N4		Word N20	
Word N5		Word N21	
Word N6		Word N22	
Word N7		Word N23	
Word N8		Word N24	
Word N9		Word N25	
Word N10		Word N26	
Word N11		Word N27	
Word N12		Word N28	
Word N13		Word N29	
Word N14		Word N30	
Word N15		Word N31	

Enable the couplings for the synchronised axes

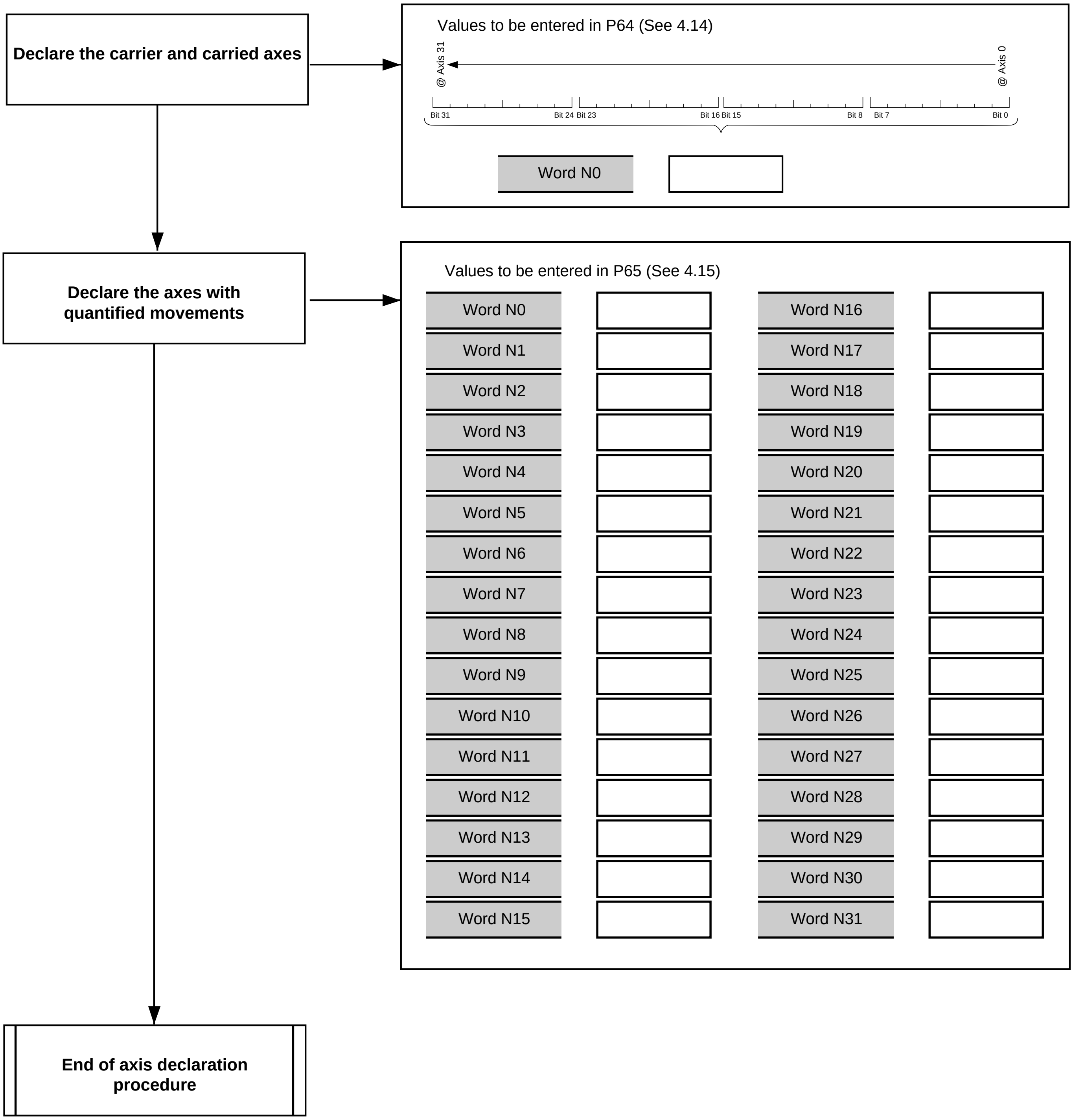
Values to be entered in P28 (See 4.13)

@ Axis 31 ← @ Axis 0

Bit 31 ... Bit 24 Bit 23 ... Bit 16 Bit 15 ... Bit 8 Bit 7 ... Bit 0

Word N0	

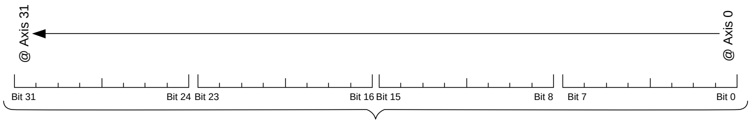
Cont'd


Declare the carrier and carried axes
Values to be entered in P64 (See 4.14)
@ Axis 31
@ Axis 0
Bit 31
Bit 24
Bit 23
Bit 16
Bit 15
Bit 8
Bit 7
Bit 0
Word N0
Declare the axes with quantified movements
Values to be entered in P65 (See 4.15)
Word N0
Word N1
Word N2
Word N3
Word N4
Word N5
Word N6
Word N7
Word N8
Word N9
Word N10
Word N11
Word N12
Word N13
Word N14
Word N15
Word N16
Word N17
Word N18
Word N19
Word N20
Word N21
Word N22
Word N23
Word N24
Word N25
Word N26
Word N27
Word N28
Word N29
Word N30
Word N31
End of axis declaration procedure

3.2 Measurement Settings

Declare the direction of axis measurement

Values to be entered in P10 (See 5.2)



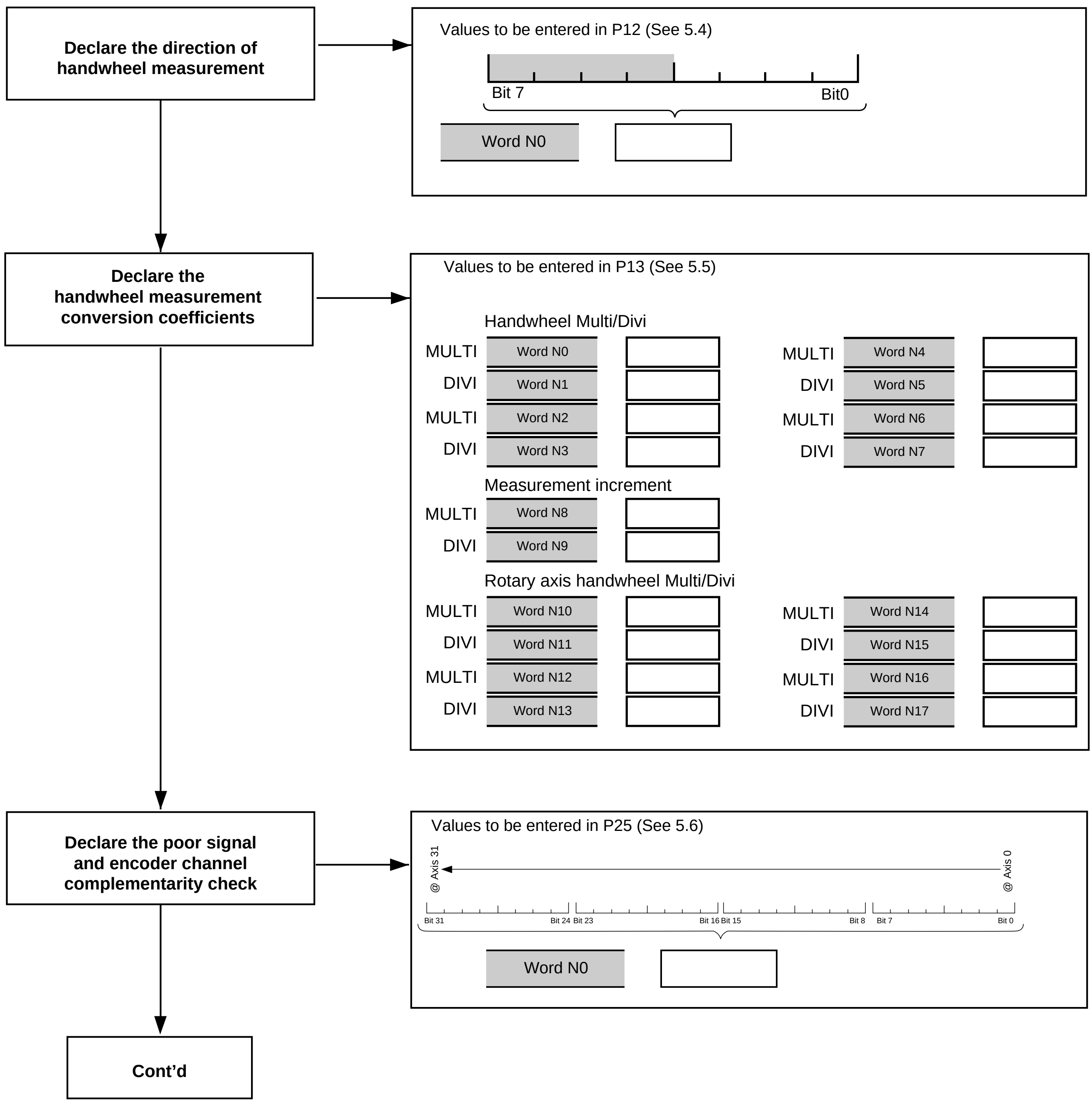
Word N0		

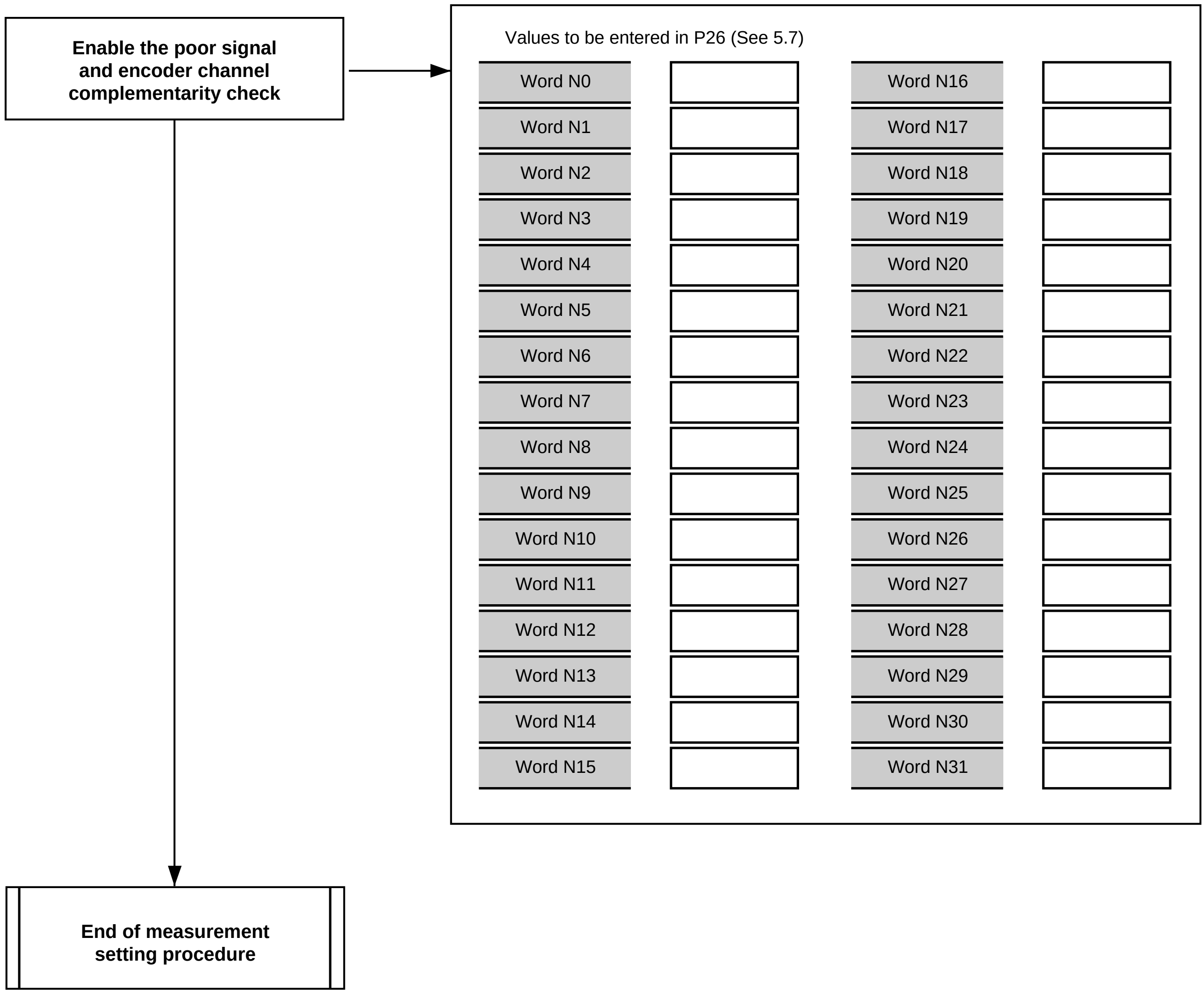
Declare axis measurement conversion coefficients

Values to be entered in P11 (See 5.3)

MULTI	Word N0		MULTI	Word N32	
DIVI	Word N1		DIVI	Word N33	
MULTI	Word N2		MULTI	Word N34	
DIVI	Word N3		DIVI	Word N35	
MULTI	Word N4		MULTI	Word N36	
DIVI	Word N5		DIVI	Word N37	
MULTI	Word N6		MULTI	Word N38	
DIVI	Word N7		DIVI	Word N39	
MULTI	Word N8		MULTI	Word N40	
DIVI	Word N9		DIVI	Word N41	
MULTI	Word N10		MULTI	Word N42	
DIVI	Word N11		DIVI	Word N43	
MULTI	Word N12		MULTI	Word N44	
DIVI	Word N13		DIVI	Word N45	
MULTI	Word N14		MULTI	Word N46	
DIVI	Word N15		DIVI	Word N47	
MULTI	Word N16		MULTI	Word N48	
DIVI	Word N17		DIVI	Word N49	
MULTI	Word N18		MULTI	Word N50	
DIVI	Word N19		DIVI	Word N51	
MULTI	Word N20		MULTI	Word N52	
DIVI	Word N21		DIVI	Word N53	
MULTI	Word N22		MULTI	Word N54	
DIVI	Word N23		DIVI	Word N55	
MULTI	Word N24		MULTI	Word N56	
DIVI	Word N25		DIVI	Word N57	
MULTI	Word N26		MULTI	Word N58	
DIVI	Word N27		DIVI	Word N59	
MULTI	Word N28		MULTI	Word N60	
DIVI	Word N29		DIVI	Word N61	
MULTI	Word N30		MULTI	Word N62	
DIVI	Word N31		DIVI	Word N63	

Cont'd


Declare the direction of handwheel measurement
Values to be entered in P12 (See 5.4)
Bit 7
Bit0
Word N0
Declare the handwheel measurement conversion coefficients
Values to be entered in P13 (See 5.5)
Handwheel Multi/Divi
MULTI
Word N0
DIVI
Word N1
MULTI
Word N2
DIVI
Word N3
MULTI
Word N4
DIVI
Word N5
MULTI
Word N6
DIVI
Word N7
Measurement increment
MULTI
Word N8
DIVI
Word N9
Rotary axis handwheel Multi/Divi
MULTI
Word N10
DIVI
Word N11
MULTI
Word N12
DIVI
Word N13
MULTI
Word N14
DIVI
Word N15
MULTI
Word N16
DIVI
Word N17
Declare the poor signal and encoder channel complementarity check
Values to be entered in P25 (See 5.6)
@ Axis 31
@ Axis 0
Bit 31
Bit 24
Bit 23
Bit 16
Bit 15
Bit 8
Bit 7
Bit 0
Word N0
Cont’d


Enable the poor signal and encoder channel complementarity check
Values to be entered in P26 (See 5.7)
Word N0
Word N1
Word N2
Word N3
Word N4
Word N5
Word N6
Word N7
Word N8
Word N9
Word N10
Word N11
Word N12
Word N13
Word N14
Word N15
Word N16
Word N17
Word N18
Word N19
Word N20
Word N21
Word N22
Word N23
Word N24
Word N25
Word N26
Word N27
Word N28
Word N29
Word N30
Word N31
End of measurement setting procedure

3

3.3 Servo-Control Settings

Declare the maximum speed on the axes

Values to be entered in P30 (See 6.2)

Word N0		Word N16	
Word N1		Word N17	
Word N2		Word N18	
Word N3		Word N19	
Word N4		Word N20	
Word N5		Word N21	
Word N6		Word N22	
Word N7		Word N23	
Word N8		Word N24	
Word N9		Word N25	
Word N10		Word N26	
Word N11		Word N27	
Word N12		Word N28	
Word N13		Word N29	
Word N14		Word N30	
Word N15		Word N31	

Declare the JOG speeds and reduced speeds

Values to be entered in P31 (See 6.3)

Slow JOG	Word N0	
Normal JOG	Word N1	
Rapid JOG	Word N2	
Reduced speed on linear axes	Word N3	
Reduced speed on rotary axes	Word N4	

Cont'd

Declare the maximum permissible accelerations

Values to be entered in P32 (See 6.4)

Work	Word N0		Work	Word N32	
Fast	Word N1		Fast	Word N33	
Work	Word N2		Work	Word N34	
Fast	Word N3		Fast	Word N35	
Work	Word N4		Work	Word N36	
Fast	Word N5		Fast	Word N37	
Work	Word N6		Work	Word N38	
Fast	Word N7		Fast	Word N39	
Work	Word N8		Work	Word N40	
Fast	Word N9		Fast	Word N41	
Work	Word N10		Work	Word N42	
Fast	Word N11		Fast	Word N43	
Work	Word N12		Work	Word N44	
Fast	Word N13		Fast	Word N45	
Work	Word N14		Work	Word N46	
Fast	Word N15		Fast	Word N47	
Work	Word N16		Work	Word N48	
Fast	Word N17		Fast	Word N49	
Work	Word N18		Work	Word N50	
Fast	Word N19		Fast	Word N51	
Work	Word N20		Work	Word N52	
Fast	Word N21		Fast	Word N53	
Work	Word N22		Work	Word N54	
Fast	Word N23		Fast	Word N55	
Work	Word N24		Work	Word N56	
Fast	Word N25		Fast	Word N57	
Work	Word N26		Work	Word N58	
Fast	Word N27		Fast	Word N59	
Work	Word N28		Work	Word N60	
Fast	Word N29		Fast	Word N61	
Work	Word N30		Work	Word N62	
Fast	Word N31		Fast	Word N63	

Declare the axis speed reference direction

Values to be entered in P20 (See 6.5)

Word N0	

Cont'd

Declare the servo-control coefficient

Values to be entered in P21 (See 6.6)

Word N0		Word N16	
Word N1		Word N17	
Word N2		Word N18	
Word N3		Word N19	
Word N4		Word N20	
Word N5		Word N21	
Word N6		Word N22	
Word N7		Word N23	
Word N8		Word N24	
Word N9		Word N25	
Word N10		Word N26	
Word N11		Word N27	
Word N12		Word N28	
Word N13		Word N29	
Word N14		Word N30	
Word N15		Word N31	

Declare the servo-loop time constant

Values to be entered in P56 (See 6.7)

Word N0	
Word N1	
Word N2	
Word N3	
Word N4	
Word N5	
Word N6	
Word N7	

Cont'd

Declare the in-position windows

Values to be entered in P22 (See 6.8)

Word N0		Word N16	
Word N1		Word N17	
Word N2		Word N18	
Word N3		Word N19	
Word N4		Word N20	
Word N5		Word N21	
Word N6		Word N22	
Word N7		Word N23	
Word N8		Word N24	
Word N9		Word N25	
Word N10		Word N26	
Word N11		Word N27	
Word N12		Word N28	
Word N13		Word N29	
Word N14		Word N30	
Word N15		Word N31	

Declare the maximum following errors

Values to be entered in P23 (See 6.9)

Word N0		Word N16	
Word N1		Word N17	
Word N2		Word N18	
Word N3		Word N19	
Word N4		Word N20	
Word N5		Word N21	
Word N6		Word N22	
Word N7		Word N23	
Word N8		Word N24	
Word N9		Word N25	
Word N10		Word N26	
Word N11		Word N27	
Word N12		Word N28	
Word N13		Word N29	
Word N14		Word N30	
Word N15		Word N31	

Cont’d

Declare dynamic movement control

Values to be entered in P57 (See 6.10)

Word N0		Word N16	
Word N1		Word N17	
Word N2		Word N18	
Word N3		Word N19	
Word N4		Word N20	
Word N5		Word N21	
Word N6		Word N22	
Word N7		Word N23	
Word N8		Word N24	
Word N9		Word N25	
Word N10		Word N26	
Word N11		Word N27	
Word N12		Word N28	
Word N13		Word N29	
Word N14		Word N30	
Word N15		Word N31	

Declare the speed anticipation coeffic ents and the number of terms for VHSM

Values to be entered in P55 (See 6.11)

Speed anticipation coefficient		Number of VHSM terms	
Word N0		Word N8	
Word N1		Word N9	
Word N2		Word N10	
Word N3		Word N11	
Word N4		Word N12	
Word N5		Word N13	
Word N6		Word N14	
Word N7		Word N15	

Cont’d

Declare the synchronised axis coupling characteristics

Values to be entered in P24 (See 6.12)

Threshold 1

Word N0	
Word N1	
Word N2	
Word N3	
Word N4	
Word N5	
Word N6	
Word N7	
Word N8	
Word N9	
Word N10	
Word N11	
Word N12	
Word N13	
Word N14	
Word N15	
Word N16	
Word N17	
Word N18	
Word N19	
Word N20	
Word N21	
Word N22	
Word N23	
Word N24	
Word N25	
Word N26	
Word N27	
Word N28	
Word N29	
Word N30	
Word N31	

Correction coefficient

Word N32	
Word N33	
Word N34	
Word N35	
Word N36	
Word N37	
Word N38	
Word N39	
Word N40	
Word N41	
Word N42	
Word N43	
Word N44	
Word N45	
Word N46	
Word N47	
Word N48	
Word N49	
Word N50	
Word N51	
Word N52	
Word N53	
Word N54	
Word N55	
Word N56	
Word N57	
Word N58	
Word N59	
Word N60	
Word N61	
Word N62	
Word N63	

Threshold 2

Word N64	

Define the tolerated servo-control error on circles

Values to be entered in P52 (See 6.13)

Word N0	

Cont'd

Very high speed machining

Values to be entered in P19 (See 6.14)

Acceleration anticipation in µs

Word N0		Word N16	
Word N1		Word N17	
Word N2		Word N18	
Word N3		Word N19	
Word N4		Word N20	
Word N5		Word N21	
Word N6		Word N22	
Word N7		Word N23	
Word N8		Word N24	
Word N9		Word N25	
Word N10		Word N26	
Word N11		Word N27	
Word N12		Word N28	
Word N13		Word N29	
Word N14		Word N30	
Word N15		Word N31	

Correction amplitude in internal units

Word N32		Word N48	
Word N33		Word N49	
Word N34		Word N50	
Word N35		Word N51	
Word N36		Word N52	
Word N37		Word N53	
Word N38		Word N54	
Word N39		Word N55	
Word N40		Word N56	
Word N41		Word N57	
Word N42		Word N58	
Word N43		Word N59	
Word N44		Word N60	
Word N45		Word N61	
Word N46		Word N62	
Word N47		Word N63	

Parameter continued on next page

Cont'd

Values to be entered in P19 (cont'd)

Time constant in hundredths of ms

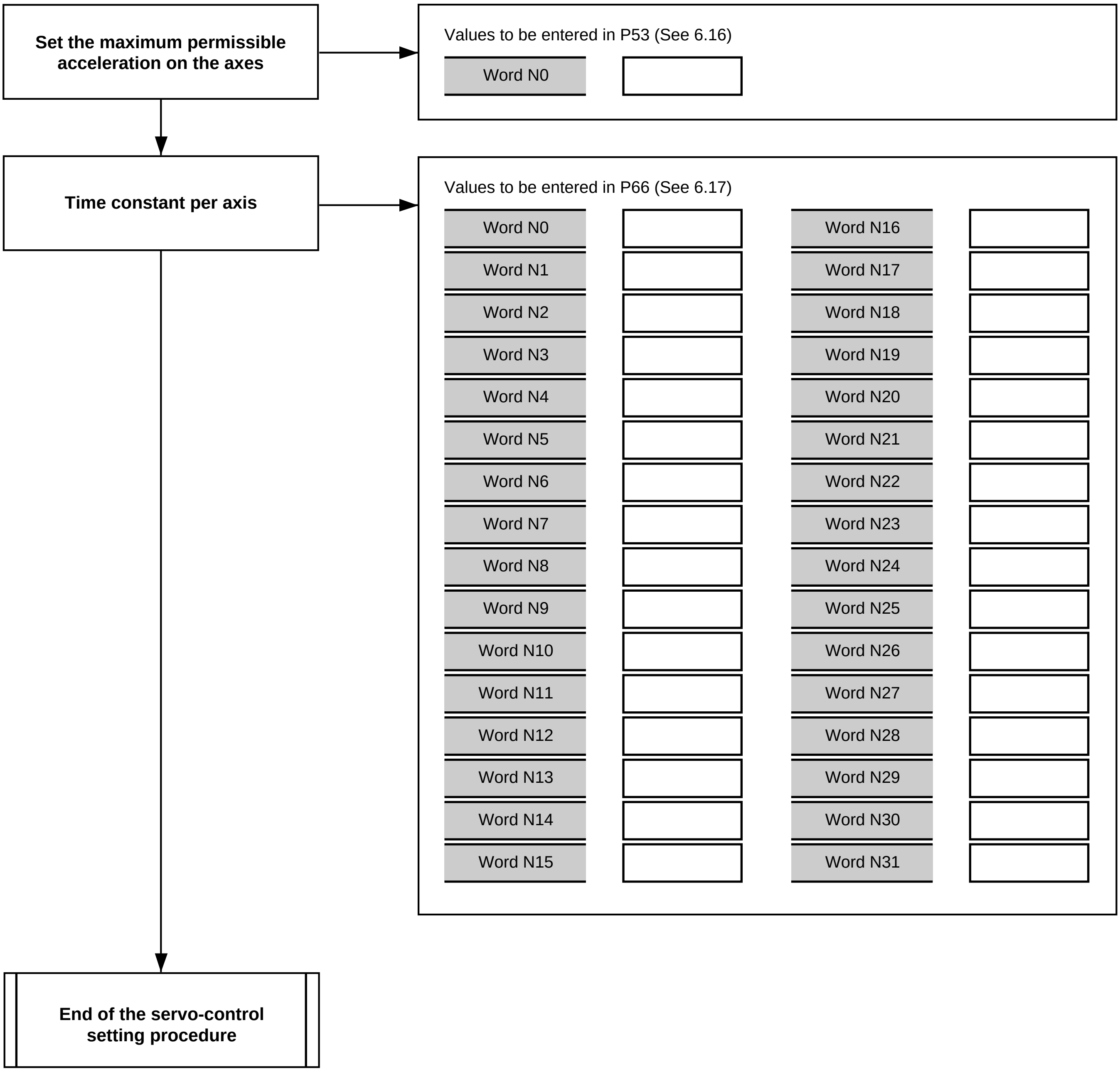
Word N64		Word N80	
Word N65		Word N81	
Word N66		Word N82	
Word N67		Word N83	
Word N68		Word N84	
Word N69		Word N85	
Word N70		Word N86	
Word N71		Word N87	
Word N72		Word N88	
Word N73		Word N89	
Word N74		Word N90	
Word N75		Word N91	
Word N76		Word N92	
Word N77		Word N93	
Word N78		Word N94	
Word N79		Word N95	

Approach speed

Values to be entered in P33 (See 6.15)

Word N0		Word N16	
Word N1		Word N17	
Word N2		Word N18	
Word N3		Word N19	
Word N4		Word N20	
Word N5		Word N21	
Word N6		Word N22	
Word N7		Word N23	
Word N8		Word N24	
Word N9		Word N25	
Word N10		Word N26	
Word N11		Word N27	
Word N12		Word N28	
Word N13		Word N29	
Word N14		Word N30	
Word N15		Word N31	

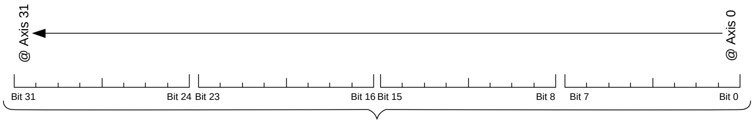
Cont'd


Set the maximum permissible acceleration on the axes
Values to be entered in P53 (See 6.16)
Word N0
Time constant per axis
Values to be entered in P66 (See 6.17)
Word N0
Word N1
Word N2
Word N3
Word N4
Word N5
Word N6
Word N7
Word N8
Word N9
Word N10
Word N11
Word N12
Word N13
Word N14
Word N15
Word N16
Word N17
Word N18
Word N19
Word N20
Word N21
Word N22
Word N23
Word N24
Word N25
Word N26
Word N27
Word N28
Word N29
Word N30
Word N31
End of the servo-control setting procedure

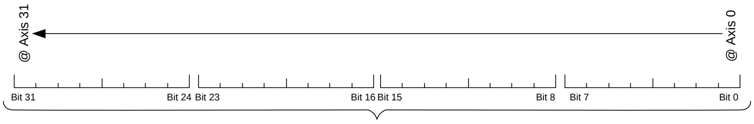
3.4 Setting of Travels

Declare the homing direction

Values to be entered in P15 (See 7.3)



Word N0	



Word N1	

Declare the switch positions in machine dimensions

Values to be entered in P16 (See 7.4)

Word N0		Word N16	
Word N1		Word N17	
Word N2		Word N18	
Word N3		Word N19	
Word N4		Word N20	
Word N5		Word N21	
Word N6		Word N22	
Word N7		Word N23	
Word N8		Word N24	
Word N9		Word N25	
Word N10		Word N26	
Word N11		Word N27	
Word N12		Word N28	
Word N13		Word N29	
Word N14		Word N30	
Word N15		Word N31	

Cont'd

Declare the axis travels

Values to be entered in P17 (See 7.5)

MIN	Word N0		MIN	Word N32	
MAX	Word N1		MAX	Word N33	
MIN	Word N2		MIN	Word N34	
MAX	Word N3		MAX	Word N35	
MIN	Word N4		MIN	Word N36	
MAX	Word N5		MAX	Word N37	
MIN	Word N6		MIN	Word N38	
MAX	Word N7		MAX	Word N39	
MIN	Word N8		MIN	Word N40	
MAX	Word N9		MAX	Word N41	
MIN	Word N10		MIN	Word N42	
MAX	Word N11		MAX	Word N43	
MIN	Word N12		MIN	Word N44	
MAX	Word N13		MAX	Word N45	
MIN	Word N14		MIN	Word N46	
MAX	Word N15		MAX	Word N47	
MIN	Word N16		MIN	Word N48	
MAX	Word N17		MAX	Word N49	
MIN	Word N18		MIN	Word N50	
MAX	Word N19		MAX	Word N51	
MIN	Word N20		MIN	Word N52	
MAX	Word N21		MAX	Word N53	
MIN	Word N22		MIN	Word N54	
MAX	Word N23		MAX	Word N55	
MIN	Word N24		MIN	Word N56	
MAX	Word N25		MAX	Word N57	
MIN	Word N26		MIN	Word N58	
MAX	Word N27		MAX	Word N59	
MIN	Word N28		MIN	Word N60	
MAX	Word N29		MAX	Word N61	
MIN	Word N30		MIN	Word N62	
MAX	Word N31		MAX	Word N63	

Cont'd

Declare the backlash error compensation

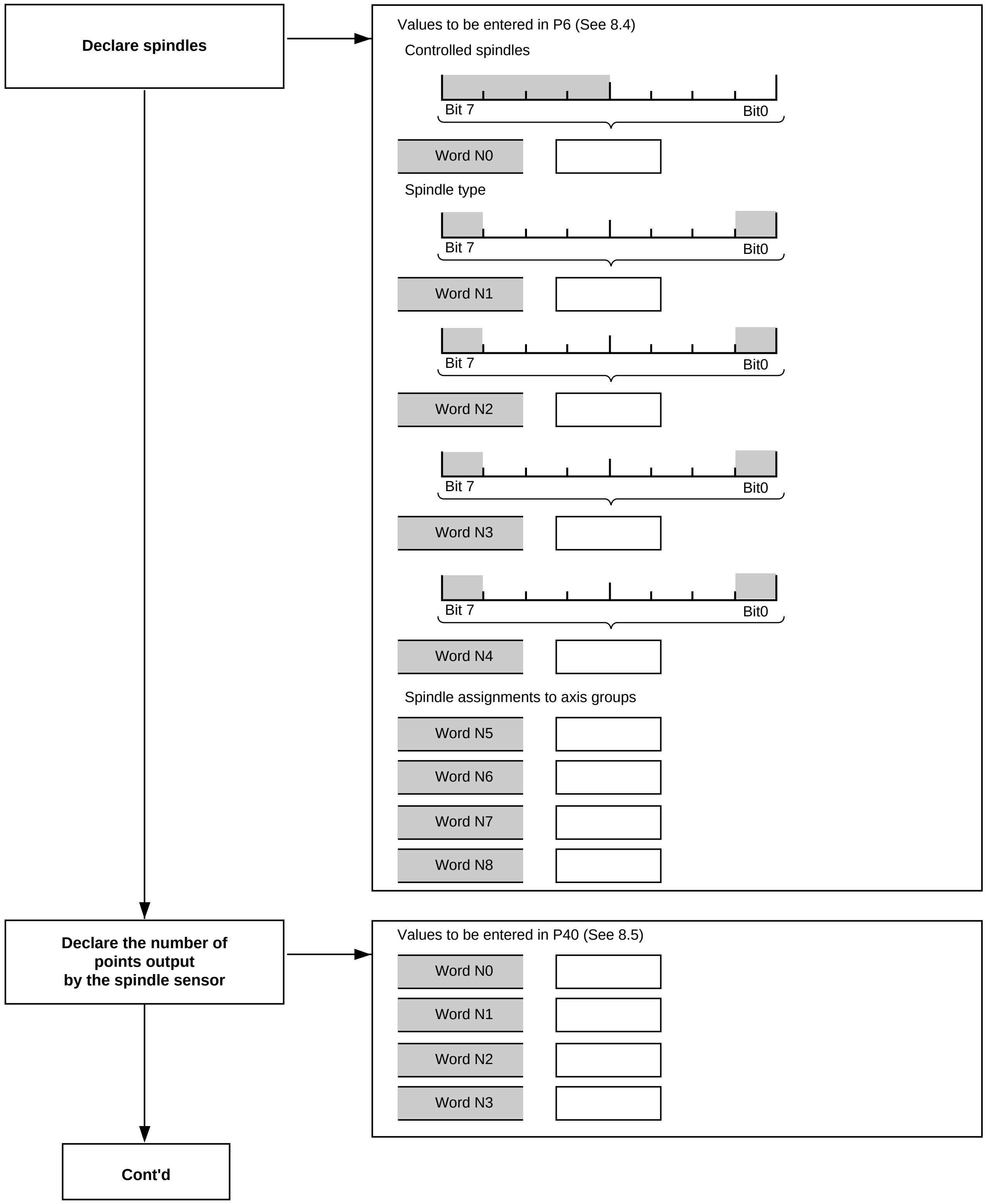
Values to be entered in P18 (See 7.6)

Word N0		Word N16	
Word N1		Word N17	
Word N2		Word N18	
Word N3		Word N19	
Word N4		Word N20	
Word N5		Word N21	
Word N6		Word N22	
Word N7		Word N23	
Word N8		Word N24	
Word N9		Word N25	
Word N10		Word N26	
Word N11		Word N27	
Word N12		Word N28	
Word N13		Word N29	
Word N14		Word N30	
Word N15		Word N31	

End of travel setting procedure

3.5 Spindle Settings

3.5.1 Spindle Definitions



Declare the spindle reference direction

Values to be entered in P41 (See 8.6)

Bit 7 Bit0

Word N0

Bit 7 Bit0

Word N1

Bit 7 Bit0

Word N2

Bit 7 Bit0

Word N3

Define the spindle 1 speed range

Values to be entered in P46 (See 8.7)

M40	MIN	Word N0		MAX	Word N1		Limit speed	Word N12
M41	MIN	Word N2		MAX	Word N3		Limit speed	Word N13
M42	MIN	Word N4		MAX	Word N5		Limit speed	Word N14
M43	MIN	Word N6		MAX	Word N7		Limit speed	Word N15
M44	MIN	Word N8		MAX	Word N9		Limit speed	Word N16
M45	MIN	Word N10		MAX	Word N11		Limit speed	Word N17

Define the spindle 2 speed range

Values to be entered in P47 (See 8.8)

M40	MIN	Word N0		MAX	Word N1		Limit speed	Word N12
M41	MIN	Word N2		MAX	Word N3		Limit speed	Word N13
M42	MIN	Word N4		MAX	Word N5		Limit speed	Word N14
M43	MIN	Word N6		MAX	Word N7		Limit speed	Word N15
M44	MIN	Word N8		MAX	Word N9		Limit speed	Word N16
M45	MIN	Word N10		MAX	Word N11		Limit speed	Word N17

Cont'd

Define the spindle 3 speed range

Values to be entered in P48 (See 8.9)

M40	MIN	Word N0		MAX	Word N1		Limit speed	Word N12	
M41	MIN	Word N2		MAX	Word N3		Limit speed	Word N13	
M42	MIN	Word N4		MAX	Word N5		Limit speed	Word N14	
M43	MIN	Word N6		MAX	Word N7		Limit speed	Word N15	
M44	MIN	Word N8		MAX	Word N9		Limit speed	Word N16	
M45	MIN	Word N10		MAX	Word N11		Limit speed	Word N17	

Define the spindle 4 speed range

Values to be entered in P49 (See 8.10)

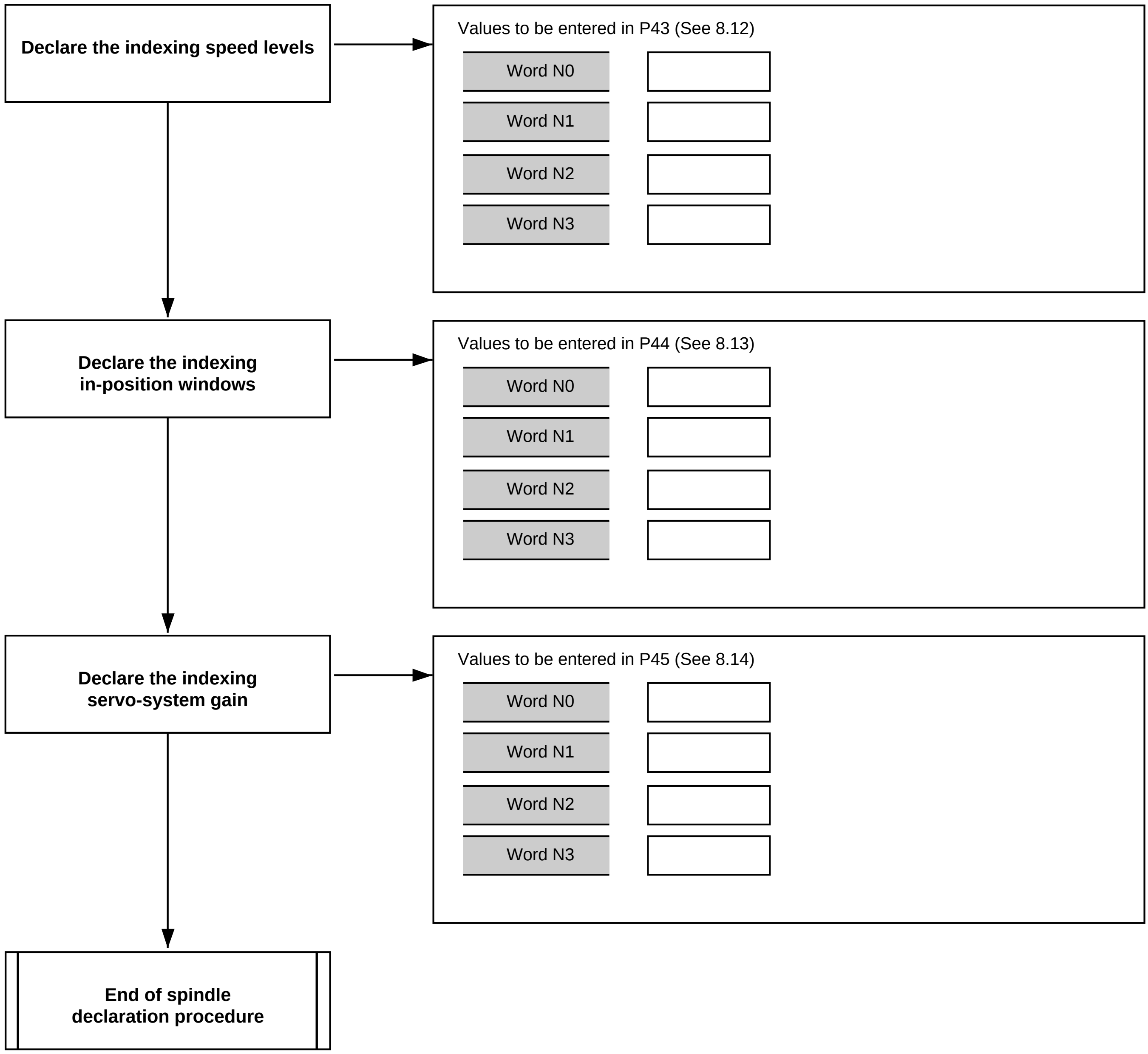
M40	MIN	Word N0		MAX	Word N1		Limit speed	Word N12	
M41	MIN	Word N2		MAX	Word N3		Limit speed	Word N13	
M42	MIN	Word N4		MAX	Word N5		Limit speed	Word N14	
M43	MIN	Word N6		MAX	Word N7		Limit speed	Word N15	
M44	MIN	Word N8		MAX	Word N9		Limit speed	Word N16	
M45	MIN	Word N10		MAX	Word N11		Limit speed	Word N17	

Declare the spindle origin

Values to be entered in P42 (See 8.11)

Word N0	
Word N1	
Word N2	
Word N3	

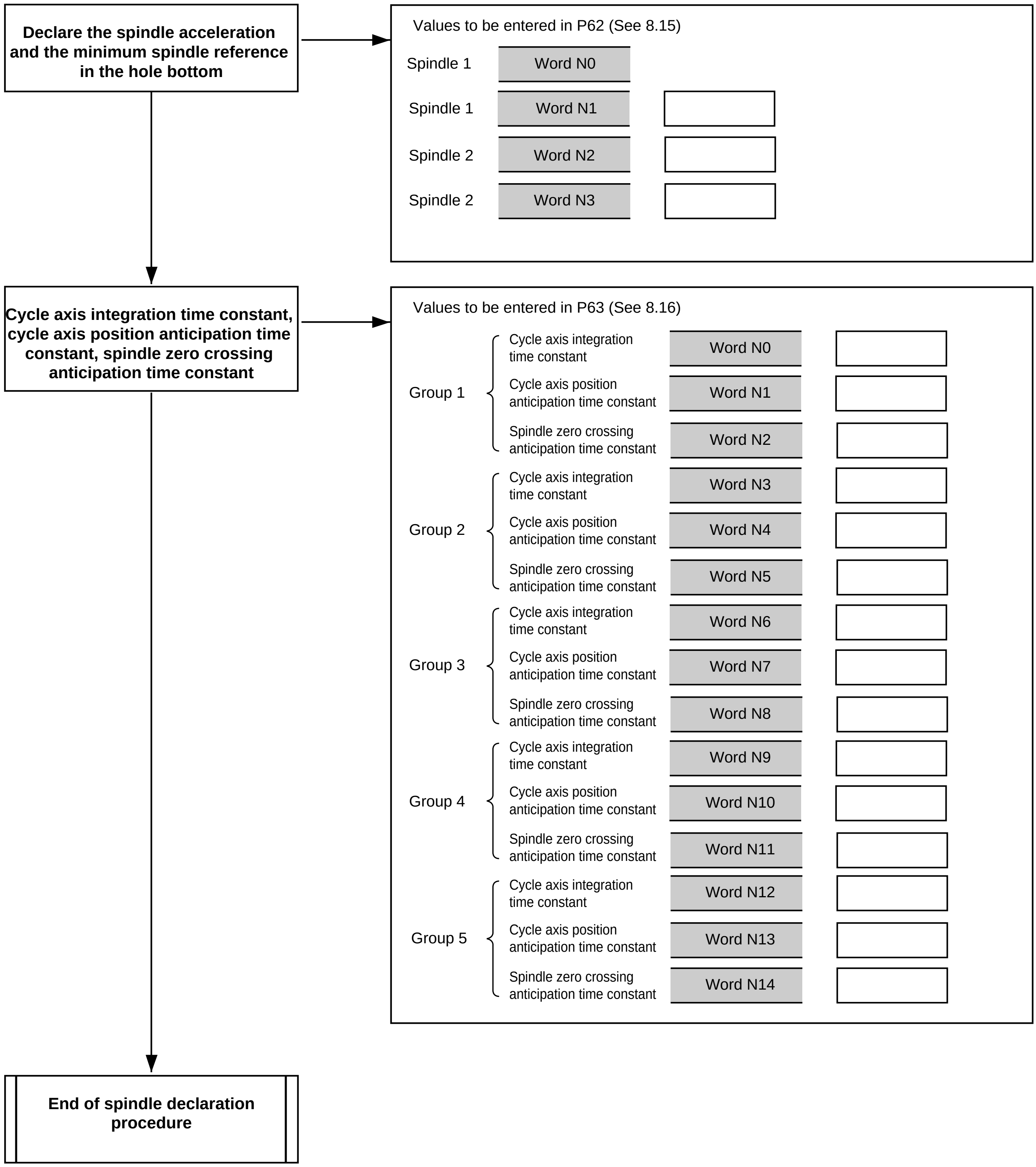
Cont'd



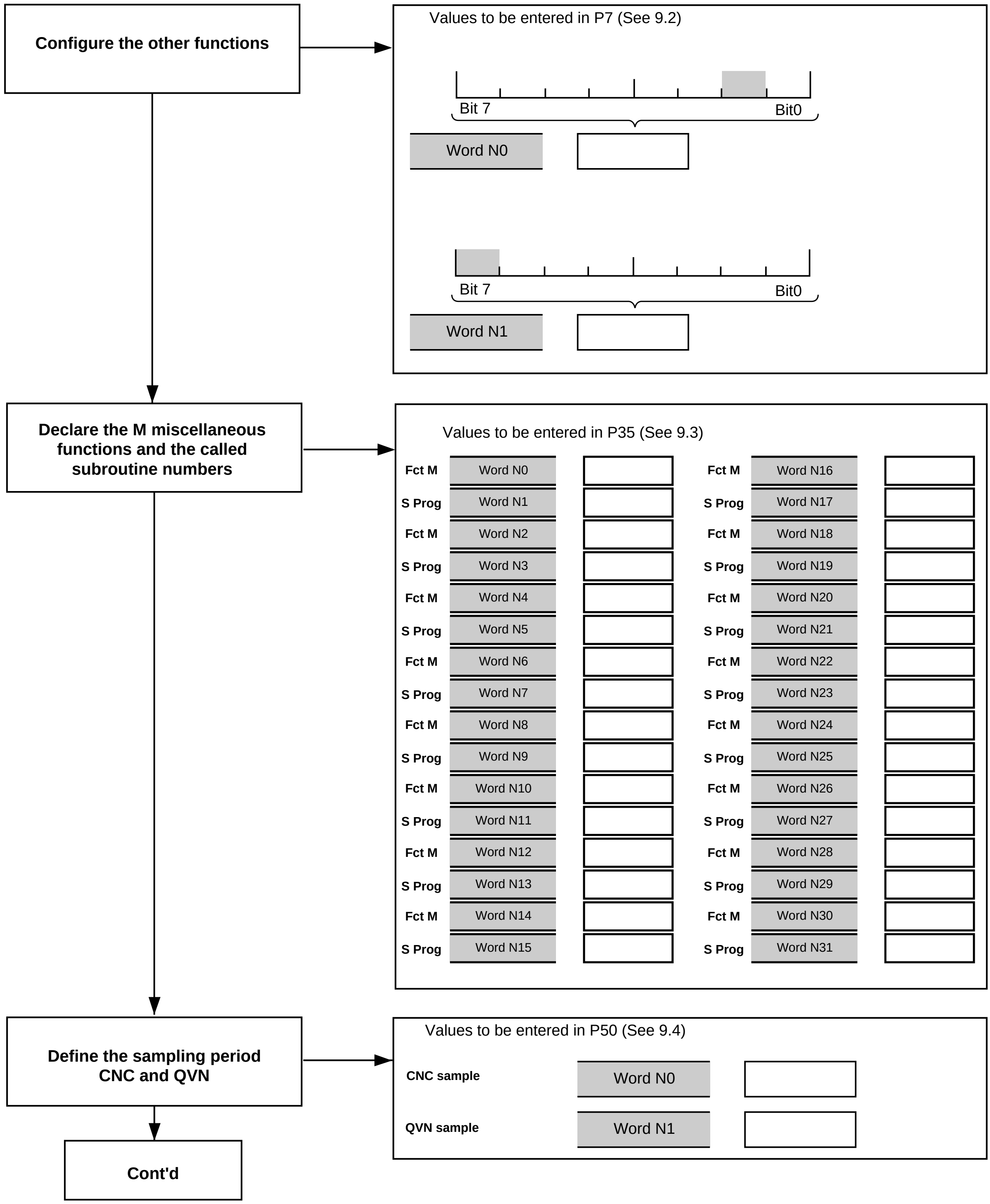
REMARK: For rigid tapping, refer to the special parameter declarations on the next page.

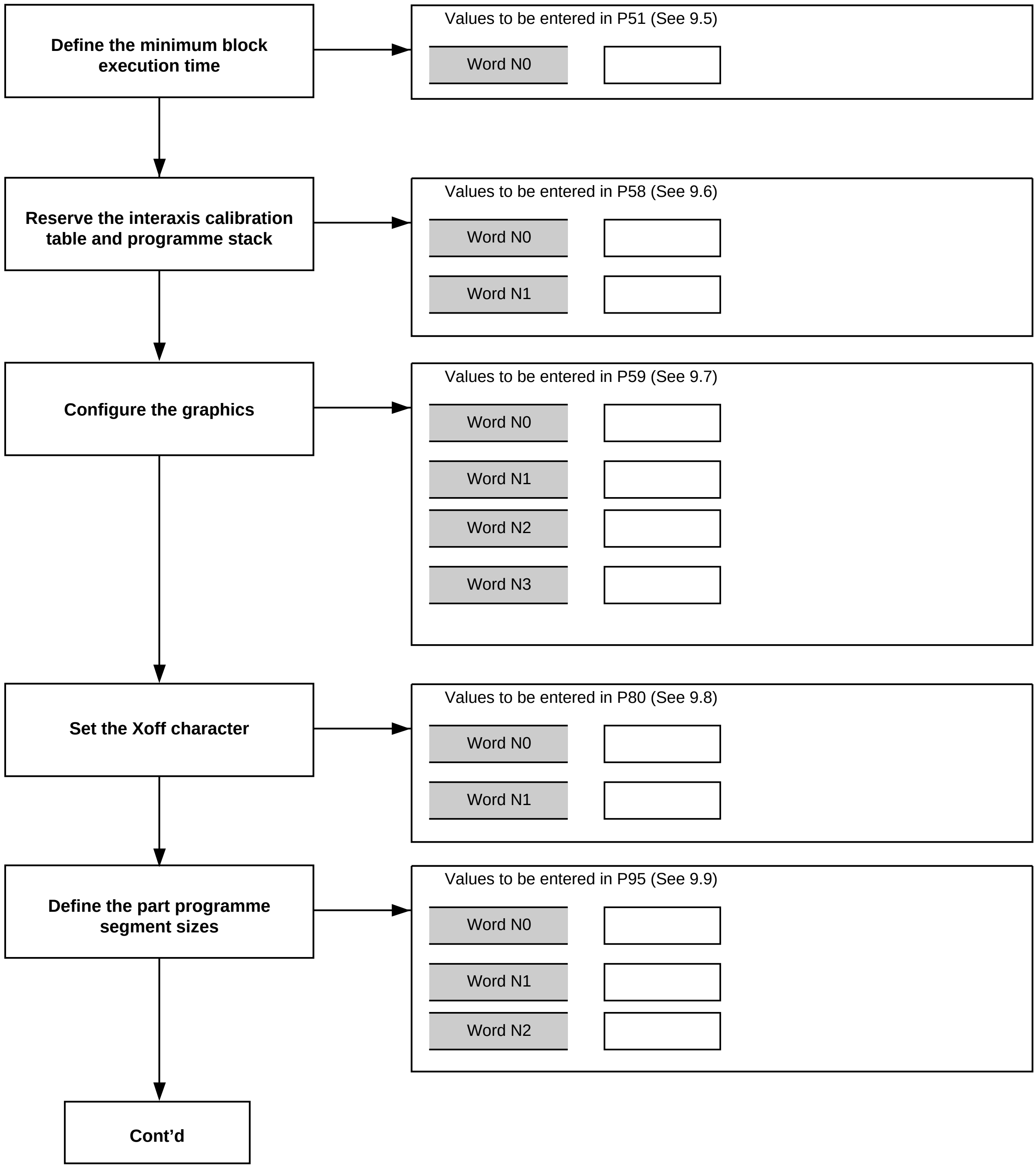
3

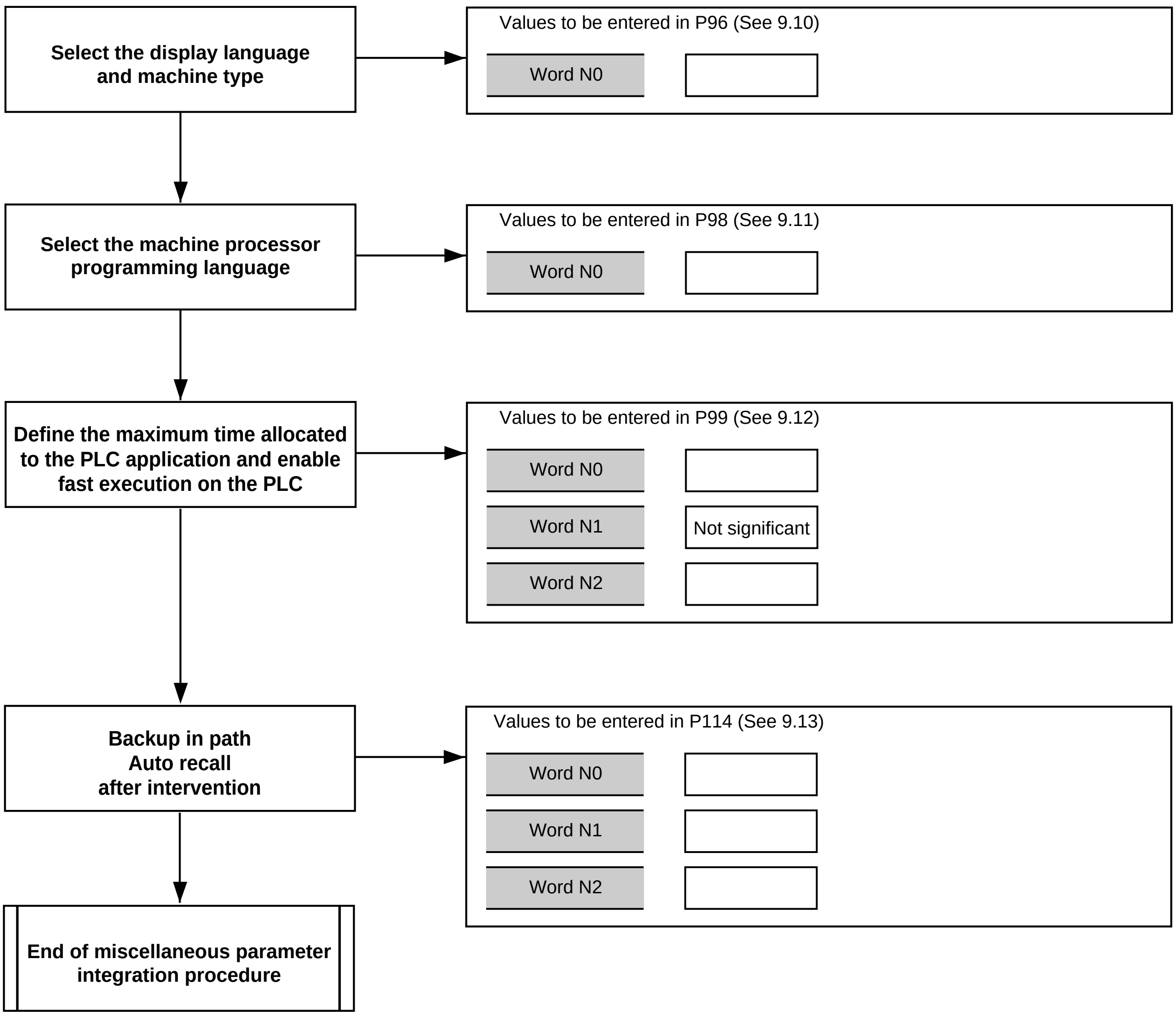
3.5.2 Rigid Tapping



3.6 Miscellaneous Parameters




Define the minimum block execution time
Values to be entered in P51 (See 9.5)
Word N0
Reserve the interaxis calibration table and programme stack
Values to be entered in P58 (See 9.6)
Word N0
Word N1
Configure the graphics
Values to be entered in P59 (See 9.7)
Word N0
Word N1
Word N2
Word N3
Set the Xoff character
Values to be entered in P80 (See 9.8)
Word N0
Word N1
Define the part programme segment sizes
Values to be entered in P95 (See 9.9)
Word N0
Word N1
Word N2
Cont'd


Select the display language and machine type
Values to be entered in P96 (See 9.10)
Word N0
Select the machine processor programming language
Values to be entered in P98 (See 9.11)
Word N0
Define the maximum time allocated to the PLC application and enable fast execution on the PLC
Values to be entered in P99 (See 9.12)
Word N0
Word N1
Not significant
Word N2
Backup in path
Auto recall
after intervention
Values to be entered in P114 (See 9.13)
Word N0
Word N1
Word N2
End of miscellaneous parameter integration procedure

3.7 Communication Parameter Settings

Set the DNC1 line and service character parameters

Values to be entered in P37 (See 10.1)

Word N0		Word N5	
Word N1		Word N6	
Word N2		Word N7	
Word N3		Word N8	
Word N4			

Define control character conversion to DNC1

Values to be entered in P38 (See 10.2)

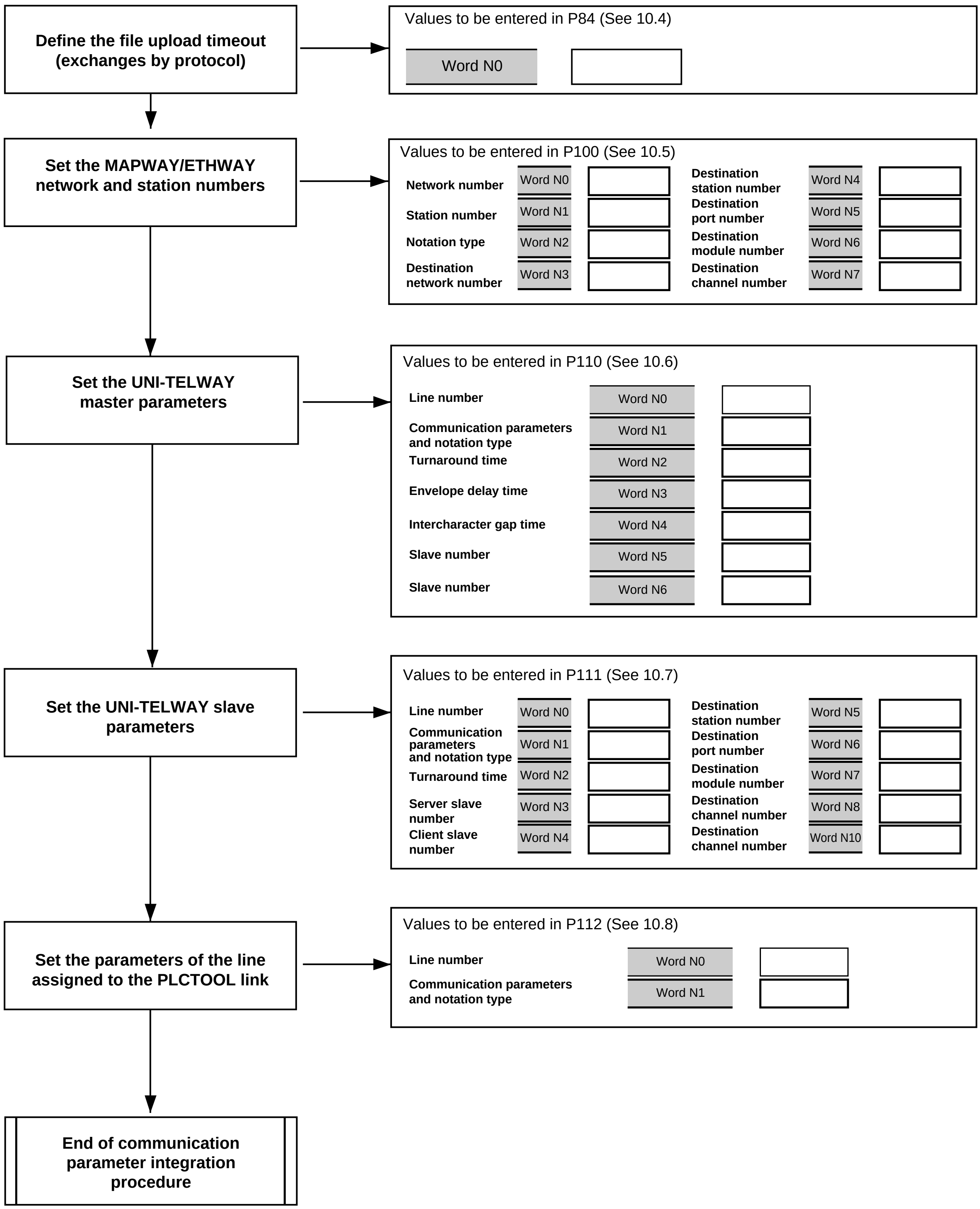
Control character sent by the CNC keyboard	Word N0		Word N10	
4-character sequence sent to the computer	Word N1		Word N11	
Control character sent by the CNC keyboard	Word N2		Word N12	
4-character sequence sent to the computer	Word N3		Word N13	
Control character sent by the CNC keyboard	Word N4		Word N14	
4-character sequence sent to the computer	Word N5		Word N15	
Control character sent by the CNC keyboard	Word N6		Word N16	
4-character sequence sent to the computer	Word N7		Word N17	
Control character sent by the CNC keyboard	Word N8		Word N18	
4-character sequence sent to the computer	Word N9		Word N19	
Computer control character code	Word N20		Word N30	
Translation of the above code sent to the CNC screen	Word N21		Word N11	
Computer control character code	Word N22		Word N32	
Translation of the above code sent to the CNC screen	Word N23		Word N33	
Computer control character code	Word N24		Word N34	
Translation of the above code sent to the CNC screen	Word N25		Word N35	
Computer control character code	Word N26		Word N36	
Translation of the above code sent to the CNC screen	Word N27		Word N37	
Computer control character code	Word N28		Word N38	
Translation of the above code sent to the CNC screen	Word N29		Word N39	

Define the DNC1 timeout

Values to be entered in P39 (See 10.3)

Word N0	
Word N1	
Word N2	Not significant

Cont'd



Define the file upload timeout (exchanges by protocol)
Values to be entered in P84 (See 10.4)
Word N0
Set the MAPWAY/ETHWAY network and station numbers
Values to be entered in P100 (See 10.5)
Network number Word N0
Station number Word N1
Notation type Word N2
Destination network number Word N3
Destination station number Word N4
Destination port number Word N5
Destination module number Word N6
Destination channel number Word N7
Set the UNI-TELWAY master parameters
Values to be entered in P110 (See 10.6)
Line number Word N0
Communication parameters and notation type Word N1
Turnaround time Word N2
Envelope delay time Word N3
Intercharacter gap time Word N4
Slave number Word N5
Slave number Word N6
Set the UNI-TELWAY slave parameters
Values to be entered in P111 (See 10.7)
Line number Word N0
Communication parameters and notation type Word N1
Turnaround time Word N2
Server slave number Word N3
Client slave number Word N4
Destination station number Word N5
Destination port number Word N6
Destination module number Word N7
Destination channel number Word N8
Destination channel number Word N10
Set the parameters of the line assigned to the PLCTOOL link
Values to be entered in P112 (See 10.8)
Line number Word N0
Communication parameters and notation type Word N1
End of communication parameter integration procedure

3.8 DISC Axis Parameter Settings

See DISC Integration Manual 938907.

4 Axis Declaration

4.1 Data Tables

Group Assignment

Axis group No.	1	2	3	4	5	6	7	8
CNC axis group								
PLC axis group								
Milling machine axis group								
Lathe axis group								
Main front turret								
Main rear turret								
Secondary turret								

Axis Assignment to the Groups

	Main linear axes			Secondary linear axes			Rotary axes		
Axis name	X	Y	Z	U	V	W	A	B	C
Group 1									
Group 2									
Group 3									
Group 4									
Group 5									
Group 6									
Group 7									
Group 8									

Definition of the Axes in Group

	Main linear axes			Secondary linear axes			Rotary axes		
Axis name	X	Y	Z	U	V	W	A	B	C
Physical address (@ 0-31)									
Measured axis									
Displayable axis									
Servo-controlled and interpolated axis									
Axis programmed by diameter									
Axis with clamps									
Modulo axis									
Axis with limited excursion									
Axis with quantified movement									
Carrier axis									
Carried axis									
Coupled (driven) axis									

Handwheel Definition

	Handwheel 1	Handwheel 2	Handwheel 3	Handwheel 4
Handwheel present				
Physical handwheel address (imposed address)	@28	@29	@30	@31

4.2 Number of Axis Groups in the System

P97

Category	Axis declaration
Type 0	8-bit hexadecimal
No. of words	2

Description

Defines the maximum number of axis groups controlled by the system.

Principle

The system can control a maximum of eight axis groups. There two types of axis groups:
- CNC axis groups,
- PLC axis groups.

Word N0 specifies the number of CNC axis groups (1 to 8).

Word N1 specifies the number of PLC axis groups (0 to 7).

Total number of groups: $N0 + N1 \leq 8$ for 1060 Series 1
Total number of groups: $N0 + N1 \leq 3$ for 1060 standard Series 2 and UCSII
Total number of groups: $N0 + N1 \leq 3$ for 1040
Total number of groups: $N0 + N1 \leq 4$ for 1040 GP

CAUTION

1060 systems equipped with a PLC function programmable in assembler language are limited to five axis groups.

No. of groups controlled	Value in N0 or N1
One group	1
Two groups	2
Three groups	3
Four groups	4
Five groups	5
Six groups	6
Seven groups	7
Eight groups	8

On the number of groups declared, the system chronologically assigns the CNC groups starting with the first axis group (group 1) then the PLC axis groups after the last CNC axis group.

Programme Parameter (see «Part Programming Manual»).

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

Example

The system controls three CNC axis groups and two PLC axis groups.

Word N0	03
Word N1	02

The groups are assigned as follows:

Group number	CNC	PLC
Group 1	X	
Group 2	X	
Group 3	X	
Group 4		X
Group 5		X
Group 6	Unassigned	
Group 7	Unassigned	
Group 8	Unassigned	

4.3 Measured Axes

P2

Category	Axis declaration
Type 6	32-bit hexadecimal
No. of words	1

Description

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

Principle

The bit position gives the physical axis address.

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

REMARK: *If the axes declared in P2 are not detected as present by the system on the bus, the message «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.

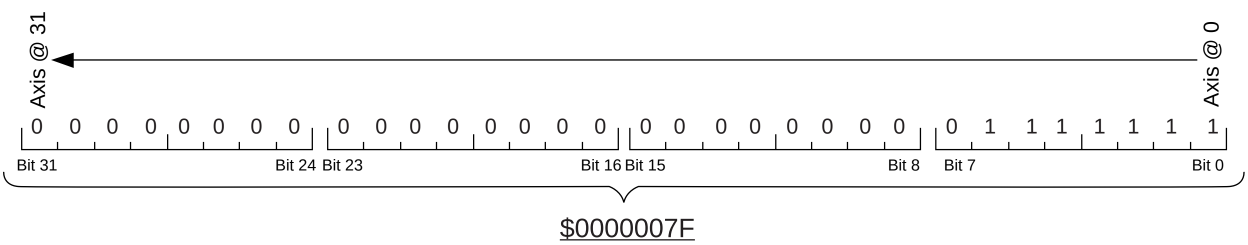
Programme Parameters (see «Part Programming Manual»).

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

4.4 Displayed Axes

P0

Category	Axis declaration
Type 6	32-bit hexadecimal
No. of words	1

Description

Defines the axes displayed on the INFO and AXIS pages.

Principle

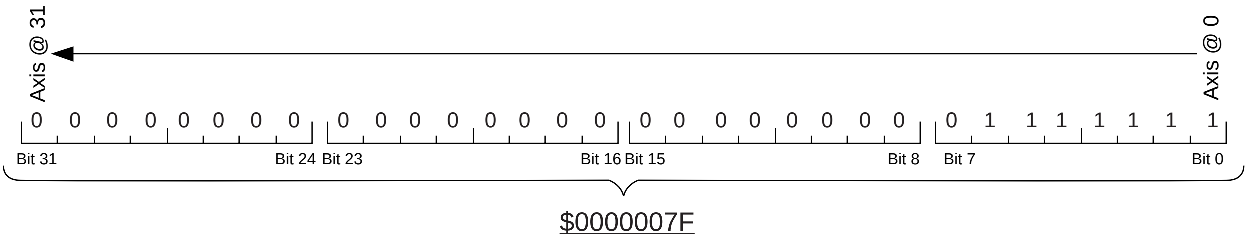
The bit position gives the physical axis address.

The bit is a 1 to declare the axis as displayable if it is assigned to a group, i.e. declared in P9.

REMARK: If the axis is declared in P2 or P3, it is displayed on the INFO and AXIS pages. If it is not declared in P2 or P3, it is only displayed on the INFO page.

Example

The axes at addresses 0 to 6 are displayable.



Word N0	0000007F

4.5 Modulo or Limited Excursion Rotary Axes

P1

Category	Axis declaration
Type 6	32-bit hexadecimal
No. of words	2

Description

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

Principle

The bit position gives the physical axis address.

<u>Word N0: Modulo 360 degrees rotary axes.</u>

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.

<u>Word N1: Limited excursion rotary axes.</u>

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.

> ⚠ **CAUTION**
>
> Only rotary axes (modulo or limited excursion) with axis names A, B or C can be assigned to a programmed group.

REMARKS: Axes declared in both lists cannot be programmed in inches and cannot be assigned a scaling factor.

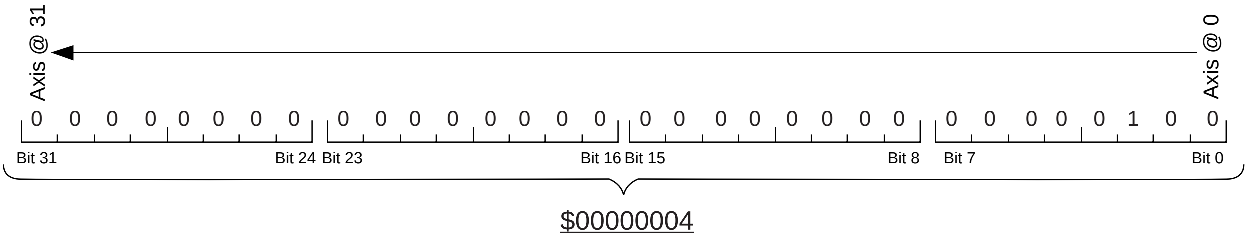
If an axis is declared modulo and with limited excursion, it is considered modulo by the system.

Programme Parameters (see «Part Programming Manual»).

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

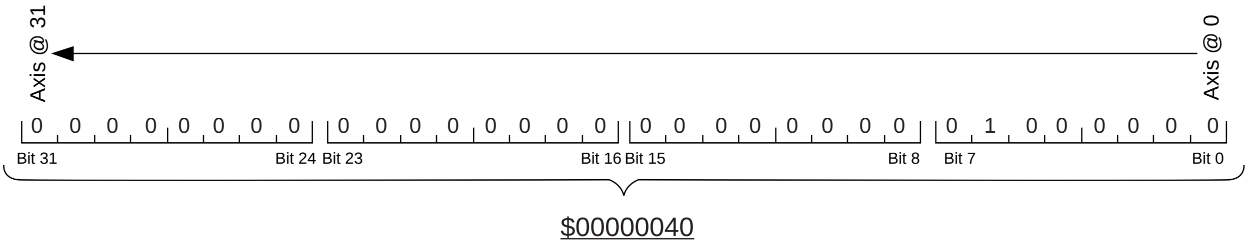
Examples

The axis with address 2 is modulo 360 degrees.



Word N0	00000004

The axis with address 6 has limited excursion.



Word N1	00000040

4.6 Servo-Controlled and Interpolated Axes

P3

Category	Axis declaration
Type 6	32-bit hexadecimal
No. of words	1

Description

Used to declare servo-controlled and interpolated axes.

Principle

The bit position gives the physical axis address.

REMARK: The system manages servo-control of the axes declared both in P3 and P2.

Programme Parameter (see «Part Programming Manual»).

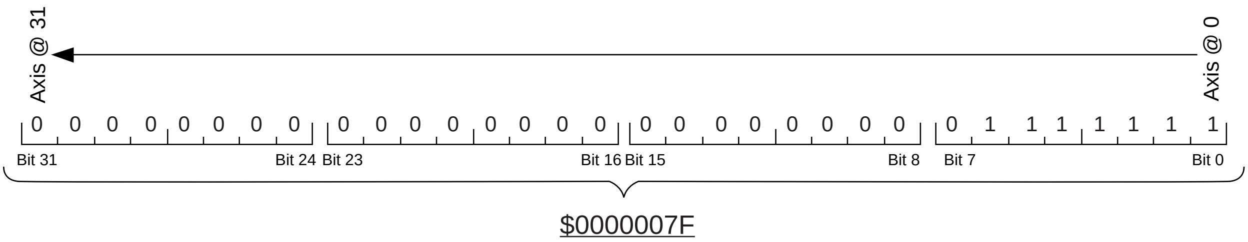
Parameter E910xx is used to read of change the state of these axes. A reset returns the CNC to the state specified by P3.

UNI-TE request (see UNI-TE Protocol User's Manual)

The state of the axes can be modified by a UNI-TE 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.



Word N0	0000007F

4.7 Axis Group Machine Configuration

P5

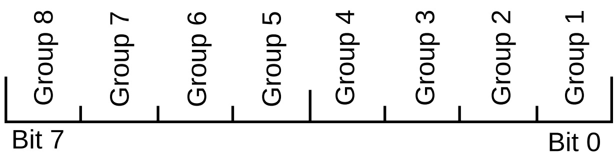
Category	Axis declaration
Type 0	8-bit hexadecimal
No. of words	2

Description

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

Principle

Each bit represents the address of an axis group.



Word N0: configuration of the initial programming functions and definition of the graphic display types.

A bit is a 0 to define a lathe group or a 1 to define a milling machine group.

Word N1: milling machine group assignment.

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

Programme Parameter (see «Part Programming Manual»).

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.

Group	**1**	**2**	**3**
N0	Lathe	Lathe	Milling machine
N1	Rear turret	Front turret	

		Bit 7							Bit 0	
Word N0	04	0	0	0	0	0	1	0	0	$04
Word N1	02	0	0	0	0	0	0	1	0	$02

4.8 Axes Programmable by Diameter and Internal Measurement **P4**

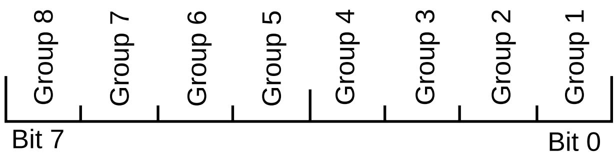
Category	Axis declaration
Type 0	8-bit hexadecimal
No. of words	5

Description

Used to:
- declare the X and U axes of each group as programmable by diameter,
- define the internal system measurement unit,
- specify that control of the X and/or U axes of each group 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 an axis group.



Word N0: characterises the X axes of the different axis groups.

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

Word N1: characterises the U axes of the different axis groups.

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

Word N2: defines the internal system measurement units.

The value entered sets the number of digits after the decimal point. The basic unit is the mm. The internal measurement units directly affect the machine travels.

The internal system measurement is defined by the following significant values:

Value of N2	Description	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 three decimal digits, i.e. µm as internal unit.

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

For manual control of the X axis of a group to be referenced to the diameter, the bit whose number corresponds to the group must be set in word 0 and word 3. For the U axis, it must be set in word 1 and word 3.

REMARK: Since the axis measurement unit is referenced to the radius, if the control increment is in this unit, the control is taken into account on the axis every two increments.

Word N4: specifies the divider for rotary axes.

This word is only valid for the CNC software at index LA or above.

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.

A value of 1 specifies the axes in degrees/10.

A value of 2 specifies the axes in degrees/100.

A value of 3 specifies the axes in degrees/1000.

A value of 4 specifies the axes in degrees/10000.

A value of 0 is the same as a value of 4 (degrees/10000).

Programme Parameters (see «Part Programming Manual»).

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.

Example

The X axes of the three groups and the U axes of groups 1 and 3 are programmed by diameter. The internal measurement is expressed in µm.

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

Word N0	07	0 0 0 0 0 1 1 1 (Bit 7 … Bit 0)	\$07
Word N1	05	0 0 0 0 0 1 0 1 (Bit 7 … Bit 0)	\$05
Word N2	03		
Word N3	02	0 0 0 0 0 0 1 0 (Bit 7 … Bit 0)	\$02
Word N4	04		

4.9 Axes with Clamps

P8

Category	Axis declaration
Type 6	32-bit hexadecimal
No. of words	1

Description

Used to declare the machine axes with clamps.

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

Principle

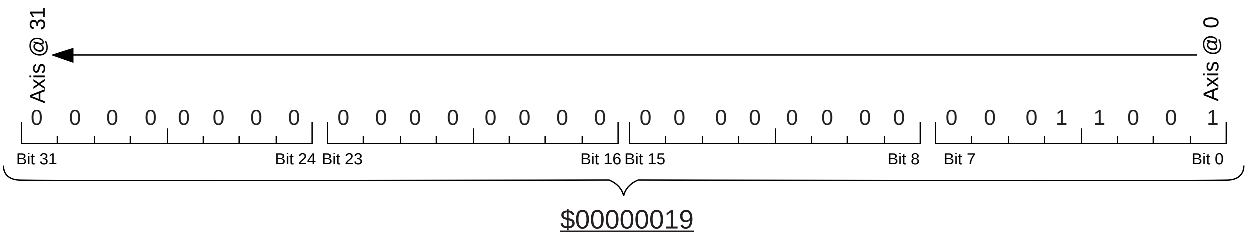
The bit position gives the physical axis address.

The bit is a 1 to declare the axis as clamped.

Example

Of the seven axes of this configuration, the axes at addresses 0, 3 and 4 are clamped.

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



Word N0	00000019

Programme Parameter (see Part Programming Manual)

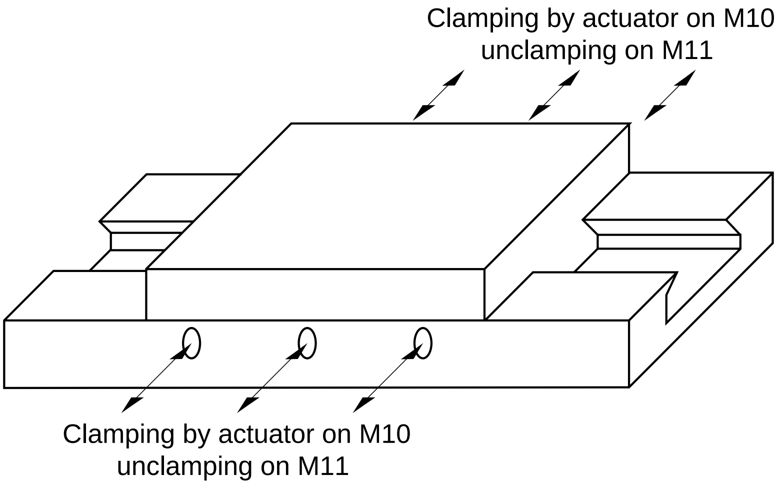
Programme parameter E9130x (x = physical axis number) is used to declare that an axis can be clamped or not. When this parameter is reset, the axes are clamped or unclamped according to the setting of machine parameter P8.

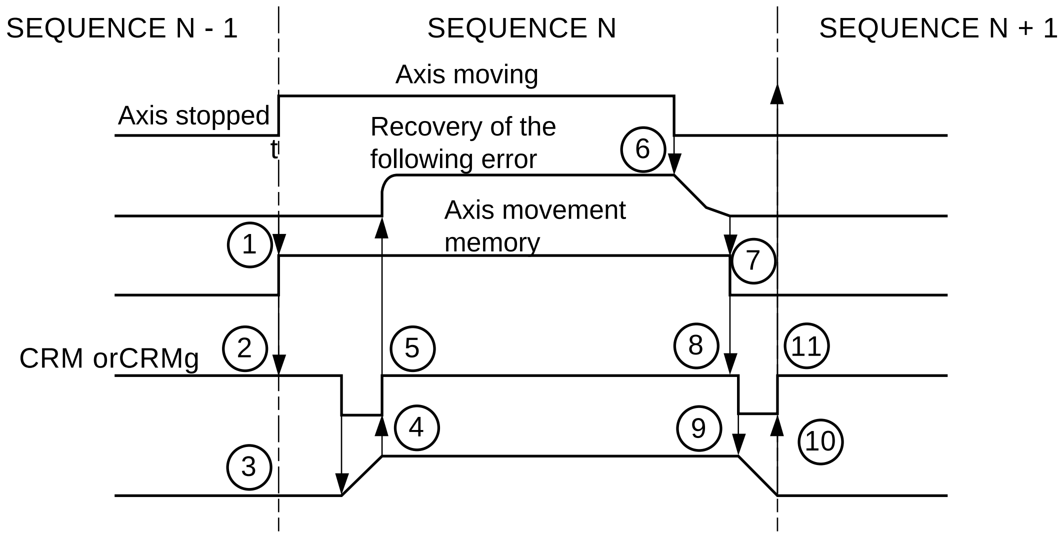
E9130x = 1 axis x with clamp

E9130x = 0 axis x without clamp.

Functional Diagram

The CNC processor requires this information since the CNC reference is not provided until the machine processor has acknowledged an end of unclamp report (CRM or CRMg (where g is the axis group number from 1 to 8)).





1 Internal timeout

2 Wait (for CRM or CRMg)

3 Axis unclamping (CRM or CRMg = 0)

4 CRM or CRMg = 1

5 Movement

6 Deceleration

7 End of movement

8 Wait (for CRM or CRMg)

9 Axis clamping

10 CRM ou CRMg = 1

11 Programme continued (unprogrammed clamped axis)

4.10 Axis Assignment to a Group

P9

Category	Axis declaration
Type 0	8-bit hexadecimal
No. of words	32

Description

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

Principle

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

The system can control three different types of axes (defined by customisation):
- CNC axes assigned to a CNC axis group,
- CNC axes assigned to a PLC axis group (axis assignment possible between CNC group and PLC group),
- PLC axes assigned to a PLC axis group (axis assignment possible between PLC groups only).

The physical axis addresses must be assigned in increasing order to the three types of axes:
- lowest addresses for the axes in the CNC axis groups,
- CNC axes assigned to PLC groups,
- the highest addresses for the axes in the PLC axis groups.

REMARKS: If the total number of axes and spindles defined in this parameter is higher than that set during 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.
If the number of axes assigned to the PLC axis groups is less than the number of PLC axes set during customisation, the message «WRONG NUMBER OF PLC AXES» is displayed at initialisation. The AUTO, SINGLE, DRYRUN and MDI modes are inhibited.

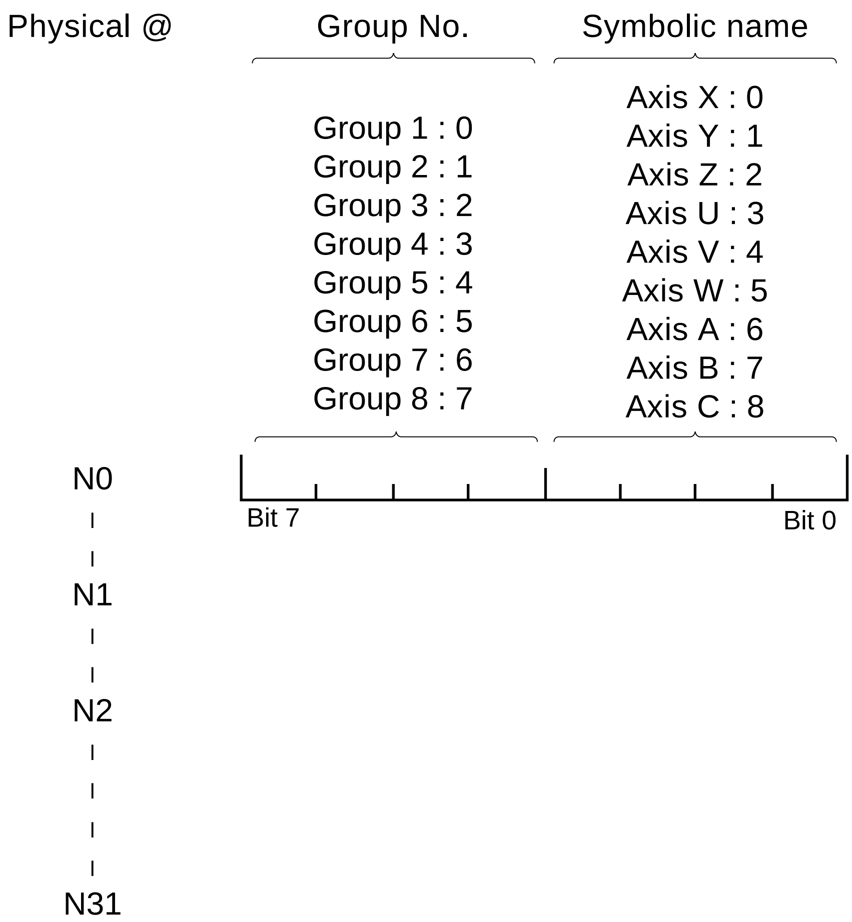
The word number gives the physical axis address. Each word includes two representative values.

Programme Parameter (see «Part Programming Manual»).

Programme parameter E7x005 is used to assign a CNC axis to a PLC axis group 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 group to which the axis belongs.



REMARK: *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 group.

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

- For T*plus* and M*plus,* parameter P9 must always be set as follows:
  T*plus*: N0 = 00 (X axis); N1 = 02 (Z axis) or N2 = 02 (Z axis)
  M*plus*: N0 = 00 (X axis); N1 = 01 (Y axis); N2 = 02 (Z axis)

Example

Machine configuration corresponding to the table below (two groups of two CNC axes, one group of three PLC axes). The customisation defines two PLC-only axes.

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

Word N0	00

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

Word N1	02

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

Word N2	10

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

Word N3	15

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

Word N4	23

The axis at address 4 (CNC axis assigned to the PLC) is the U axis of group 3.

Word N5	25

The axis at address 5 (PLC axis) is the W axis of group 3.

Word N6	28

The axis at address 6 (PLC axis) is the C axis of group 3.

Word N7	FF

The axes at address 7 and on do not have a symbolic name and do not belong to any group.

4.11 Handwheel Declaration

P14

Category	Axis declaration
Type 0	8-bit hexadecimal
No. of words	2

Description

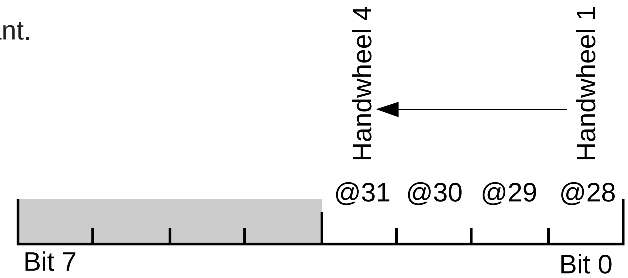
Used to declare handwheels and filter the handwheel measurement.

Principle

Word N0: Handwheel declaration

The bit position gives the physical handwheel address.

Only the four LSBs are significant.



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

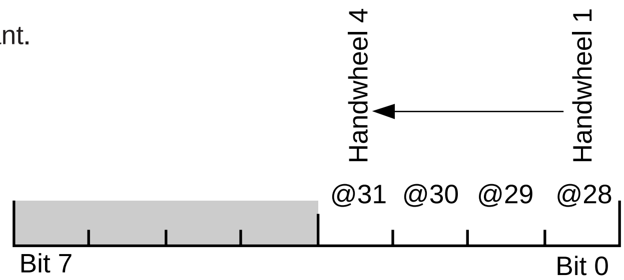
REMARK: If an encoder is assigned both to a handwheel (by P14) and to an axis (by P2), it is assigned by priority to the handwheel.

Word N1: 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.

Only the four LSBs are significant.

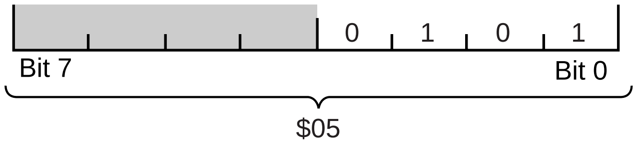


When the bit is a 1, filtering is enabled on the handwheel addressed.

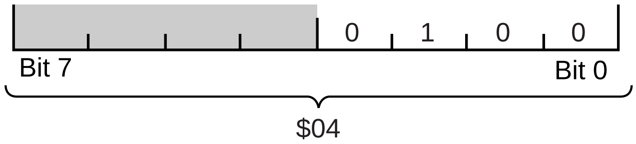
Example

Two handwheels are assigned to addresses 28 and 30 and the measurement of the handwheel at address 30 is filtered.

Word N0	05



Word N1	04



4.12 Coupling Assignment for Duplicated and Synchronised Axes

P27

Category	Axis declaration
Type 0	8-bit hexadecimal
No. of words	32

Description

Used to assign a driven axis to a drive 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 + 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.

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

Programme Parameters (see «Part Programming Manual»).

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.

Example

Driven axis @	**0**	**1**	**2**
Axis name	X	Y	Z
Drive axis @	None	0	0

Word N0	FF

The X axis at address 0 does not have a drive axis.

Word N1	00

The Y axis at address 1 is driven by the X axis (@ 00).

Word N2	00

The Z axis at address 2 is driven by the X axis (@ 00).

4.13 Synchronised Axis Coupling Enable

P28

Category	Axis declaration
Type 6	32-bit hexadecimal
No. of words	1

Description

Enables synchronisation of a driven axis with a drive axis.

Principle

The synchronised axes are duplicated axes and the driven axis is synchronised by 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 a 1, the driven axis is synchronised with its drive axis declared in P27.

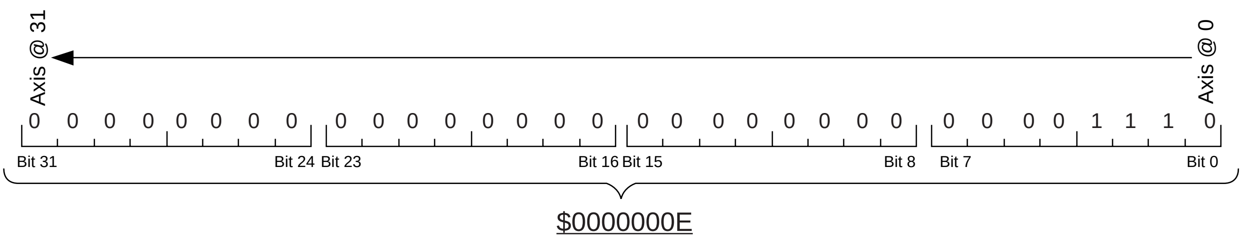
> ⚠ **CAUTION**
>
> A coupling cannot be both synchronised and symmetric.

Programme Parameters (see «Part Programming Manual»)

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.



Word N0	0000000E

4.14 Carrier or Carried Axes

P64

Category	Axis declaration
Type 6	32-bit hexadecimal
No. of words	1

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 as it is declared in P9 as primary (X, Y, Z) or secondary (U, V, W).

REMARKS: 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 «Part Programming Manual»)

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

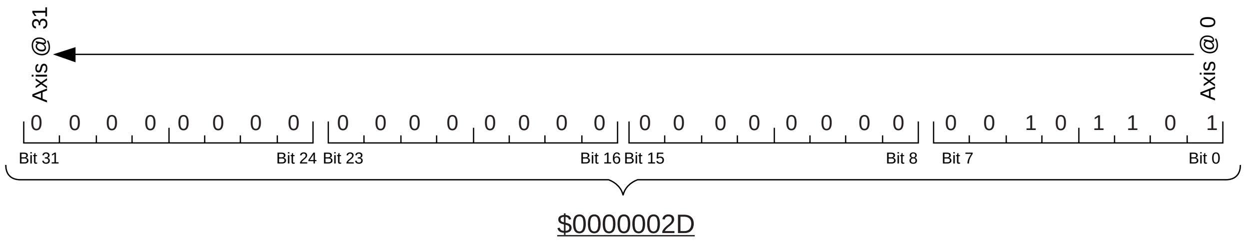
E70006 = 0 The X and U axes are independent.

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



Word N0	0000002D

4.15 Axes with Quantified Movements

P65

Category	Axis declaration
Type 5	Unsigned decimal
No. of words	32

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 ten-thousandths of a degree. A value of zero corresponds to axis movement by increments of one ten-thousandths of a degree.

The word number gives the physical axis address.

REMARK: 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

5 Measurement

5.1 Data Tables

Measurements of the Axes in Group

	Main linear axes			Secondary linear axes			Rotary axes		
Axis name	X	Y	Z	U	V	W	A	B	C
Physical axis address (@ 0 to 31)									
Measured axes									
Measurement reversal									
Measurement conversion coefficient (MULTI/DIVI)									
Encoder error check									

Handwheel Definition

		Handwheel 1	Handwheel 2	Handwheel 3	Handwheel 4
Physical handwheel address (imposed address)		@28	@29	@30	@31
Measurement reversal					
Measurement conversion coefficient	MULTI				
	DIVI				
Overspeed coefficient in G12	MULTI				
	DIVI				

5.2 Axis Measurement Direction

P10

Category	Measurements
Type 6	32-bit hexadecimal
No. of words	1

Description

Used to define the direction of axis measurement.

Principle

The bit position gives the physical axis address.

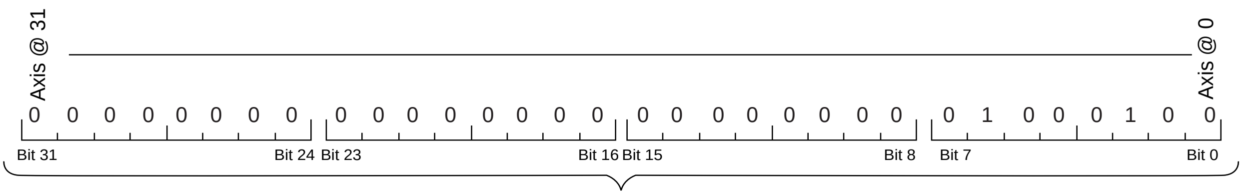
If the bit is a zero, the encoder for the corresponding axis is mounted in the forward direction (the «measurement step» supplied by the encoder increases as the tool moves in the positive direction of the axis).

If the bit is a 1, the axis encoder is reverse-mounted (the measurement decreases as the tool 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



$00000044

Word N0	00000044

UNI-TE request (see UNI-TE Protocol User's *Manual*)

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

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

5.3 Axis Measurement Conversion Coefficient

P11

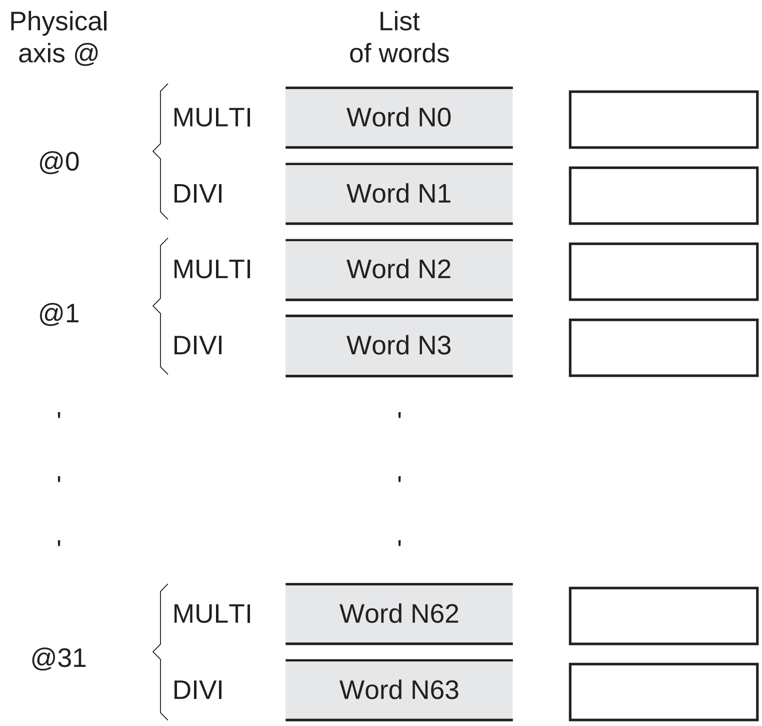
Category	Measurements
Type 5	Unsigned decimal
No. of words	64

Description

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).



5.3.1 Axis Measured by a Sensor without SSI Serial Transmission

The conversion formula is as follows:

Movement (internal measurement unit) = Encoder measurement x Multi/Divi

Where Encoder measurement = Number of periods (of channel A or channel B) on the CNC input x 4 for the movement considered.

Yielding

Multi/Divi = Movement (Internal measurement unit)/Encoder measurement

The internal measurement is expressed in mm/100, µm, µm/10 or µm/100 for linear axes (see 4.8) and thousandths of a degree for rotary axes.

⚠ CAUTION

For optimum accuracy, ratio MULTI/DIVI must be $\leq$ 1
(e.g. An internal unit of 1 µm with MULTI/DIVI = 10 is equivalent to a measurement to 0.01 mm).

The MULTI and DIVI coefficients must be < 32768.

UNI-TE request (see UNI-TE Protocol User's *Manual*)

The axis measurement conversion coefficients can be modified by a UNI-TE request. A reset (CNC reset) resets the CNC to the state specified by P11.

Examples

Case of a rotary encoder

Evaluate the movement and encoder measurement for one encoder revolution.

Encoder Encoder measurement = Number of lines x 4 x interpolation of the consecutive electronics.

Encoder ROD456 (2500 lines) and consecutive electronics EXE610B (interpolation by 10) with an internal unit of 0.1 µm, movement 5 mm, i.e. the 50,000 internal units.

Encoder measurement = 2500 x 4 x 10 = 100,000

MULTI/DIVI = 50,000/100,000 = 1/2

MULTI/DIVI = 50000/100000 = 1/2 (MULTI = 1 and DIVI = 2 after simplifying the fraction).

Case of a linear encoder

The movement and encoder measurement are given for a rule division period.

Encoder Encoder measurement = 1 x 4 x interpolation of the consecutive electronics.

Rule LS107 (period 20 µm) and consecutive electronics EXE630 (interpolation by 5) with an internal unit of 1 µm.

Measurement = 1 x 4 x 5 = 20

MULTI/DIVI = Period/Measurement = 20/20 = 1/1.

5.3.2 Axis Measured by Sensor with SSI Serial Transmission

If the sensor measurement is sent on fewer than 16 bits, it is systematically rescaled by the system, which shifts it 16-n spaces left to obtain a measurement without discontinuity.

The odd words (DIVI) must be modified accordingly:

Number of bits of the SSI frame	P11 odd word DIVI
32-16	unchanged
15	*2
14	*4
13	*8
etc ...	
N<16	$*2^{(16-n)}$

<u>Example</u>

A machine is equipped with axis 1 with 12-bit SSI encoder.

If the measurement increment is 1 µm, we would have:

With a conventional incremental encoder:

Word 2 (MULTI) = 1

Word 3 (DIVI) = 1

With the above encoder, since the measurement is on 12 bits, DIVI is multiplied by 2^4

Word 2 (MULTI) = 1

Word 3 (DIVI) = 16

⚠ CAUTION

DIVI must be divisible by 16, or the measurement will not be correctly scaled.
No check is made.

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

5.4 Handwheel Measurement Direction

P12

Category	Measurements
Type 0	8-bit hexadecimal
No. of words	1

Description

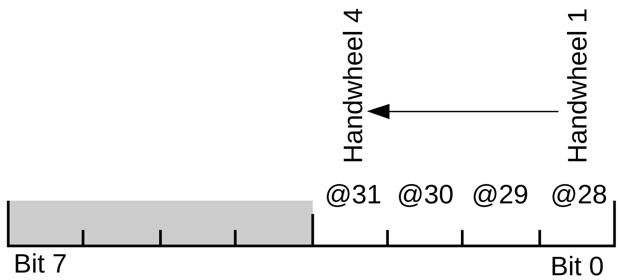
Used to define the handwheel measurement direction.

Principle

Only the four LSBs are significant.

UNI-TE request (see UNI-TE Protocol User's *Manual*)

The parameter can be modified by a UNI-TE request. A reset (CN reset) resets the CNC to the state specified by P3.



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

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

Example

The handwheels at addresses @28 and @30 are equipped with reverse-mounted encoders.

Word N0	05

5.5 Handwheel Measurement Conversion Coefficient

P13

Category	Measurements
Type 5	Unsigned decimal
No. of words	18

Description

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

Principle

Words N0 to N7

The first four pairs of words (N0 to N7) correspond to the values assigned to the handwheels. The first word of the pair indicates the multiplier coefficient (MULTI) and the second 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 4.8).

Words N8 and N9

These words define the measurement increment for the handwheel in automatic mode so 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 (used with CNC software at index J)

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 Parameter (see «Part Programming *Manual*»).

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.

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

5.6 Poor Signal and Encoder Channel Complementarity Check Declaration

P25

Category	Measurements
Type 6	32-bit hexadecimal
No. of words	2

Description

Used to declare the axes that can be affected by the poor signal and encoder channel complementarity checks and the fault filtering timeout.

Used to declare the QVN motor encoder which can be affected by the poor signal and encoder channel complementarity checks.

Word N0

This word is used for all the position encoders (axis, spindle) and all the handwheels connected to analogue axis cards or QVN cards.

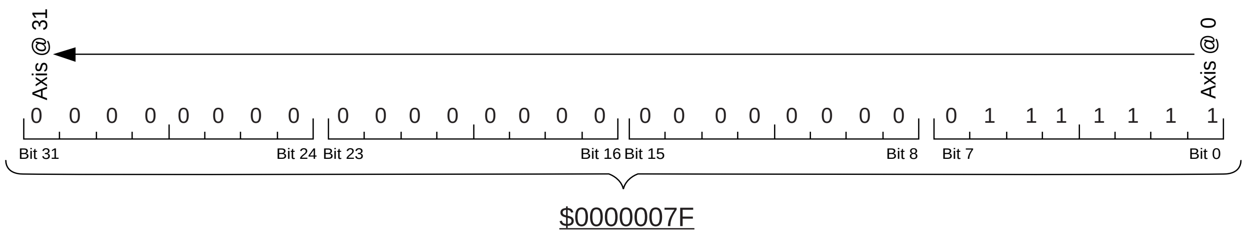
This word has a default value of $00000000 (no position encoder or handwheel checked).

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

If the bit is a 0, the check is inhibited on the axis concerned. If the bit is a 1, the check is enabled on the axis concerned.

Example

Check enabled on the axes at addresses 0 to 6.



Word N0	0000007F

Word N1

This word inhibits or enables the check on the QVN motor encoder.

Operation of a QVN application is enabled only if the check on the motor encoder is enabled.

For further details, refer to DISC Integration Manual 938907.

If word N1 = 0 (with DISC axes), the torque is not enabled even if +24 V is applied to connector on the amplifier front panel and variables %W34.L are set.

REMARK: Parameter P25 is meaningless with DISC NT.

5.7 Poor Signal and Encoder Channel Complementarity Check

P26

Category	Measurements
Type 0	8-bit hexadecimal
No. of words	32

Description

Activates the encoder channel complementarity and poor signal check and assigns the timeout for filtering faults.

Principle

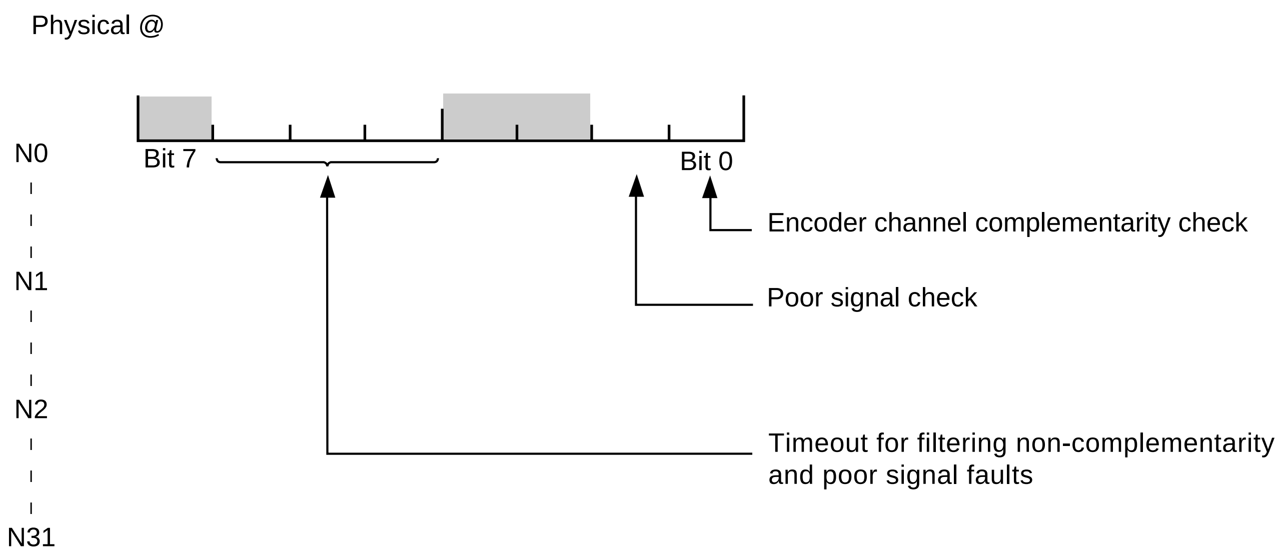
The word number gives the physical axis address.

If the bit is a 1, the check is active.

Default value: 00

> ⚠ **CAUTION**
>
> The check cannot be activated in P26 unless it is first declared in P25.

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

REMARK: Parameter P26 is meaningless with DISC NT.

5.8 Definition of Measurement Sensor Type and Parameters

P34

Category	Measurements
Type 6	32-bit hexadecimal
No. of words	32

Description

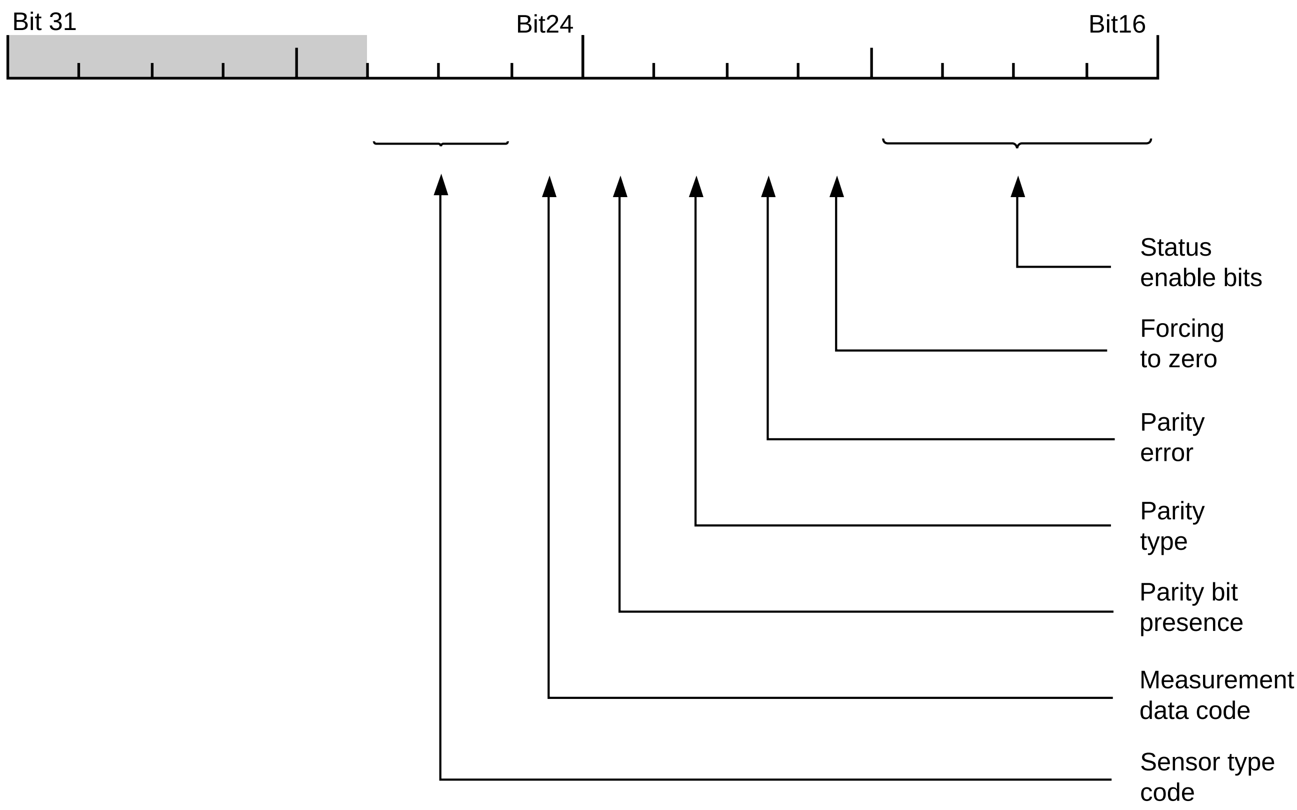
Specifies the type of sensor (incremental, absolute, etc.) used to measure an axis and the sensor parameters.

Principle

Each long word per axis specifies the contents of two control registers:
- high word (bits 31-16): serial control register 1
- low word (bits 15-0): serial control register 2.

Default values initialised: 0000 0000

Serial Control Register 1 Format (bits 31-16)



Bits 31-27

Not used.

Bits 26-25

Sensor type code

Bit 26	25	
0	0	incremental sensor
0	1	absolute sensor with SSI serial transmission
1	0	so-called combined absolute encoder
1	1	incremental rule with encoded reference marks.

Programme parameter E936xx (xx = axis No.) is the image of these bits.

Bit 24
- 0: gray code
- 1: measurement data binary code

Bit 23
- 0: not significant
- 1: parity bit present at the end of the transmission frame

Bit 22
- 0: even parity
- 1: odd parity

Bit 21
- 0: not significant
- 1: "parity error" processed

Bit 20
- 0: not significant
- 1: measurement "forcing to zero" error processing

Bits 19 à 16

Setting of one of these bits enables the corresponding status bit.

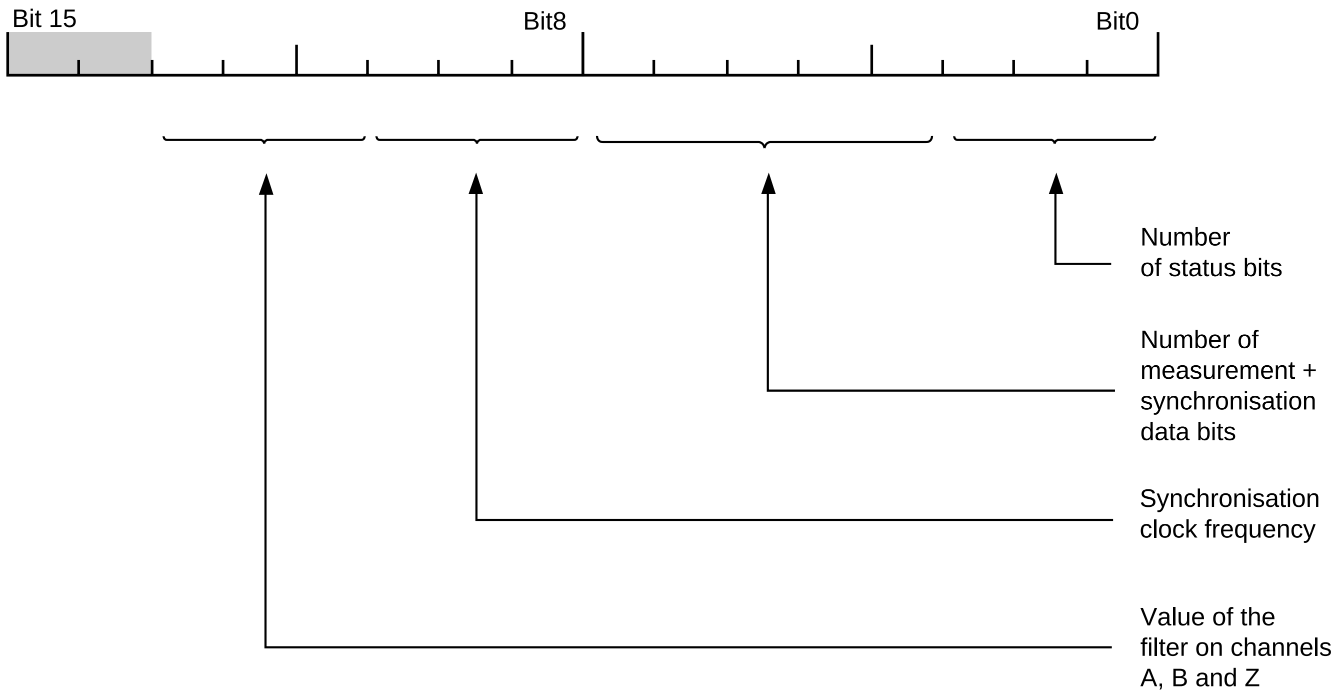
The sensors can transmit up to four different states, depending on the model. These bits are used to define the bits to be processed by the CNC. For use, do not omit to programme bits 2-0.

REMARK *If an enabled status bit goes high, the following message is displayed on the CNC:*

Error: 210 + x where x = axis No.

Poor signal or channel loss or serial feedback fault on axis 0.

Serial Control Register 2 Format (bits 15-0)



Bits 15 et 14

Not used

Bits 13 à 11

Select the timeout on channels A, B and Z

Bit 13	12	11		
0	0	0	:	120 ns (default value)
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

Bits 10 to 8

Select the transmission clock frequency

Bit 10	9	8	
0	0	0	:100 KHz
0	0	1	:200 KHz
0	1	0	:400 KHz
0	1	1	:500 KHz
1	0	0	:800 KHz
1	0	1	: 1 MHz
1	1	0	:1.6 MHz
1	1	1	: 2 MHz

Bits 7 to 3

Number of measurement and synchronisation data bits in the frame

Examples:

- The Stegmann AG66 SSI encoder transmits the measurement on 12 bits (resolution 4096 pulses per revolution, single revolution).

  The bit codes are:

7	6	5	4	3
0	1	1	0	0

- The Stegmann AG661 SSI encoder transmits the measurement on 24 bits (resolution 4096 pulses per revolution times 4096 revolutions).

  The bit codes are:

7	6	5	4	3
1	1	0	0	0

Bits 2 to 0

Number of status bits transmitted at the end of the transmission frame. These bits must be programmed for use by bits 19-16.

Example:

The Stegmann AG66 and AG661 SSI encoders transmit a so-called PFB (power failure bit) status bit which is set in the event of a power failure.

- Bit codes:

2	1	0
0	0	1

⚠ CAUTION

- Linear SSI encoders and combined sensors
- The zero point must be outside the machine travel limits (P17).
- For SSI measurements, rules with encoded reference marks, etc. Do not encode for additional measurements for DISC NT.

5.9 Measurement Sensor Graduation and Number of Graduations

P36

Category	Measurements
Type 5	Unsigned decimal
No. of words	64

Description

Used to adapt the measurement sensor to the CNC axis cards.

Principle

Two words are assigned to each axis: N0 and N32 to axis 0; and N1 and N33 to axis 1; N2 and N34 to axis 2, etc.

The meaning of words 00-31 and 32-63 depends on the sensor type.

Default values initialised:
N00-N31: 1
N32-N63: 1

5.9.1 Incremental Rules with Encoded Reference Marks

Words 00-31: graduation in measurement increments p taking multi/divi into account.

Words 32-63: Number of graduations N between two marks (manufacturer data).

Calculation of a Word 0 to 31

Let:
- P be the manufacturer's graduation
- Exe be the EXE multiplier
- N be the number of graduations between two marks
- P11 be the MULTI/DIVI coefficient applied to the axis considering a multiplier k depending on the machine kinematics and the internal resolution used.

We have:

- $$P11 = k \cdot \frac{P}{4 \cdot Exe}$$

We calculate the graduation p in measurement increments as follows:

$$\text{P36, word 0 - 31}: \quad (p) = \frac{4 \cdot Exe}{k} = \frac{P}{P11}$$

P36, word 32-63: N graduations

Example

Axis 0 is equipped with a Heidenhain LS 776C rule with encoded reference marks = EXE 612.

Manufacturer data:
- multiplier due to EXE: Exe = 10
- rule graduated every 20 µm: P = 20 (graduation)
- law applied: 1000*P, i.e. N = 1000 (number of graduations).

If the internal resolution unit is the micrometre and if the measurement increments are not multiplied or divided by the machine kinematics, k = 1.

$$P11 = \frac{20}{4 \cdot 10} = \frac{1}{2}$$

P36, word 0 = 20*2 = 40

P36, word 32 = 1000

5.9.2 Combined Sensor: Incremental and SSI

Word 00-31: **Numerator** of the ratio $\frac{\text{Value of the SSI step}}{\text{Value of the incremental step}}$

Word 32-63: **Denominator** of the ratio $\frac{\text{Value of the SSI step}}{\text{Value of the incremental step}}$

Conventional Case

The incremental measurement uses an IBV unit which generates square signals. This unit generally has a multiplier. If there is no IBV, set words 00-31 to 1 (default value).

The SSI measurement is applied directly to the axis card.

Example

The Heidenhain ROC 413 encoder outputs frames of 13 bits and 1024 * 4 periods.
This encoder is connected via an IBV with a multiplier of 10.

For instance, if each encoder revolution corresponds to a distance of 10 mm, increment P11 is determined as follows:

$$1024p * 4 * 10_{ibv} * \frac{MULTI}{DIVI} = 10 * 1000\ \mu$$

$$\frac{MULTI}{DIVI} = \frac{1000}{1024 * 4} = \frac{250}{1024}$$

P36 is determined as follows:
If the frame is 13 bits wide, there are 2^{13} = 8192 SSI steps per revolution, which must be equal to 1024*4*10 *incremental steps.*

2^{13} = 8192 *SSI steps* = 1024*4*10 *incremental steps*

The ratio is then equal to:

$$\frac{\textit{SSI steps}}{\textit{inc steps}} = \frac{8192}{1024 * 4 * 10} = \frac{8}{40}$$

or any other combination giving an equivalent ratio.

5.9.3 Other Type of Sensor (Incremental and SSI)

Words 00-63 are not used. They can be set to any value.

⚠ CAUTION

For SSI measurements, rules with encoded reference marks, etc. Do not encode for additional measurements for DISC NT.

5.10 Measurement Setting

REMARK: The values can be recorded in the tables of Chapter 3.

Requirements

Emergency stop switch depressed.

CNC on.

Utility 5 displayed on the CNC screen (see 13.2.1).

Action

Set the following parameters:

Declare all the measured axes in P2.

Declare the poor signal check as active on all axes in P25 and P26.

Declare no servo-controlled axes in P3.

Set the Multi/Divi measurement conversion coefficient for each measured axis.

Enter the computed Multi/Divi values in parameter P11.

Enter the sensor type and parameters in parameter P34.

Enter the graduation and number of graduations in parameter P36.

Exit from utility 5.

Return to the «AXIS» page.

If an error of type 210 to 241 (axis fault) appears on the «AXIS» page:

Check the poor signal error (Pin 7) for the faulty axis.

Check display of the axes on the «AXIS» page.

Only the measurement is displayed.

Check of the Measurement

Rotate each axis by hand in both directions.

REMARK: If this check is not possible, it can be carried out after applying power.

Check how the measurement varies.

An increase in the measurement should correspond to positive movement of the axes.

If this is not the case:

Reverse the measurement direction by changing the state of the bit
at the corresponding axis address in P10.

Rotate each axis by a value that is easy to check.

If the measured movement does not correspond to the effective movement:

Check the Multi/Divi calculation in P11.

6 Servo-Control

6.1 Data Tables

Servo-Control Characteristics of the Axes in Group

	Main linear axes			Secondary linear axes			Rotary axes		
Axis name	X	Y	Z	U	V	W	A	B	C
Position servo-loop time constant (ms)									
MAXimum feed rate (mm/min, or deg/min)									
Servo-drive refe-rence for VMAX (9 V recommended)									
Feed rate buildup time TV (ms) (time measured at 63% VMAX)									
Maximum acceleration DDM=VITMAX/TV (mm/s^2, or deg/s^2)									
Maximum work rate on the path VTmax									
Acceleration for movement at the work rate DDm = VTmax/TV (mm/s^2, or deg/s^2)									
Stopping accuracy at the work rate (µm)									
Servo-control error tolerated on circles (µm/smp^2)									

Speeds and Accelerations of the Axes in Group

	Main linear axes			Secondary linear axes			Rotary axes		
Axis name	X	Y	Z	U	V	W	A	B	C
Physical axis address (@ 0-31)									
Fast JOG speed									
Normal JOG speed									
Slow JOG speed									
Acceleration for movement at the work rate									
Speed reference direction									
KVAR servo-control coefficient									
Maximum following error EMAX (in internal units or deg/10000)									

6.2 Maximum Axis Traverse Rates

P30

Category	Servo-control
Type 4	Unsigned decimal
No. of words	32

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 Parameter

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

UNI-TE request (see UNI-TE Protocol User's Manual)

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

6.3 JOG Speeds and Reduced Speeds

P31

Category	Servo-control
Type 4	Unsigned decimal
No. of words	5

Description

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

Principle

Words N0 to N2

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

Words N3 and N4 are 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.

6.4 Maximum Permissible Acceleration

P32

Category	Servo-control
Type 5	Unsigned decimal
No. of words	64

Description

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

Principle

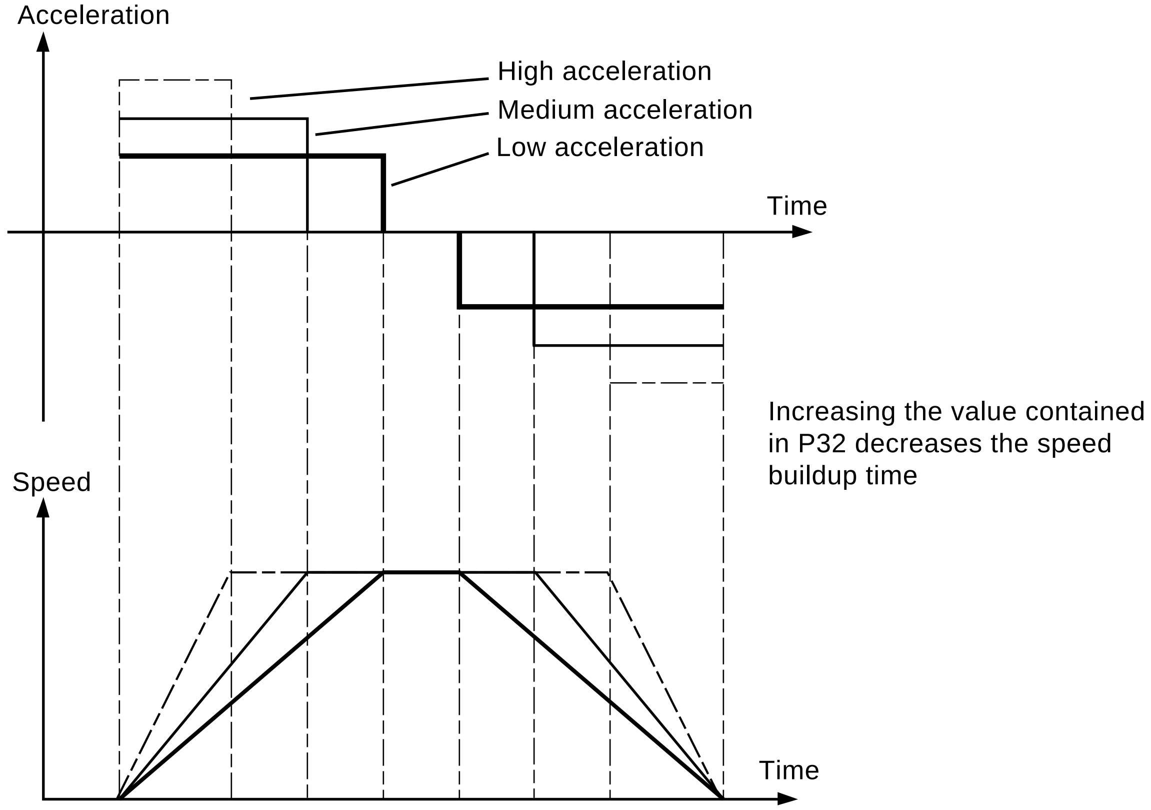
Each pair of words defines the accelerations on an axis. The first word gives the acceleration for the work rates and the second gives the accelerations for the fast speeds: «DRYRUN, FAST JOG and G0 modes».

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

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

Physical axis @		List of words	
@0	Acceleration Work rate	Word N0	
	Acceleration Fast speed	Word N1	
@1	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



REMARK *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 for indexing with constant acceleration (parameter P6, words 1 to 4, bit 5 equal to 0).

Programme Parameter

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.

UNI-TE request (see UNI-TE Protocol User's Manual)

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

6.5 Direction of Axis Speed Reference

P20

Category	Servo-control
Type 6	32-bit hexadecimal
No. of words	1

Description

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

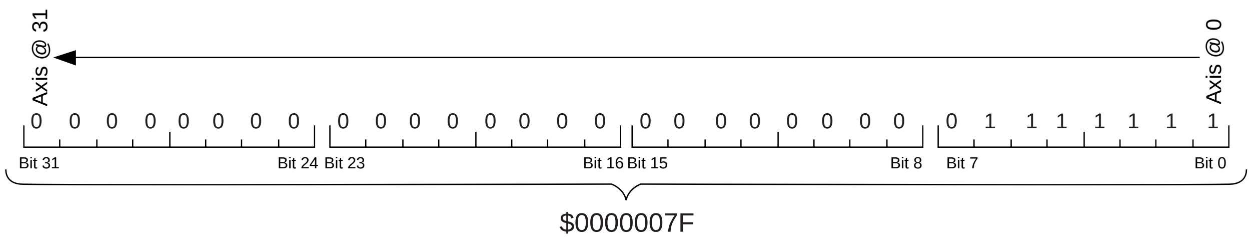
Principle

The bit position gives the physical axis address.

If the bit is a 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.



Word N0	0000007F

UNI-TE request (see UNI-TE Protocol User's Manual)

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

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

6.6 Servo-System Loop Gain Coefficient

P21

Category	Servo-control
Type 4	Unsigned decimal
No. of words	32

Description

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

Principle

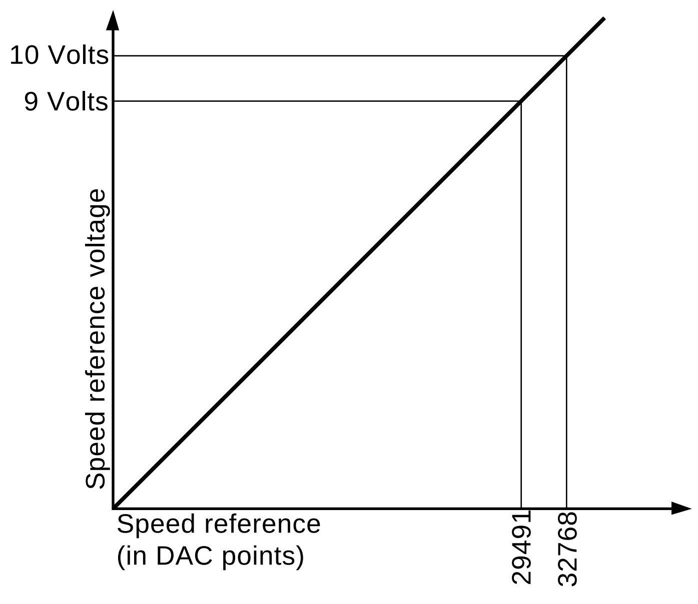
Each word corresponds to the physical address of an axis:
- the value in word N0 is assigned to the axis with physical address 0,
- the value in word N1 is assigned to the axis with physical address 1,
- the value in word N2 is assigned to the axis with physical address 2,
- etc.

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

KVAR Calculation

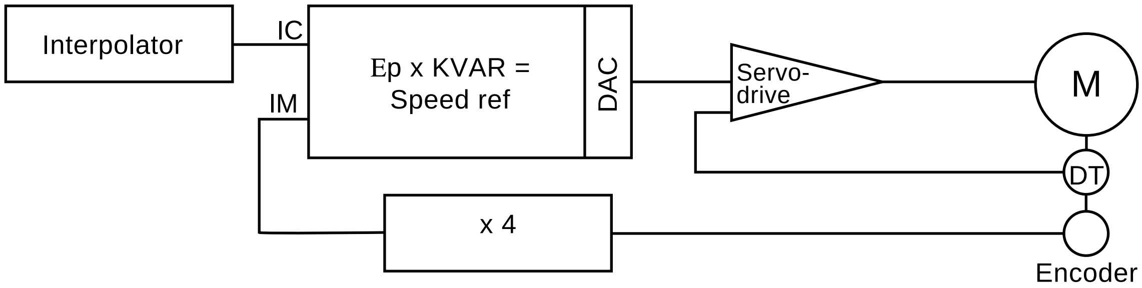
The CNC function generates a theoretical speed reference voltage of 10 V maximum. This voltage is provided by a DAC on an axis encoder card. Generally, the servo-drives are set so that the axis speed is obtained for a servo-drive input voltage of 9 V.

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	Coefficient k
1/10 mm	100
1/100 mm	10
µm	1
1/10 µm	0.1
1/100 µm	0.01
1/10000 deg	0.1

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)

yielding:

Ep max = VMAX/Gain

Yielding 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.

CAUTION

The following error displayed on the CNC is in internal units (ten-thousandths of a degree or from 0.01 µm to 0.01 µm).
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 x 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

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.

UNI-TE request (see UNI-TE Protocol User's Manual)

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

Differences Related to Use of DISC Axes

See DISC Integration Manual 938907.

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

6.7 Servo-Loop Time Constant

P56

Category	Servo-control
Type 5	Unsigned decimal
No. of words	8

Description

Defines the position servo-loop time constant.

Principle

Each word corresponds to an axis group:
- word N0 for axis group 1,
- word N1 for axis group 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:

$$T = 1/\ Gain = Ep\ /VIT$$

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

Example

For group 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

REMARK: 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.

Programme Parameter

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

UNI-TE request (see UNI-TE Protocol User's Manual)

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

6.8 In-Position Window

P22

Category	Servo-control
Type 5	Unsigned decimal
No. of words	32

Description

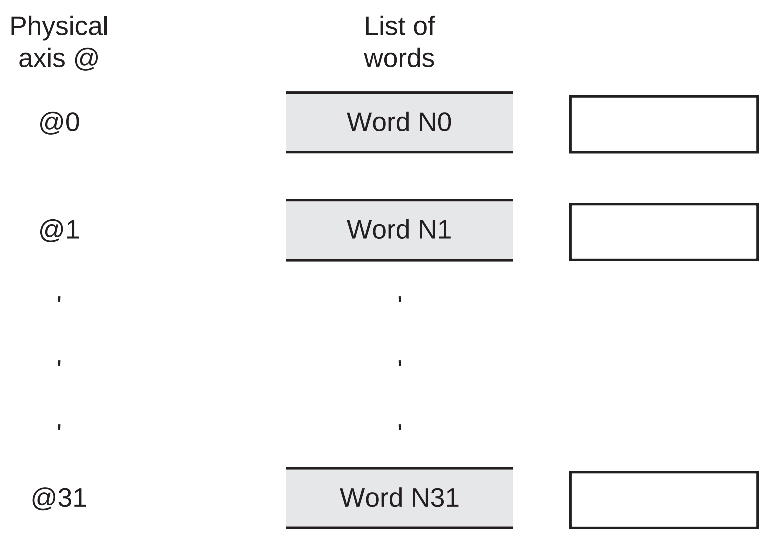
A distinction is made between two cases:

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.



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/100, µm, µm/10 or µm/100 for linear axes (see 4.8) and deg/10000 for rotary axes.

6.9 Maximum Following Error

P23

Category	Servo-control
Type 4	Unsigned decimal
No. of words	32

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 (see 4.8) and deg/10000 for rotary axes.

REMARK: 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	Coefficient k
1/10 mm	100
1/100 mm	10
µm	1
1/10 µm	0.1
1/100 µm	0.01
1/10000 deg	0.1

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

UNI-TE request (see UNI-TE Protocol User’s Manual)

The maximum following errors can be modified by a UNI-TE request. A reset (CNC reset) resets the CNC to the state specified by P23.

6.10 Dynamic Movement Control

P57

Category	Servo-control
Type 5	Unsigned decimal
No. of words	32

Description

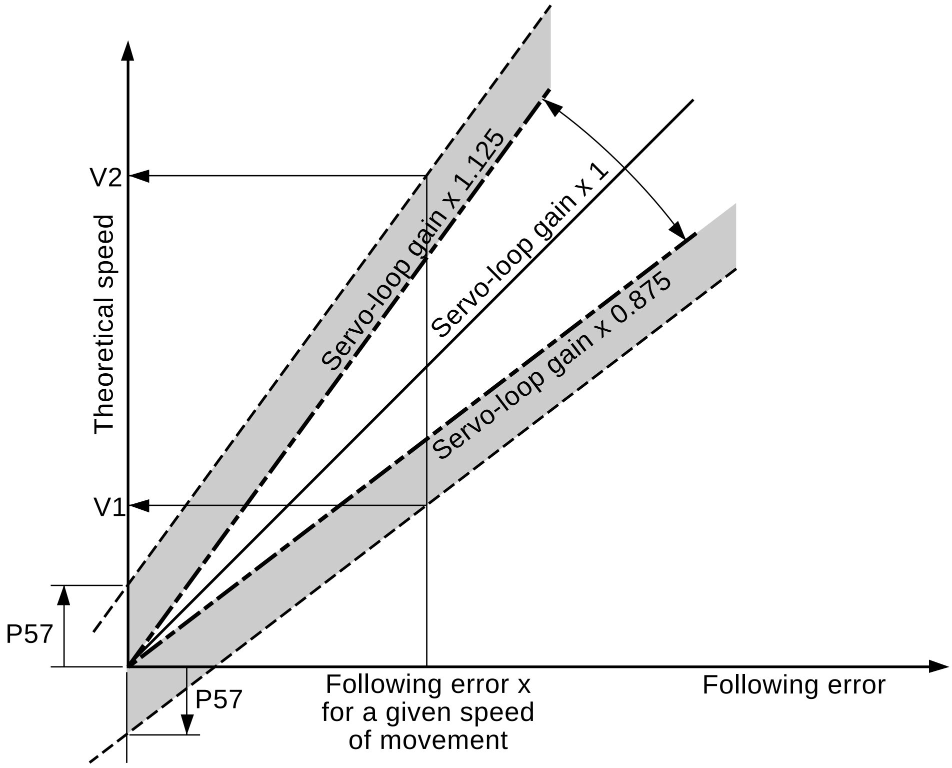
Used each sample (RTC in P50) to compare the real speed of an axis with its theoretical speed deduced from the following error.

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 expressed 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.

REMARKS: *For correct use of the settings of this parameter, the position servo-loop time constants must be specified in parameter P56 (see 6.7).*
The value of zero in a word disables dynamic control on the corresponding axis.

UNI-TE request (see UNI-TE Protocol User's Manual)

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

6.11 Speed Anticipation Coefficient and Number of Filtered Terms for Very High Speed Machining

P55

Category	Servo-control
Type 5	Unsigned decimal
No. of words	8

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.

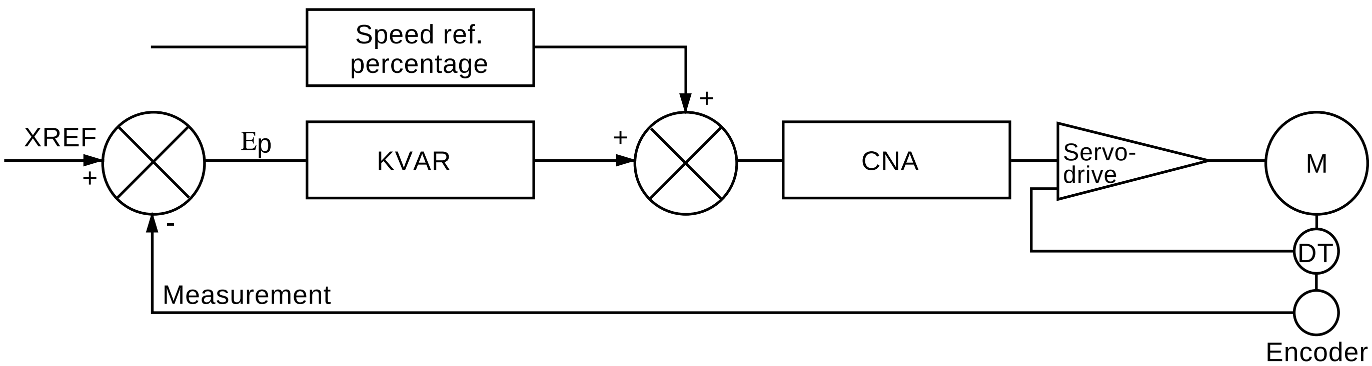
Principle

Each word represents an axis group:
- Words N0 and N8 for axis group 1,
- Words N1 and N9 for axis group 2,
- etc.

Words No to N7

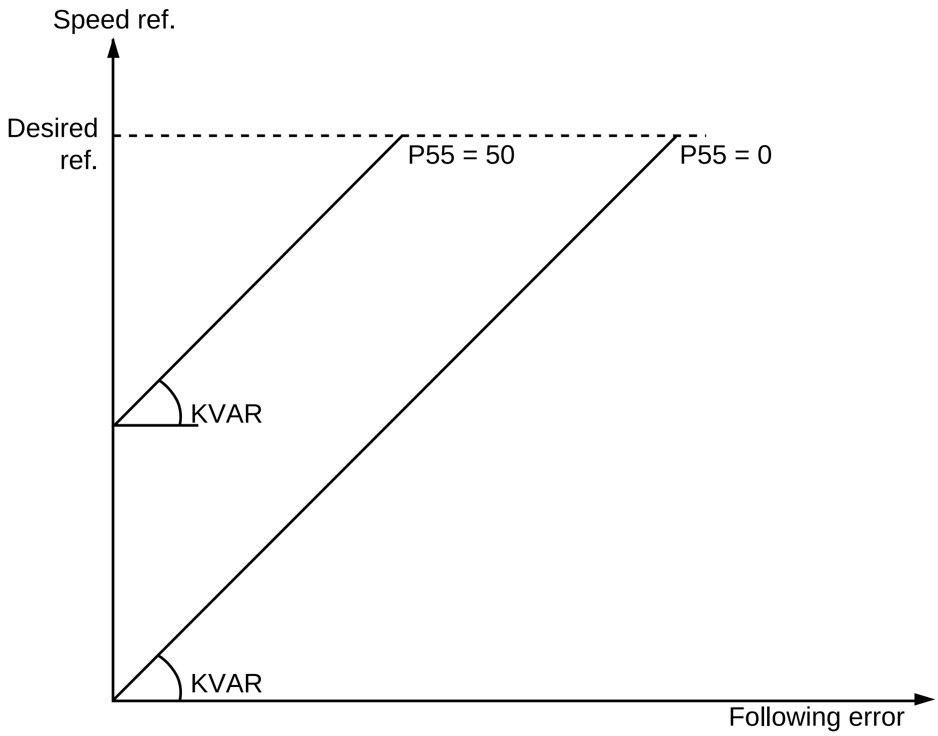
Each word contains a speed reference expressed as a percentage between 0 and 100.

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



CAUTION

A value of zero has no effect on the system.



Anticipation does not change the gain of the speed loop.

CAUTION

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

Words N8 to N15

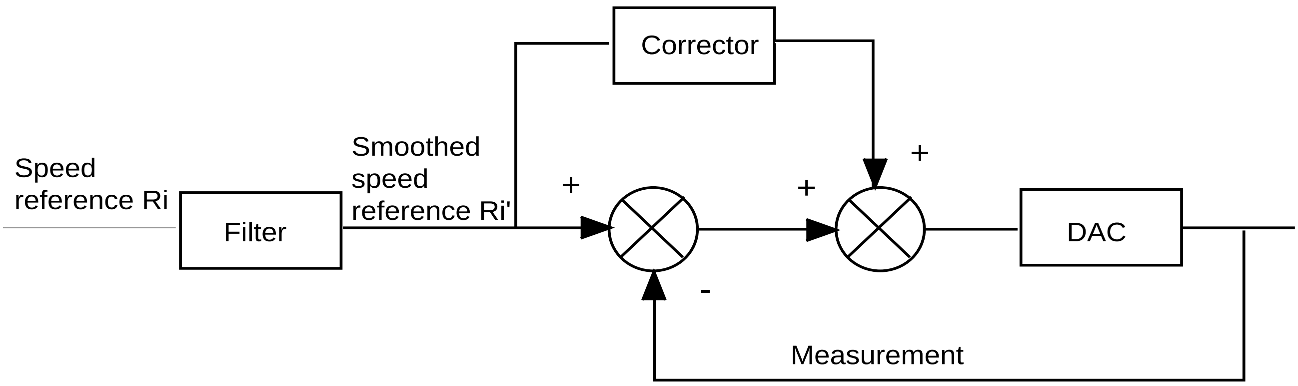
For very high speed machining, a filter smoothes the speed reference. This filter averages several consecutive settings.

The number of terms used for the filter specifies the number of calculated references averaged to calculate the 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 around 5.



Programme Parameter

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.

UNI-TE request (see UNI-TE Protocol User's Manual)

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

6.12 Characteristics of Synchronised Axis Pairs

P24

Category	Servo-control
Type 5	Unsigned decimal
No. of words	65

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.

Driven axis
physical @
N0
.
.
.
.
.
N31 — Threshold 1 (1 word per axis)

Driven axis
physical @ + 32
N32
.
.
.
.
.
N63 — Correction coefficient (1 word per axis)

N64 Threshold 2 (1 word for all the axes)

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 basic 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 basic 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 basic value is 600 µm.

The internal units can be mm/10, mm/100, µm, µm/10 or µm/100 for linear axes (see 4.8) and deg/1000 for rotary axes.

REMARK: *The error generated when thresholds 1 and 2 are exceeded causes reset of CN_PRET and display of error 39 on the «AXIS» page.*

6.13 Servo-Loop Error Tolerated on Circles

P52

Category	Miscellaneous
Type 5	Unsigned decimal
No. of words	1

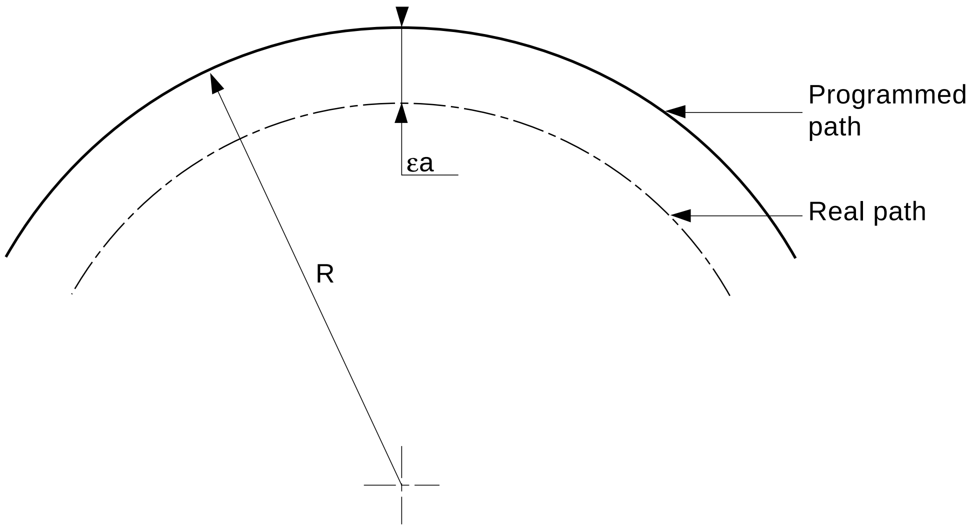
Description

Defines the maximum servo-loop error tolerated on circles.

The basic value is 50 µm.

Principle

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



εa is expressed 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:

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

Example :

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

We have:

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

Programme Parameter

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.

6.14 Very High Speed Machining

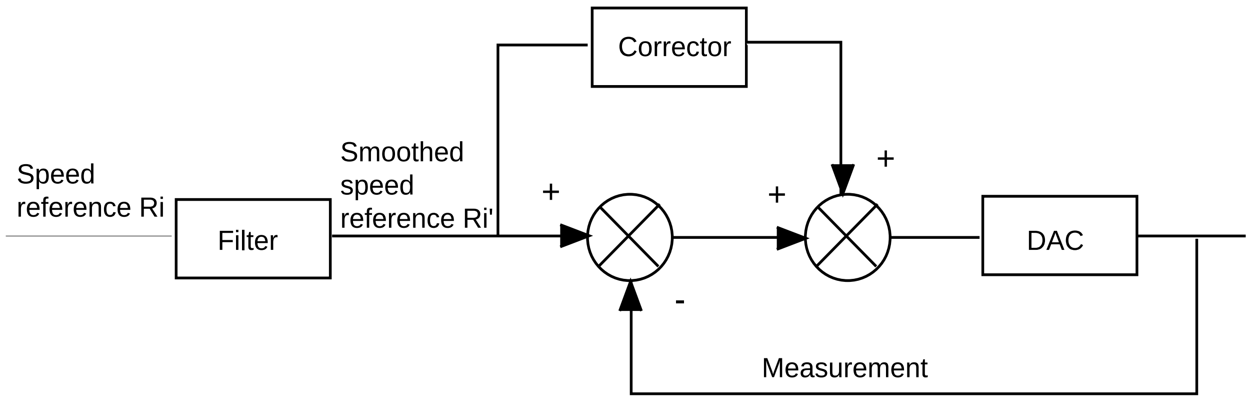
P19

Category	Servo-control
Type 1	Signed decimal
No. of words	96

Description

This parameter is used to set:
- the acceleration anticipation for each axis,
- the dry friction correction amplitude for each axis,
- tThe dry friction correction time constant for each axis.

Principle



Words N0 to N31: 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:

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

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

Words N32 to N63: 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: Dry friction correction time constant

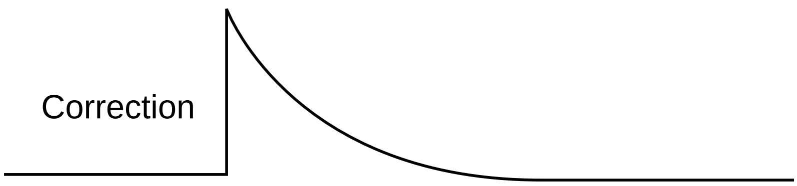
Each word corresponds to the physical address of an axis. The dry friction correction time constant is expressed in hundredths of ms. The order of magnitude is 5000 hundredths of ms.

Dry friction elimination

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

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

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



Programme Parameter

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.

6.15 Approach Speed

P33

Category	Servo-control
Type 5	Unsigned decimal
No. of words	32

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).

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

Example

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

Va = γ x T = 1000 x 60 x 10^{-3} = 60 mm/s, i.e. 3600 mm/min

UNI-TE Request (see UNI-TE Protocol User's Manual).

This parameter can be modified by a UNI-TE request.

Programme Parameter

Programme parameters E973xx are the image in mm/min of machine parameter P33. They can be read and written and retain the last value programmed after a reset.

6.16 Setting of the Maximum Permissible Acceleration on the Axes

P53

Category	Servo-control
Type 5	Unsigned decimal
No. of words	1

Description

The $\sin^2$ function consists of modulating the average acceleration for small speed variations so that the speed variation is completed within a time greater than or equal to the minimum time declared in this parameter.

This time is expressed in ms and is common to all axis groups.

Principle

Determination of the average acceleration:
– Speed variation Dv greater than Dt*Gmax/2: Gaver = Gmax/2
– Speed variation Dv less than Dt*Gmax/2: Gaver = Dv/Dt
where Dv = speed variation
and Dt = minimum time.

When P53 is set to 0, function $\sin^2$ continues to be processed in the old way (Gaver = Gmax/2 regardless of the speed variation).

Programme Parameter

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

6.17 Time Constant per Axis

P66

Category	Servo-control
Type 5	Unsigned decimal
No. of words	32

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.

For CNC software at index LS and above. 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 group. With total speed anticipation, different time constants can be used on the axes in a group, 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 group P56 continues to be used for speed limiting based on path curve radii to control the servo-control error.

As in versions before the CNC software at index K, it 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.

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

The values are expressed in milliseconds.

6.18 JOG Speed Setting

The JOG speeds are set by parameter P31 (the values can also be recorded in the tables of Section 3.3).

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 normal 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 machine processor for signals VIT MAN1 and VIT MAN2.

6.19 Setting the Maximum Speeds and Accelerations

Actions

Apply the following equation and record the computed values in the table.

DDM = VITMAX / TV

Where:

VITMAX: Maximum speed on the axis (in mm/min or deg/min).

REMARK: VITMAX (in mm/min) is equivalent to VMAX (in mm/s).

TV : Speed buildup time (from 0 to VMAX) (in ms).

DDM : Maximum acceleration (in mm/s^2 or deg/s^2).

Axis @	VITMAX (in mm/min or deg/min)	TV (in ms)	DDM (in mm/s^2 or deg/s^2)
0			
1			
2			
3			
4			
5			
6			
7			
8			

Enter the values of VITMAX in the words of P30 (the values can also be recorded in the table of Section 3.3).

Enter the values of DDM in the words of P32 (the values can also be recorded in the table of Section 3.3).

CAUTION

A value other than zero must be entered in P32 for each axis declared in P2.

6.20 Position Servo-Loop Settings

REMARK: The values can be recorded in the tables of Chapter 3.

Actions

Check that the limit switches and machine errors cut off power.

Declare a positive speed reference in P20 (a value of 1 in each bit of the word).

Declare a positive loop direction for the axes in P10 (value 0 in each word).

Enter the time constants in P56.

Check that the servo-drive is set to maximum speed for a 9 V reference.

REMARK: If the servo-drive reference voltage is different from 9 V, replace this voltage by the suitable value in the equation for KVAR.

Calculate the axis servo-loop gain coefficient and maximum following error.

Calculation of the Axis Servo-Loop Gain Coefficient KVAR (See 6.6)

Apply the following equation for a maximum voltage of 9 V:

KVAR = 29491 / (VMAX x T) x k

Where:
- T : Servo-loop time constant (in s),
- VMAX : Maximum speed on the axis (in mm/s or deg/s)..

Coefficient k depends on the internal measurement units used by the system.

Calculation of the Maximum Following Error (See 6.9)

Apply the following equation:

EMAX = VMAX x T x 11/10/k

Where:
- T : Servo-loop time constant (in ms).
- VMAX : Maximum speed on the axis (in mm/s or deg/s).
- EMAX : Maximum following error (according to the internal measurement units used: mm/10, mm/100, µm, µm10 or µm/100 for linear axes (See 4.8) and deg/10000 for rotary axes.

Coefficient k depends on the internal measurement units used by the system.

REMARK: A coefficient of 11/10 is applied to obtain a 10 percent margin.

Axis @	VMAX	EMAX
0		
1		
2		
3		
4		
5		
6		
7		
8		

Enter the values of KVAR in the words of P21.

Enter the values of VMAX in the words of P23.

Declare the first axis to be integrated as servo-controlled in P3.

Turn on the CNC.

The axis must not move and the CNC must not display an error of type 40 to 71: «excessive following error on the axis».

If this is not the case:

Turn off the CNC and reverse the axis speed reference direction in P10.

Turn on the CNC and select the continuous JOG speed.

Set the feed rate potentiometer to 20 percent.

Manually control the axis.

The axis should move in the required direction and stop when the jog control is released.

If this is not the case:

Turn off the power and adjust P20 and P10 to obtain movement in the required direction.

Turn on the power and check the two directions of movement.

Set the speed reference offset.

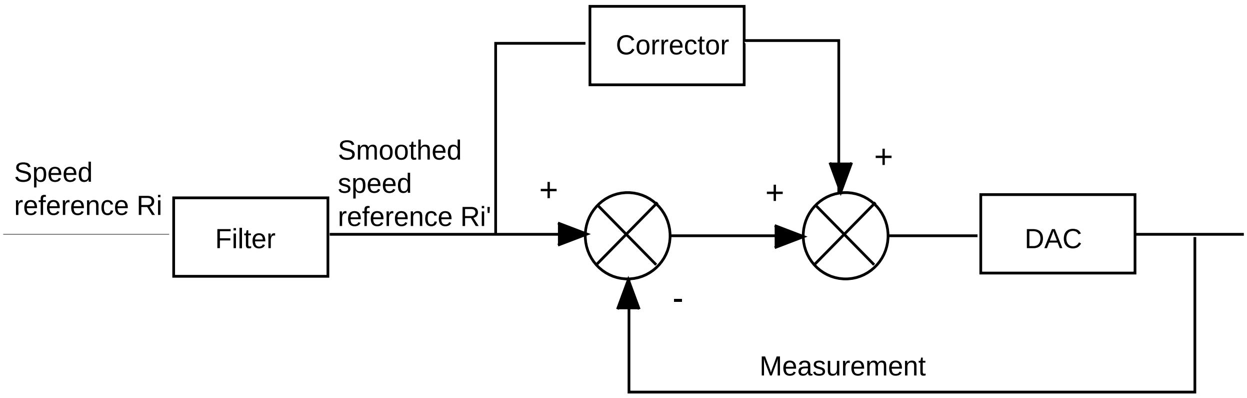
With the axis not moving, action on the zero offset servo-drive potentiometer cancels the positioning error displayed on the «AXIS» page in the error column.

If it was not possible to check the axis measurement, repeat the procedure of Sec. 6.13.

Turn off the power.

Declare a new servo-controlled axis if necessary and repeat the above procedure.

6.21 Setting for Very High Speed Machining



Actions

Set the speed anticipation to 100 percent, proceeding as follows:

Set the same gain on the all the axes.

Calculate the time constants and enter them in P56 (See 6.7).

Adjust the acceleration anticipation using the Ball-Bar trace to have the real circle as close as possible to the theoretical circle. The check can be made on a circle with a radius of 20 mm at a speed of 3 m/min. It is advisable to check that the trace remains acceptable at higher or lower speeds.

The acceleration anticipation can be accurately set in P19 (See 6.14) dynamically by the Ball-Bar part programme.

The overshoot due to cycle hold action during execution of the circle generated by the Ball-Bar programme allows the number of filtered terms to be adjusted as accurately as possible (words N8 to N15 of P55 (See 6.11)).

Example of Use of the Ball-Bar Programme

Create a part programme.

% 1020

M997 G77 H9997.9

G80

REMARK: Programme %9997.9 is a firmware macro.

Preset P19 and P55.

Programme function G120 in MDI mode followed by the interpolation plane, the direction of rotation of the circle, the radius and the speed (e.g. G120 G18 G3 R.. F..).

Display the «graphic parameters» page.

Select programme %9997.9.

Scale planes 1 and 2.

A target is displayed on the screen. It consists of a solid circle representing the theoretical circle and dotted concentric circles delimiting errors of 10, 100 and 1000 measurement increments. The measured position of the two axes of the plane are positioned on this target and adjustment of the acceleration anticipation is proposed.

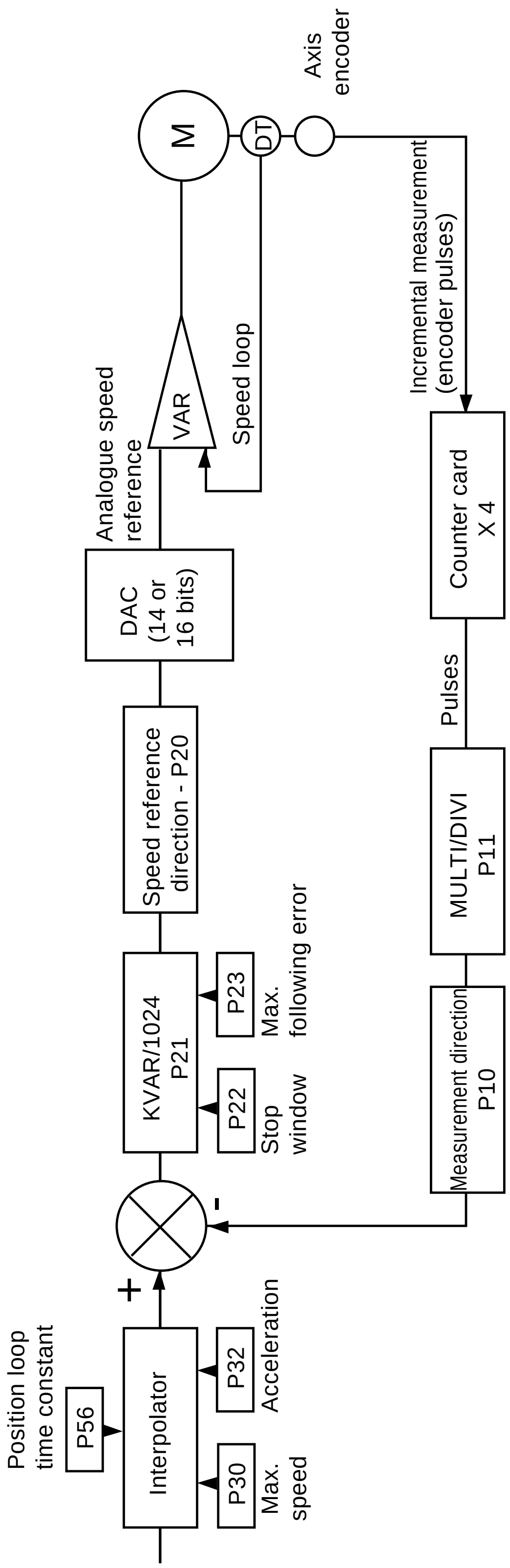
Enter the results in P19 (See 6-14).

Ball-Bar Programme Operating Mode

The Ball-Bar programme has two operating modes:

- in the default mode, each point is read on the fly and displayed immediately
- when function P is programmed in the calling block (G120 G18 G3 R.. F.. P1), the system reads 512 points in sequence then displays them; it then reads 512 more points and displays them and so forth.

6.22 General Block Diagram of Servo-Control



7 Axis Travels

7.1 Data Tables

Travels and Reversal Error Compensations for the Axes in Group

	Main linear axes			Secondary linear axes			Rotary axes		
Axis name	X	Y	Z	U	V	W	A	B	C
Physical axis addresses (@ 0 to 31)									
Maximum travel on the axis (internal units or deg/10000)									
Negative homing									
Position of the ORPOM switch (internal units or deg/10000)									
Minimum or negative travel (internal units or deg/10000)									
Maximum or positive travel (internal units or deg/10000)									
Reversal error compensation (internal units or deg/10000)									

7.2 Reminders

The measurement origin «OM» and machine zero point «Om» coincide.

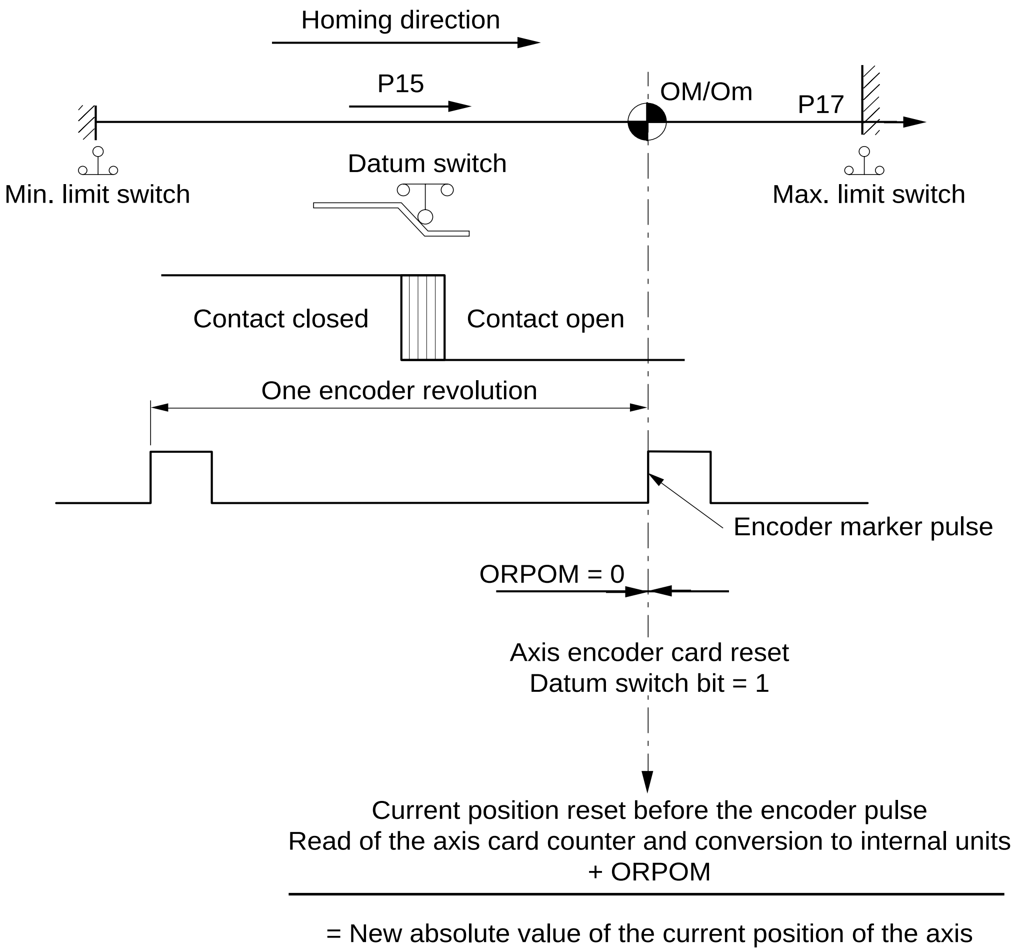
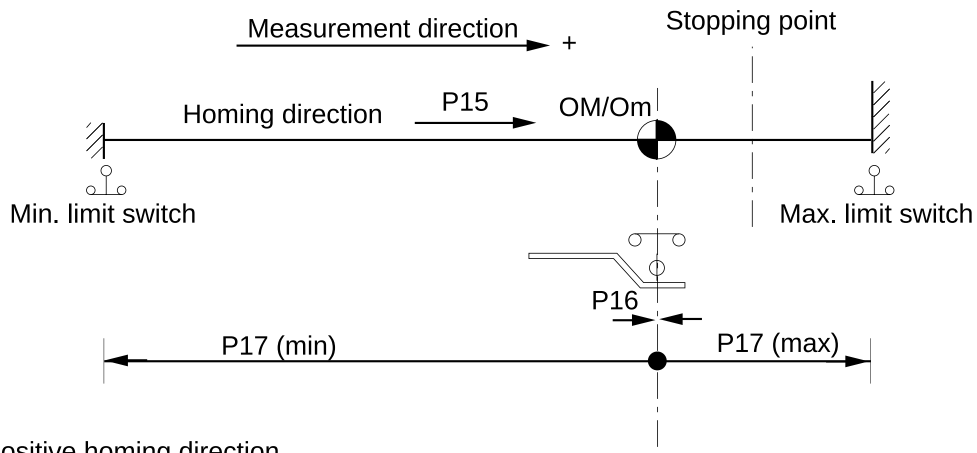


Figure 7.1 - Detail of homing

Summary of the Different Cases of Homing on the Datum switch

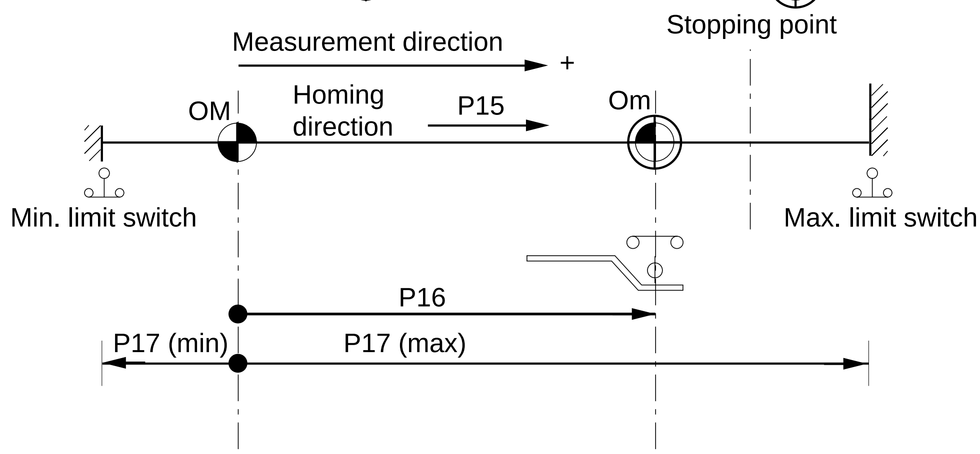
Case 1: Positive homing direction

The measurement origin «OM » and machine zero point«Om » coincide



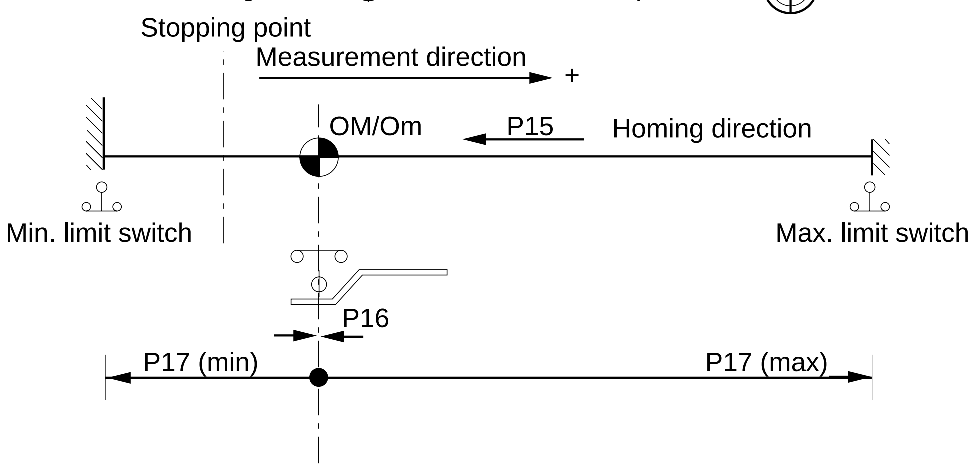
Case 2: Positive homing direction

The measurement origin «OM » and machine zero point «Om » coincide

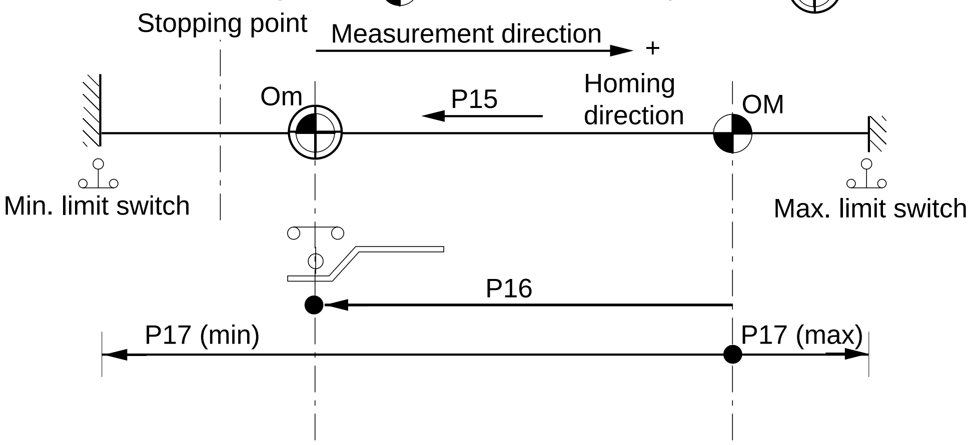


Case 3: Negative homing direction

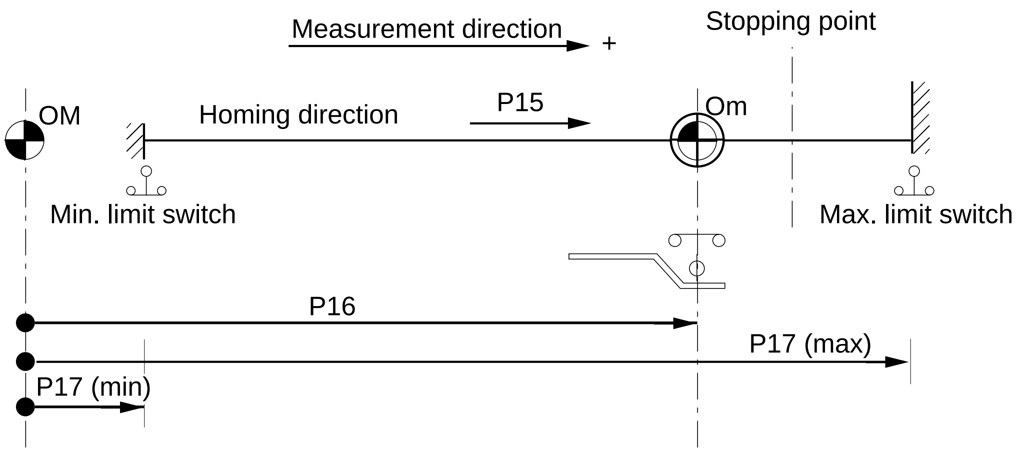
The measurement origin «OM » and machine zero point «Om » coincide



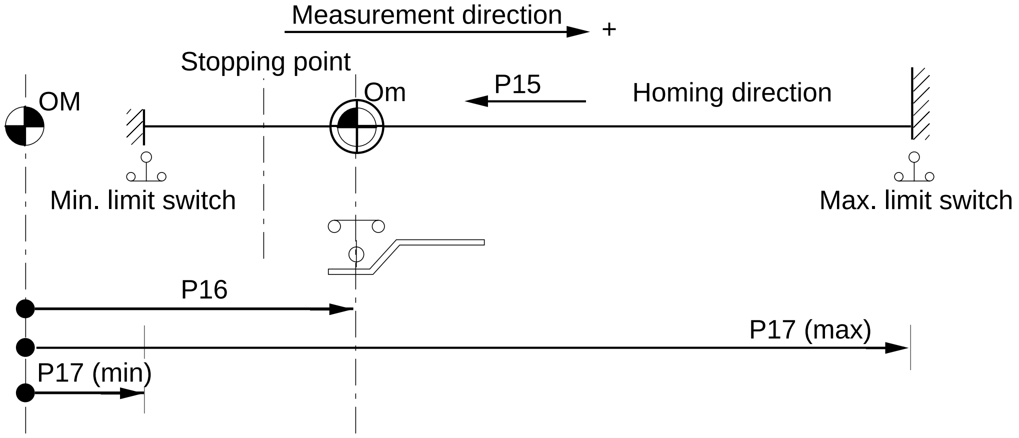
Case 4: Negative homing direction
The measurement origin «OM » and machine zero point«Om » do not coincide



Case 5: Positive homing direction
The measurement origin «OM » is not accessible



Case 6: Negative homing direction
The measurement origin «OM » is not accessible



7.3 Homing Direction

P15

Category	Travels on axes
Type 6	32-bit hexadecimal
No. of words	2

Description

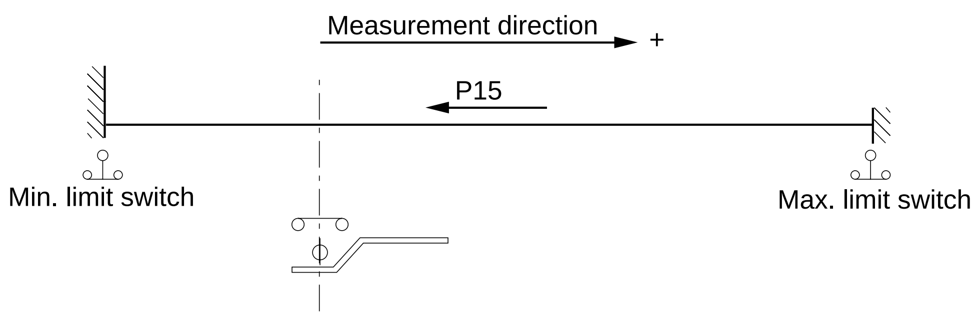
Defines the direction from which the datum switch is approached in homing mode.

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 a 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 a 1, the datum switch is not tested on the corresponding axis.

In homing mode, homing occurs on the first marker pulse encountered.

Programme Parameter

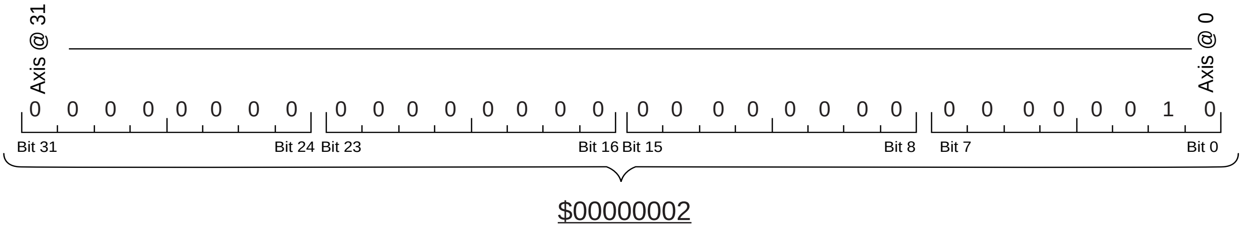
Programme parameter E933xx is used to read the value of the homing direction.

UNI-TE request (see UNI-TE Protocol User’s Manual)

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

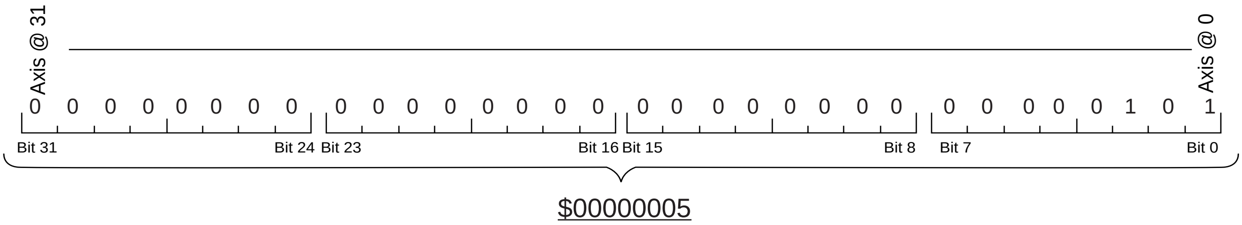
Example

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



Word N0	00000002

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



Word N1	00000005

7.4 Datum Switch Position in Machine Dimensions

P16

Category	Travels on axes
Type 2	Signed decimal
No. of words	32

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 (see 4.8) and deg/10000 for rotary axes.

To determine the sign of ORPOM, it is necessary to go to the desired measurement origin OM and set the initialisation point.

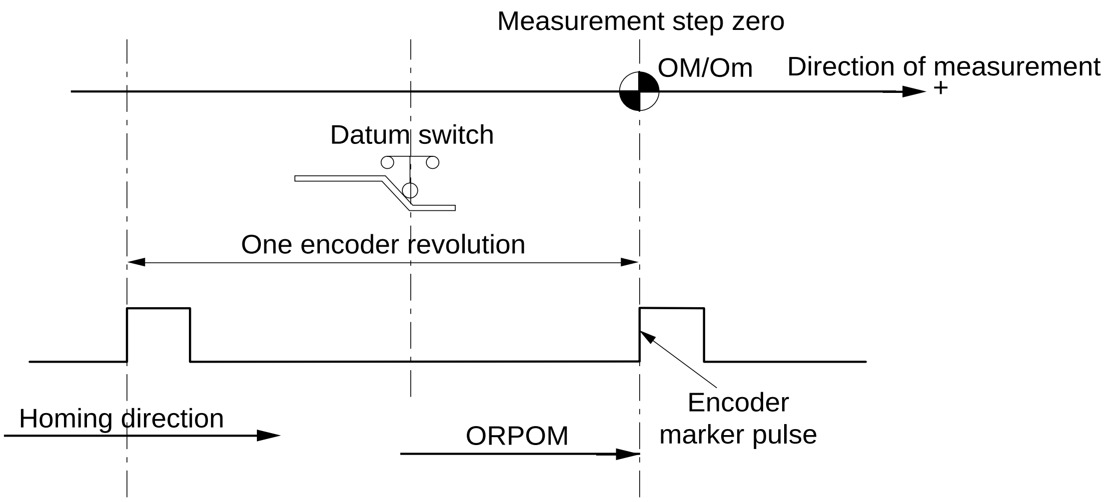
Each word contains the position of the datum switch on the corresponding axis in machine dimensions. The sign is determined by the direction from the measurement origin OM to the datum switch, taking into account the normalised axis direction.

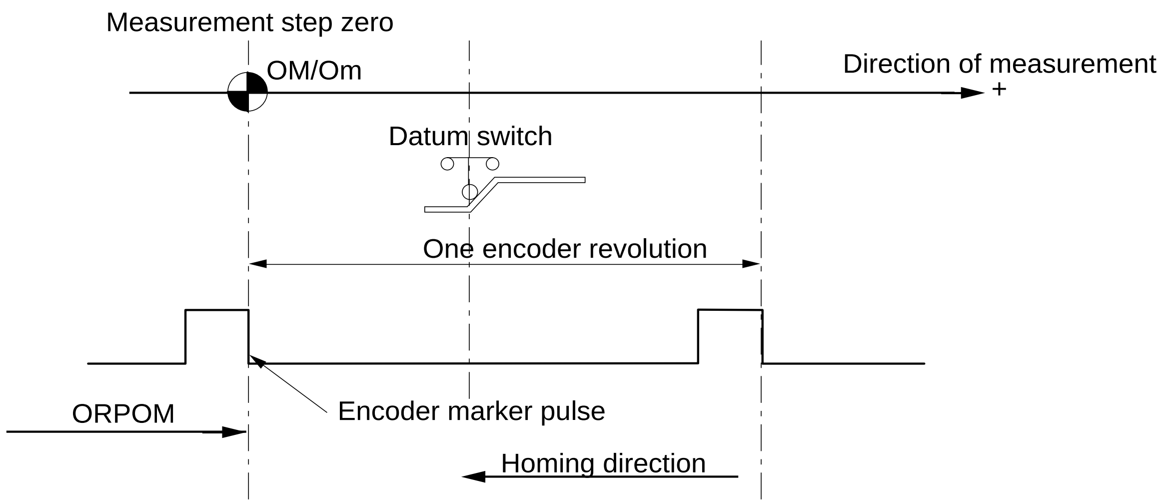
UNI-TE request (see UNI-TE Protocol User’s Manual)

The measurement origin on an axis can be modified by a UNI-TE 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 operation.





7.5 Axis Travel Limits

P17

Category	Travels on axes
Type 2	Signed decimal
No. of words	64

Description

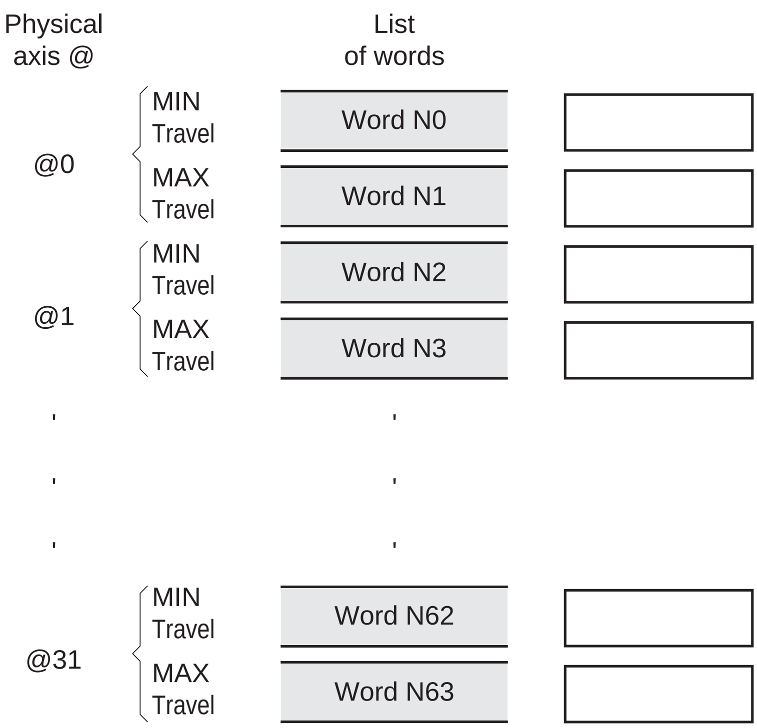
Defines the axis travel limits. Used to limit software travel in the measurement space.

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.

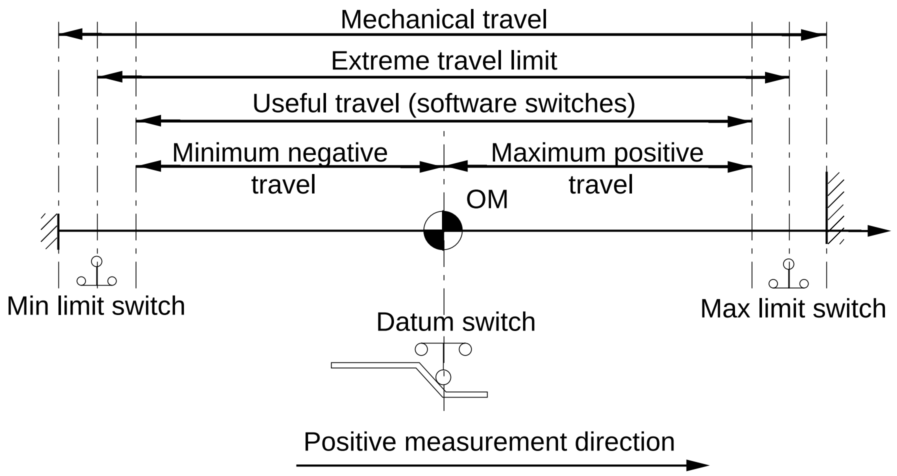
The second word of the pair gives the upper limit (maximum or positive) of MAX travel.



The values are expressed in the internal measurement unit: mm/10, mm/100, µm, µm/10 or µm/100 for linear axes (see 4.8) and deg/10000 for rotary axes.

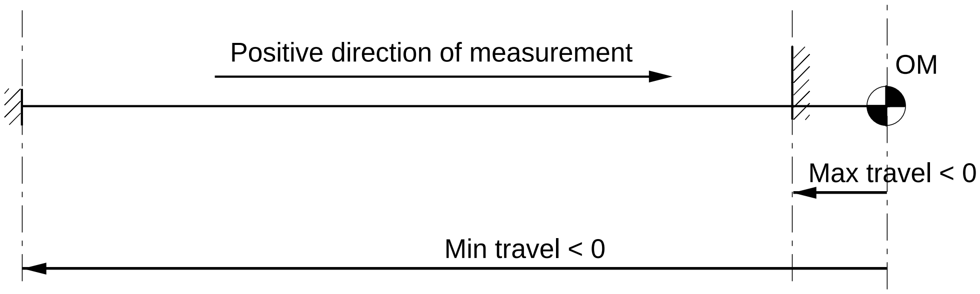
UNI-TE request (see UNI-TE Protocol User's Manual)

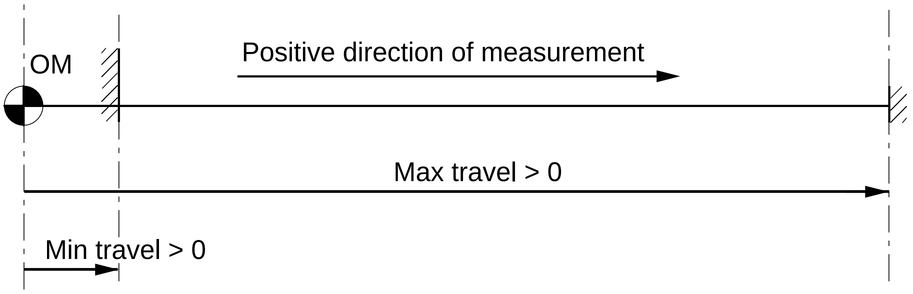
The travels on an axis can be modified by a UNI-TE request. A reset (CNC reset) resets the CNC to the state specified by P17.



To determine the sign of travel, it is necessary to go to the measurement origin and determine the minimum and maximum travel limits.

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





7.6 Backlash Error Compensation

P18

Category	Travels on axes
Type 1	Signed decimal
No. of words	32

Description

Used to correct positioning errors due to backlash.

Principle

The word number gives the physical axis address.

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 (see 4.8) and deg/10000 for rotary axes.

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.

UNI-TE request (see UNI-TE Protocol User's Manual)

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

7.7 Setting the Zero Points

Set the homing direction by P15 according to the location of the switches.

Axis @	Homing direction
Axis 0	
Axis 1	
Axis 2	
Axis 3	
Axis 4	
Axis 5	
Axis 6	
Axis 7	
Axis 8	

Turn on the CNC.

Home each axis by moving it manually to the datum switch.

Repeat the procedure for each of the axes.

REMARK: The state of the datum switch can be displayed by external parameter E930xx (where xx is the physical axis address).

7.8 Check of Homing

This test is used to check that the machine homing is repeatable.

It requires the use of a dial comparator.

Enter the following programme (for the X axis) from the CNC keyboard (see «Operator Manual»).

CAUTION

The programme below is given for guidance but must be adapted to the machine used.

%9990	
N1 G77 N5 N60	
N2 G9 @X500 F30	Position for the check
N3 M0 $CHECK THE POSITION	
N4 L10 = L10+1 G79 L10=<10 N1	
(TEN CHECKS)	
M02	
N5	
@X=X L0=0	Address equivalence
	(@X: Axis name; L0: Axis number)
N10	
L6 = 91100+L0	Homing not carried out (E911XX)
L1 = 90000+L0	Measurement of the axis (E900XX)
L2 = 91000+L0	Servo-controlled axis (E910XX)
L3 = 92000+L0	Enable datum switch (E920XX)
L4 = 93000+L0	Datum switch status (E930XX)
EL6 = 0	Datum switch declared for the axis
EL2=0 EL1=-1000 EL2=1 L5=EL1/1000	Measurement initialised at -1 (or 359 deg for a modulo axis)
G79 L5>0 N40	Jump to N40 is axis modulo
$LINEAR AXIS	
N20 G79 EL4=0 N30	Jump to N30 is the axis is not on datum switch
G52 G00 L5=L5-1 @XL5 G79 N20	Move off the switch by 1 mm
	Sign depending on homing direction
N30 EL2=0 EL1=-50000000 EL3=1 EL2=1	Measurement initialised at 50 m
	Sign depending on homing direction
G1 G52 @X0 G10 @L0>0 N60	Move to measurement origin until the value is zero

`$MODULO AXIS`	
`N40 G79 EL4=0 N50`	Jump to N50 if the axis is not on the datum switch
`L5=L5+1 G00 G52 @X-L5 G79 N18`	Move off the switch by 1 degree
	Sign depending on homing direction
`N50 EL2=0 EL3=1 EL1=100000 EL2=1`	Initialise the measurement
`G01 G52 G93 F0.1 @XL5 G10 L5=EL1/1000+180`	Move on the measurement origin by 180 degrees
`@L0<50 N60`	until the value > 0 degrees
`G79 N50`	
`N60 G94 F1000`	

Execute the programme for the axis concerned.

Modify and rerun the programme for each of the axes.

7.9 Setting the Travels on the Axes

Home all the axes.

Determine the travel limits from the data of Section 7.7 or by manual movement to the electrical switches.

Axis @	MIN travel	MAX travel
Axis 0		
Axis 1		
Axis 2		
Axis 3		
Axis 4		
Axis 5		
Axis 6		
Axis 7		
Axis 8		

The values are expressed in the internal measurement unit: mm/10, mm/100, µm, µm/10 or µm/100 for linear axes (see 4.8) and deg/10000 for rotary axes.

Enter the axis travel limits in the words of parameter P17.

8 Spindles

8.1 Data Tables

Spindle Definition

	Spindle 1	Spindle 2	Spindle 3	Spindle 4
Spindle present				
Spindle physical address (imposed address)	@24	@25	@26	@27
Reversed spindle reference				
Spindle origin (expressed in internal measurement units)				
Number of increments per encoder revolution				
Spindle origin (expressed in internal measurement units)				
Maximum spindle speed (expressed in internal measurement units)				
In-position window (expressed in internal measurement units)				
Spindle servo-loop gain (in rpm/rev)				

Spindle Speed Ranges

	Spindle 1		Spindle 2		Spindle 3		Spindle 4	
Spindle speed range (rpm/rev)	Min	Max	Min	Max	Min	Max	Min	Max
Range M40								
Range M41								
Range M42								
Range M43								
Range M44								
Range M45								

Spindle Definition

	Spindle 1	Spindle 2	Spindle 3	Spindle 4
Spindle present				
Physical spindle address (imposed address)	@24	@25	@26	@27
Lathe spindle				
Milling machine spindle				
Measured spindle				

8.2 Principle of Bidirectional Spindle Indexing

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

8.2.1 Spindle Indexing

Spindle indexing is activated by miscellaneous function M19 (see «Part Programming Manual»). 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 machine processor at the beginning of the indexing cycle and POSBR1 (spindle 1), POSBR2 (spindle 2), POSBR3 (spindle 3) or POSBR4 (spindle 4) 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.

8.2.2 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 as default.

A value for the indexed stop position is programmed:
- with the C address for a lathe group,
- with the EC address for a milling machine group.

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).

8.2.3 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).

8.2.4 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

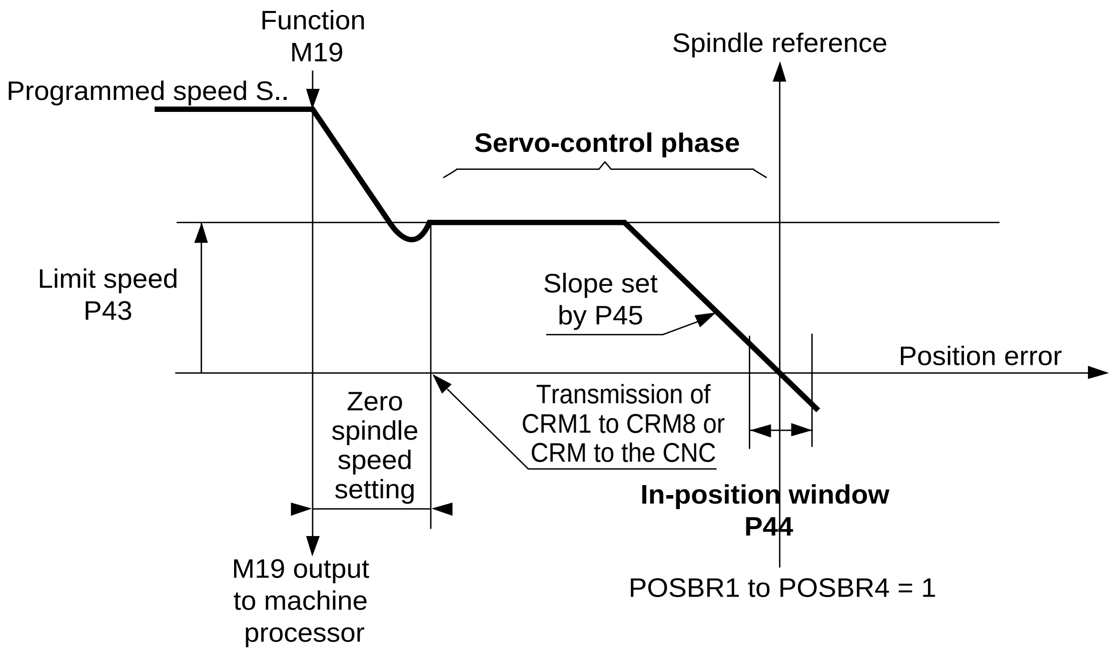
P45 is expressed in rpm/rev, i.e. 1/min.

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 a 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.



8.2.5 Spindle Indexing at Constant Acceleration

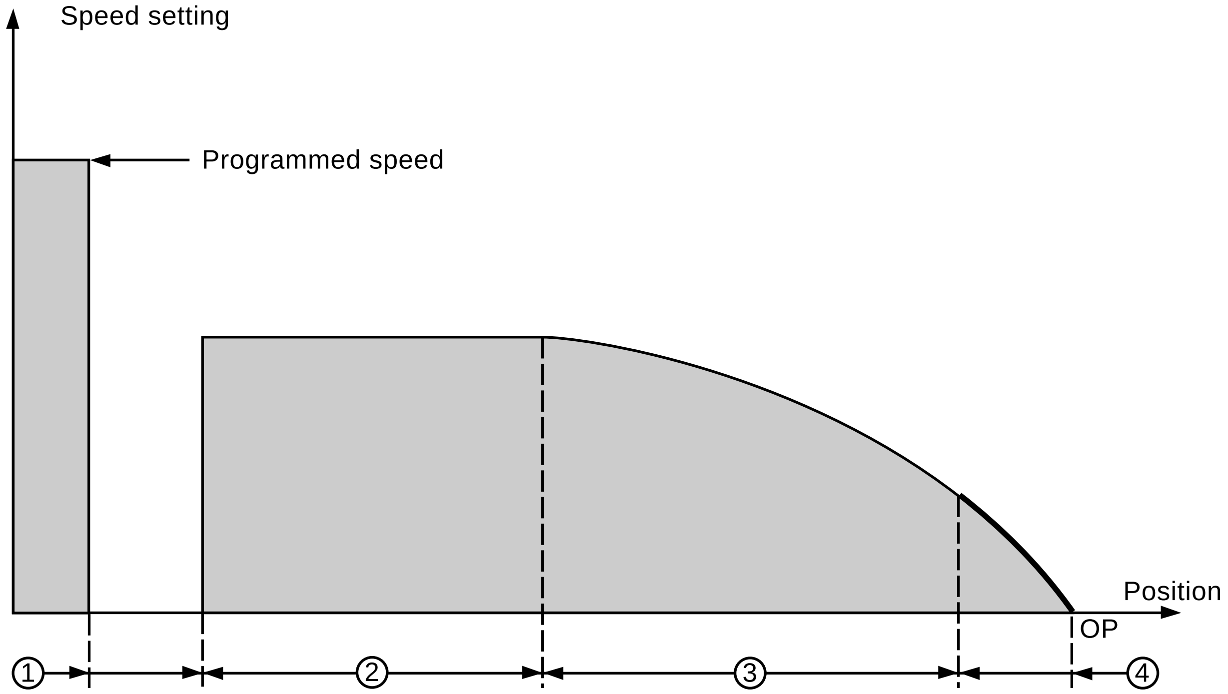
With spindle indexing, when bit 5 of words N1 to N4 of P6 is a 1, the positioning speed is at constant deceleration.

The deceleration is expressed in deg/s^2 in parameter P32 (see 6.4).

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 - setting = 0 up to real speed
< limit and CRM or CRMg = 1

2 - setting = limit speed until reaching
the deceleration region

3 - deceleration to the indexing point at
constant «gamma»

4 - servo-control at constant gain on
the indexing point

8.3 Rigid Tapping

This function allows tapping to be performed without using an extensible 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 performing 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 groups 1 to 5.

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

8.3.1 Procedure

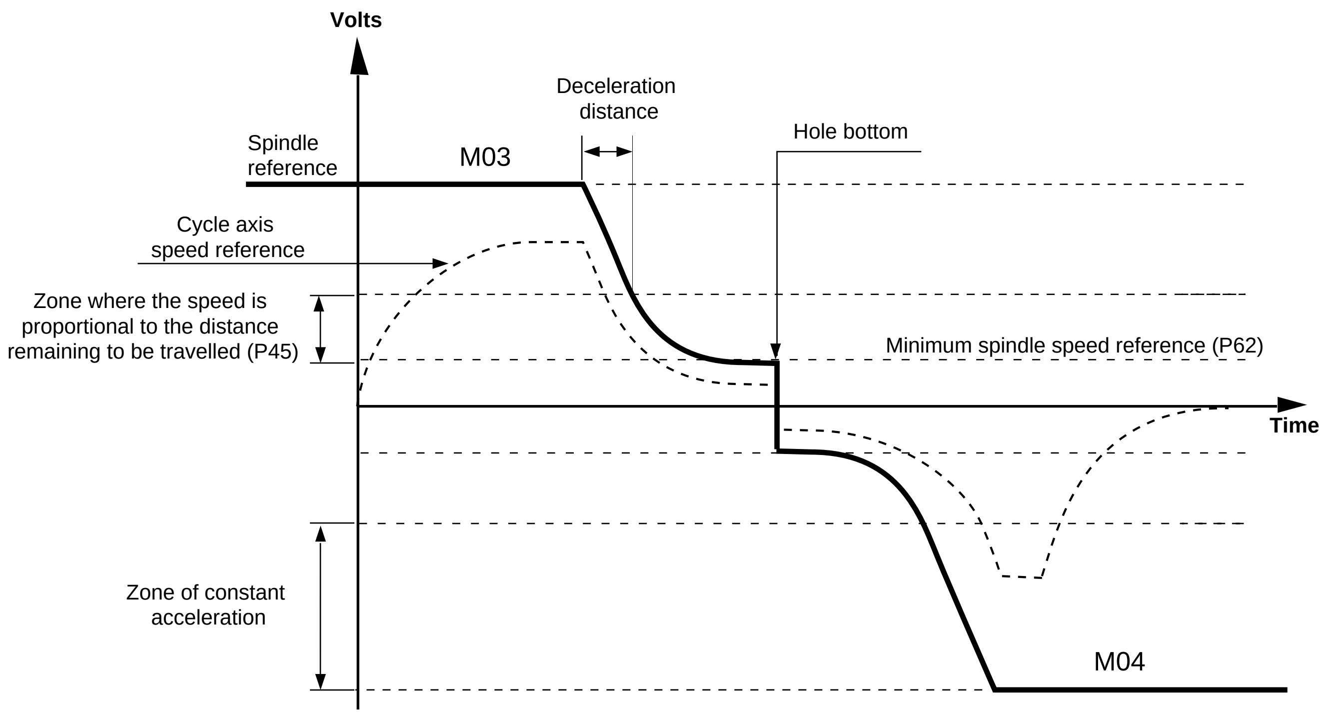
At the beginning of infeed, the cycle axis is synchronised on the spindle zero pulse. This synchronisation is repeated each cycle.

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).

In the hole bottom, a minimum spindle speed is applied to effectively reach 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.



8.3.2 Feed Stop (CYHLD)

The action of CYHLD for rigid tapping depends on machine parameter P7 word 1 bit 4:
- Bit = 0: CYHLD causes detapping before arriving in the hole bottom
- Bit = 1: CYHLD inhibited during tapping; at the end of the cycle, the system goes into CYHLD state.

8.4 Spindle Declaration

P6

Category	Axis declaration
Type 0	8-bit hexadecimal
No. of words	9

Description

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

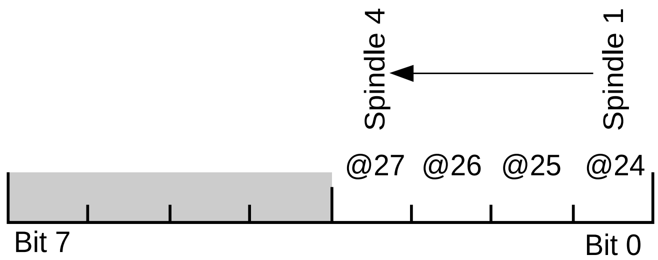
Principle

Word N0 - 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:



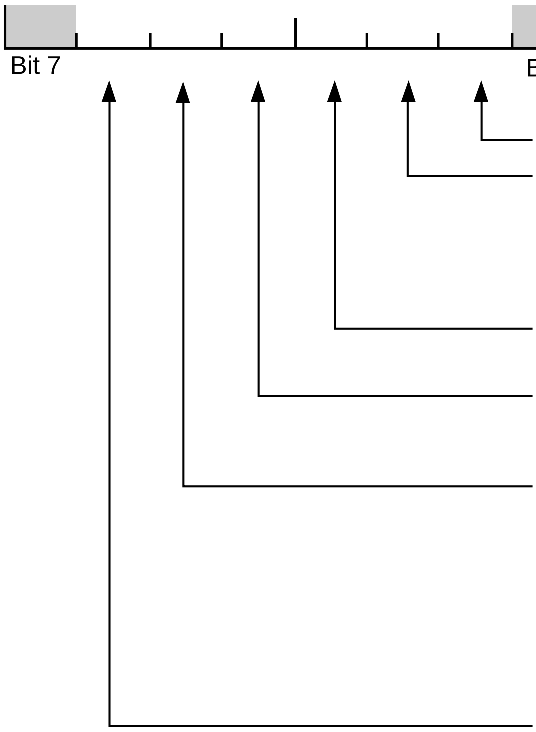
The bit is a 1 to indicate that the spindle is measured.

REMARKS: 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 - 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.

REMARK: Only bits 1 to 6 are significant.



1 = 1: spindle measurement reversal

Spindle type indexed by the CNC
- 0 for a lathe spindle (indexed on the position of the C axis)
- 1 for a milling machine spindle (indexed on the position addressed by function EC)

Bit 2 Measurement direction inoperative for DISC NT spindle

1: spindle can be indexed by the CNC (function M19 recognised)

Spindle speed programming format
- 0: S05
- 1: S32

1: with indexing, indicates that the positioning speed must be at constant acceleration. During spindle speed build-up, modification of programmed speed, stop programmed by M05 or with spindle indexing by M19, the system generates a ramp. The ramp is determined by the acceleration declared in P32, which must not be zero. The first word of the pair of words is assigned to the axis number used by the spindle.

1: declaration of machine speed parameters as rpmx10 and accelerations as deg/s^{2}x10.

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

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.

> ⚠ **CAUTION**
>
> 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: Spindle assignments to axis groups

- 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 group assigned:
- 0 for group 1
- 1 for group 2
- etc.

UNI-TE request (see UNI-TE Protocol User’s Manual)

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

8.5 Spindle Measurement Conversion

P40

Category	Spindles
Type 4	Unsigned decimal
No. of words	4

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	

REMARK: 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.

REMARK: The spindle modulo cannot be less than 4096.

UNI-TE request (see UNI-TE Protocol User's Manual)

The spindle measurement conversion can be modified by a UNI-TE 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)	$\longrightarrow$	360/4096 deg
If P40 =	5000 pts/rev The spindle modulo is equal to:	2^{13} (8192)	$\longrightarrow$	360/8192 deg
If P40 =	20000 pts/rev The spindle modulo is equal to:	2^{15} (32768)	$\longrightarrow$	360/32768 deg
If P40 =	65536 pts/rev The spindle modulo is equal to:	2^{16} (65536)	$\longrightarrow$	360/65536 deg

8.6 Spindle Reference Reversal

P41

Category	Spindles
Type 0	8-bit hexadecimal
No. of words	4

Description

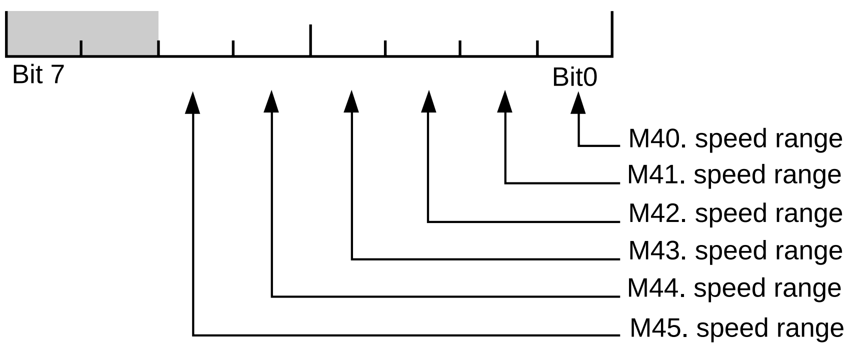
Used to define the spindle reference direction.

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 a 1 to indicate that the spindle reference is reversed.



UNI-TE request (see UNI-TE Protocol User's Manual)

The spindle reference directions can be modified by a UNI-TE request. A reset (CNC reset) resets the CNC to the state specified by P41.

8.7 Spindle 1 Speed Ranges

P46

Category	Spindles
Type 5	Unsigned decimal
No. of words	18

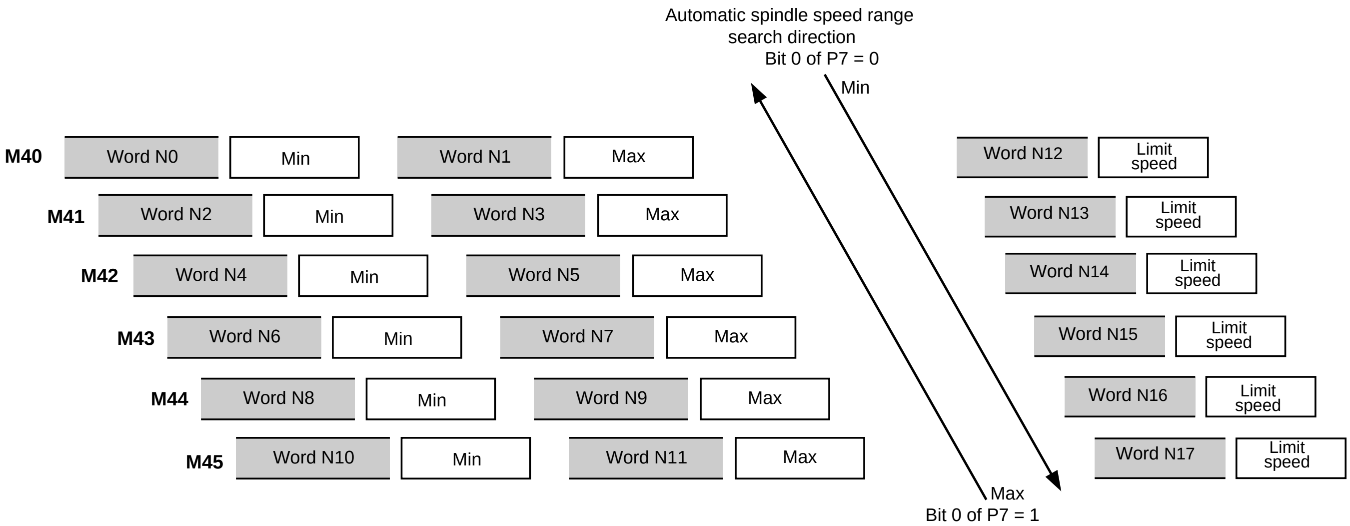
Description

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 in rpm.

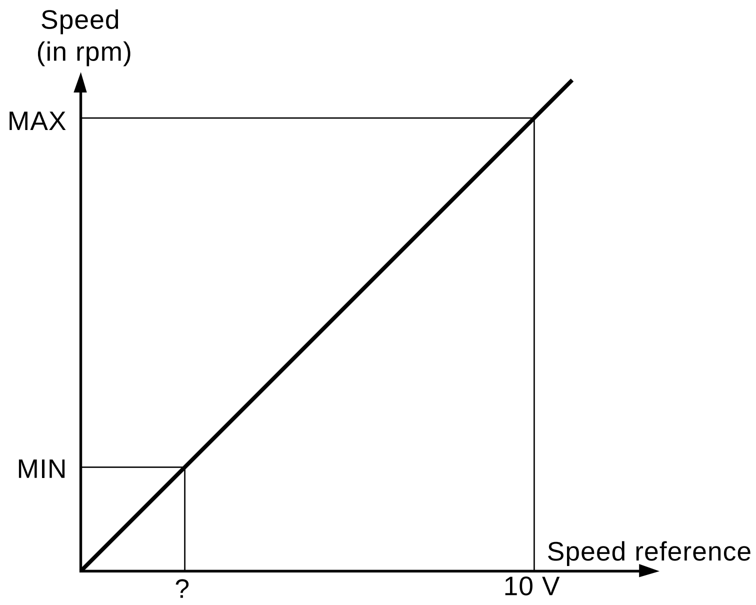
If the spindle speed programming format declared in P6 is S32, the speeds are expressed in hundredths of a revolution per minute.

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).

REMARK: 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.

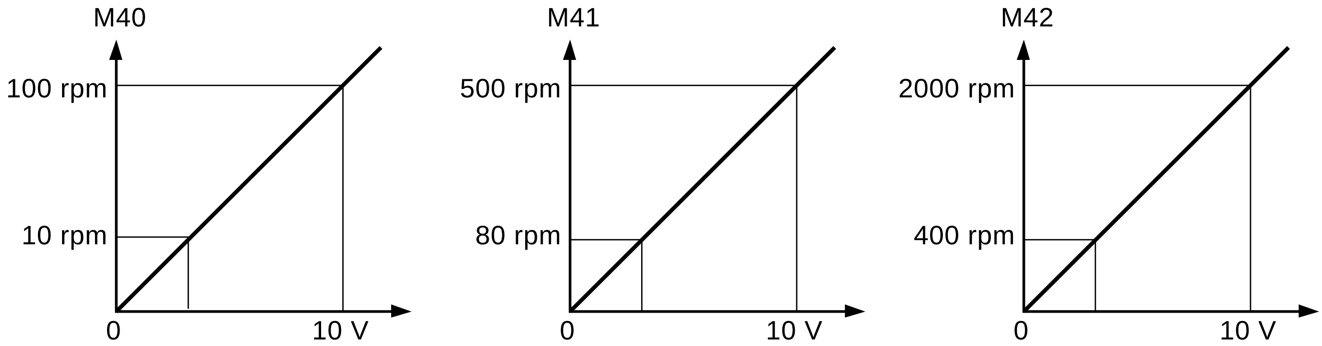


UNI-TE request (see UNI-TE Protocol User's Manual)

The speed ranges of spindle 1 can be modified by a UNI-TE 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



8.8 Spindle 2 Speed Ranges

P47

Category	Spindles
Type 5	Unsigned decimal
No. of words	18

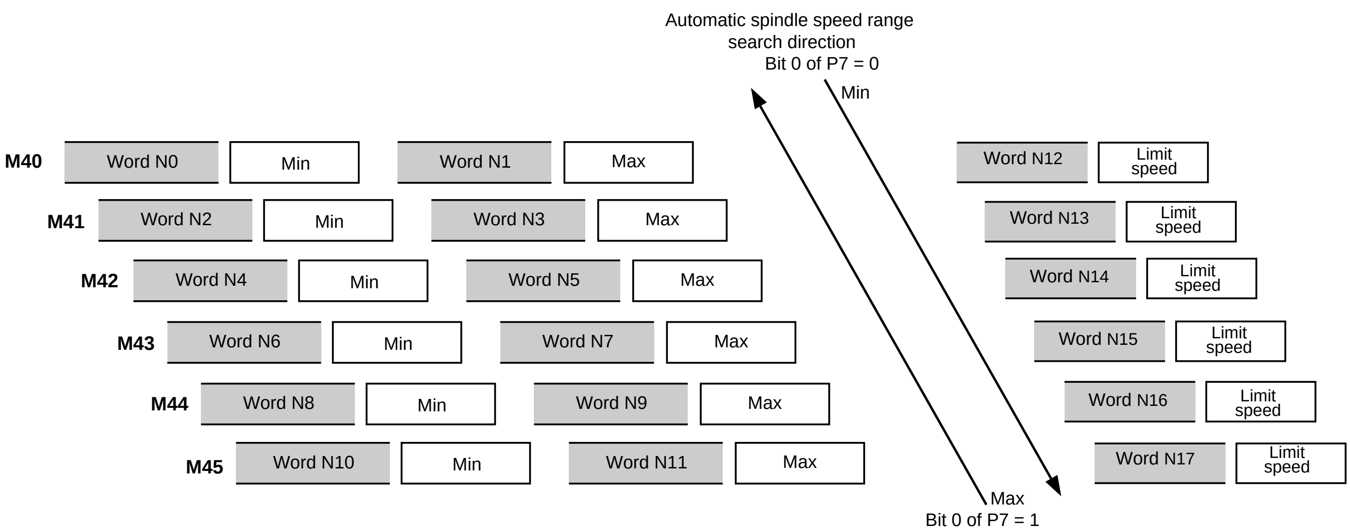
Description

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 in rpm.

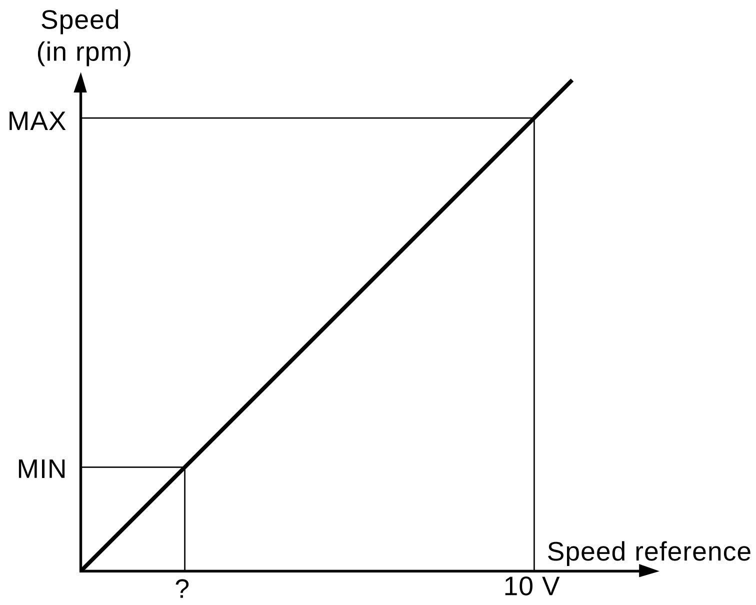
If the spindle speed programming format declared in P6 is S32, the speeds are expressed in hundredths of a revolution per minute.

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).

REMARK: 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.

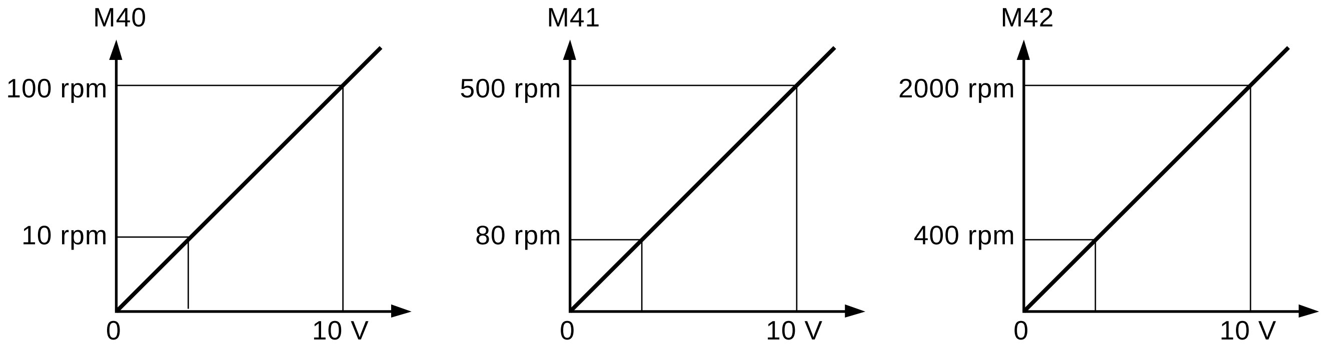


UNI-TE request (see UNI-TE Protocol User's Manual)

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

Example

	Min	Max
M40	10	100
M41	80	500
M42	400	2000



8.9 Spindle 3 Speed Ranges

P48

Category	Spindles
Type 5	Unsigned decimal
No. of words	18

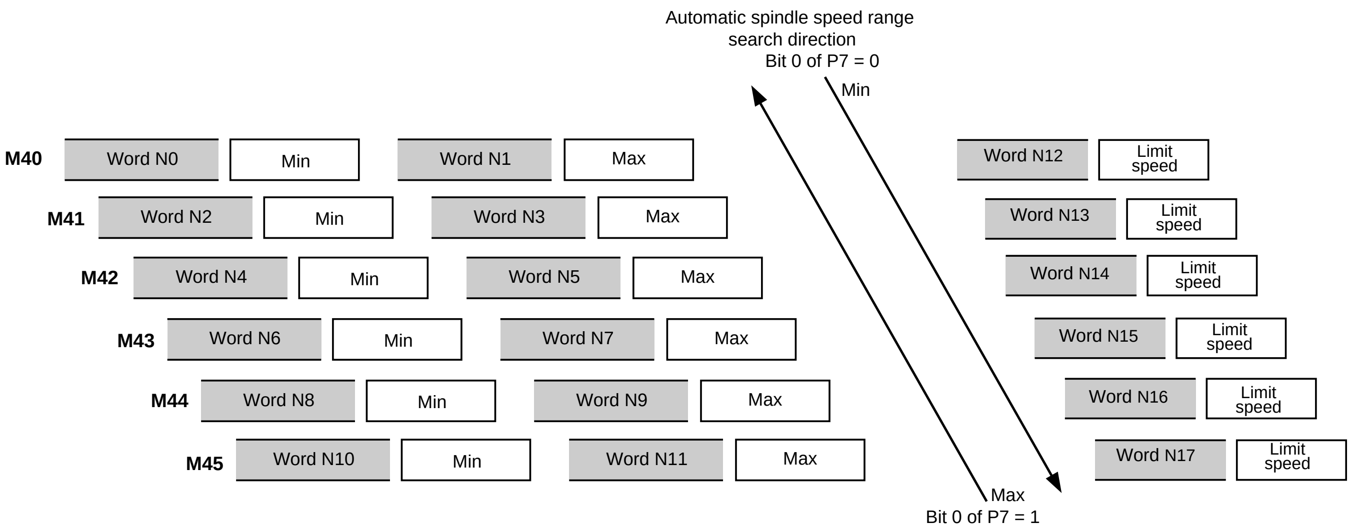
Description

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 in rpm.

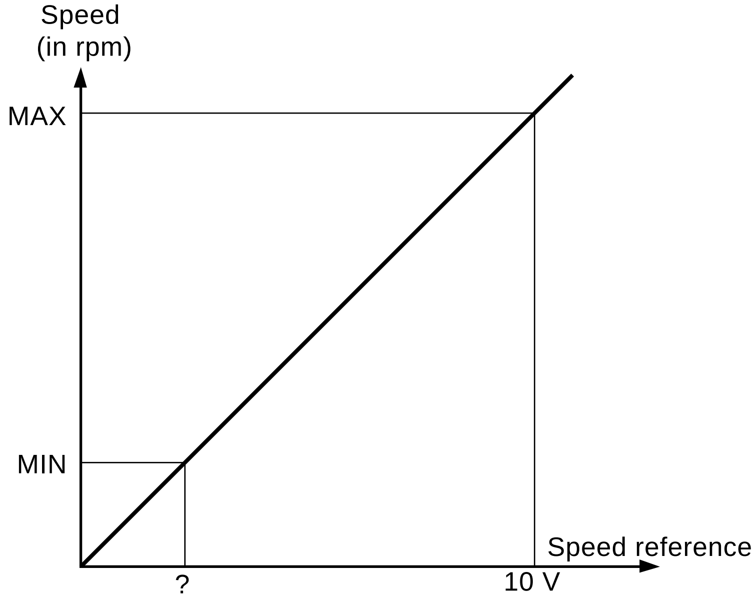
If the spindle speed programming format declared in P6 is S32, the speeds are expressed in hundredths of a revolution per minute.

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).

REMARK: 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.

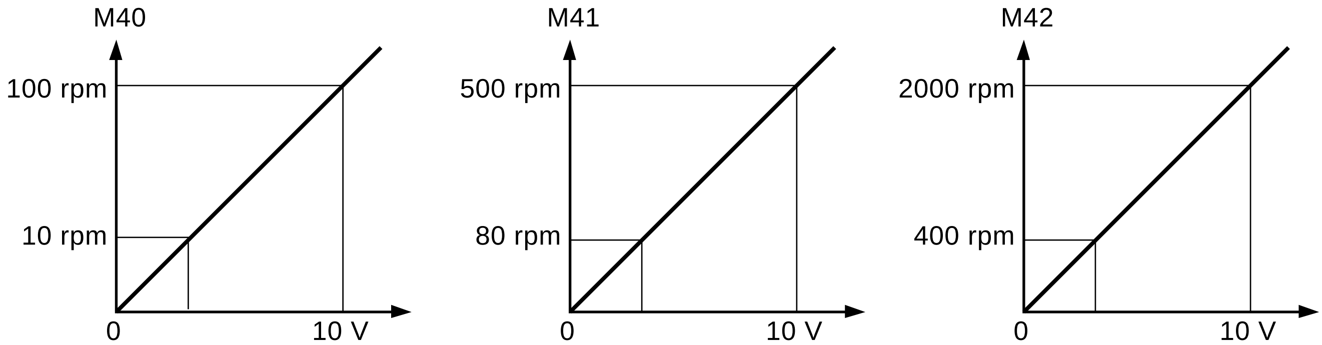


UNI-TE request (see UNI-TE Protocol User's Manual)

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

Example

	Min	Max
M40	10	100
M41	80	500
M42	400	2000



8.10 Spindle 4 Speed Ranges

P49

Category	Spindles
Type 5	Unsigned decimal
No. of words	18

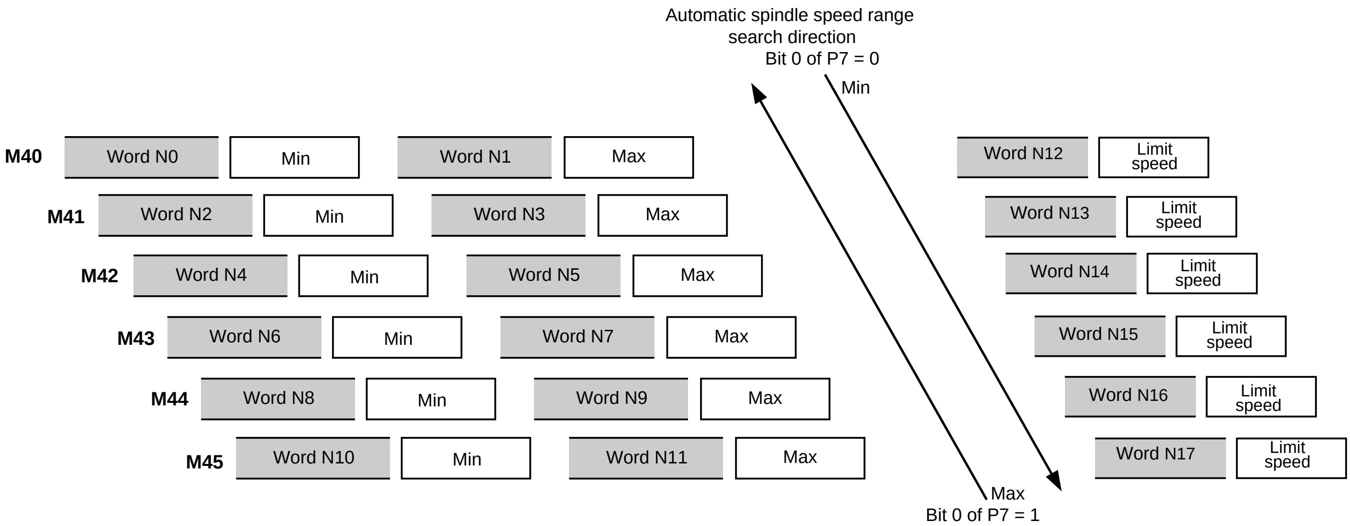
Description

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 in rpm.

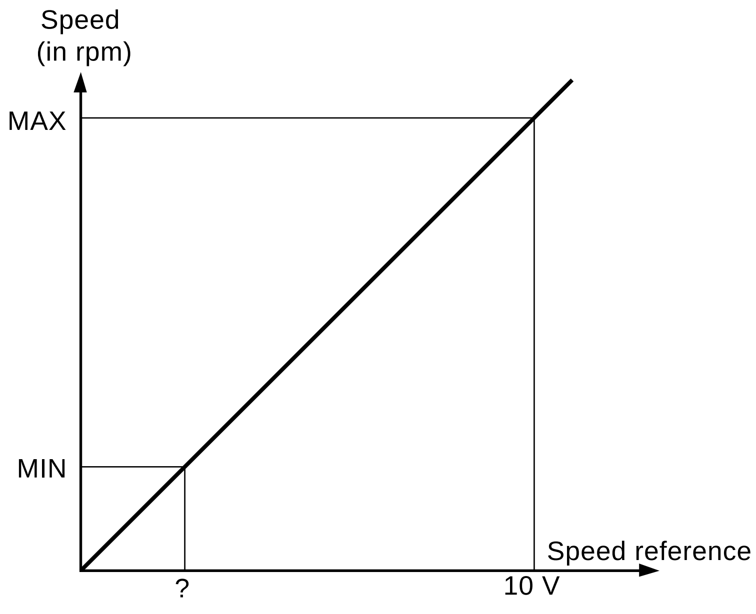
If the spindle speed programming format declared in P6 is S32, the speeds are expressed in hundredths of a revolution per minute.

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).

REMARK: 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.

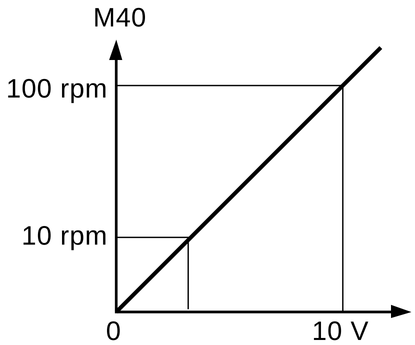


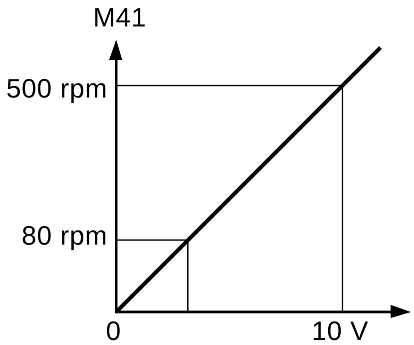
UNI-TE request (see UNI-TE Protocol User's Manual)

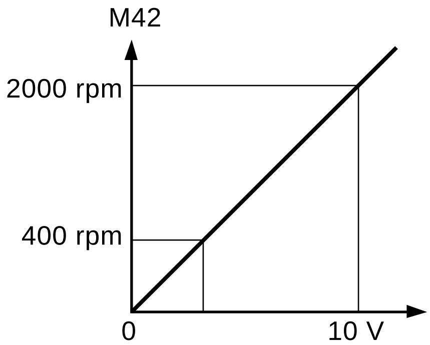
The speed ranges of spindle 4 can be modified by a UNI-TE request. A reset (CNC reset) resets the CNC to the state specified by P49.

Example

	Min	Max
M40	10	100
M41	80	500
M42	400	2000







8.11 Spindle Origins

P42

Category	Spindles
Type 4	Unsigned decimal
No. of words	4

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 the 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.

UNI-TE request (see UNI-TE Protocol User’s Manual)

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

8.12 Maximum Speed During Spindle Indexing

P43

Category	Spindles
Type 5	Unsigned decimal
No. of words	4

Description

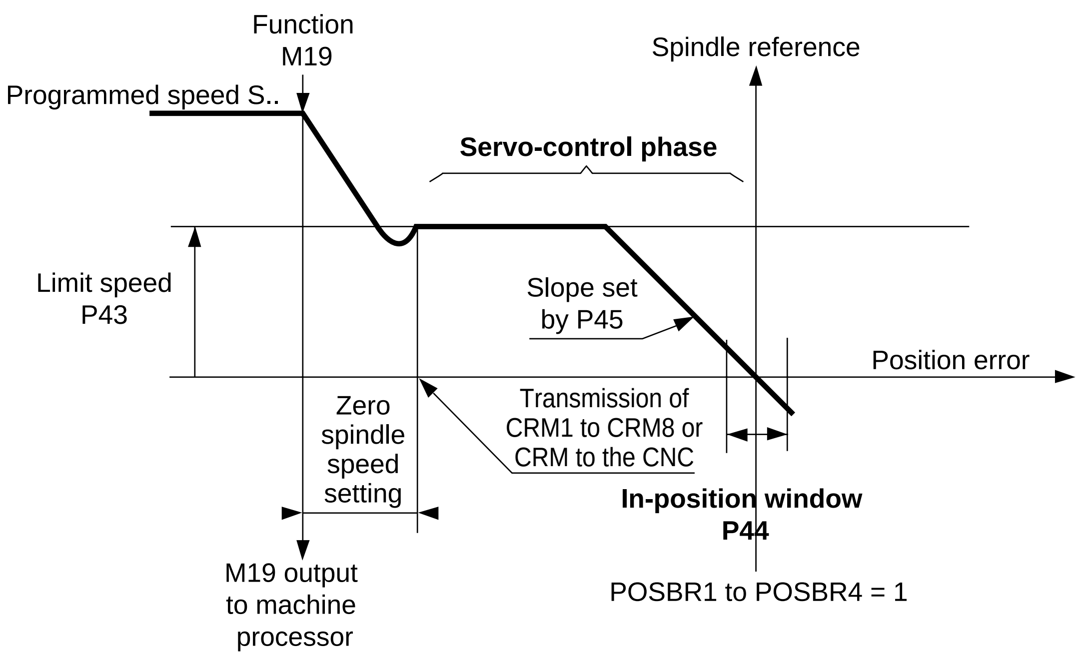
Used to define the maximum spindle speed during indexing.

Principle

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 in rpm.



Programme Parameter

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.

UNI-TE request (see UNI-TE Protocol User’s Manual)

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

8.13 Spindle Indexing In-Position Window

P44

Category	Spindles
Type 5	Unsigned decimal
No. of words	4

Description

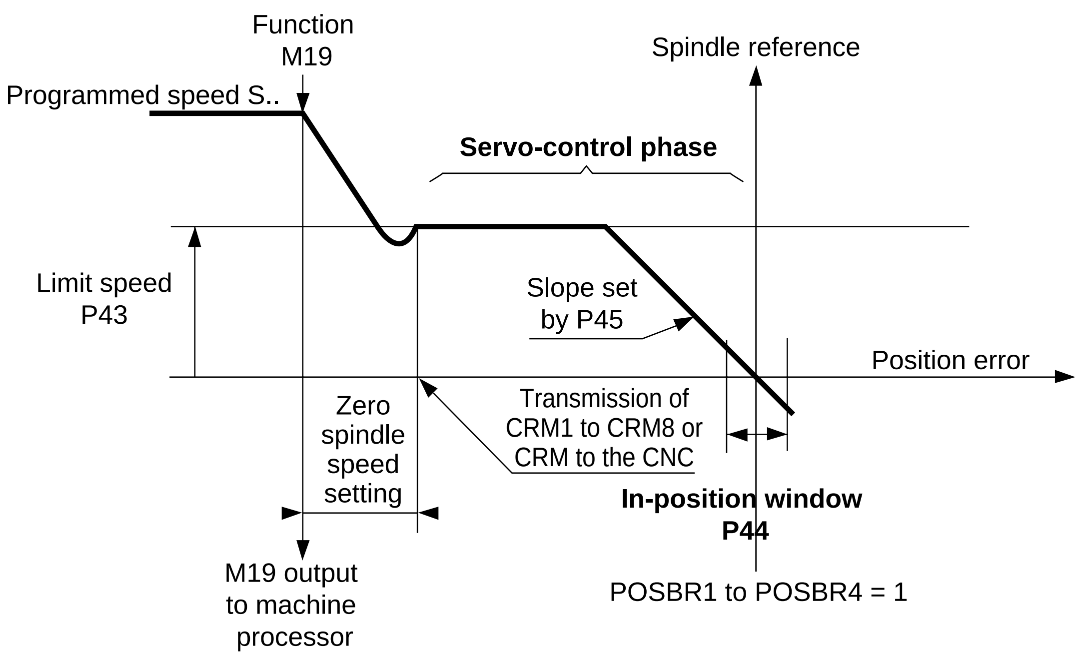
Used to define the spindle servo-loop accuracy.

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 (see 8.5).



Programme Parameter

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.

UNI-TE request (see UNI-TE Protocol User’s Manual)

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

8.14 Spindle Indexing Servo-Loop Gain

P45

Category	Spindles
Type 5	Unsigned decimal
No. of words	4

Description

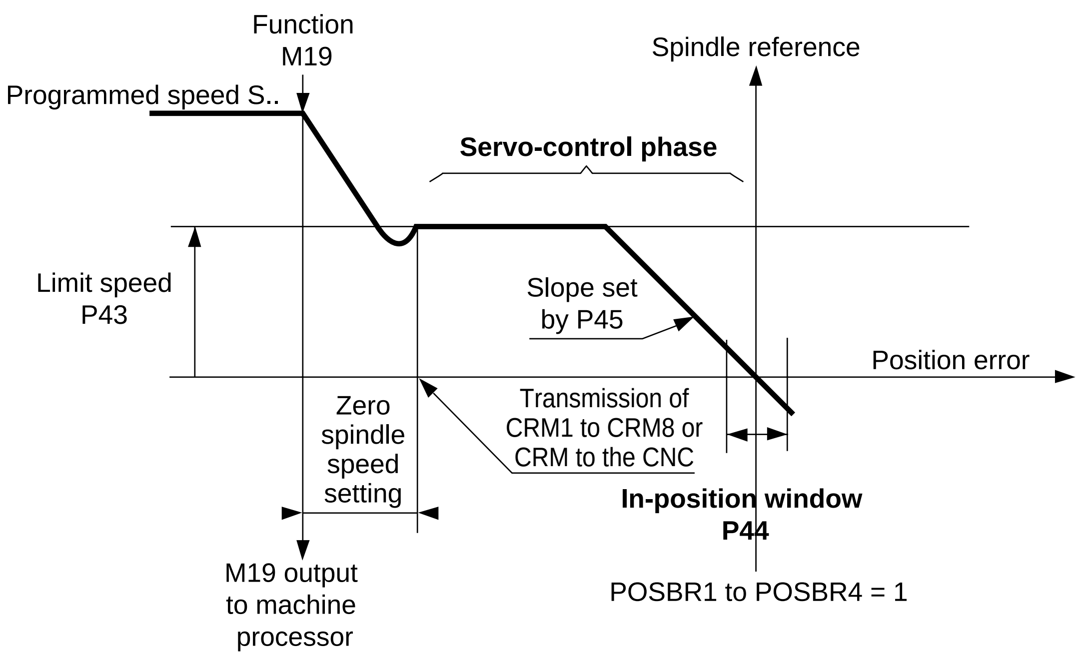
Used to define the spindle servo-loop ramp.

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 in rpm/rev.



Programme Parameter

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.

UNI-TE request (see UNI-TE Protocol User's Manual)

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

8.15 Spindle Acceleration and Minimum Spindle Reference in Hole Bottom

P62

Category	Spindles
Type 1	Signed decimal
No. of words	8

Remark: Concerns only rigid tapping.

Description

Used to define the spindle acceleration and minimum spindle speed reference in the hole bottom for spindles 1 and 2.

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 in 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

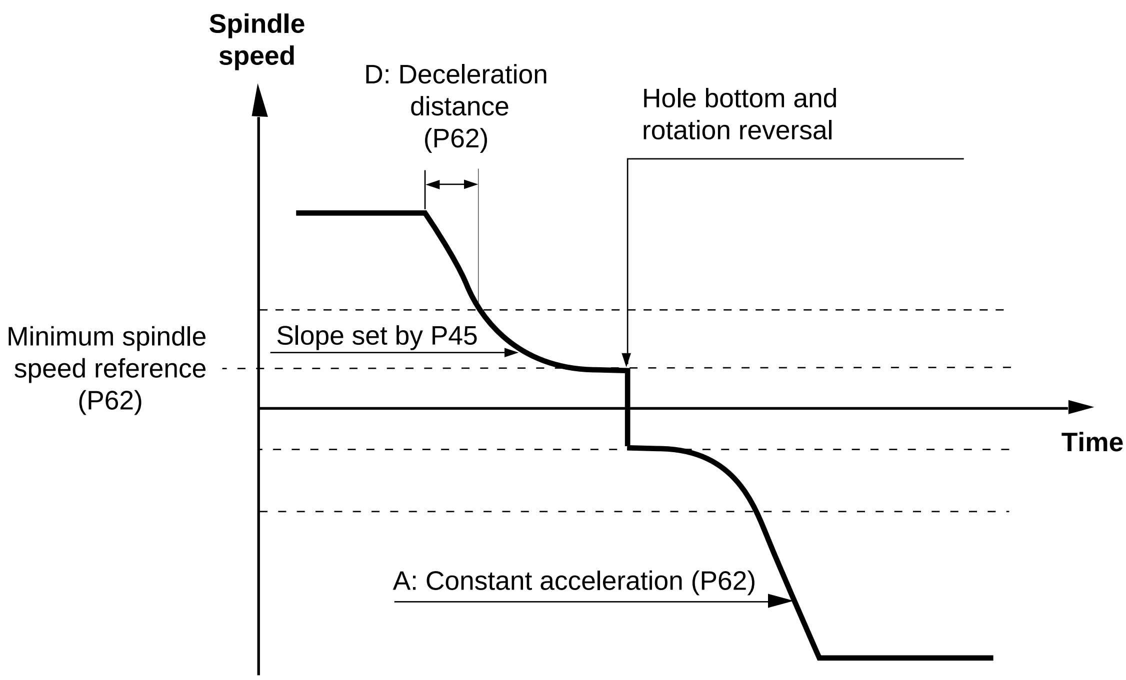
Word N1, Word N3, Word N5, Word N7

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.

REMARK: Refer to 8.16 (parameter P63) for the method used to set this parameter.



8.16 Cycle Axis Integration Time Constant, Cycle Axis Position Anticipation Time Constant and Spindle Zero Crossing Anticipation Time Constant

P63

Category	Spindles
Type 1	Signed decimal
No. of words	24

Remark: Concerns only rigid tapping.

Description

This parameter includes eight groups of three words, i.e. one per axis group.

Axis Group		List of words	
Group 1	Cycle axis integration tc	Word N0	
	Cycle axis position anticipation tc	Word N1	
	Spindle zero crossing anticipation tc	Word N2	
'		'	
'		'	
Group 8	Cycle axis integration tc	Word N22	
	Cycle axis position anticipation tc	Word N23	
	Spindle zero crossing anticipation tc	Word N24	

Word N0

This word 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

This word 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 $\Sigma p/V$. 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

This word 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/s^2.

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

CAUTION

The values entered are constant for all the axes of a group. 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 axis group (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.

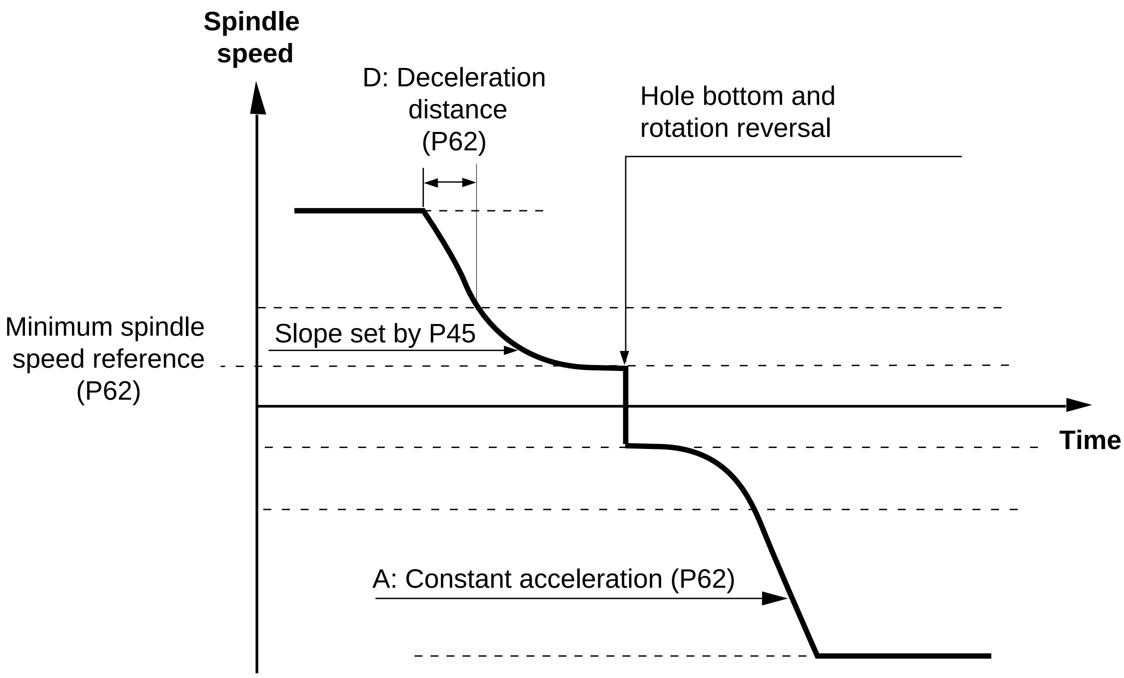
Setting P62
- Word 0: keep the default value of 10,
- Word 1: set the value supplied by the servo-drive manufacturer or keep the default value of 100.

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 upon reversal.

⚠ **CAUTION**
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.

9 Miscellaneous Functions

9.1 Data Tables

Automatic spindle speed range search direction	**Increasing**	**Decreasing**

Transmission of M and T functions	**Yes**	**No**
In SEQNO mode		
In TEST mode		

Choice of feed rates	**mm/min**	**mm/rev**

Measurement units	**Inches**	**Metric**

Sampling period	µs
Minimum block execution time	ms
Number of E8xxxx parameters used for interaxis calibration	
Programme stack size bytes	bytes

CNC Screen Hard Copy

Line No.	

Print type	**Black and White**	**Grey levels**	**Colour**

Colour chart	**Colour monitor**	**B&W monitor**	**B&W monitor (with grey levels)**

9

Colour for PROCAM	**Black and White**	**Colour**

Graphics on lathe	**Normal**	**With mirroring**

Dynamic graphic drawing from:	**Reference**	**Measurement**

Xoff replacement character	
BS replacement character	
Area 1 memory size (Customer area)	kbytes
Area 2 memory size (OEM area)	kbytes
Area 3 memory size (NUM area)	kbytes

Type of machine for PROCAM	**Milling machine**	**Lathe**	**Combined machine**

Single slide or multislide lathe for PROCAM	**Single**	**Multi**

Display languages	**French**	**English**	**German**	**Italian**	**Spanish**	**Swedish**

Machine processor programming language	**Ladder**	**Assembler**

9.2 Miscellaneous Function Parameter Setting

P7

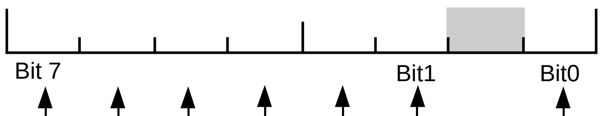
Category	Miscellaneous
Type 0	8-bit hexadecimal
No. of words	2

Description

Definition of miscellaneous functions.

Principle

Word N0



Automatic spindle speed range search direction.
If bit 0 = 0, the search for the speed ranges (parameters P46 to P49) is made in the increasing direction.
If bit 0 = 1, the search for the speed ranges is made in the decreasing direction.

Transmission of decoded M and T functions to the machine processor in SEQNO mode.
If this bit equals 1, the functions are transmitted to the machine processor.

For CNC software at index L1 and above, this bit can be set to access indexed or unindexed programmes called by M functions
If this bit = 0, unindexed programme
If this bit = 1, indexed programme

Measurement units
If this bit equals 0, the system is initialised in metric units (G71) by a reset or M02.
If this bit a 1, the system is initialised in inches (G70).

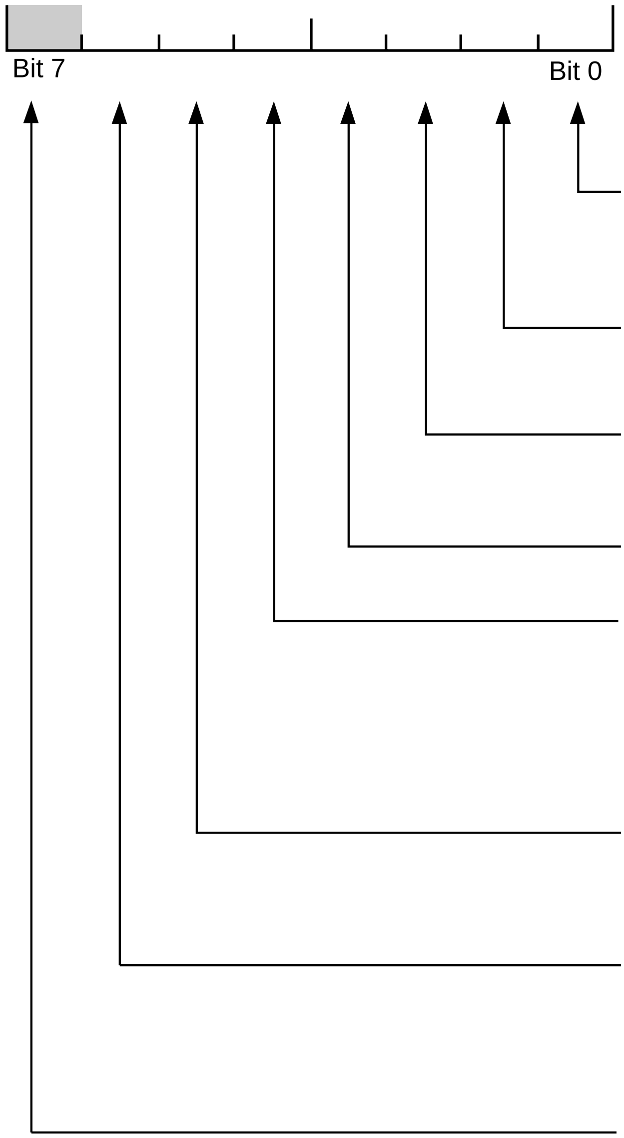
Forces the T functions to 0 during a reset (reset, M02, power on).
If this bit equals 1, the value is forced to 0.

Enables write and selection of the interaxis calibration tables.
If this bit equals 1, write of parameter E94000 by programming other than exchanges by protocols is inhibited.
If this bit equals 0, write of parameter E94000 by programming is enabled.

Transmission of M and T functions to the machine processor in TEST mode.
If this bit equals 1, the functions are transmitted to the machine processor.

Automatic spindle speed range search enable
If this bit = 0, automatic speed range search is enabled
If this bit = 1, automatic speed range search is inhibited.

Word N1



Choice of feed rate programming unit.
If this bit equals 1, the system is initialised in mm/rev (G94).
If this bit equals 0, the system is initialised in mm/min (G95).

Forcing of MANUAL mode.
If this bit is a 1, miscellaneous function M12 forcing the MANUAL mode is processed.

Homing selection
If this bit is a 0, homing from the keyboard is enabled.
If this bit is a 1, homing from the keyboard is inhibited.

Subroutine call on a reset
If this bit is a 1, subroutine call on a reset is enabled.

Feed stop for rigid tapping
If this bit is a 0, detapping is commanded before reaching the hole bottom.
If this bit is a 1, feed stop is inhibited during the tapping phase; where applicable, the system is in Feed Stop state at the end of the cycle.

Speed variation in S
If this bit is a 0, linear speed variation is enabled.
If this bit is a 1, gradual acceleration is enabled.

G52 prohibited when an inclined plane is valid.
If this bit is a 0, programming an inclined plane axis in G52 returns error 24.
If this bit is a 1, programming an inclined plane axis in G52 is enabled.

For CNC software at index LS and above
This bit enables or inhibits detection of error 149
By default, the bit = 0 and detection is enabled.

Programme Parameter

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

9.3 Subroutine Call by M Function

P35

Category	Miscellaneous
Type 5	Unsigned decimal
No. of words	32

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.

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

Example

Word 0	6
Word 1	9206
Word 2	85
Word 3	9285

Function M6 calls subroutine %9206.

Function M85 calls subroutine %9285.

CAUTION

If the system has at least one PLC axis group, the subroutines for the CNC axis groups must be indexed in the part programmes (applies to CNC software up to index H; unnecessary above).

9.4 Sampling Period

P50

Category	Miscellaneous
Type 5	Unsigned decimal
No. of words	2

Description

Defines the CNC sampling period and the sampling period for the QVN cards.

Word N0

Defines the CNC sampling period (T) in µs.
- The period is 5000 µs for 1060 SI and 1060 SII.
- The period is 6000 µs for 1020, 1040 and 1050.

With THE DISC NT function, 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 Parameter

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

Programme parameter E41007 gives the recommended value of the RTC in µs.

Differences Related to the UC SII CPU

The sampling period must be a multiple of 2 ms. The default value is 6 ms.

If the sampling period is not a multiple of 2 ms, the following message appears when the system is initialised:

```
WARNING : SAMPLING PERIOD
IS NOT A MULTIPLE OF 2 ms
```

The system operates normally, but the sampling period is rounded down to the next lower multiple of 2 ms.

Word N1 (for CNC software at index H and above - Only for 1060 SI and 1060 SII)

Defines the sampling period for all the QVN cards in the CNC rack. The range of permissible values is from 400 µs to 1000 µs in steps of 50 µs.

Any value outside this range is limited to 400 µs or 1000 µs.

> ⚠ **CAUTION**
>
> The ratio of the CNC sampling period to the QVN sampling period must be an integer value.

If one of these requirements is not satisfied, one of the following error messages is displayed at initialisation and mode change is inhibited:

- Sampling period outside range or not in the step:
  SAMPLING PERIOD QVN MUST
  BE BETWEEN 0.4 ms and 1 ms
  BY STEP OF 0.05 ms
- Ratio not an integer:
  SAMPLING PERIOD MUST BE A
  MULTIPLE OF PERIOD QVN AND EQUAL
  AT LEAST AT 2 ms FOR AXIS QVN.

9.5 Minimum Block Execution Time

P51

Category	Miscellaneous
Type 5	Unsigned decimal
No. of words	2

Description

Defines the minimum time for executing a block by interpolation.

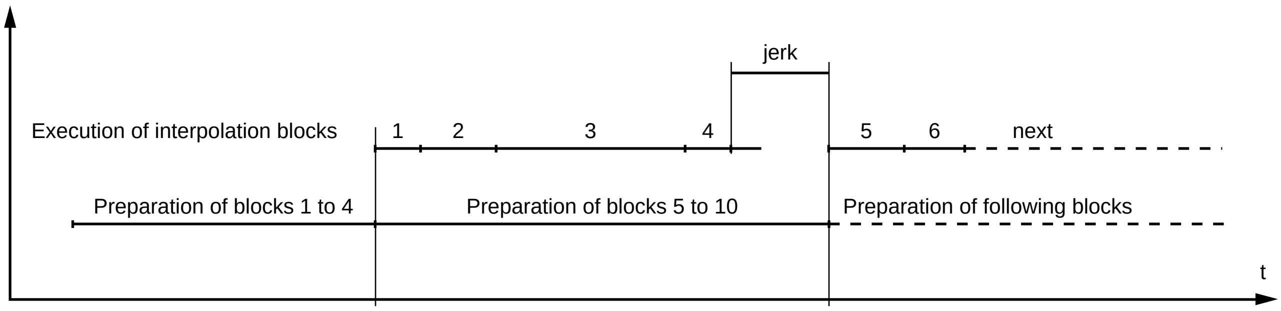
Principle

Word N0

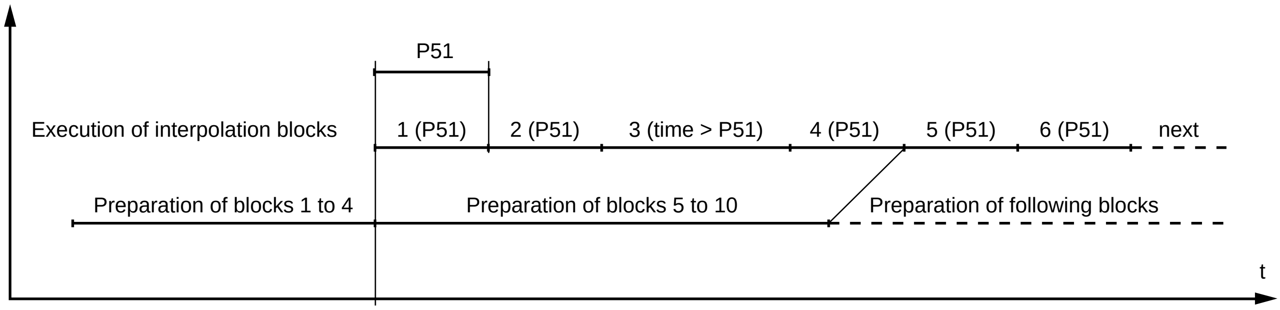
The block execution time is displayed in milliseconds. The default value is 20 ms.

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 axis groups and axes, the longer the time required to prepare the blocks. The minimum block execution time must be set according to the system configuration:

Number of axes	2-8	9-32
Number of axis groups	1-3	4-8
P51	6-12 ms	20 ms (default)

It should be noted that too high a minimum block execution time can penalise system performance during execution of short blocks.

Word N1

Not significant.

Programme Parameter

Programme parameter E32000 is used to modify the minimum block execution time in a part programme.

9.6 Interaxis Calibration Table and Programme Stack Reservation

P58

Category	Miscellaneous
Type 5	Unsigned decimal
No. of words	3

Description

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

Principle

Word N0

Used to allocate the interaxis calibration table.

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

The parameters are reserved by increasing order from E81000 to E81xxx and E82000 to E82xxx.

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

REMARK: 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 graphic group and axis groups.

The value entered corresponds to the size in word (2 bytes) for one group. The minimum stack size must be above 10 kbytes.

REMARK: The total programme stack size is displayed on the «DIR.» page.

Example

Word 0	200

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

Word 1	20480

The programme stack has a size of 40 kbytes.

Word N2

Size in KB of the AP and MMI programme execution area (default 512).

9.7 Graphic Element Configuration

P59

Category	Miscellaneous
Type 0	8-bit hexadecimal
No. of words	4

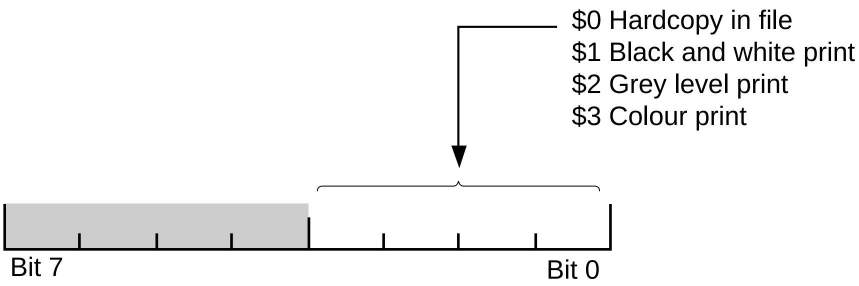
Description

Used to configure the graphic elements of the system.

Principle

Word N0

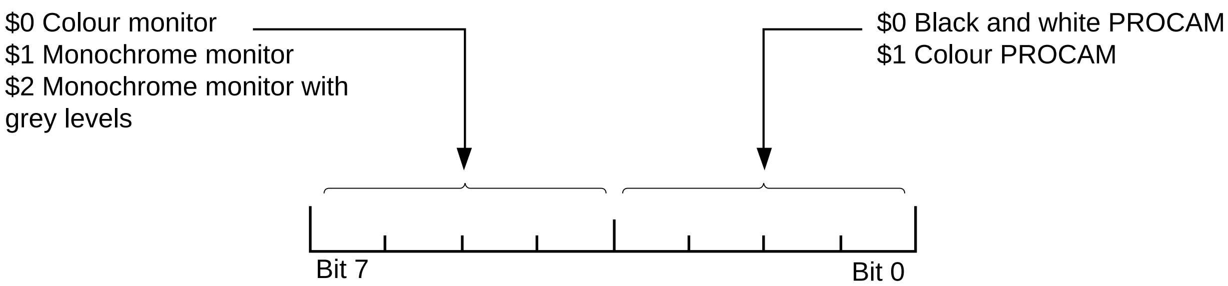
The low half-byte is used to define the print type for hard copies.



Word N1

Used to select, according to the monitor:
- the colour chart,
- colour or black and white display for PROCAM.

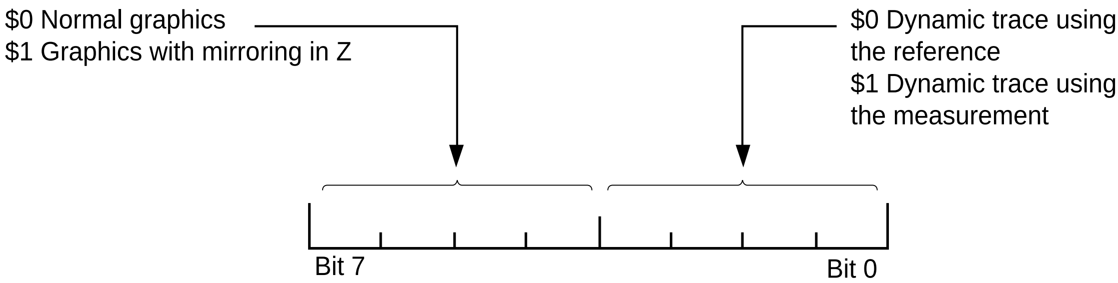
The high bits represent the colour chart and the low bits determine the choice of colours for PROCAM.



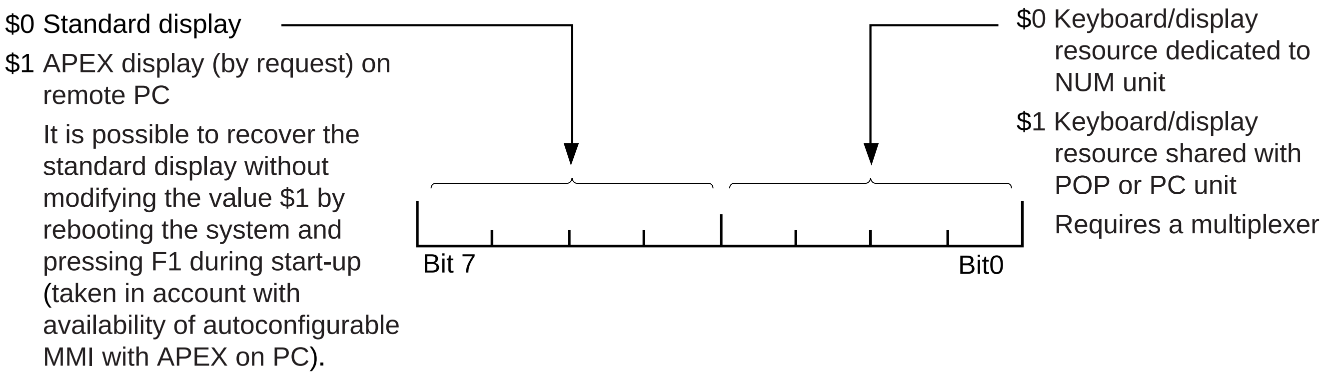
Word N2

This word assigns the graphic display on the system.

The high bytes are valid only for lathe axis groups.



Word N3



9.8 Potentiometer Min-Max Range

P67

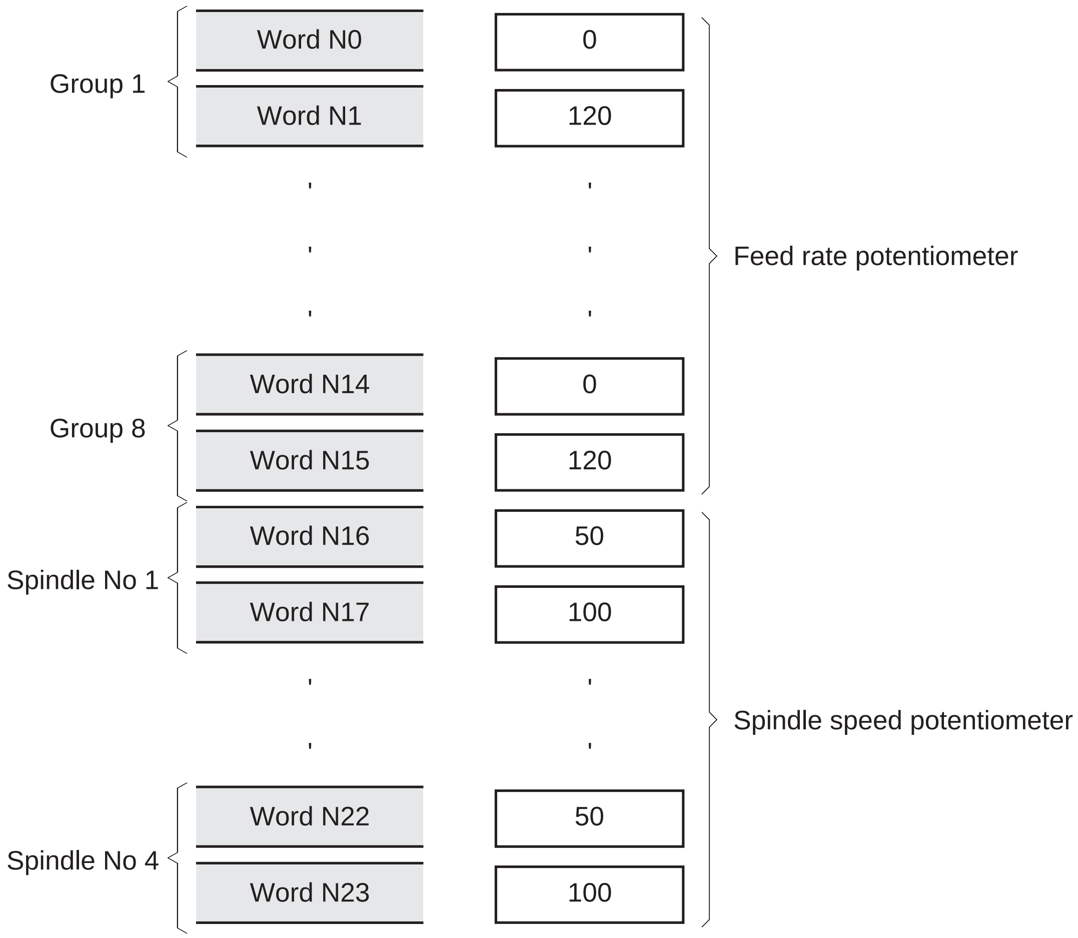
Category	Miscellaneous
Type 5	Unsigned decimal
No. of words	24

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.



9.9 Xoff Character Setting

P80

Category	Miscellaneous
Type 0	8-bit hexadecimal
No. of words	2

Description

Sets the Xoff character.

Principle

Used to replace the ISO Xoff character and the EIA BS character by another ISO or EIA character.

The Xoff or BS character setting is used for the following modes:
- part programme load/unload on peripheral,
- tool file load/unload on peripheral.

REMARK: This parameter is not used for loading the utilities.

Each word is loaded with the hexadecimal code of the replacement character.

Word N0

Used to replace Xoff ($93) by the hexadecimal code ($xx) of another ISO character.

Word N1

Used to replace BS ($2A) by the hexadecimal code ($xx) of another EIA character.

9.10 Memory Area Allocation

P95

Category	Miscellaneous
Type 5	Unsigned decimal
No. of words	3

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).

REMARK: Any remaining area not allocated by this parameter is assigned to area 0 «Free access user area».

Word N0

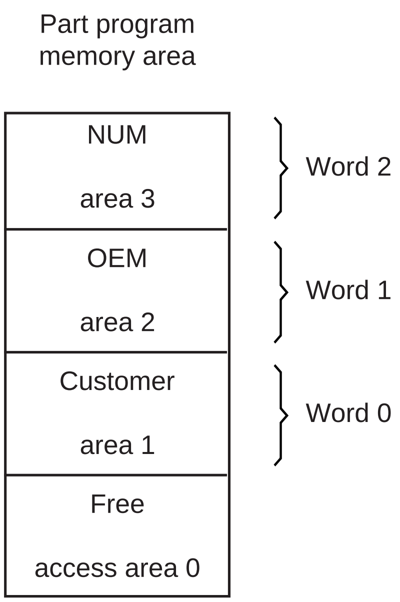
Used to define the size allocated to area 1 (Customer area).

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
Word 2
OEM
area 2
Word 1
Customer
area 1
Word 0
Free
access area 0

9.11 Display Language and Machine Type Selection

P96

Category	Miscellaneous
Type 8	Character string
No. of words	1

Description

Defines the display language for the software and the type of machine for PROCAM.

Principle

Word N0 is a string of four ASCII characters.

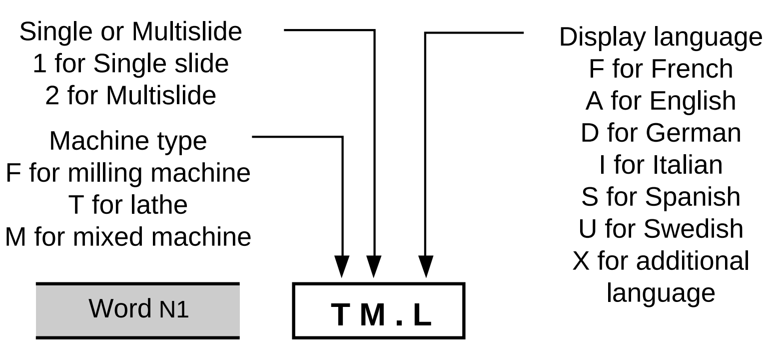
The first character specifies the type of machine. This code is used by PROCAM to find the macros specific to the machine.

The second character is used by PROCAM for lathes and mixed machines. It specifies whether the machine is a single or multislide machine.

The third character is not significant. It is preferably a «.».

The fourth character defines the display language. This code is interpreted by the CNC software and by PROCAM.

REMARK: The default language is French.



9.12 Built-in PLC Programming Language Selection

P98

Category	Miscellaneous
Type 0	8-bit hexadecimal
No. of words	1

Description

Used to select the built-in PLC programming language.

Principle

The value 00 indicates that the PLC function is programmed in assembler. The PLC program is accessible via utility 6 «PLC PROG. IN ASSEMBLER».

The value 01 indicates that the PLC function is programmed in ladder language. The PLC program is accessible via utility 7 «PLC PROG. IN LADDER».

REMARK: Selecting a language inhibits access to the other utility. The message «UTILITY IS NOT COMPATIBLE WITH THE SELECTED TYPE OF APPLICATION» is displayed if it is attempted to call the utility.

Message at Initialisation

The default value of parameter P98 is 1.

If the PLC application loaded is programmed in assembler, the message «DO YOU WANT TO DESTROY PLC ASSEMBLER PROGRAMM, (Y/N)» is displayed.

<u>Application in assembler</u>

- Answer «N» to the question
- Call utility 5 and set P98 to 0.

<u>Ladder application</u>

- Answer «Y» to the question.

9.13 Miscellaneous Times

P99

Category	Miscellaneous
Type 5	Unsigned decimal
No. of words	3

Description

Defines the maximum time allocated to the PLC application for a system equipped with the UC SII central processing unit.

Word N0

Used to define the maximum time allocated to the PLC for the customer PLC programme application. This time is expressed in ms. The default value is 10 ms.

REMARK: For P99 to operate correctly, word N0 must be higher than word N0 of P50. The default value is twice as high.

The maximum time cannot be less than 4 ms nor greater than 14 ms. It must be a multiple of 2 ms. Values outside these limits are forced to the limit values.

The time must be set to the minimum value allowing execution of the sequential tasks but no greater than necessary so as to prevent the background tasks from penalising analysis of the part programmes.

Setting the Time Allocated to the PLC Application

Two cases can occur:

- If there are no background tasks, set P99 to the minimum value required to prevent the PLC from going into fault status (the sequential tasks must be executed cyclically). The extra time will be allocated to the CNC.
- If there are background tasks, set P99 to the minimum value required to prevent the PLC from going into fault status then increase P99 if necessary to improve the time allocated to the PLC background tasks (transparent mode).

⚠ CAUTION

Setting a higher value than necessary can degrade the CNC performance (block preparation time and maximum speed).

Word N1

Not significant

Word N2

Fast execution on PLC, default value 1.

Optimises hardware task execution time.

Word N3

Specifies the refresh rate of the pages displayed on the CNC.

A timeout in seconds is set in this word. It can return CNC error "P50 too low".

= 0 : Check of minimum RTC disabled during execution.

= Value : Check of minimum RTC enabled during execution.

Value = Display refresh rate in seconds.

Reminder

Programme parameter E41007 contains the recommended RTC value in µs (P50).

9.14 Backup in Path, Auto Recall after INTERV

P114

Category	Miscellaneous
Type 5	Unsigned decimal
No. of words	3

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 INTERV 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 Operator Manual).

A maximum of 100 blocks can be stored.

Word N1

This word defines the number of points to be stored during axis retraction in INTERV mode to allow axis recall on the same path (see Operator Manual).

A maximum of 10 points can be stored.

REMARK: Auto recall can be combined with backup and return in the path.

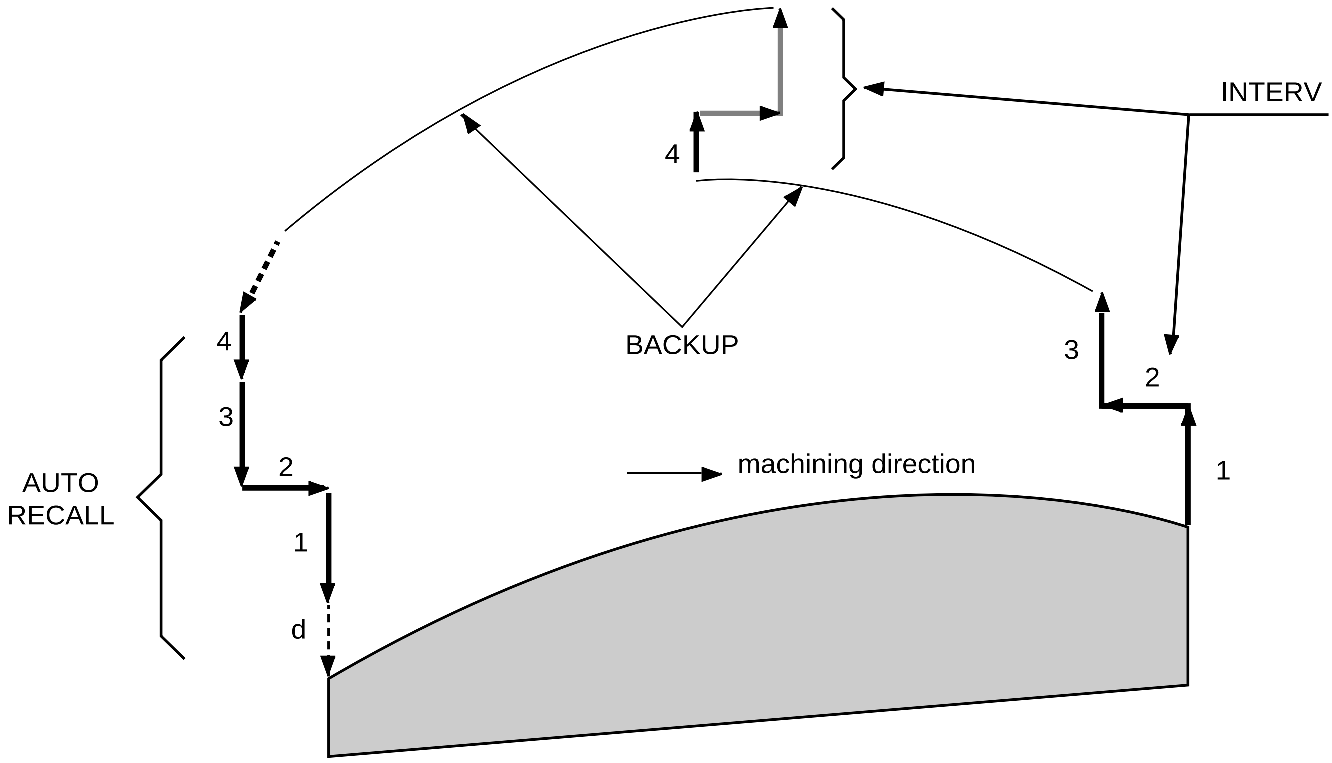
Word N2

Auto recall is carried out at the JOG rate, but approach to the recall point can be carried at the work rate. To do so, word N2 specifies the distance to be covered at the work rate.

Example

Four stored points (1 to 4). The thin line is the rest of INTERV that was not stored.

d = recall distance at the work rate



10 Communication

10.1 Character Formatting for DNC1

P37

Category	Communication
Type 0	8-bit hexadecimal
No. of words	9

Description

Used with the DNC1 link procedure to set the communication parameters and service characters.

Principle

Word N0 is used to set the communication parameters. Words N1 to N8 set the service character codes.

WORD N0

Sets the communication parameters on the DNC1 line.

Word number	Value	Character	Special character
N1	$02	STX (Start of text)	
N2	$03	ETX (End of text)	On IBM PC: BEL = $07
N3	$16	SYN (Synchronisation)	
N4	$06	ACK (Positive acknowledgement)	On IBM PC: SOH = $01
N5	$15	NAK (Negative acknowledgement)	
N6	$04	EOT (End of transmission)	
N7	$05	ENQ (Retransmission request)	
N8	$1D	GS (Group separator)	

10.2 Translation of Special Characters for DNC1

P38

Category	Communication
Type 6	32-bit hexadecimal
No. of words	40

Description

Used with the DNC1 link procedure to translate the special characters controlling the dialogue.

Principle

This table contains the translations of the codes used to control the dialogue between the CNC operator and the computer.

The codes generally correspond to the cursor control key codes.

This 40-word table is divided into two parts.

Words N0 to N19

These words translate the characters sent from the keyboard to the computer.

Even words: Control character codes sent by the CNC keyboard.

Odd words: Sequence of four characters maximum sent to the computer as translation of the code contained in the preceding word.

Words N20 to N39

These words translate the characters sent from the computer to the CNC.

Even words: Computer control character codes.

Odd words: Translation of the code contained in the preceding word. It is this translation that is transmitted to the CNC screen.

In this context, the translation includes only one character.

Example

For the CNC «line delete» (hex code $08) keyboard code, the sequence «ESC[C» is sent to the computer.

The «cursor left» function code is $89 on the computer and $91 on the CNC keyboard.

Word N0	00000008
Word N1	001B9143
Word N20	00000089
Word N21	00000091

10.3 DNC1 Timeouts

P39

Category	Communication
Type 4	Unsigned decimal
No. of words	3

Description

Used to set timeouts for the DNC1 link procedure.

Principle

Word N0

Defines the minimum time (in ms) that the CNC must wait between receiving a message or an acknowledgement and sending back an acknowledgement or message.

This timeout allows the computer to go into reception mode.

Word N1

Defines the maximum time (in ms) allotted to the computer to answer after the CNC sends a message or acknowledgement (timeout). The default value is 1000 ms.

Word N2

Not significant.

10.4 Setting the File Up/Download Timeout

P84

Category	Communication
Type 5	Unsigned decimal
No. of words	1

Description

Sets the timeout for “Part programme” type file upload or download.

Download is carried out by the “Read-download-segment” request and upload by the “Read-upload-segment” request in the framework of exchanges by protocol.

“Part programme” type files are automatically closed by the CNC if the requester does not take action within the time set by P84.

Principle

The timeout is expressed in seconds. A value of zero sets an indefinite timeout.

10.5 MAPWAY/ETHWAY Network and Station Number

P100

Category	Communication
Type 0	Hexadecimal on 8 bits
No. of words	8

Description

Configures the station according to the Telemecanique series 7 addressing system.

The network and station numbers physically identify a unit on the MAPWAY or ETHWAY network. Refer to the MAPWAY-ETHWAY documentation for further details.

Specifies the notation type (Intel or Motorola) used for transmission of words or long words over the network.

REMARK: *If the interface is present in the rack and parameter P100 is not set, the DEF LED on the front of the interface flashes and the message «PARAMETER P100 WORD 0 INCORRECT» is displayed in the «Error message» page.*

Principle

Word N0

Sets the network number (Telemecanique series 7 addressing). The default value is $FF.

Word N1

Sets the station number (Telemecanique series 7 addressing). The default value is $FF.

Word N2

Sets the notation type (Intel or Motorola).

Set to 0 for Intel notation.

Set to 1 for Motorola notation.

Order of Byte Transmission Over the Network

		Byte 1	Byte 2	Byte 3	Byte 4
Intel notation	Word	Low byte, LSBs	Low byte, MSBs		
	Long word	Low byte, LSBs	Low byte, MSBs	High byte, LSBs	High byte, MSBs
Motorola notation	Word	Low byte, MSBs	Low byte, LSBs		
	Long word	High byte, MSBs	High byte, LSBs	Low byte, MSBs	Low byte, LSBs

Unsolicited Data Destination Address

The next five bytes define the series 7 destination address to which unsolicited data are sent.

Reminder

Unsolicited data are messages which can be sent by:
- A part programme (instruction $n). This instruction is described in detail in the programming manual, or
- The operator from the panel in the Communication page, "Operator Messages" menu, "Network messages" page (refer to the Operator Manual).

Word N3

Defines the destination network number.

Word N4

Defines the destination station number.

Word N5

Defines the destination port number.

Word N6

Defines the destination module number.

Word N7

Defines the destination channel number.

REMARK: *Encoding of a series 7 address is described in Chapter 5 of the UNI-TELWAY Integration Manual.*

10.6 UNI-TELWAY Master Settings

Category	Communication
Type 0	8-bit hexadecimal
No. of words	8

Description

Sets the UNI-TELWAY master function:
- line assigned to UNI-TELWAY,
- communication parameters,
- notation type (Intel or Motorola),
- timeouts (turnaround, envelope delay, intercharacter gap),
- number of slaves,
- the number of slave to which the unsolicited data are sent,
- the number of polling cycles.

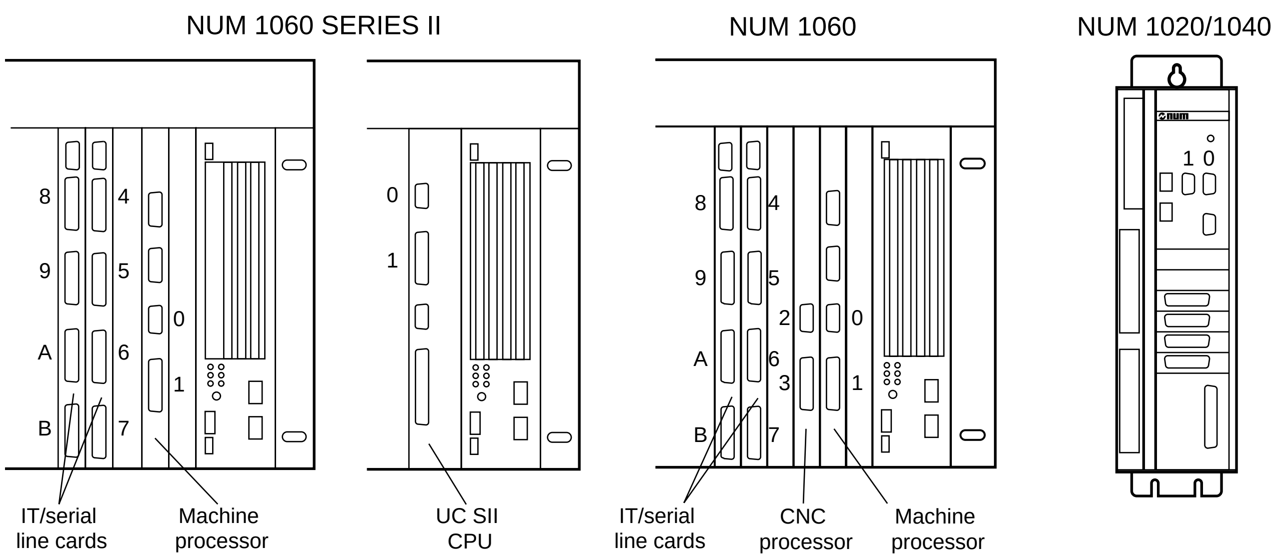
Principle

Word N0

Sets the line number assigned to UNI-TELWAY. The default value is $FF.

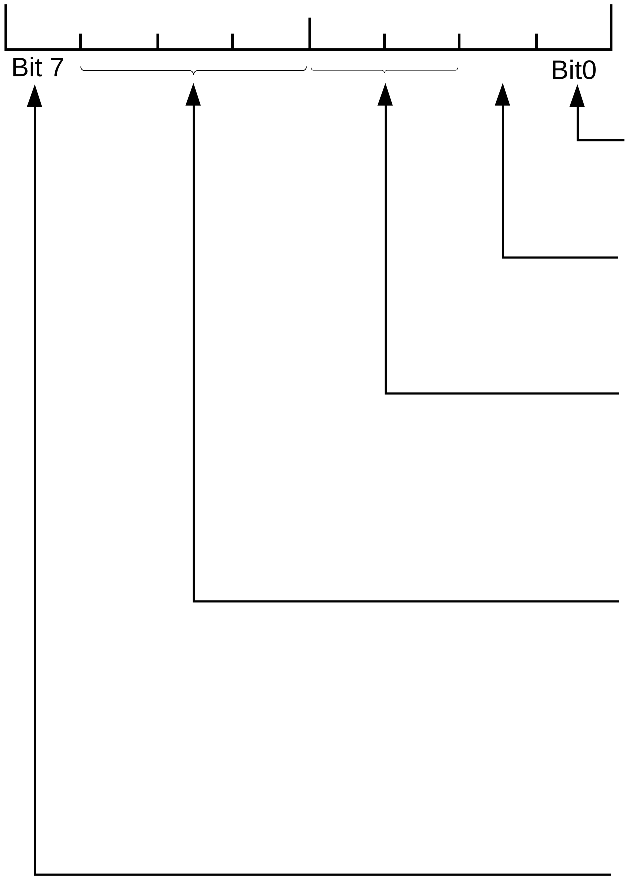
For a NUM 1060 CNC, the possible values are:
- $0 for the machine processor TTL line or COMM1 on the UC SII CPU,
- $1 for the machine processor RS232C or COMM2 on the US SII CPU,
- $4 to $B for the lines of the IT/serial line cards.

For a NUM 1020/1040 CNC, the value is $01 for the serial line.



Word N1

Sets the communication parameters and notation type (Intel or Motorola). The default value is $15.



Parity:
- bit a 1: parity,
- bit a 0: no parity.

Parity type:
- bit a 1: even parity,
- bit a 0: odd parity.

Number of stop bits:

Bit 3	Bit 2	
0	0	1 stop bit
0	1	1 stop bit
1	0	1.5 stop bits
1	1	2 stop bits

Data rates

Bit 6	Bit 5	Bit 4	
0	0	0	19200 bauds
0	1	0	4800 bauds
0	1	1	2400 bauds
1	0	0	1200 bauds
Other values			9600 bauds

Notation type
- bit a 0: Intel format,
- bit a 1: Motorola format.

Order of Byte Transmission Over the Network

		Byte 1	Byte 2	Byte 3	Byte 4
Intel notation	Word	Low byte, LSBs	Low byte, MSBs		
	Long word	Low byte, LSBs	Low byte, MSBs	High byte, LSBs	High byte, MSBs
Motorola notation	Word	Low byte, MSBs	low byte, LSBs		
	Long word	High byte, MSBs	High byte, LSBs	Low byte, MSBs	Low byte, LSBs

10

Word N2

Sets the turnaround time (as a number of characters). The default value is $1.

The turnaround time is the minimum time that a unit must wait before beginning to send a message to allow the remote unit to turn around (switch from transmission to reception). The minimum time that can be set is the time required to send one character.

Word N3

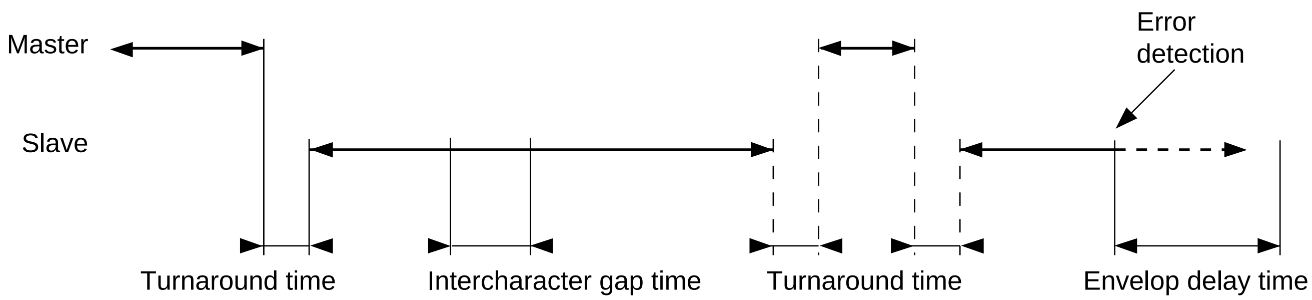
Sets the envelope delay time (as a number of characters). The default value is $5.

If a received frame is recognised as incorrect, the master considers the end of the envelope delay time as the end of transmission.

Word N4

Sets the intercharacter gap time (as a number of characters). The default value is $5.

During transmission of a frame, the characters must follow one another with a maximum gap between characters equal to the time required to send five characters.



Word N5

Sets the number of slaves accessible by the master on the network. This number is limited to 64 ($40).

Word N6

Specifies the number of the destination slave to which the unsolicited data are sent. The default value is $FF = no destination slave.

Reminder

Unsolicited data are messages which can be sent by:
- A part programme (instruction $n). This instruction is described in detail in the programming manual, or
- The operator from the panel in the Communication page, "Operator Messages" menu, "Network messages" page (refer to the Operator Manual).

Word N7

Defines the number of polling cycles during which a slave that did not answer is not polled. The default value is $14.

10.7 UNI-TELWAY Slave Settings

P111

Category	Communication
Type 0	8-bit hexadecimal
No. of words	10

Description

Sets the UNI-TELWAY slave function
- line assigned to UNI-TELWAY,
- communication parameters,
- notation type (Intel or Motorola),
- timeouts (turnaround, envelope delay, intercharacter gap),
- number of slaves (server or client),
- the destination address of the unsolicited data.

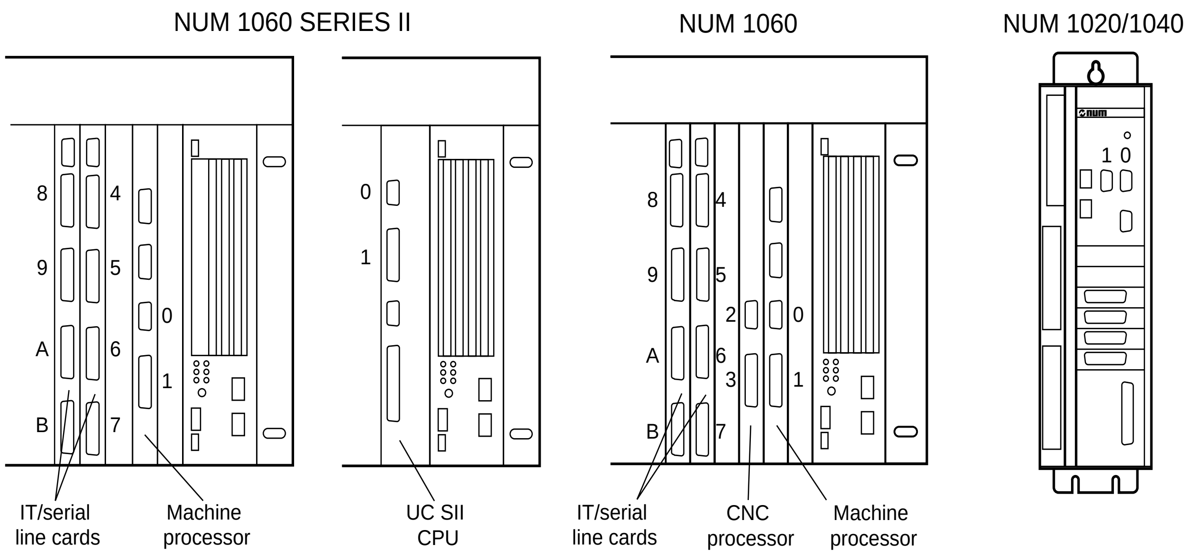
Principle

Word N0

Sets the line number assigned to UNI-TELWAY. The default value is $FF.

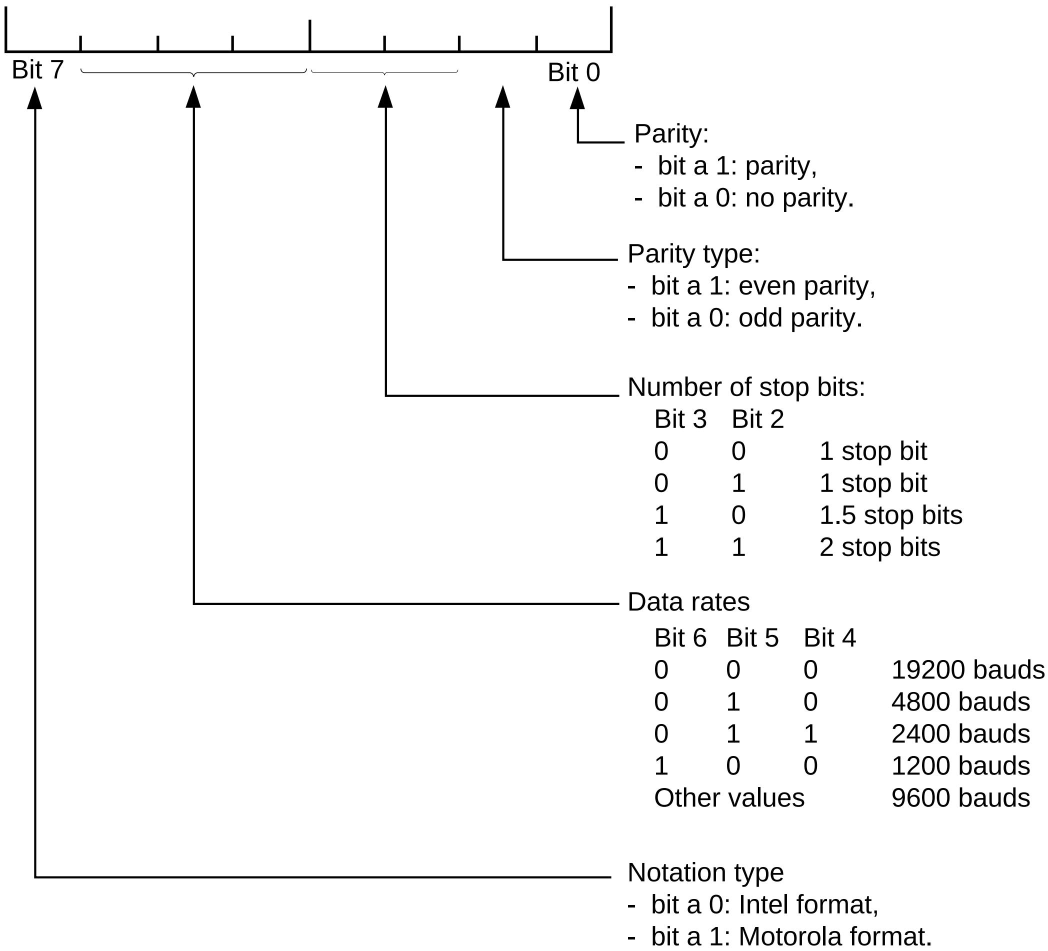
For a NUM 1060 CNC, the possible values are:
- $0 for the machine processor TTL line or COMM1 on the UC SII CPU,
- $1 for the machine processor RS232C or COMM2 on the US SII CPU,
- $4 to $B for the lines of the IT/serial line cards.

For the NUM 1020/1040 CNC, the possible values are $0 for line COM1 or $1 for the Serial line.



Word N1

Sets the communication parameters and notation type (Intel or Motorola). The default value is $15.



Order of Byte Transmission Over the Network

		Byte 1	Byte 2	Byte 3	Byte 4
Intel notation	Word	Low byte, LSBs	Low byte, MSBs		
	Long word	Low byte, LSBs	Low byte, MSBs	High byte, LSBs	High byte, MSBs
Motorola notation	Word	Low byte, MSBs	low byte, LSBs		
	Long word	High byte, MSBs	High byte, LSBs	Low byte, MSBs	Low byte, LSBs

Word N2

Sets the turnaround time (as a number of characters). The default value is $1.

The turnaround time is the minimum time that a unit must wait before beginning to send a message to allow the remote unit to turn around (switch from transmission to reception). The minimum time that can be set is the time required to send one character.

For further details, refer to the drawing in Section 10.6.

Word N3

Sets the server slave number. The default value is $0.

Word N4

Specifies the client slave number. The default value is $0.

Destination Address of the Unsolicited Data

The next five bytes define the series 7 destination address to which unsolicited data are sent.

Reminder

Unsolicited data are messages which can be sent by:
- A part programme (instruction $n). This instruction is described in detail in the programming manual, or
- The operator from the panel in the Communication page, "Operator Messages" menu, "Network messages" page (refer to the Operator Manual).

Word N5

Defines the destination network number.

Word N6

Defines the destination station number.

Word N7

Defines the destination port number.

Word N8

Defines the destination module number.

Word N9

Defines the destination channel number.

REMARK: *Encoding of a series 7 address is described in Chapter 5 of the UNI-TELWAY Integration Manual.*

10.8 Settings of the Line Assigned to the PLCTOOL Link

P112

Category	Communication
Type 0	8-bit hexadecimal
No. of words	3

Description

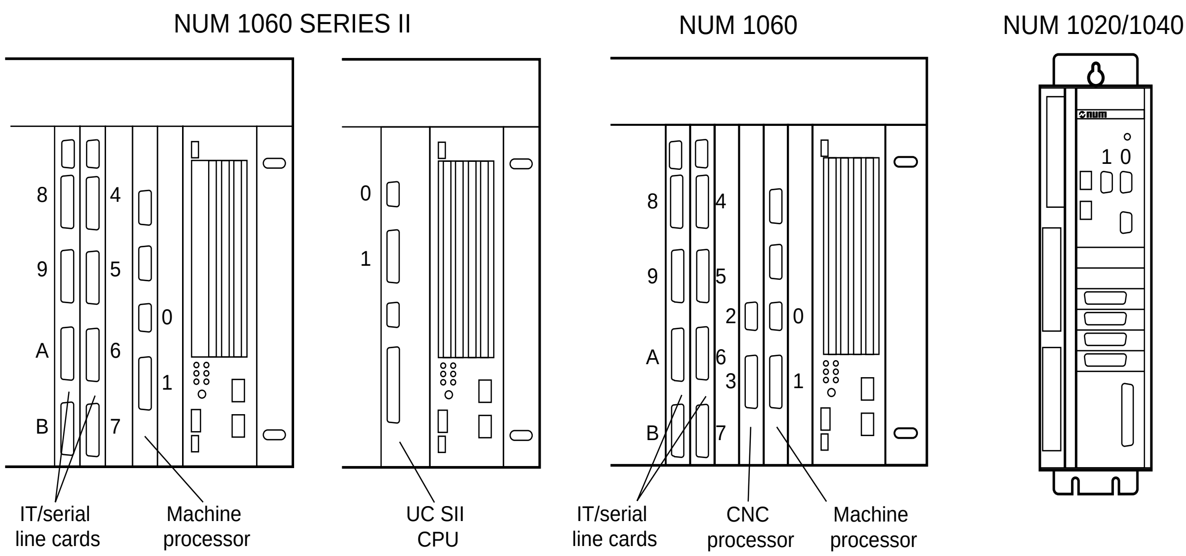
Sets the line assigned to the PLCTOOL link:
- line number assigned to PLCTOOL,
- communication parameters.

Principle

Word N0

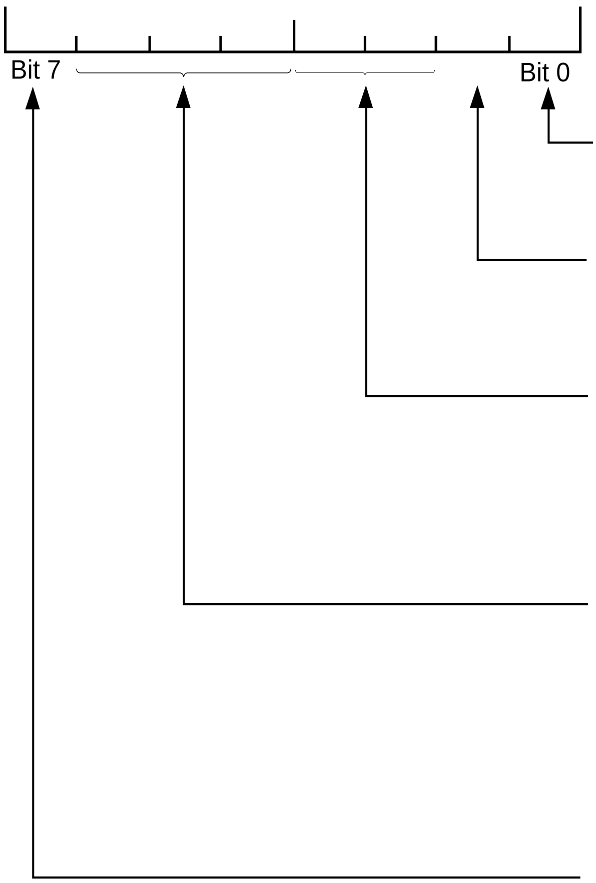
Sets the line number assigned to PLCTOOL. The default value is $FF.

The possible values are:
- $0 for the machine processor TTL line or COMM1 on the UC SII CPU,
- $1 for the machine processor RS232C or COMM2 on the US SII CPU,
- $2 for the CNC processor DNC line,
- $3 for the CNC processor PERIPH line,
- $4 to $B for the lines of the IT/Serial line cards.



Word N1

Sets the communication parameters and notation type (Intel or Motorola). The default value is $15.



Parity:
- bit a 1: parity,
- bit a 0: no parity.

Parity type:
- bit a 1: even parity,
- bit a 0: odd parity.

Number of stop bits:

Bit 3	Bit 2	
0	0	1 stop bit
0	1	1 stop bit
1	0	1.5 stop bits
1	1	2 stop bits

Data rates

Bit 6	Bit 5	Bit 4	
0	0	0	19200 bauds
0	1	0	4800 bauds
0	1	1	2400 bauds
1	0	0	1200 bauds
Other values			9600 bauds

Notation type
- bit a 0: Intel format,
- bit a 1: Motorola format.

Although the notation type can be set, the value of bit 7 of this word must always be 0 (Intel format).

Word N2

Not significant.

11 Improving the Settings

11

11.1 Checking the Maximum Speeds

Write the following program for the first axis to be checked (e.g. X) (see «Operator Manual»).

The axis must be moved at low speed (10 percent VMAX).

```
N10 G9 X100 F...
N15 G9 X-100
N20 G79 N10
N25 M02
```

REMARK: For each axis to be checked, change the axis name and speed in the above program.

Turn on the CNC.

Set the feed rate potentiometer to 100 percent.

Home the axis to be checked.

Select the AUTO mode.

Run the program by pressing the «CYCLE» key on the machine panel.

If error number 40 to 71 «Following error too large on the axis» is displayed, check the setting of parameter P56.

Check following for the axis in movement on the «AXIS» page in the «ERROR» column.

The value must be:

EMAX (in internal measurement units or deg/10000) = VMAX x T x 11/10/k

REMARK: In case of tachometer asymmetry or incorrect adjustment of the servo-drive, the value may not be the same in both directions.

If the following error is not corrected:

Check that the servo-drive reference voltage is equal to 9 V.

Adjust the servo-drive scale setting if necessary using the potentiometer provided.

Suspend execution of the program.

Modify the program by programming the maximum speed on the axis.

Rerun the program.

The following error should stabilise at (VMAX x T) and not reach the limit set in parameter P23.

If an error number 40 to 71 «Following error too large on the axis» is displayed before the limit set in P23:

Check the setting of parameter P56.

If the speed reference is not saturated at 10 V:

Recompute the setting of P23.

CAUTION

If these settings do not affect operation on the system, the maximum speed required can probably not be reached (mistake in manufacturers data, problem with the motor, servo-drive power supply voltage, etc.).

Check the isolation between axes.

The following error should remain stable on the axes not programmed.

If this is not the case:

Check the screenings and earthing of the machine.

After completing the check of an axis:

Change the axis name and speed in the program.

Test the next axis by repeating the maximum speed checking procedure from the beginning.

11.2 Improving the Servo-Control Settings

When all the machine axes have been integrated, the settings can be improved.

11.2.1 Time Constant

It is recommended to use the same time constant on all the main axes. The largest time constant is used for these axes.

The equations to be used are:

KVAR (for a 9 V reference) = 29491 / VMAX x T x k

EMAX = VMAX x T x 11/10/k

Axis @	T	VMAX	KVAR	EMAX
0				
1				
2				
3				
4				
5				
6				
7				
8				

Modify the settings on each axis if these values are optimistic.

Enter the values of KVAR in the words of P21.

Enter the values of EMAX in the words of P23.

Enter the values of T in the words of P56.

11.2.2 In-Position Window

The in-position window gives the maximum positioning accuracy. It is determined from the number of encoder pulses.

In-position window > MULTI/DIVI

Set the servo-drive offsets as accurately as possible.

Move the axes in both directions.

Note the value displayed in the «FOLLOWING» column when the axis has stabilised.

P22 = 2 x (Value)

Axis @	MULTI	DIVI	Window
0			
1			
2			
3			
4			
5			
6			
7			
8			

11.2.3 Dynamic Movement Control

Parameter P57 is used to check operation of the axes whatever their speed. Parameter P57 gives an operational tolerance. Like P22, it is determined from the number of encoder pulses.

P57 = 2 to 4 x P22

Axis @	P22	P57
0		
1		
2		
3		
4		
5		
6		
7		
8		

Enter the values in the words of P22.

Enter the values in the words of P57.

12 DISC Axes

The parameters concerning the DISC axes are described in the DISC Integration Manual 938907.

The following parameters are concerned:
- P21 servo-control loop gain coefficient
- P70 card mapping
- P71 encoder configuration
- P72 motor direction of rotation
- P73 maximum motor speed
- P74 speed servo-loop proportional action coefficient
- P75 speed servo-loop integral action coefficient
- P76 speed sensor measurement increment average (motor sensor)
- P77 speed measurement filter
- P78 torque reference filter
- P79 static current limiting
- P85 declaration of a torque slave QVN application
- P86 torque slave rotation direction same as or different from master
- P87 preload current of a master/slave pair

13 Parameter Integration Tool

13

13.1 Integration Tool on IBM PC or Compatible

Refer to the SETTOOL Parameter Integration Tool documentation.

13.2 Utility 5

Resident utility 5 includes:
- an editor used to:
. display the machine parameters,
. edit the machine parameters,
- a parameter table load module,
- a parameter table unload module.

13.2.1 Utility Access Procedure

Requirements

There no special access requirements for utility 5.

Action

Select the utility menu — UTIL

The «UTILITIES PRESENT» menu is displayed.

If necessary, select the language in which the utilities are to be edited.

Enter «A» for English.

Or

Enter «F» for French.

The menu is displayed in the language chosen.

Select utility 5. — % 5

13

The «MACHINE SETUP DATA» main menu is displayed.

```
            MACHINE SETUP DATA

> 0 DISPLAY
1 CHANGE
2 LOAD
3 UNLOAD
4 CHECK

?▮
..!..                                          EXIT
```

Exit from the Procedure

Key in the command ☞ CTRL S X OFF

Return to the «AXIS» page.

13.2.2 Using of the Editor

13.2.2.1 Using the Commands

The two editor modules contain commands used to move around in the parameter pages and modify the parameter values.

Requirements

Parameter to be checked or edited displayed on the screen.

Actions

Enter the command corresponding to the operation
(see table below).

OPERATION	LINE SELECTED	COMMAND SYNTAX	MODULE CONCERNED
Display of the parameters	Any word	P [n] [↵] where [n] is the parameter number	Display Change
Select a word in a parameter	Any word	P [n]_N [n] [↵] where [n] is the parameter number and word number	Display Change
Display the next parameter	Any word	[↵] or [.../...] or [Pg Dn]	Display Change
Move the cursor to the next word	Previous word	[↓]	Display Change
Move the cursor to any word	Any word	N [n] [↵] where [n] is the word number	Display Change
Display the value of a word on the dialogue line	Word to be edited	[SHIFT] [# 3] [↵]	Change
Edit the value of a word	Word to be edited	[SHIFT] [# 3] + [Value] [↵]	Change
Edit a string of words (maximum 10 consecutive values)	First word to be edited	[SHIFT] [# 3] + [String] [↵]	Change
Move the cursor one character right on the dialogue line	Dialogue line	[→]	Change

13

OPERATION	LINE SELECTED	COMMAND SYNTAX	MODULE CONCERNED
Move the cursor one character left on the dialogue line	Dialogue line	←	Change
Move the cursor to the end of the dialogue line	Dialogue line	SHIFT →	Change
Move the cursor to the beginning of the dialogue line	Dialogue line	SHIFT ←	Change
Delete the dialogue line	Dialogue line	⇐	Change
Delete the character to the left of the cursor	Dialogue line	⇦	Change

13.2.2.2 Displaying Parameters

The parameter editor is used to display the machine parameters.

Requirements

«MACHINE SETUP DATA» menu displayed.

Actions

Select « DISPLAY». ☞) 0 ⏎

«PARAMETER?» is displayed on the dialogue line.

Enter the command corresponding to the operation to be performed (see table below).

OPERATION	COMMAND SYNTAX
Display P0	⏎ or .../... or Pg Dn
Display any parameter	P [n] ⏎ where [n] is the parameter number
Select a word in a parameter	P [n]_N [n] ⏎ where [n] is the parameter number and the word number

The display page for the parameter selected is displayed (P9 in the example).

```
(DISPL)   PARA: 9   NB WORDS: 32   TYPE: 0
                                   (BIN. BYTE)



>N 0   00
 N 1   02
 N 2   08
 N 3   10
 N 4   11

../..                                   EXIT
```

The «PARA» field contains the parameter number.

The «TYPE» field contains the parameter type (see 1.2).

The «NB WORDS «field specifies the number of words in the parameter.

Nn (where n = word number) followed by a value gives the word number and the value assigned to it.

Moving around the editor

Enter the command corresponding to the operation to be performed (see 13.2.2.1).

Exit from the Procedure

Return to the «MACHINE SETUP DATA» menu. EXIT

13

13.2.2.3 Editing Parameters

The parameter editor is used to edit the machine parameters.

Requirements

«MACHINE SETUP DATA» menu displayed.

Actions

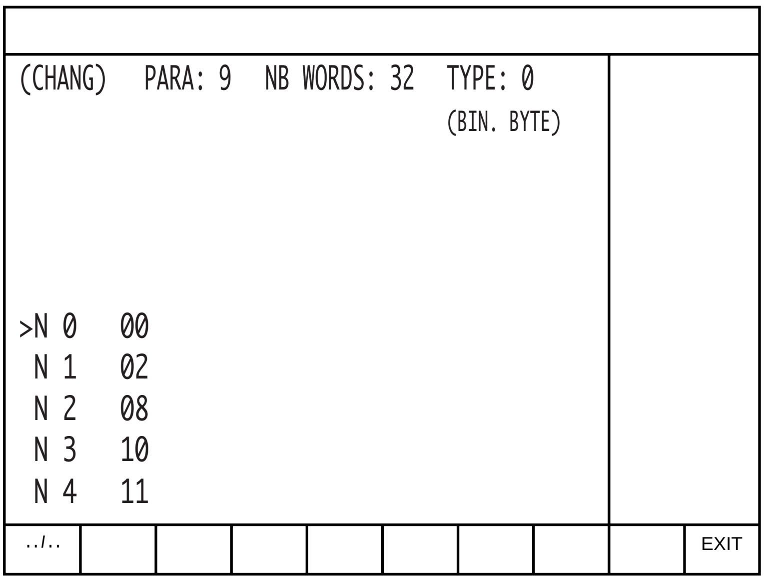
Select «CHANGE». ☞ !1

«PARAMETER?» is displayed on the dialogue line.

Enter the command corresponding to the operation to be performed (see table below).

OPERATION	COMMAND SYNTAX
Display P0	or ../.. or Pg Dn
Display any parameter	P [n] where [n] is the parameter number
Select a word in a parameter	P [n]_N [n] where [n] is the parameter number and the word number

The display page for the parameter selected is displayed (P9 in the example).



The «PARA» field gives the parameter number.

The «TYPE» field gives the parameter type (see 1.2).

The «NB WORDS «field specifies the number of words in the parameter.

Nn (where n = word number) followed by a value gives the word number and the value assigned to it.

Editing a word

Move the cursor to the word to be edited (see 13.2.2.1).

Display the value in the dialogue line. — SHIFT # 3 ↵

Edit the value. — ↵

Or

Open the dialogue line. — SHIFT # 3

Key in the new value. — ↵

Editing a string of words

Move the cursor to the first word to be edited (see 13.2.2.1).

Open the dialogue line. — SHIFT # 3

Enter the string of new values
[Value 1]_[Value 2]_[Value 3], etc. — ↵

CAUTION

After editing, it is recommended to store the parameter table or print out a hard copy (see 13.2.4).

Exit from the Procedure

Return to the «MACHINE SETUP DATA» menu. — EXIT

Confirming the Changes

Enter. — CTRL S X OFF

Resetting the CNC

Acknowledge the messages displayed.

Return to the «AXIS» page.

13.2.3 Loading a Parameter Table

The table can be loaded from a PC with the NUM software (APA10, NUMPC, PLCTOOL or SETTOOL).

Requirements

«MACHINE SETUP DATA» menu displayed.

Load peripheral on and connected to one of the serial lines of the CPU.

Serial line compatible with the peripheral selected (see Operator Manual).

Actions

Set the peripheral to unload mode.

Select «PERIPHERAL LOAD». @ 2

The cursor moves to line 2 and the message «READY?» is displayed on the dialogue line.

Enter. Y

Start unloading from the peripheral.

If the identification number of the parameter table to be loaded does not correspond to the job reference of the CNC, transmission is stopped and the identification number is displayed on the dialogue line.

Replace the identification number by the job reference.

Transmission is resumed.

The load pages are displayed for each parameter loaded and the sequence of character strings is displayed in the dialogue line (see P9 below).

```
(LOAD)   PARA: 9   NB WORDS: 32   TYPE: 0
                                  (BIN. BYTE)




>N 0   00
 N 1   02
 N 2   08

#FF FF FF FF FF FF▮
../..                                          EXIT
```

The display stops on the page of the last parameter loaded.

Return to the «MACHINE SETUP DATA» menu.

Confirming the Modifications.

Enter. ☞ CTRL S X OFF

Resetting the CNC

Acknowledge the messages displayed.

Return to the «AXIS» page.

13.2.4 Unloading a Parameter Table

The parameter table can be unloaded to a PC via the RS232 link using the NUM software (APA10, NUMPC, PLCTOOL or SETTOOL).

Requirements

«MACHINE SETUP DATA» menu displayed.

Load peripheral on and connected to one of the serial lines of the CPU.

Serial line compatible with the peripheral selected (see Operator Manual).

Actions

Set the peripheral to load mode.

Select «PERIPHERAL UNLOAD».

The cursor moves to line 3 and the message «READY?» is displayed on the dialogue line.

Enter

The CNC job reference (%nnnnnnnn) is displayed on the dialogue line.

You may enter a comment after the job reference.

You can also edit the job reference.

Start the unload procedure on the CNC side.

Start the load procedure on the peripheral side.

The parameter table is unloaded with a return to the «MACHINE SETUP DATA» menu at the end of the operation.

13.2.5 Check of a Parameter Table Unloaded Into a Peripheral

After unloading a parameter table into a peripheral, it is necessary to check that the data saved are the same as the initial data.

Requirement

Peripheral ready to send the data saved.

«MACHINE SETUP DATA» menu displayed.

Actions

Set the peripheral to unload mode.

Select «VERIFY» ☞ [$ 4] [⟼]

The cursor moves to line 4 and the message «READY?» is displayed on the dialog line.

Enter. ☞ [Y] [⟼]

Run the unload procedure on the peripheral side.

The parameters are listed on the CNC screen by increasing order, preceded by «(VERI)».

At the end of the check, if no errors were detected, the «MACHINE SETUP DATA» menu is redisplayed with the cursor on item 0, «DISPLAY».

Otherwise, the system stops on the parameter with an error and the message «FILE ERROR» (INCORRECT TAPE?) is displayed on the dialog line. Repeat the table unload procedure (see 13.2.4). If the error persists, check the link between the CNC and the peripheral and the communication parameters.

13.2.6 Parameter Table Syntax

A parameter table can be created or edited under one of the NUM editors for PC (APA10, NUMPC, PLCTOOL or SETTOOL). The file created must be saved as an ASCII file.

Proceed as follows to create a table under a PC editor:

Begin the table with the «%» character and end it with «Xoff» (ALT19).

The eight digits following «%» are the job reference of the CNC.

At the end of each line, the two digits after the semicolon indicate the number of characters in the line (including spaces and «#»). The value is hexadecimal.

Characters PaNb specify the parameter concerned and the first word on the next data line. This is followed by the type and number of words in the parameter: tcnd (starting at CNC software index K). For instance, P9N0:t0n32 = Parameter P9, first word on line 0, type 0 and 32 words.

A data line starts with the character «#».

A data line cannot exceed ten values. Parameters with more than ten words must be written on several lines.

Example

`%00000000;09`	Start of table, job reference (8 characters)
`P0N0:t6n1;09`	
`#00000007;09`	
`P1N0:t6n2;09`	Parameter P1, Words starting at N0, Type 6, 2 words (9 characters)
`#00000000 00000000;12`	Data string, Words N0 and N1 (18 characters)
`P2N0:t6n1;09`	
`#00000007;09`	
`P3N0:t6n1;09`	
`#00000007;09`	
`P4N0:t0n4;09`	Parameter P4, Words starting at N0, Type 0, 4 words (9 characters)
`#0 0 0;06`	Data string, Words N0 to N3 (6 characters)
`P5N0:t0n2;09`	
`#1 0;04`	
`P6N0:t0n9;09`	
`#0 0 0 0 0;0A`	
`P7N0:t0n2;09`	
`#54 A;05`	
`P8N0:t6n1;09`	
`#00000000;09`	
`P9N0:t0n32;0A`	Parameter P9, Words starting at N0, Type 0, 32 words (10 characters)
`#0 1 2 FF FF FF FF FF FF FF;1B`	Data string, Words N0 to N9 (27 characters)
`P9N10:t0n32;0B`	Parameter P9, Words starting at N10, Type 0, 32 words (11 characters)
`#FF FF FF FF FF FF FF FF FF FF;1E`	Data string, Words N10 to N19 (30 characters)
`P9N20:t0n32;0B`	Parameter P9, Words starting at N10, Type 0, 32 words (11 characters)
`#FF FF FF FF FF FF FF FF FF FF;1E`	Data string, Words N20 to N29 (30 characters)
`P9N30:t0n32;0B`	
`#FF FF;06`	
`P10N0:t6n1;0A`	
`#00000000;09`	
`...`	
`...`	
`X off`	End of table, Xoff character (ALT19)

Index

To contact NUM

Convention

The phone and fax numbers are in the following format in the list below:

WW/ [X](YY)ZZ.ZZ.ZZ

WW/	Country code
[X]	Digit(s) to be dialed for domestic calls only
(YY)	City or area code

The [ ] and ( ) are used only if the digits are not always dialed in the number

NUM in the UK and USA

List updated on 27-11-97

Country	Address	Phone E-Mail	Fax
UK	NUM UK Ltd - Coventry	44/ [0]12.03.44.42.33 Num-uk@Num-uk.com	44/ [0]12.03.44.41.17
USA **Canada**	NUM CORPORATION - Naperville (Il.)	1/ (630)505.72.37 and 77.22 Numengineer@Ameritech.net	1/ (630)505.77.54

NUM in the World

Country	Address	Phone E-Mail	Fax
HEAD OFFICE			
France	NUM S.A. - 21, Avenue du Maréchal Foch BP 68 - 95101 Argenteuil Cedex	33/ [0]1.34.23.66.66 Telex: 609 611	33/ [0]1.39.47.25.19
SUBSIDIARIES and REPRESENTATIVES			
Australia	EMTS PTY. LTD. - Thomastown	61/ [0](3)94.60.69.00 emts@emts.com.au	61/ [0](3)94.62.12.14
China	NUM S.A. Beijing Office - Beijing	86/ (10)64.10.64.51 and 52 beijing_num@mail.schneider.fr chiieng_tchen@mail.schneider.fr	86/ (10)64.10.64.50
Germany	NUM-GÜTTINGER GmbH - Stuttgart / Ostfildern	49/ 71.13.48.06.0	49/ 71.13.48.06.10
Netherlands /	NUM-GÜTTINGER GmbH - Ratingen	49/ 21.02.43.21.80 to 83	49/ 21.02.43.21.90
Czech Republic	NUM-GÜTTINGER GmbH - Chemnitz / Hartmannsdorf	49/ 37.22.63.61.31 and 32	49/ 37.22.63.61.30
Italy	NUM Spa - Milano	39/ (2)27.00.09.63 CASTIGLIOLAM@NUM.IT	39/ (2)27.00.15.68
	NUM Spa - Bologna / Zola Predosa	39/ (51)75.41.18	39/ (51)75.40.82
	NUM Spa - Cuggiono	39/ (2)97.96.95.00 albertinic@num.it	39/ (2)97.96.92.51
Scandinavia	NUM NORDEN AB - Västerås (Sweden)	46/ [0]21.13.11.31 num@quicknet.se	46/ [0]21.13.12.30
	NUCOS OY - Pirkkala (Finland)	358/ (3)342.71.00	358/ (3)342.71.30
	MOTION CONTROL SENTERET AS - Oslo (Norway)	47/ 22.72.48.70 salg@motioncontrol.no	47/ 22.72.26.53
Spain **Portugal**	TELENUM S.A. - Irún	34/ [9](43)62.35.55 Telenum@interplanet.es	34/ [9](43)62.20.12
Switzerland **Austria**	NUM-GÜTTINGER AG - Teufen	41/ [0](71)335.04.11 prodman@num.ch	41/ [0](71)333.35.87
	NUM-GÜTTINGER SA - Biel	41/ [0](32)323.53.73	41/ [0](32)323.55.25
Taiwan	NUM TELEMECANIQUE TAIWAN - Taichung	886/ (4)328.20.81 and 82 numtwn@tcts1.globalnet.com.tw	886/ (4)328.20.83

NUM in the World

List updated on 27-11-97

Country	Address	Phone E-Mail	Fax
DISTRIBUTION and AFTER-SALES SERVICE			
Algeria	R.M.O - Algiers	213/ (2)56.42.65	213/ (2)56.42.65
Belgium **Luxemburg**	SCHNEIDER MGTE - Brussels	32/ (2)373.77.72	32/ (2)373.79.80
India	SCHNEIDER ELECTRIC INDIA PVT. LTD. - New Delhi	91/ (11)681.28.20 and 80.07 seipl.del@gndel.globalnet.ems.vsnl.net.in	91/ (11)646.50.80
Japan	SCHNEIDER JAPAN - Tokyo	81/ (35)474.44.74	81/ (35)474.44.70 and 80
Malaysia	IME TRADING SDN. BHD. - Petaling jaya	60/ (3)793.33.50 and 68.66 ime@pc.jaring.my	60/ (3)793.43.84
South Africa	MACHINE TOOL TECHNOLOGIES - Jeppestown	27/ [0](11)614.65.31 MACHINE@ICON.CO.ZA	27/ [0](11)614.25.15
Thailand	N.R. AUTOMATION Co. Ltd - Samutprakam	66/ (2)758.14.88 to 90	66/ (2)384.19.45
Turkey	SCHNEIDER ELEKTRIK A.S. - Istanbul	90/ (216)306.60.82 p 140 ocakir@compuserve.com	90/ (216)353.47.24
TRAINING CENTRES			
Belgium	TECHNIFUTUR - Liège	(41)52.12.32	(41)52.14.08
China	WUHAN INSTITUTE OF TECHNOLOGY - Wuhan	(27)783.27.65	(27)781.48.07
France	CITEF - Cergy Pontoise	33/ [0]130.75.32.00	33/ [0]130.30.35.49
Thailand	THAI FRENCH INNOVATION CENTER - Bangkok	66/ (2)587.76.70 arn@kmitnb.ac.th	66/ (2)586.90.14

NUM
PARAMETER MANUAL
en-938818/9

You can help us improve the quality of our documentation by filling in this questionnaire and sending or faxing it to:

NUM SA
Direction Marketing/Documentation
21, avenue du Maréchal Foch
BP 68 - 95101 Argenteuil Cedex

Fax : 33(0)1.34.23.66.00

We thank you for your cooperation.

		Unsatis-factory	Satisfactory
Layout			
	Format, binding	❑	❑
	Page layout	❑	❑
Accessibility (outline, table of contents, index)		❑	❑
Technical contents			
	Clarity	❑	❑
	Correctness	❑	❑
	Completeness	❑	❑
Understandability of the subject matter			
	Incremental approach	❑	❑
	Illustrations	❑	❑
	Examples	❑	❑

Comments and Suggestions

NAME: ______________________ FUNCTION: ______________________
COMPANY: ______________________ TELEPHONE: ______________________
ADDRESS: ____
ZIP CODE: ________ CITY: ______________ COUNTRY: ______________