

CNC

8060

8065

Programming manual

(Ref. 1402)



FAGOR AUTOMATION

MACHINE SAFETY

It is up to the machine manufacturer to make sure that the safety of the machine is enabled in order to prevent personal injury and damage to the CNC or to the products connected to it. On start-up and while validating CNC parameters, it checks the status of the following safety elements. If any of them is disabled, the CNC shows a warning message.

- Feedback alarm for analog axes.
- Software limits for analog and sercos linear axes.
- Following error monitoring for analog and sercos axes (except the spindle) both at the CNC and at the drives.
- Tendency test on analog axes.

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC resulting from any of the safety elements being disabled.

HARDWARE EXPANSIONS

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC resulting from any hardware manipulation by personnel unauthorized by Fagor Automation.

If the CNC hardware is modified by personnel unauthorized by Fagor Automation, it will no longer be under warranty.

COMPUTER VIRUSES

FAGOR AUTOMATION guarantees that the software installed contains no computer viruses. It is up to the user to keep the unit virus free in order to guarantee its proper operation.

Computer viruses at the CNC may cause it to malfunction. An antivirus software is highly recommended if the CNC is connected directly to another PC, it is part of a computer network or floppy disks or other computer media is used to transmit data.

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC due a computer virus in the system.

If a computer virus is found in the system, the unit will no longer be under warranty.



All rights reserved. No part of this documentation may be transmitted, transcribed, stored in a backup device or translated into another language without Fagor Automation's consent. Unauthorized copying or distributing of this software is prohibited.

The information described in this manual may be changed due to technical modifications. Fagor Automation reserves the right to make any changes to the contents of this manual without prior notice.

All the trade marks appearing in the manual belong to the corresponding owners. The use of these marks by third parties for their own purpose could violate the rights of the owners.

It is possible that CNC can execute more functions than those described in its associated documentation; however, Fagor Automation does not guarantee the validity of those applications. Therefore, except under the express permission from Fagor Automation, any CNC application that is not described in the documentation must be considered as "impossible". In any case, Fagor Automation shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC if it is used in any way other than as explained in the related documentation.

The content of this manual and its validity for the product described here has been verified. Even so, involuntary errors are possible, thus no absolute match is guaranteed. Anyway, the contents of the manual is periodically checked making and including the necessary corrections in a future edition. We appreciate your suggestions for improvement.

The examples described in this manual are for learning purposes. Before using them in industrial applications, they must be properly adapted making sure that the safety regulations are fully met.

INDEX

About the product	9
Declaration of conformity	13
Declaration of conformity	15
Version history	17
Safety conditions	19
Warranty terms	23
Material returning terms.....	25
CNC maintenance	27

CHAPTER 1 CREATING A PROGRAM.

1.1 Programming languages.....	29
1.2 Program structure.....	30
1.2.1 Program body.....	31
1.2.2 The subroutines.....	32
1.3 Program block structure.....	33
1.3.1 Programming in ISO code.....	34
1.3.2 High-level language programming.....	36
1.4 Programming of the axes.....	37
1.5 List of "G" functions.....	38
1.6 List of auxiliary (miscellaneous) M functions.....	42
1.7 List of statements and instructions.....	43
1.8 Comment programming.....	46
1.9 Variables and constants.....	47
1.10 Arithmetic parameters.....	48
1.11 Arithmetic and logic operators and functions.....	49
1.12 Arithmetic and logic expressions.....	51

CHAPTER 2 MACHINE OVERVIEW

2.1 Axis nomenclature	53
2.2 Coordinate system	55
2.3 Reference systems	56
2.3.1 Origins of the reference systems	57
2.4 Home search.....	58
2.4.1 Definition of "Home search".....	58
2.4.2 "Home search" programming.....	59

CHAPTER 3 COORDINATE SYSTEM

3.1 Programming in millimeters (G71) or in inches (G70).....	61
3.2 Absolute (G90) or incremental (G91) coordinates	62
3.2.1 Rotary axes.....	63
3.3 Programming in radius (G152) or in diameters (G151).....	65
3.4 Coordinate programming	66
3.4.1 Cartesian coordinates	66
3.4.2 Polar coordinates	67

CHAPTER 4 WORK PLANES.

4.1 About work planes on lathe and mill models.....	70
4.2 Select the main new work planes.....	71
4.2.1 Mill model or lathe model with "trihedron" type axis configuration.....	71
4.2.2 Lathe model with "plane" type axis configuration.....	72
4.3 Select any work plane and longitudinal axis.....	73
4.4 Select the longitudinal axis of the tool.....	75

CHAPTER 5 ORIGIN SELECTION

5.1 Programming with respect to machine zero.....	78
5.2 Set the machine coordinate (G174).....	80
5.3 Fixture offset	81
5.4 Coordinate preset (G92)	82

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

5.5	Zero offsets (G54-G59/G159)	83
5.5.1	Variables for setting zero offsets	85
5.5.2	Incremental zero offset (G158)	86
5.5.3	Excluding axes in the zero offset (G157)	88
5.6	Zero offset cancellation (G53)	89
5.7	Polar origin preset (G30)	90

CHAPTER 6**TECHNOLOGICAL FUNCTIONS**

6.1	Machining feedrate (F)	93
6.2	Feedrate related functions	95
6.2.1	Feedrate programming units (G93/G94/G95)	95
6.2.2	Feedrate blend (G108/G109/G193)	96
6.2.3	Constant feedrate mode (G197/G196)	98
6.2.4	Cancellation of the % of feedrate override (G266)	100
6.2.5	Acceleration control (G130/G131)	101
6.2.6	Jerk control (G132/G133)	103
6.2.7	Feed-Forward control (G134)	104
6.2.8	AC-Forward control (G135)	105
6.3	Spindle speed (S)	106
6.4	Tool number (T)	107
6.5	Tool offset number (D)	110
6.6	Auxiliary (miscellaneous) functions (M)	112
6.6.1	List of "M" functions	113
6.7	Auxiliary functions (H)	114

CHAPTER 7**THE SPINDLE. BASIC CONTROL.**

7.1	The master spindle of the channel	116
7.1.1	Manual selection of a master spindle	118
7.2	Spindle speed	119
7.2.1	G192. Turning speed limit	120
7.2.2	Constant surface speed	121
7.3	Spindle start and stop	122
7.4	Gear change	124
7.5	Spindle orientation	126
7.5.1	The turning direction for spindle orientation	128
7.5.2	M19 function with an associated subroutine	130
7.5.3	Positioning speed	131
7.6	M functions with an associated subroutine	132

CHAPTER 8**TOOL PATH CONTROL**

8.1	Rapid traverse (G00)	133
8.2	Linear interpolation (G01)	135
8.3	Circular interpolation (G02/G03)	138
8.3.1	Cartesian coordinates (Arc center programming)	140
8.3.2	Cartesian coordinates (Radius programming)	141
8.3.3	Polar coordinates	143
8.3.4	Temporary polar origin shift to the center of arc (G31)	146
8.3.5	Arc center in absolute coordinates (G06/G261/G262)	147
8.3.6	Arc center correction (G264/G265)	148
8.4	Arc tangent to previous path (G08)	149
8.5	Arc defined by three points (G09)	151
8.6	Helical interpolation (G02/G03)	152
8.7	Electronic threading with constant pitch (G33)	154
8.7.1	Programming examples for a mill	156
8.7.2	Programming examples for a lathe	157
8.8	Rigid tapping (G63)	159
8.9	Manual intervention (G200/G201/G202)	161
8.9.1	Additive manual intervention (G201/G202)	162
8.9.2	Exclusive manual intervention (G200)	163
8.9.3	Jogging feedrate	164

CHAPTER 9**GEOMETRY ASSISTANCE**

9.1	Square corner (G07/G60)	167
9.2	Semi-rounded corner (G50)	168
9.3	Controlled corner rounding, radius blend, (G05/G61)	169
9.3.1	Types of corner rounding	170
9.4	Corner rounding, radius blend, (G36)	174
9.5	Corner chamfering, (G39)	176
9.6	Tangential entry (G37)	178
9.7	Tangential exit (G38)	179



CNC 8060
CNC 8065

(REF. 1402)

9.8	Mirror image (G11, G12, G13, G10, G14)	180
9.9	Coordinate system rotation, pattern rotation, (G73).....	184
9.10	General scaling factor	186

CHAPTER 10 ADDITIONAL PREPARATORY FUNCTIONS

10.1	Dwell (G04)	189
10.2	Software limits by program (G198-G199)	190
10.3	Hirth axes (G170-G171).....	191
10.4	Changing of parameter range of an axis (G112)	192

CHAPTER 11 TOOL COMPENSATION

11.1	Tool radius compensation	195
11.1.1	Location code (shape or type) of the turning tools	196
11.1.2	Functions associates with radius compensation	199
11.1.3	Beginning of tool radius compensation	202
11.1.4	Sections of tool radius compensation	205
11.1.5	Change of type of radius compensation while machining	209
11.1.6	Cancellation of tool radius compensation	211
11.2	Tool length compensation	214

CHAPTER 12 SUBROUTINES.

12.1	Executing subroutines from RAM memory.	218
12.2	Definition of the subroutines	219
12.3	Subroutine execution.	220
12.3.1	LL. Call to a local subroutine	221
12.3.2	L Call to a global subroutine.....	221
12.3.3	#CALL. Call to a global or local subroutine.	221
12.3.4	#PCALL. Call to a global or local subroutine initializing parameters.	222
12.3.5	#MCALL. Modal call to a local or global subroutine.	223
12.3.6	#MDOFF. Turning the subroutine into non-modal.....	225
12.3.7	#RETDSBLK. Execute subroutine as a single block.....	226
12.4	#PATH. Define the location of the global subroutines.	227
12.5	OEM subroutine execution.....	228
12.6	Generic user subroutines (G500-G599).....	230
12.7	Assistance for subroutines.....	233
12.7.1	Subroutine help files.....	233
12.7.2	List of available subroutines.....	234
12.8	Interruption subroutines.	235
12.8.1	Repositioning axes and spindles from the subroutine (#REPOS).....	236
12.9	Subroutine "program_start".....	237

CHAPTER 13 EXECUTING BLOCKS AND PROGRAMS

13.1	Executing a program in the indicated channel.	239
13.2	Executing a block in the indicated channel.	241
13.3	Abort the execution of the program and resume it in another block or program.....	242

CHAPTER 14 "C" AXIS

14.1	Activating the spindle as "C" axis.....	246
14.2	Machining of the face of the part.....	248
14.3	Machining of the turning side of the part.....	250

CHAPTER 15 ANGULAR TRANSFORMATION OF AN INCLINE AXIS.

15.1	Turning angular transformation on and off.....	255
15.2	Freezing (suspending) the angular transformation	256
15.3	Obtaining information on angular transformation.....	257

CHAPTER 16 TANGENTIAL CONTROL.

16.1	Turning tangential control on and off.	261
16.2	Freezing tangential control.....	264
16.3	Obtaining information on tangential control.	266

CHAPTER 17 COORDINATE TRANSFORMATION

17.1	Movement in an inclined plane	269
17.2	Kinematics selection (#KIN ID)	271



CNC 8060
CNC 8065

(REF. 1402)

17.3	Coordinate systems (#CS) (#ACS)	272
17.3.1	Coordinate system definition MODE 1	275
17.3.2	Coordinate system definition MODE 2	277
17.3.3	Coordinate system definition MODE 3	279
17.3.4	Coordinate system definition MODE 4	280
17.3.5	Coordinate system definition MODE5	281
17.3.6	Coordinate system definition MODE6	282
17.3.7	Operation with 45° spindles (Huron type)	285
17.4	How to combine several coordinate systems	286
17.5	Tool perpendicular to the plane (#TOOL ORI)	288
17.6	Using RTCP (Rotating Tool Center Point)	290
17.6.1	Considerations about the RTCP function	293
17.7	Tool length compensation (#TLC)	294
17.8	Kinematics related variables	295
17.9	How to withdraw the tool when losing the plane	296

CHAPTER 18 HSC. HIGH SPEED MACHINING

18.1	HSC mode. Optimization of surface error	298
18.2	HSC mode. Optimizing the contouring error	299
18.3	HSC mode. Optimizing the machining speed	301
18.4	Canceling the HSC mode	303

CHAPTER 19 LASER.

19.1	Synchronized switching	305
19.1.1	Activate synchronized switching	306
19.1.2	Cancel synchronized switching	307
19.1.3	Variables related to synchronized switching	308
19.2	PWM (Pulse-Width Modulation)	309
19.2.1	Activate the PWM	310
19.2.2	Cancel the PWM	312
19.2.3	PWM variables	313

CHAPTER 20 VIRTUAL TOOL AXIS.

20.1	Activate the virtual tool axis	316
20.2	Cancel the virtual tool axis	317
20.3	Variables associated with the virtual tool axis	318

CHAPTER 21 STATEMENTS AND INSTRUCTIONS

21.1	Programming statements	320
21.1.1	Display instructions. Display an error on the screen	320
21.1.2	Display instructions. Display a warning on the screen	322
21.1.3	Display instructions. Display a message on the screen	324
21.1.4	Display instructions. Define the size of the the graphics area	325
21.1.5	Enabling and disabling instructions	326
21.1.6	Electronic axis slaving	327
21.1.7	Axis parking	328
21.1.8	Modifying the configuration of the axes of a channel	330
21.1.9	Modifying the configuration of the spindles of a channel	335
21.1.10	Spindle synchronization	338
21.1.11	Selecting the loop for an axis or a spindle. Open loop or closed loop	342
21.1.12	Collision detection	344
21.1.13	Spline interpolation (Akima)	346
21.1.14	Polynomial interpolation	349
21.1.15	Acceleration control	350
21.1.16	Definition of macros	352
21.1.17	Block repetition	354
21.1.18	Communication and synchronization between channels	356
21.1.19	Movements of independent axes	359
21.1.20	Electronic cams	363
21.1.21	Additional programming instructions	366
21.2	Flow controlling instructions	367
21.2.1	Jump to a block (\$GOTO)	367
21.2.2	Conditional execution (\$IF)	368
21.2.3	Conditional execution (\$SWITCH)	370
21.2.4	Block repetition (\$FOR)	371
21.2.5	Conditional block repetition (\$WHILE)	372
21.2.6	Conditional block repetition (\$DO)	373



CNC 8060
CNC 8065

(REF. 1402)

CHAPTER 22 CNC VARIABLES.

22.1	Understanding how variables work	375
22.1.1	Accessing numeric variables from the PLC.	377
22.2	Variables in a single-channel system.....	378
22.3	Variables in a multi-channel system.	381
22.4	Variables related to general machine parameters.	384
22.5	Variables related to the machine parameters of the channels.....	405
22.6	Variables related to axis and spindle machine parameters.	428
22.7	Variables related to the sets of machine parameters.....	465
22.8	Variables related to machine parameters for JOG mode.....	519
22.9	Variables related to machine parameters for M functions.....	523
22.10	Variables related to kinematic machine parameters.	525
22.11	Variables related to machine parameters for the tool magazine.....	529
22.12	Variables related to OEM machine parameters.	532
22.13	Variables associated with the status and resources of the PLC.	534
22.14	PLC consulting logic signals; general.	538
22.15	PLC consulting logic signals; axes and spindles.	549
22.16	PLC consulting logic signals; spindles.	554
22.17	PLC consulting logic signals; independent interpolator.	556
22.18	PLC consulting logic signals; tool manager.	558
22.19	PLC consulting logic signals; keys.....	561
22.20	PLC modifiable logic signals; general.....	562
22.21	PLC modifiable logic signals; axes and spindles.	570
22.22	PLC modifiable logic signals; spindles.....	576
22.23	PLC modifiable logic signals; independent interpolator.	578
22.24	PLC modifiable logic signals; tool manager.	579
22.25	PLC modifiable logic signals; keys.....	584
22.26	Variables related to the machine configuration.....	585
22.27	Variables related to volumetric compensation.	593
22.28	Variables associated with the Mechatrolink bus.	594
22.29	Variables related to synchronized switching.	596
22.30	PWM related variables.....	597
22.31	Variables related to cycle time.....	599
22.32	Variables associated with the feedback inputs for analog axes.....	601
22.33	Variables associated with the analog inputs and outputs.	603
22.34	Variables associated with the velocity command and the feedback of the drive.	604
22.35	Variables related to the change of gear and set of the Sercos drive.	606
22.36	Variables related to loop adjustment.....	607
22.37	Variables related to the loop of the axis or of the tandem spindle.	615
22.38	Variables related to user tables (zero offset table).	617
22.39	Variables related to user tables (fixture table).	622
22.40	Variables related to user tables (arithmetic parameters table).	624
22.41	Variables related to the position of the axes.	628
22.42	Variables related to spindle position.	634
22.43	Feedrate related variables.	636
22.44	Variables associated with acceleration and jerk on the tool path.	641
22.45	Variables related to managing the feedrate in HSC mode.....	642
22.46	Variables related to spindle speed.....	646
22.47	Variables associated with the tool manager.	654
22.48	Variables related to managing the tool magazine and the tool changer arm.....	656
22.49	Variables related to the active tool and to the next one.	658
22.50	Variables associated with any tool.....	670
22.51	Variables associated with the tool being prepared.	679
22.52	Variables related to jog mode.	687
22.53	Variables related to the programmed functions.	693
22.54	Variables related to the electronic cam.....	720
22.55	Variables related to the independent axes.....	722
22.56	Variables associated with the virtual tool axis.....	729
22.57	Variables defined by the user.	730
22.58	General variables of the CNC.	731
22.59	Variables related to CNC status.....	734
22.60	Variables associated with the part-program being executed.	739
22.61	Interface related variables.....	743



CNC 8060
CNC 8065

(REF. 1402)

ABOUT THE PRODUCT

BASIC CHARACTERISTICS.

Basic characteristics.	8060 M	8060 T	8065 M	8065 T
PC-based system.	Closed system		Open system	
Operating system.	Windows XPe		Windows XP	
Number of axes.	3 a 5		3 to 28	
Number of spindles.	1	2	1 to 4	
Number of tool magazines.	1	2	1 to 4	
Number of execution channels.	1	2	1 to 4	
Number of handwheels.	1 a 3		1 to 12	
Type of servo system.	Analog / Sercos Digital		Analog / Digital Sercos / Digital Mechatrolink	
Communications.	RS485 / RS422 / RS232 Ethernet			
Integrated PLC.				
PLC execution time.	< 1ms/K			
Digital inputs / Digital outputs.	1024 / 1024			
Marks / Registers.	8192 / 1024			
Timers / Counters.	512 / 256			
Symbols.	Unlimited			
Block processing time.	< 1.5 ms		< 1 ms	

Remote modules.	RIOW	RIO5	RIO70	RCS-S
Communication with the remote modules.	CANopen	CANopen	CANfagor	Sercos
Digital inputs per module.	8	16 / 32	16	---
Digital outputs per module.	8	24 / 48	16	---
Analog inputs per module.	4	4	8	---
Analog outputs per module.	4	4	4	4
Inputs for PT100 temperature sensors.	2	2	---	---
Feedback inputs.	---	---	4 (*)	4 (**)

(*) Differential TTL / Sinusoidal 1 Vpp (**) TTL / Differential TTL / Sinusoidal 1 Vpp / SSI protocol

Customizing (on an open system only).

PC-based open system, fully customizable.
 INI configuration files.
 FGUIM visual configuration tool.
 Visual Basic®, Visual C++®, etc.
 Internal databases in Microsoft® Access.
 OPC compatible interface



CNC 8060
CNC 8065

(REF. 1402)

SOFTWARE OPTIONS.

Bear in mind that some of the features described in this manual depend on the software options that are installed. The information of the following table is informative only; when purchasing the software options, only the information provided in the ordering handbook is valid.

Software options (.M· model).

	8060 M	8065 M		8065 M Power	
		Basic	Pack 1	Basic	Pack 1
Open system. Access to the administrator mode.	---	---	---	Option	Option
Number of execution channels	1	1	1	1	1 to 4
Number of axes	3 a 5	3 to 6	5 to 8	5 to 12	8 to 28
Number of spindles	1	1	1 to 2	1 to 4	1 to 4
Number of tool magazines	1	1	1	1 to 2	1 to 4
Limited to 4 interpolated axes	Option	Option	Option	Option	Option
IEC 61131 language	---	---	Option	Option	Option
HD graphics	Option	Option	Option	Standard	Standard
Conversational IIP	Option	Option	Option	Option	Option
Dual-purpose machines (M-T)	---	---	---	Option	Standard
"C" axis	Option	Standard	Standard	Standard	Standard
Dynamic RTCP	Option	---	Option	Option	Standard
HSSA machining system.	Option	Standard	Standard	Standard	Standard
Probing canned cycles	Option	Option	Standard	Standard	Standard
Tandem axes	Option	---	Option	Standard	Standard
Synchronism and cams	---	---	---	Option	Standard
Tangential control	---	---	Standard	Standard	Standard
Volumetric compensation (up to 10 m ³).	---	---	---	Option	Option
Volumetric compensation (more than 10 m ³).	---	---	---	Option	Option
ProGTL3 language.	Option	Option	Option	Option	Option
Part-program translator.	Option	Option	Option	Option	Option



CNC 8060
CNC 8065

(REF. 1402)

Software options (-T- model).

	8060 T	8065 T		8065 T Power	
		Basic	Pack 1	Basic	Pack 1
Open system. Access to the administrator mode.	---	---	---	Option	Option
Number of execution channels	2	1	1 to 2	1 to 2	1 to 4
Number of axes	3 a 5	3 to 5	5 to 7	5 to 12	8 to 28
Number of spindles	2	2	2	3 to 4	3 to 4
Number of tool magazines	2	1	1 to 2	1 to 2	1 to 4
Limited to 4 interpolated axes	Option	Option	Option	Option	Option
IEC 61131 language	---	---	Option	Option	Option
HD graphics	Option	Option	Option	Standard	Standard
Conversational IIP	Option	Option	Option	Option	Option
Dual-purpose machines (T-M)	---	---	---	Option	Standard
"C" axis	Option	Option	Standard	Standard	Standard
Dynamic RTCP	---	---	---	Option	Standard
HSSA machining system.	Option	Option	Standard	Standard	Standard
Probing canned cycles	Option	Option	Standard	Standard	Standard
Tandem axes	Option	---	Option	Standard	Standard
Synchronism and cams	Option	---	Option	Option	Standard
Tangential control	---	---	---	Option	Standard
Volumetric compensation (up to 10 m³).	---	---	---	Option	Option
Volumetric compensation (more than 10 m³).	---	---	---	Option	Option
ProGTL3 language.	---	---	---	---	---
Part-program translator.	Option	---	---	---	---



**CNC 8060
CNC 8065**

(REF. 1402)

DECLARATION OF CONFORMITY

The manufacturer:

Fagor Automation S. Coop.
Barrio de San Andrés Nº 19, C.P.20500, Mondragón -Guipúzcoa- (Spain).

Declares:

The manufacturer declares under their exclusive responsibility the conformity of the product:

8060 CNC

Consisting of the following modules and accessories:

CN60-10H, CN60-10HT, CN60-10V, CN60-10VT
OP-PANEL, OP-PANEL+SPDL RATE
Remote Modules RIOW, RIO5, RIO70, RCS-S, RIOR.

Note.Some additional characters may follow the model references indicated above. They all comply with the directives listed here. However, compliance may be verified on the label of the unit itself.

Referred to by this declaration with following directives:

Low-voltage regulations.

IEC 60204-1:2005/A1:2008 Electrical equipment on machines. Part1. General requirements.

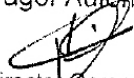
Regulation on electromagnetic compatibility.

EN 61131-2: 2007 PLC. Part 2. Equipment requirements and tests.

According to the European Community Directives 2006/95/EC on Low Voltage and 2004/108/EC on Electromagnetic Compatibility and their updates.

In Mondragón, February 1st, 2014.

Fagor Automation, S. Coop.


Director Gerente
Pedro Ruiz de Aguirre

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

DECLARATION OF CONFORMITY

The manufacturer:

Fagor Automation S. Coop.
Barrio de San Andrés Nº 19, C.P.20500, Mondragón -Guipúzcoa- (Spain).

Declares:

The manufacturer declares under their exclusive responsibility the conformity of the product:

8065 CNC

Consisting of the following modules and accessories:

8065-M-ICU, 8065-T-ICU
MONITOR-LCD-10K, MONITOR-LCD-15, MONITOR-SVGA-15
HORIZONTAL-KEYB, VERTICAL-KEYB, OP-PANEL
BATTERY
Remote Modules RIOW, RIO5, RIO70, RCS-S.

Note.Some additional characters may follow the model references indicated above. They all comply with the directives listed here. However, compliance may be verified on the label of the unit itself.

Referred to by this declaration with following directives:

Low-voltage regulations.

IEC 60204-1:2005/A1:2008 Electrical equipment on machines. Part1. General requirements.


Regulation on electromagnetic compatibility.

EN 61131-2: 2007 PLC. Part 2. Equipment requirements and tests.

According to the European Community Directives 2006/95/EC on Low Voltage and 2004/108/EC on Electromagnetic Compatibility and their updates.

In Mondragón, September 1st, 2013.

Fagor Automation, S. Coop.


Director Gerente
Pedro Ruiz de Aguirre

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

VERSION HISTORY

Here is a list of the features added to each manual reference.

Ref. 1402

Software V01.00 (8060)

8060. First version.



CNC 8060
CNC 8065

(REF. 1402)

SAFETY CONDITIONS

Read the following safety measures in order to prevent harming people or damage to this product and those products connected to it. Fagor Automation shall not be held responsible of any physical damage or defective unit resulting from not complying with these basic safety regulations.



Before start-up, verify that the machine that integrates this CNC meets the 89/392/CEE Directive.

PRECAUTIONS BEFORE CLEANING THE UNIT

If the CNC does not turn on when actuating the start-up switch, verify the connections.

- Do not get into the inside of the unit.** Only personnel authorized by Fagor Automation may manipulate the inside of this unit.
- Do not handle the connectors with the unit connected to AC power.** Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

PRECAUTIONS DURING REPAIR

In case of a malfunction or failure, disconnect it and call the technical service.

- Do not get into the inside of the unit.** Only personnel authorized by Fagor Automation may manipulate the inside of this unit.
- Do not handle the connectors with the unit connected to AC power.** Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

PRECAUTIONS AGAINST PERSONAL DAMAGE

- Interconnection of modules.** Use the connection cables provided with the unit.
- Use proper cables.** To prevent risks, use the proper cables for mains, Sercos and Bus CAN recommended for this unit.
In order to avoid electrical shock at the central unit, use the proper power (mains) cable. Use 3-wire power cables (one for ground connection).
- Avoid electrical overloads.** In order to avoid electrical discharges and fire hazards, do not apply electrical voltage outside the range selected on the rear panel of the central unit.
- Ground connection.** In order to avoid electrical discharges, connect the ground terminals of all the modules to the main ground terminal. Before connecting the inputs and outputs of this unit, make sure that all the grounding connections are properly made.
In order to avoid electrical shock, before turning the unit on verify that the ground connection is properly made.
- Do not work in humid environments.** 8060. In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 50 °C (122 °F).
8065. In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45 °C (113 °F).
- Do not work in explosive environments.** In order to avoid risks or damages, do no work in explosive environments.



CNC 8060
CNC 8065

(REF. 1402)

PRECAUTIONS AGAINST PRODUCT DAMAGE

Working environment.	This unit is ready to be used in industrial environments complying with the directives and regulations effective in the European Community. Fagor Automation shall not be held responsible for any damage suffered or caused by the CNC when installed in other environments (residential or homes).
Install the unit in the right place.	It is recommended, whenever possible, to install the CNC away from coolants, chemical product, blows, etc. that could damage it. This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as: <ul style="list-style-type: none"> Powerful loads connected to the same AC power line as this equipment. Nearby portable transmitters (Radio-telephones, Ham radio transmitters). Nearby radio/TV transmitters. Nearby arc welding machines. Nearby High Voltage power lines.
Enclosures.	The manufacturer is responsible of assuring that the enclosure involving the equipment meets all the currently effective directives of the European Community.
Avoid disturbances coming from the machine.	The machine must have all the interference generating elements (relay coils, contactors, motors, etc.) uncoupled.
Use the proper power supply.	Use an external regulated 24 Vdc power supply for the keyboard and the remote modules.
Grounding of the power supply.	The zero volt point of the external power supply must be connected to the main ground point of the machine.
Analog inputs and outputs connection.	Use shielded cables connecting all their meshes to the corresponding pin.
Ambient conditions.	8060. The storage temperature must be between +5 °C and +55 °C (41 °F and 131 °F). 8065. The storage temperature must be between +5 °C and +45 °C (41 °F and 113 °F). The storage temperature must be between -25 °C and 70 °C (-13 °F and 158 °F).
Central unit enclosure.	Make sure that the needed gap is kept between the central unit and each wall of the enclosure. Use a DC fan to improve enclosure ventilation.
Main AC power switch.	This switch must be easy to access and at a distance between 0.7 and 1.7 m (2.3 and 5.6 ft) off the floor.

PROTECTIONS OF THE UNIT ITSELF

Remote modules.	All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
------------------------	--



CNC 8060
CNC 8065

(REF. 1402)

SAFETY SYMBOLS

Symbols that may appear on the manual.



Danger or prohibition symbol.

It indicates actions or operations that may hurt people or damage products.



Warning symbol.

It indicates situations that certain operations could cause and the suggested actions to prevent them.



Obligation symbol.

It indicates actions and operations that must be carried out.



Information symbol.

It indicates notes, warnings and advises.

Symbols that the product may carry.



Ground protection symbol.

It indicates that that point must be under voltage.

WARRANTY TERMS

INITIAL WARRANTY

All products manufactured or marketed by FAGOR carry a 12-month warranty for the end user which could be controlled by the our service network by means of the warranty control system established by FAGOR for this purpose.

In order to prevent the possibility of having the time period from the time a product leaves our warehouse until the end user actually receives it run against this 12-month warranty, FAGOR has set up a warranty control system based on having the manufacturer or agent inform FAGOR of the destination, identification and on-machine installation date, by filling out the document accompanying each FAGOR product in the warranty envelope. This system, besides assuring a full year of warranty to the end user, enables our service network to know about FAGOR equipment coming from other countries into their area of responsibility.

The warranty starting date will be the one appearing as the installation date on the above mentioned document. FAGOR offers the manufacturer or agent 12 months to sell and install the product. This means that the warranty starting date may be up to one year after the product has left our warehouse so long as the warranty control sheet has been sent back to us. This translates into the extension of warranty period to two years since the product left our warehouse. If this sheet has not been sent to us, the warranty period ends 15 months from when the product left our warehouse.

This warranty covers all costs of material and labour involved in repairs at FAGOR carried out to correct malfunctions in the equipment. FAGOR undertakes to repair or replace their products within the period from the moment manufacture begins until 8 years after the date on which it disappears from the catalogue.

It is entirely up to FAGOR to determine whether the repair is or not under warranty.

EXCLUDING CLAUSES

Repairs will be carried out on our premises. Therefore, all expenses incurred as a result of trips made by technical personnel to carry out equipment repairs, despite these being within the above-mentioned period of warranty, are not covered by the warranty.

Said warranty will be applied whenever the equipment has been installed in accordance with instructions, has not be mistreated, has not been damaged by accident or by negligence and has not been tampered with by personnel not authorised by FAGOR. If, once servicing or repairs have been made, the cause of the malfunction cannot be attributed to said elements, the customer is obliged to cover the expenses incurred, in accordance with the tariffs in force.

Other warranties, implicit or explicit, are not covered and FAGOR AUTOMATION cannot be held responsible for other damages which may occur.



CNC 8060
CNC 8065

(REF. 1402)

WARRANTY ON REPAIRS

In a similar way to the initial warranty, FAGOR offers a warranty on standard repairs according to the following conditions:

PERIOD	12 months.
CONCEPT	Covers parts and labor for repairs (or replacements) at the network's own facilities.
EXCLUDING CLAUSES	The same as those applied regarding the chapter on initial warranty. If the repair is carried out within the warranty period, the warranty extension has no effect.

When the customer does not choose the standard repair and just the faulty material has been replaced, the warranty will cover just the replaced parts or components within 12 months.

For sold parts the warranty is 12 months length.

SERVICE CONTRACTS

The SERVICE CONTRACT is available for the distributor or manufacturer who buys and installs our CNC systems.



CNC 8060
CNC 8065

(REF. 1402)

MATERIAL RETURNING TERMS

When sending the central nit or the remote modules, pack them in its original package and packaging material. If the original packaging material is not available, pack it as follows:

- 1 Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).
- 2 Attach a label indicating the owner of the unit, person to contact, type of unit and serial number. In case of malfunction also indicate symptom and a brief description of the problem.
- 3 Wrap the unit in a polyethylene roll or similar material to protect it. When sending a central unit with monitor, protect especially the screen.
- 4 Pad the unit inside the cardboard box with poly-utherane foam on all sides.
- 5 Seal the cardboard box with packing tape or industrial staples.



CNC 8060
CNC 8065

(REF. 1402)

CNC MAINTENANCE

CLEANING

The accumulated dirt inside the unit may act as a screen preventing the proper dissipation of the heat generated by the internal circuitry which could result in a harmful overheating of the unit and, consequently, possible malfunctions. Accumulated dirt can sometimes act as an electrical conductor and short-circuit the internal circuitry, especially under high humidity conditions.

To clean the operator panel and the monitor, a smooth cloth should be used which has been dipped into de-ionized water and /or non abrasive dish-washer soap (liquid, never powder) or 75° alcohol. Do not use highly compressed air to clean the unit because it could generate electrostatic discharges.

The plastics used on the front panel are resistant to grease and mineral oils, bases and bleach, dissolved detergents and alcohol. Avoid the action of solvents such as chlorine hydrocarbons, benzole, esters and ether which can damage the plastics used to make the unit's front panel.

PRECAUTIONS BEFORE CLEANING THE UNIT

Fagor Automation shall not be held responsible for any material or physical damage derived from the violation of these basic safety requirements.

- Do not handle the connectors with the unit connected to AC power. Before handling these connectors (I/O, feedback, etc.), make sure that the unit is not connected to main AC power.
- Do not get into the inside of the unit. Only personnel authorized by Fagor Automation may manipulate the inside of this unit.
- If the CNC does not turn on when actuating the start-up switch, verify the connections.



CNC 8060
CNC 8065

(REF. 1402)

CREATING A PROGRAM.

1

1.1 Programming languages.

The CNC has its own programming language described in this manual. The program is edited block by block and each one may be written in ISO language or in High level language. See "[1.3 Program block structure.](#)" on page 33.

When editing high level commands, the editor offers a list of available commands.

8055 language.

Programs can also be edited in the 8055 CNC language. Programming in 8055 CNC language is enabled from the part-program editor. Refer to the operating manual to enable this option.

This manual does not describe the 8055 language; refer to the specific documentation for this product. Obviously, since this CNC and the 8055 are two functionally different products, some concepts may be different.



CNC 8060
CNC 8065

(REF. 1402)

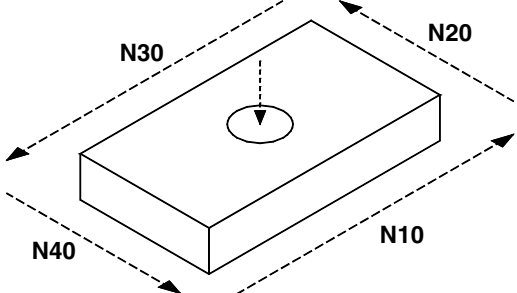
1.2 Program structure.

A CNC program consists of a set of blocks or instructions that properly organized, in subroutines or in the program body, provide the CNC with the necessary data to machine the desired part.

Each block contains all the functions or command necessary to execute an operation that may be machining, preparing the cutting conditions, controlling the elements of the machine, etc.

1.

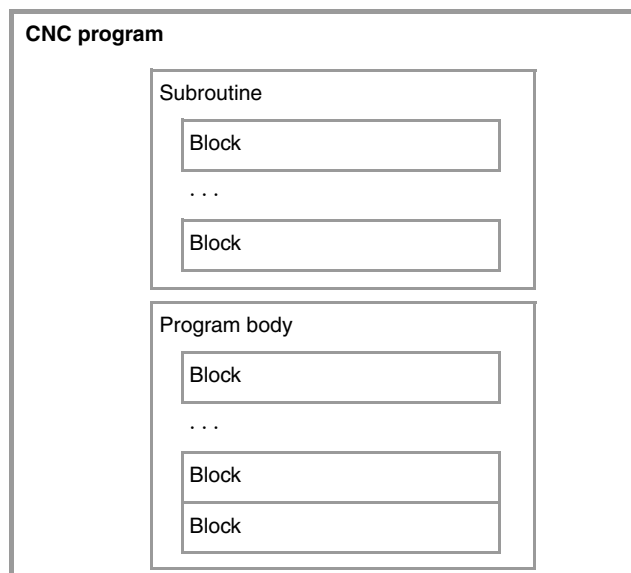
CREATING A PROGRAM.
Program structure.



```

%example
(Name of the program)
N5 F550 S1000 M3 M8 T1 D1
(Sets the machining conditions)
N6 GO X0 Y0
(Positioning)
N10 G1 G90 X100
N20 Y50
N30 X0
N40 Y0
(Machining)
N50 M30
(End of program)
    
```

The CNC program may consist of several local subroutines and the body of the program. The local subroutines must be defined at the beginning of the program.



CNC 8060
CNC 8065

(REF. 1402)

1.2.1 Program body.

The body of the program has the following structure.

Header	The header indicates the beginning of the body of the program. The header must be programmed when the program has local subroutines.
Program blocks	It is the main part of the program, the one containing movements, operations, etc.
End of program	

Program header.

The header of the program is a block consisting of the "%" character followed by the name of the program. The name of the program may be up to 14 characters long and may consist of uppercase and lowercase characters as well as numbers (no blank spaces are allowed).

```
%O123
%PROGRAM
%PART923R
```

The header must be programmed when the program contains local subroutines; otherwise, programming the header is optional.

The name defined in the header has nothing to do with the name of the file. The two may be different.

Program body.

The body of the program consists of blocks in charge of executing operations, movements, etc.

End of the program.

The end of the program body is defined by functions "M02" or "M30" and they are equivalent. There is no need to program these functions; when reaching the end of the program without executing any of them, the CNC ends the execution and shows a warning indicating that they are missing.

```
M30
M02
```

The CNC behaves differently when reaching the end of the program depending on whether the M02 / M30 has been programmed or not

	With M02/M30	Without M02/M30
The CNC selects the first block of the program.	Yes	Yes
The CNC stops the spindle.	Yes	No
The CNC assumes the initial conditions.	Yes (*)	No
The CNC initializes the cutting conditions.	Yes	No

(*) Stopping the spindle depends on the setting of machine parameter SPDLSTOP.

1.

CREATING A PROGRAM.
Program structure.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

1.2.2 The subroutines.

A subroutine is a set of blocks that, once properly identified, may be called upon several times from another subroutine or from the program. Subroutines are normally used for defining a bunch of operations or movements that are repeated several times throughout the program. See chapter "12 Subroutines."

Types of subroutines.

The CNC has two types of subroutines, namely local and global. There is also a third type available, OEM subroutines, that are a special case of a global subroutine defined by the OEM.

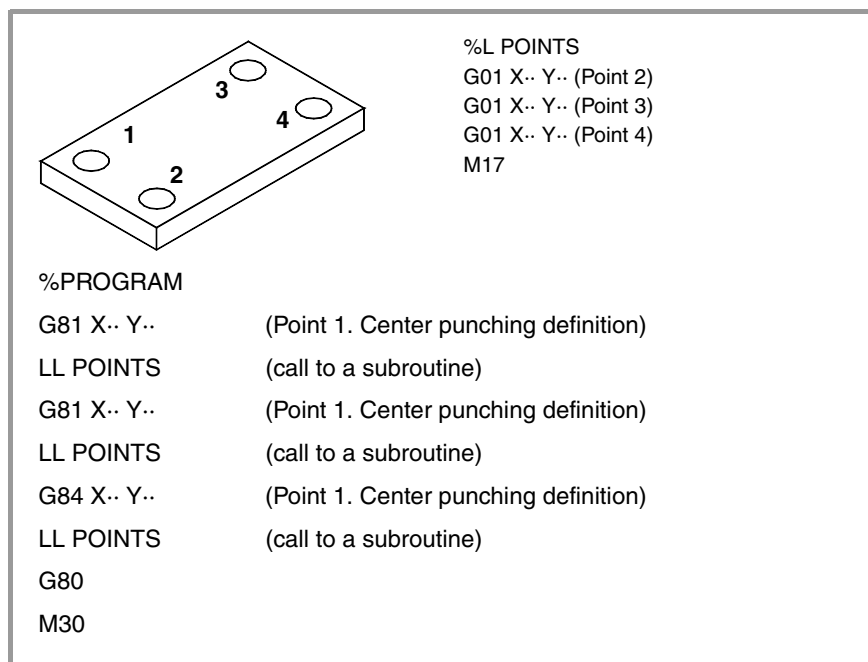
Global subroutines.

The global subroutine is stored in CNC memory as an independent program. This subroutine may be called upon from any program or subroutine being executed.

Local subroutines.

The local subroutine is defined as part of a program. This subroutine may only be called upon from the program where it has been defined.

A program can have several local subroutines; but they all must be defined before the body of the program. A local subroutine can call a second local subroutine with the condition that the calling subroutine be defined after the one being called.



1.3 Program block structure.

The blocks comprising the subroutines or the program body may be defined by commands in ISO code or in high-level language. Each block must be written in either language but not mixed; a program may combine blocks written in both languages. Empty blocks (empty lines) are also allowed.

In either language, it is also possible to use any type of arithmetic, relational or logic expression.

Programming in ISO code.

It is especially designed to control the movement of the axes because it provides movement data and conditions as well as feedrate and speed. Some of the available commands are:

- Preparatory functions for movement establishing the geometry and work conditions such as linear and circular interpolations, threading, canned cycles, etc.
- Functions to control cutting conditions such as feedrate of the axes, spindle speed and accelerations.
- Functions to control the tools.
- Additional functions containing technological instructions.
- Definition of position values.

High-level language programming.

This language provides the user with a set of control commands with a terminology similar to the one used by other languages, such as \$IF, \$GOTO, #MSG, #HSC, etc. Some of the available commands are:

- Programming instructions.
- Flow controlling instructions to make loops and jumps within the program.
- To define and call upon subroutines with local parameters where a local variable is the one only known to the subroutine where it has been defined.

It is also possible to use any type of arithmetic, relational or logic expression.

Arithmetic parameters, variables, constants and arithmetic expressions.

Constants, arithmetic parameters, variables and arithmetic expressions may be used from ISO blocks as well as from high level commands.

1.

CREATING A PROGRAM.
Program block structure.

FAGOR 

**CNC 8060
CNC 8065**

(REF. 1402)

1.3.1 Programming in ISO code.

ISO-coded functions consist of letters and numbers. The letters are "N", "G", "F", "S", "T", "D", "M", "H", "NR" plus those identifying the axes.

The numbers include digits "0" through "9", the "+" and "-" signs and the decimal point ".". Likewise, the numerical format may be replaced by a parameter, variable or arithmetic expression whose result is a number.

Programming allows blank spaces between letters, numbers and a sign as well as not using the sign with positive values.

Block structure.

A block may have the following functions, but needs not contain all of them. The data has no set order, it may be programmed anywhere in the block. The only exception being the block-skip condition and the block identification which must always be programmed at the beginning.

```
/ N— G— G— X..C— F— S— T— D— M— H— NR—
```

·/· Block skip condition.

If the block-skip mark is active, the CNC will skip the blocks having this character (not executing them) and will go on to the next block.

The CNC reads several blocks ahead of the one in execution, in order to calculate in advance the path to travel. The block-skip condition is examined at the time when the block is read.

·N· Block identification.

The block identification must be programmed when the block is used as the destination of references or jumps. In this case, it is recommended to program it alone in the block. It may be represented in two ways:

- The letter "N" followed by the block number (0-4294967295) and the ":" character (only when the label is used as the destination of a block jump); they need not follow a particular order or be consecutive.

If the label is not a jump target and is programmed without ":", it may go in any position of the block, not necessarily at the beginning.

- "[<name>]" type labels, where <name> may be up to 14 characters long and may consist of uppercase and lowercase characters as well as numbers (no blank spaces are allowed).

Both types of data may be programmed in the same block.

```
N10: X12 T1 D1
[CYCLE] G81 I67
X34 N10 S100 M3
```

·G· Preparatory functions.

G functions set the geometry and work conditions such as linear and circular interpolations, chamfers, canned cycles, etc. See "[1.5 List of "G" functions.](#)" on page 38.

·X..C· Coordinates of the point

These functions set the movement of the axes. See "[1.4 Programming of the axes.](#)" on page 37.

Depending on the units, the programming format will be:

- In millimeters, format ± 5.4 (5 integers and 4 decimals).
- In inches, format ± 4.5 (4 integers and 5 decimals).

·F· Axis feedrate.

The feedrate is represented by the letter "F" followed by the desired feedrate value.

1.

CREATING A PROGRAM.
Program block structure.



CNC 8060
CNC 8065

(REF. 1402)

·S· Spindle speed

This function sets the spindle speed.

The spindle name is defined by 1 or 2 characters. The first character is the letter S and the second character is optional, it must be a numerical suffix between 1 and 9. This way, the name of the spindles may be within the range S, S1 ... S9.

The feedrate is represented by the axis letter followed by the target position for the axis. For spindles like S1, S2, etc. the "=" sign must be included between the axis name and the speed.

```
S1000
S1=334
```

·T· Tool number.

This function selects the tool to be used to carry out the programmed machining operation. The tool is represented by the letter "T" followed by the tool number (0-4294967295).

·D· Tool offset number.

This function selects the tool offset. The tool offset is represented by the letter "D" followed by the tool offset number. The number of offsets available for each tool is defined in the tool table.

·M H· Auxiliary functions.

With the auxiliary functions, it is possible to control machine elements such as spindle turning direction, coolant, etc. These functions are represented by the letters "M" or "H" followed by the function number (0-65535)

·NR· Number of block repetitions.

It indicates the number of times the block will be executed. It can only be programmed in blocks containing a movement.

If the block is under the influence of a modal canned cycle, the latter will be repeated as many times as the block repetition has been programmed. When programming NR0, the movements will be executed, but the modal canned cycle is not executed at the end of each one.

```
G91 G01 X34.678 F150 NR4
```

Block comment .

Any comment may be associated with the blocks. When executing the program, the CNC ignores this information.

The CNC offers various methods to include comments in the program. See ["1.8 Comment programming."](#) on page 46.

1.

CREATING A PROGRAM.
Program block structure.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

1.3.2 High-level language programming.

The commands of high level language are made up of control instructions "#" and flow control instructions "\$".

Block structure.

A block may have the following commands, but needs not contain all of them.

```
/ N— <rest of commands>
```

·/· Block skip condition.

If the block-skip mark is active, the CNC will skip the blocks having this character (not executing them) and will go on to the next block.

The CNC reads several blocks ahead of the one in execution, in order to calculate in advance the path to travel. The block-skip condition is examined at the time when the block is read.

·N· Block identification.

The block identification must be programmed when the block is used as the destination of references or jumps. In this case, it is recommended to program it alone in the block. It may be represented in two ways:

- The letter "N" followed by the block number (0-4294967295) and the ":" character (only when the label is used as the destination of a block jump); they need not follow a particular order or be consecutive.

If the label is not a jump target and is programmed without ":", it may go in any position of the block, not necessarily at the beginning.

- "[<name>]" type labels, where <name> may be up to 14 characters long and may consist of uppercase and lowercase characters as well as numbers (no blank spaces are allowed).

Both types of data may be programmed in the same block.

·# \$· High-level language commands.

The high-level commands comprise the instructions and flow control instructions.

- Instructions are programmed preceded by the "#" sign and they can only be programmed one per block. They are used to carry out various functions.
- Flow control instructions are programmed preceded by the "\$" sign and can only be programmed one per block. They are used to make loops and program jumps.

Assigning values to parameters and variables can also be considered as high-level commands.

Block comment .

Any comment may be associated with the blocks. When executing the program, the CNC ignores this information.

The CNC offers various methods to include comments in the program. See ["1.8 Comment programming."](#) on page 46.

1.

CREATING A PROGRAM.
Program block structure.



CNC 8060
CNC 8065

(REF. 1402)

1.4 Programming of the axes.

Programming using the name of the axis.

The axis name is defined by 1 or 2 characters. The first character must be one of the letters X - Y - Z - U - V - W - A - B - C. The second character is optional and will be a numerical suffix between 1 and 9. This way, the name of the spindles may be within the range X, X1...X9,...C, C1...C9.

The movements are represented by the axis letter followed by the target position for the axis. For axes like X1, Y2, etc. the "=" sign must be included between the axis name and the coordinate.

```
X100
Z34.54
X2=123.4
A5=78.532
```

Programming using wild cards.

The axes can also be programmed using wild cards. The wild cards may be used to program and refer to the axes of the channel using their position in it, including the empty spaces. The wild card is represented with the "?" character followed by the position number of the axis, ?1 for the first axis, ?2 for the second one, and so forth. If the position of an empty space is programmed, the CNC will display an error message.

```
Y 00000.0000
X 00000.0000
? * * * * . * * * *
Z 00000.0000
```

In a channel with the following distribution of axes, the wild cards refer to the following axes.

- The ?1 wild card corresponds to the Y axis.
- The ?2 wild card corresponds to the X axis.
- The ?3 wild card issues an error, there is no axis in that position.
- The ?4 wild card corresponds to the Z axis.

Using these wild cards, the user can program a movement as follows.

```
?1 = 12345.1234
?2 = 50.34
```

Besides for programming movements, the wild cards can also be used to refer to the axes in the following G functions and instructions.

G functions.		Instructions.	
G14	G134	#MOVE ABS	#LINK
G45	G135	#MOVE ADD	#UNLINK
G74	G145	#MOVE INF	#PARK
G92	G158	#CAM ON	#UNPARK
G100	G170	#CAM OFF	#SERVO ON
G101	G171	#FOLLOW ON	#SERVO OFF
G112	G198	#FOLLOW OFF	
G130	G199	#TOOL AX	
G132			



CREATING A PROGRAM.
Programming of the axes.



CNC 8060
CNC 8065

(REF. 1402)

1.5 List of "G" functions.

The following tables show a list of "G" functions available at the CNC. The meaning of the "M", "D" and "V" fields of the table is the following:

- M· Modal function.
- D· Default function.
- V· Displayed function.

Next to each function, it indicates which chapter of this manual describes it; if no chapter is indicated, the function is described in another manual.

·M· Modal function.

A modal function, once programmed, remains active until an incompatible "G" code is programmed or an M02 or an M30 or until an EMERGENCY or a RESET is carried out or the CNC is turned off and back on.

Those cases indicated with "!", mean the function remains active even after an M02, M30 or a reset and after the CNC is powered off and back on.

·D· Default function.

It is the function that is activated by default; in other words, the function assumed by the CNC on power-up, after executing an M02 or M30 and after an EMERGENCY or a RESET.

Those cases indicated with "?" mean that the default quality of the function depends on the settings of the CNC machine parameters.

·V· Displayed function.

The function is displayed in automatic and jog modes next to the current machining conditions.

Function	M	D	V	Meaning	
G00	*	?	*	Rapid traverse.	8.1
G01	*	?	*	Linear interpolation.	8.2
G02	*		*	Clockwise circular (helical) interpolation.	8.3 / 8.6
G03	*		*	Counterclockwise circular (helical) interpolation.	8.3 / 8.6
G04			*	Dwell	10.1
G05	*	?	*	Controlled corner rounding (modal).	9.3
G06			*	Arc center in absolute coordinates (not modal).	8.3.5
G07	*	?	*	Square corner (modal).	9.1
G08			*	Arc tangent to previous path.	8.4
G09			*	Arc defined by three points.	8.5
G10	*	*		Mirror image cancellation.	9.8
G11	*		*	Mirror image on X.	9.8
G12	*		*	Mirror image on Y.	9.8
G13	*		*	Mirror image on Z.	9.8
G14	*		*	Mirror image in the programmed directions.	9.8
G17	*	?	*	Main plane X-Y, and longitudinal axis Z.	4.2
G18	*	?	*	Main plane Z-X, and longitudinal axis Y.	4.2
G19	*		*	Main plane Y-Z, and longitudinal axis X.	4.2
G20	*		*	Main plane by two directions and longitudinal axis.	4.3
G30			*	Polar origin preset.	5.7
G31			*	Temporary polar origin shift to the center of arc.	8.3.4
G33	*		*	Electronic threading with constant pitch.	8.7
G36			*	Automatic radius blend.	9.4
G37			*	Tangential entry.	9.6
G38			*	Tangential exit.	9.7
G39			*	Automatic chamfer blend.	9.5
G40	*	*		Cancellation of tool radius compensation.	11.1
G41	*		*	Left-hand tool radius compensation.	11.1
G42	*		*	Right-hand tool radius compensation.	11.1
G45				Turn tangential control on and off.	16.1
G50	*	?		Semi-rounded corner.	9.2
G53	*			Zero offset cancellation.	5.6
G54	!		*	Absolute zero offset 1.	5.5
G55	!		*	Absolute zero offset 2.	5.5
G56	!		*	Absolute zero offset 3.	5.5
G57	!		*	Absolute zero offset 4.	5.5

1.

CREATING A PROGRAM.
List of "G" functions.



CNC 8060
CNC 8065

(REF. 1402)

Function	M	D	V	Meaning	
G58	!		*	Absolute zero offset 5.	5.5
G59	!		*	Absolute zero offset 6.	5.5
G60			*	Square corner (not modal).	9.1
G61			*	Controlled corner rounding (not modal).	9.3
G63	*		*	Rigid tapping.	8.8
G70	*	?	*	Programming in inches.	3.1
G71	*	?		Programming in millimeters.	3.1
G72			*	Scaling factor.	9.10
G73	*		*	Coordinate system rotation.	9.9
G74			*	Home search	2.4
G90	*	?		Programming in absolute coordinates.	3.2
G91	*	?	*	Programming in incremental coordinates.	3.2
G92	!		*	Coordinate preset.	5.4
G93	*		*	Machining time in seconds.	6.2.1
G94	*	?		Feedrate in millimeters/minute (inches/minute).	6.2.1
G95	*	?	*	Feedrate in millimeters/revolution (inches/revolution).	6.2.1
G96	*		*	Constant surface speed.	7.2.2
G97	*	*		Constant turning speed.	7.2.2
G108	*	*		Feedrate blending at the beginning of the block.	6.2.2
G109			*	Feedrate blending at the end of the block.	6.2.2
G112	*			Changing of parameter range of an axis.	10.4
G130	*		*	Percentage of acceleration to be applied per axis or spindle.	6.2.5
G131	*		*	Percentage of acceleration to be applied to all the axes.	6.2.5
G132	*		*	Percentage of jerk to be applied per axis or spindle.	6.2.6
G133	*		*	Percentage of jerk to be applied to all the axes.	6.2.6
G134	*		*	Percentage of Feed-Forward to be applied.	6.2.7
G135	*		*	Percentage of AC-Forward to be applied.	6.2.8
G136	*		*	Circular transition between blocks.	11.1.2
G137	*	*		Linear transition between blocks.	11.1.2
G138	*		*	Direct activation/cancellation of tool compensation.	11.1.2
G139	*	*		Indirect activation/cancellation of tool compensation.	11.1.2
G145				Freeze tangential control.	16.2
G151	*	*	*	Programming in diameters.	3.3
G152	*			Programming in radius.	3.3
G157	*		*	Excluding axes in the zero offset.	5.5.3
G158	*		*	Incremental zero offset.	5.5.2
G159	!		*	Additional absolute zero offsets.	5.5
G170	*			Hirth axes OFF	10.3
G171	*	*		Hirth axes ON	10.3
G174	*			Set the machine coordinate.	5.2
G180			*	OEM subroutine execution.	12.5
G189					
G380			*	OEM subroutine execution.	12.5
G399					
G192	*		*	Turning speed limit.	7.2.1
G193	*		*	Interpolating the feedrate.	6.2.2
G196	*		*	Constant cutting point feedrate.	6.2.3
G197	*	*		Constant tool center feedrate.	6.2.3
G198				Setting of lower software travel limits	10.2
G199				Setting of upper software travel limits	10.2
G200				Exclusive manual intervention.	8.9.2
G201	*			Activation of additive manual intervention.	8.9.1
G202	*	*		Cancellation of additive manual intervention.	8.9.1
G261	*		*	Arc center in absolute coordinates (modal).	8.3.5
G262	*	*		Arc center referred to starting point.	8.3.5
G263	*		*	Arc radius programming.	8.3.2
G264	*		*	Cancel arc center correction.	8.3.6
G265	*	*		Activate arc center correction.	8.3.6
G266			*	Feedrate override at 100%	6.2.4
G500			*	Generic user subroutines	12.6
G599					

1.

CREATING A PROGRAM.

List of "G" functions.



CNC 8060
CNC 8065

(REF. 1402)

Probing.

Function	M	D	V	Meaning
G100			*	Probing until making contact.
G101	*			Include probe offset.
G102	*			Exclude probe offset.
G103			*	Probing until not making contact.
G104				Probe movement up to the programmed position.

Machining canned cycles. ·M· model (milling).

Function	M	D	V	Meaning
G80	*	*		Canned cycle cancellation.
G81	*		*	Drilling canned cycle.
G82	*		*	Drilling canned cycle with a variable peck.
G83	*		*	Deep-hole drilling canned cycle with constant peck.
G84	*		*	Tapping canned cycle.
G85	*		*	Reaming canned cycle.
G86	*		*	Boring canned cycle.
G87	*		*	Rectangular pocket canned cycle.
G88	*		*	Circular pocket canned cycle.
G98	*	*		Withdrawal to the starting plane.
G99	*		*	Withdrawal to the reference plane.
G160			*	Multiple machining in straight line.
G161			*	Multiple machining in rectangular pattern.
G162			*	Multiple machining in grid pattern.
G163			*	Multiple machining in a full circle.
G164			*	Multiple machining in arc pattern.
G165			*	Machining programmed with an arc-chord.
G210	*		*	Bore milling canned cycle.
G211	*		*	Inside thread milling cycle.
G212	*		*	Outside thread milling cycle.

1.

CREATING A PROGRAM.
List of "G" functions.



CNC 8060
CNC 8065

(REF. 1402)

Machining canned cycles. ·T· model (lathe).

Function	M	D	V	Meaning
G66			*	Pattern repeat canned cycle.
G68			*	Stock removal cycle along X axis.
G69			*	Stock removal canned cycle along Z axis.
G81			*	Turning canned cycle with straight sections.
G82			*	Facing canned cycle with straight sections.
G83			*	Drilling / tapping canned cycle.
G84			*	Turning canned cycle with arcs.
G85			*	Facing canned cycle with arcs.
G86			*	Longitudinal threading canned cycle.
G87			*	Face threading canned cycle.
G88			*	Grooving canned cycle along the X axis.
G89			*	Grooving canned cycle along the Z axis.
G160			*	Drilling / tapping canned cycle on the face of the part.
G161			*	Drilling / tapping canned cycle on the side of the part.
G162			*	Slot milling canned cycle along the side of the part.
G163			*	Slot milling canned cycle along the face of the part.

1.**CREATING A PROGRAM.**

List of "G" functions.

FAGOR **CNC 8060
CNC 8065**

(REF. 1402)

1.6 List of auxiliary (miscellaneous) M functions.

The following table shows a list of "M" functions available at the CNC. Next to each function, it indicates which chapter of this manual describes it; if no chapter is indicated, the function is described in another manual.

Function	Meaning	
M00	Program stop.	6.6.1
M01	Conditional program stop.	6.6.1
M02	End of program.	1.2.1
M03	Start the spindle clockwise.	7.3
M04	Start the spindle counterclockwise.	7.3
M05	Stop the spindle.	7.3
M06	Tool change.	6.6.1
M17	End of a global or local subroutine.	12.2
M19	Spindle orientation.	7.5
M29	End of a global or local subroutine.	12.2
M30	End of program.	1.2.1
M41	Selects gear ·1·.	7.4
M42	Selects gear ·2·.	7.4
M43	Selects gear ·3·.	7.4
M44	Selects gear ·4·.	7.4

1.

CREATING A PROGRAM.

List of auxiliary (miscellaneous) M functions.



CNC 8060
CNC 8065

(REF. 1402)

1.7 List of statements and instructions.

The following tables show a list of statements and instructions functions available at the CNC. Next to each of them, it indicates which chapter of this manual describes it; if no chapter is indicated, the function is described in another manual.

Instruction	Meaning	
\$GOTO	Block skip.	21.2.1
\$IF \$ELSEIF \$ELSE \$ENDIF	Conditional execution.	21.2.2
\$SWITCH \$CASE \$BREAK \$DEFAULT \$ENDSWITCH	Conditional execution.	21.2.3
\$FOR \$BREAK \$CONTINUE \$ENDFOR	Block repetition.	21.2.4
\$WHILE \$BREAK \$CONTINUE \$ENDWHILE	Conditional block repetition.	21.2.5
\$DO \$BREAK \$CONTINUE \$ENDDO	Conditional block repetition.	21.2.6

Statement	Meaning	
L	Call to a global subroutine.	12.3.2
LL	Call to a local subroutine.	12.3.1
#ABORT	Abort the execution of the program and resume it in another block or program.	13.3
#ACS	Fixture coordinate system.	17.3
#ANGAX OFF	Turn angular transformation off.	15.1
#ANGAX ON	Turn angular transformation on.	15.1
#ANGAX SUSP	Freeze angular transformation.	15.2
#ASPLINE ENDTANG	Akima splines. Type of final tangent.	21.1.13
#ASPLINE MODE	Akima splines. Selection of tangent type.	21.1.13
#ASPLINE STARTTANG	Akima splines. Type of starting tangent.	21.1.13
#AXIS	Axis upon which the additive manual intervention is applied.	8.9
#CALL	Call to a global or local subroutine.	12.3.3
#CALL AX	Add a new axis to the configuration.	21.1.8
#CALL SP	Add a spindle to the configuration.	21.1.9
#CAM ON	Activate the electronic cam (real coordinates).	21.1.20
#CAM OFF	Cancel the electronic cam.	21.1.20
#CAX	"C" axis. Activating the spindle as ·C· axis.	14.1
#CD OFF	Cancel collision detection.	21.1.12
#CD ON	Activating collision detection.	21.1.12
#CLEAR	Channels. It clears the synchronism marks of the channel.	21.1.18
#CONTJOG	Manual intervention. Feedrate in continuous jog.	8.9.3
#COMMENT BEGIN	Beginning of comment.	1.8
#COMMENT END	End of comment.	1.8
#CS	Machining coordinate system.	17.3
#CYL	"C" axis. Machining of the turning side of the part.	14.3
#DEF	Macros. Definition of macros.	21.1.16
#DELETE	It initializes the global user variables.	1.9
#DFHOLD	Disable the feed-hold signal.	21.1.5
#DGWZ	It defines the graphic display area.	21.1.4
#DSBLK	End of the single-block treatment.	21.1.5
#DSTOP	Disable the cycle stop signal.	21.1.5
#EFHOLD	Disable the feed-hold signal.	21.1.5
#ERROR	Display an error on the screen.	21.1.1
#ESBLK	Beginning of the single-block treatment.	21.1.5
#ESTOP	Enable the cycle stop signal.	21.1.5
#EXBLK	It executes a block in the indicated channel.	13.2
#EXEC	It executes a program in the indicated channel.	13.1
#FACE	"C" axis. Machining of the face of the part.	14.2
#FLUSH	Interrupt block preparation.	21.1.21

1.

CREATING A PROGRAM.

List of statements and instructions.


FAGOR
CNC 8060
CNC 8065

(REF. 1402)

1.

CREATING A PROGRAM.

List of statements and instructions.

Statement	Meaning	
#FOLLOW OFF	Independent axis. End the synchronization movement.	21.1.19
#FOLLOW ON	Independent axis. Begin the synchronization movement (real coordinates).	21.1.19
#FREE AX	Free an axis from the configuration.	21.1.8
#FREE SP	Free a spindle from the configuration.	21.1.9
#HSC OFF	It cancels the HSC mode.	18.4
#HSC ON	HSC mode. Optimizing the contouring error.	18.2
#HSC ON [FAST]	HSC mode. Optimizing the machining speed.	18.3
#INCJOG	Manual intervention. Feedrate in incremental jog.	8.9.3
#INIT MACROTAB	Macros. Initialize the table of macros.	21.1.16
#KIN ID	Selection of the kinematics.	17.2
#LINK	Activate the electronic coupling (slaving) of axes.	21.1.6
#MASTER	Selecting the master spindle of the channel.	7.1.1
#MCALL	Modal call to a local or global subroutine initializing parameters.	12.3.5
#MCS	Program a movement referred to machine zero.	5.1
#MCS OFF	Cancel the machine coordinate system.	5.1
#MCS ON	Activate the machine coordinate system.	5.1
#MDOFF	Turning the subroutine into non-modal.	12.4
#MEET	Channels. It activates the mark in the indicated channel.	21.1.18
#MOVE	Independent axis. Positioning move.	21.1.19
#MPG	Manual intervention. Resolution of the handwheels.	8.9.3
#MSG	Display a message on the screen.	21.1.3
#PARK	Park an axis.	21.1.7
#PATH	Define the location of the global subroutines.	12.4
#PCALL	Call to a global or local subroutine initializing parameters.	12.3.4
#POLY	Polynomial interpolation.	21.1.14
PWMOUT ON	Activate the PWM.	19.2.1
PWMOUT OFF	Cancel the PWM.	19.2.2
#RENAME AX	Rename the axes.	21.1.8
#RENAME SP	Rename the spindles.	21.1.9
#REPOS	Repositioning axes and spindles from an OEM subroutine.	12.8.1
#RET	End of a global or local subroutine.	12.2
#RETDSBLK	Execute subroutine as a single block.	12.3.7
#ROUNDPAR	Type of corner rounding.	9.3.1
#ROTATEMZ	Positioning a turret magazine.	6.4
#RPT	Block repetition.	21.1.17
#RTCP	RTCP transformation.	17.6
#SCALE	Scaling factor.	9.10
#SERVO ON	Activates the closed loop mode.	21.1.11
#SERVO OFF	Activates the open loop mode.	21.1.11
#SET AX	Set axis configuration.	21.1.8
#SET OFFSET	Manual intervention. Moving limits.	8.9.3
#SET SP	Set spindle configuration.	21.1.9
#SIGNAL	Channels. It activates the mark in its own channel.	21.1.18
#SLOPE	Acceleration control.	21.1.15
#SPLINE OFF	Akima splines. It cancels spline adaptation.	21.1.13
#SPLINE ON	Akima splines. It activates spline adaptation.	21.1.13
#SWTOUT ON	Activate synchronized switching.	19.1.1
#SWTOUT OFF	Cancel synchronized switching.	19.1.2
#SYNC	Spindle synchronization. Synchronization of the real coordinate.	21.1.10
#SYNC POS	Manual intervention. Coordinate synchronization.	8.9.3
#TANGCTRL OFF	Cancel tangential control.	16.1
#TANGCTRL ON	Activate tangential control.	16.1
#TANGCTRL SUSP	Freeze tangential control.	16.2
#TANGFEED RMIN	Minimum contouring radius for applying constant feedrate	6.2.3
#TCAM ON	Activate the electronic cam (theoretical coordinates).	21.1.20
#TFOLLOW ON	Independent axis. Begin the synchronization movement (theoretical coordinates).	21.1.19
#TIME	Dwell	10.1
#TLC	Tool length compensation.	17.7
#TOOL AX	Longitudinal tool axis selection.	4.4
#TOOL ORI	Tool perpendicular to the plane.	17.5
#TSYNC	Spindle synchronization. Synchronization of the theoretical coordinate.	21.1.10
#UNLINK	Cancel the electronic coupling (slaving) of axes.	21.1.6
#UNPARK	Unpark an axis	21.1.7
#UNSYNC	Spindle synchronization. Decouple the spindles.	21.1.10
#VIRTAX ON	Activate the virtual tool axis.	20.1
#VIRTAX ON	Cancel the virtual tool axis.	20.2



CNC 8060
CNC 8065

(REF. 1402)

Statement	Meaning	
#WAIT	Channels. It waits for a mark to be activated in the indicated channel.	21.1.18
#WAIT FOR	Wait for an event.	21.1.21
#WARNING	Display a warning on the screen.	21.1.2
#WARNINGSTOP	Display a warning on the screen and interrupt the program.	21.1.2

Probing.

#SELECT PROBE	Probe selection.

Probing canned cycles. ·M· model (milling).

#PROBE 1	Tool calibration (dimensions and wear).
#PROBE 2	Probe calibration
#PROBE 3	Surfacing measuring.
#PROBE 4	Outside corner measuring.
#PROBE 5	Inside corner measuring.
#PROBE 6	Measuring the angle with the abscissa axis.
#PROBE 7	Outside corner and angle measuring.
#PROBE 8	Hole measuring.
#PROBE 9	Circular boss measuring.
#PROBE 10	Rectangular part centering.
#PROBE 11	Circular part centering.
#PROBE 12	Tabletop probe calibration

Probing canned cycles. ·T· model (lathe).

#PROBE 1	Tool calibration.
#PROBE 2	Tabletop probe calibration
#PROBE 3	Part measuring along the ordinate axis.
#PROBE 4	Part measuring along the abscissa axis.

1.

CREATING A PROGRAM.

List of statements and instructions.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

1.8 Comment programming.

Any comment may be associated with the blocks. When executing the program, the CNC ignores this information.

The CNC offers various methods to include comments in the program.

Programming comments in parenthesis "(" and ")".

The comment must go in parenthesis "(" and ")". Comments programmed this way need not go at the end of the block; it may go in the middle and there may be more than one comment in the same block.

```
N10 G90 X23.45 F100 (comment) S200 M3 (comment)
```

Programming comments with the ";" character.

The information to be considered as comment must go after the ";" character. The comment may be programmed alone in the block or may be added at the end of a block.

```
N10 G90 X23.45 T1; comment
```

Programming comments with the #COMMENT instruction.

The instructions #COMMENT BEGIN and #COMMENT END indicate the beginning and end of a comment. The blocks programmed between them are considered by the CNC as a single comment and are ignored when executing the program.

```
#COMMENT BEGIN
P1 : Machining width.
P2 : Machining length.
P3 : Machining depth
#COMMENT END
```

1.

CREATING A PROGRAM.
Comment programming.



CNC 8060
CNC 8065

(REF. 1402)

1.9 Variables and constants.

Constants.

They are fixed values that cannot be modified by program; constants are numbers in decimal, binary and hexadecimal system and read-only tables and variables because their value cannot be changed within a program.

Hexadecimal values are represented preceded by the \$ symbol.

Hexadecimal	Decimal	Binary
\$4A	74	0100 1010

Variables.

The CNC has a number of internal variables that may be accessed from the user program, from the PLC or from the interface.

User variables.

The user can create his own variables. These are read-write variables and are evaluated during block preparation.

The mnemonics of the variables are the following. Replace the suffix *name* with the name of the variable.

- V.P.*name* - Local user variable
- V.S.*name* - Global user variable

```
V.P.mylocalvar
V.S.myglobalvar
```

Local user variables may only be accessed from the program or subroutine where they have been programmed. Global user variables will be shared by the program and the subroutines of the channel.

Global user variables maintain their value after a reset.

Initialize the user variables.

Variables are deleted when the CNC is turned off and they can also be deleted from the part-program using the #DELETE instruction. This statement may be used to initialize the global and local variables stored in the CNC, even if they are not being used by the program. The #DELETE instruction must always go with some variable; it must not be programmed alone in the block.

```
#DELETE V.P.localvar1
#DELETE V.S.globalvar1 V.S.globalvar2
```



1.10 Arithmetic parameters.

Arithmetic parameters are general purpose variables that the user may utilize to create his/her own programs. The CNC has global, local and common arithmetic parameters. The range of available parameters of each type is defined in the machine parameters.

Arithmetic parameters are programmed with the "P" code followed by the parameter number. There are some tables for consulting the value of these parameters; refer to the operating manual to learn how to handle these tables.

The user may use the arithmetic parameters when editing its own programs. During execution, the CNC will replace these parameters with the values assigned to them at the time.

```
P0=0 P1=1 P2=20 P3=50 P4=3
P10=1500 P100=800 P101=30
...
GP0 XP0 YP0 SP10 MP4      ==>   G0 X0 Y0 S1500 M3
GP1 XP2 YP3 FP100         ==>   G1 X20 Y50 F800
MP101                     ==>   M30
```

Local arithmetic parameters.

Local parameters may only be accessed from the program or subroutine where they have been programmed. There are seven groups of local parameters in each channel.

The maximum range of local parameters is P0 to P99, the typical range being P0 to P25.

When the parameters are used in the block calling a subroutine may also be referred to by the letters A-Z (except Ñ and Ç) so "A" is the same as P0 and "Z" the same as P25.

Global arithmetic parameters.

Global parameters may be accessed from any program and subroutine called upon from the program. The value of these parameters is shared by the program and the subroutines. There is a group of global parameters in each channel.

The maximum range of global parameters is P100 to P9999, the typical range being P100 to P299.

Common arithmetic parameters.

The common parameters may be accessed from any channel. The value of these parameters is shared by all the channels. Reading and writing these parameters interrupts block preparation.

The maximum range of common parameters is P10000 to P19999, the typical range being P10000 to P10999.

Programming the arithmetic parameters.

In blocks programmed in ISO code, it is possible to define the values of all the fields "N", "G", "F", "S", "T", "D", "M", "H", "NR" and axis coordinates using parameters. Using indirect addressing, it is also possible to define the number of a parameter with another parameter; "P[P1]", "P[P2+3]".

In blocks having statements, the values of any expression may be defined with parameters.

1.

CREATING A PROGRAM.
Arithmetic parameters.



CNC 8060
CNC 8065

(REF. 1402)

1.11 Arithmetic and logic operators and functions.

An operator is a symbol that indicates the mathematical or logic operations to carry out. The CNC offers the following types of operators.

Arithmetic operators.

To perform arithmetic operations.

+	Add	$P1 = 3+4$	$P1=7$
-	Subtract Change sign	$P2 = 5-2$ $P2 = -(3+4)$	$P2=3$ $P2 = -7$
*	Multiply	$P3 = 2*3$	$P3=6$
/	Divide	$P4 = 9/2$	$P4=4.5$
MOD	Module or remainder of a division	$P5 = 5 \text{ MOD } 2$	$P5=1$
**	Exponent	$P6 = 2**3$	$P6=8$

In the operation, when using the parameter or variable storing the result, the add, subtract, multiply and divide operators may be used as follows:

+=	Compounded addition	$P1 += 3$	$P1=P1+3$
-=	Compounded subtraction	$P2 -= 5$	$P2=P2-5$
*=	Compounded multiplication	$P3 *= 2$	$P3=P3*2$
/=	Compounded division	$P4 /= 9$	$P4=P4/9$

Relational operators.

Used for doing comparisons.

==	Equal to	$P1 == 4$
!=	Different from, other than	$P2 != 5$
>=	Greater than or equal to	$P3 >= 10$
<=	Smaller than or equal to	$P4 <= 7$
>	Greater than	$P5 > 5$
<	Smaller than	$P6 < 5$

Binary operators.

Used for doing binary comparisons between constants and/or arithmetic expressions.

&	Binary AND	$P1 = P11 \& P12$
	Binary OR	$P2 = P21 P22$
^	Exclusive OR (XOR)	$P3 = P31 \wedge P32$
INV[...]	Inverse	$P4 = \text{INV}[P41]$

If the constant or the result of the arithmetic expression is a decimal number, the decimal portion will be ignored.

Logic operators.

Used for doing logic comparisons between conditions.

*	Logic AND	$\$IF [P11 == 1] * [P12 >=5]$
+	Logic OR	$\$IF [P21 != 0] + [P22 == 8]$

Each condition should go between brackets, otherwise, an undesired comparison may be done due to the priority between operators.



CREATING A PROGRAM.

Arithmetic and logic operators and functions.



CNC 8060
CNC 8065

(REF. 1402)

Boolean constants.

TRUE	True	\$IF V.S.VAR == TRUE
FALSE	Not true	\$IF V.S.VAR == FALSE

Trigonometric functions.

SIN[...]	Sine	P1 = SIN[30]	P1 = 0.5
COS[...]	Cosine	P2 = COS[30]	P2 = 0.866
TAN[...]	Tangent	P3 = TAN[30]	P3 = 0.5773
ASIN[...]	Arc-sine	P4 = ASIN[1]	P4 = 90
ACOS[...]	Arc-cosine	P5 = ACOS[1]	P5 = 0
ATAN[...]	Arc-tangent	P6 = ATAN[1]	P6 = 45
ARG[...]	Arctangent y/x	P7=ARG[-1,1]	P7=225

In these type of functions the following must be borne in mind:

- In the "TAN" function, the argument cannot take the values ...-90°, 90°, 270°...
- In the "ASIN" and "ACOS" functions, the argument must always be within ±1.
- There are two functions to calculate the arctangent:

"ATAN" It returns the result between ±90°.

"ARG" It returns the result between 0° and 360°.

Mathematical functions.

ABS[...]	Absolute value	P1 = ABS[-10]	P1 = 10
SQR[...]	Square function	P2 = SQR[4]	P2 = 16
SQRT[...]	Square root	P3 = SQRT[16]	P3 = 4
LOG[...]	Decimal logarithm	P4 = LOG[100]	P4 = 2
LN[...]	Neperiam logarithm	P5 = LN[100]	P5 = 4.6051
EXP[...]	"e" function	P6 = EXP[1]	P6 = 2.7182
DEXP[...]	Decimal exponent	P6 = DEXP[2]	P7 = 100

In these type of functions the following must be borne in mind:

- In the "LN" and "LOG" functions, the argument must be grater than zero.
- In the "SQRT" function, the argument must be positive.

Other functions.

INT[...]	Returns the integer	P1 = INT[4.92]	P1 = 4
FRACT[...]	Returns decimal portion	P2 = FRACT[1.56]	P2 = 0.56
ROUND[...]	Rounds up or down to the nearest integer	P3 = ROUND[3.12] P4 = ROUND[4.89]	P3 = 3 P4 = 5
FUP[...]	Returns the integer plus one. (If the number is an integer, it returns it)	P5 = FUP[3.12] P6 = FUP[9]	P5 = 4 P6 = 9
EXIST[...]	It checks whether the selected variable or parameter exists or not	\$IF EXIST[P1] \$IF EXIST[P3] == FALSE	

In the "EXIST" function, programming "\$IF EXIST[P1] == TRUE" is the same as programming "\$IF EXIST[P1]".

1.

CREATING A PROGRAM.

Arithmetic and logic operators and functions.



CNC 8060
CNC 8065

(REF. 1402)

1.12 Arithmetic and logic expressions.

An expression is any valid combination of operators, constants, parameters and variables. Expressions may be used to program the numerical portion of any function, statement, etc.

The priorities of the operators and the way they can be associated determine how these expressions are calculated:

Priority from highest to lowest	They are associated
Functions, - (change sign)	from right to left.
** (exponent), MOD (remainder)	from left to right.
* (multiplication, logic AND), / (division)	from left to right.
+ (suma, OR lógico), - (resta)	from left to right.
Relational operators	from left to right.
& (AND), ^ (XOR)	from left to right.
(OR)	from left to right.

Brackets should be used in order to clarify the order in which the expression is to be evaluated. Using redundant or additional brackets will neither cause errors nor slow down the execution.

$$P3 = P4/P5 - P6 * P7 - P8/P9$$

$$P3 = [P4/P5] - [P6 * P7] - [P8/P9]$$

Arithmetic expressions.

Their result is a numerical value. They consist of a combination of arithmetic and binary operators with constants, parameters and variables.

This type of expressions may also be used to assign values to parameters and variables:

```
P100 = P9   P101 = P[P7]   P102 = P[P8 + SIN[P8*20]]
P103 = V.G.TOOL
V.G.FIXT[1].X=20   V.G.FIXT[1].Y=40   V.G.FIXT[1].Z=35
```

Relational expressions.

Their result is a TRUE or a FALSE. They combine relational and logic operators with arithmetic expressions, constants, parameters and variables.

```
... [P8==12.6] ...
```

It compares if the value of P8 is equal to 12.6.

```
... ABS[SIN[P4]] > 0.8 ...
```

It compares if the absolute value of the sine of P4 is greater than 0.8.

```
... [[P8<=12] + [ABS[SIN[P4]] >=0.8] * [V.G.TOOL==1]] ...
```

1.

CREATING A PROGRAM.
Arithmetic and logic expressions.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

1.

CREATING A PROGRAM.

Arithmetic and logic expressions.



CNC 8060
CNC 8065

(REF. 1402)

2.1 Axis nomenclature

With this CNC, the manufacturer may select up to 28 axes (that must be properly defined as linear, rotary, etc. by setting machine parameters), without no limitation as how to program them and they may all be interpolated at the same time.

The DIN 66217 standard denomination for the axes is:

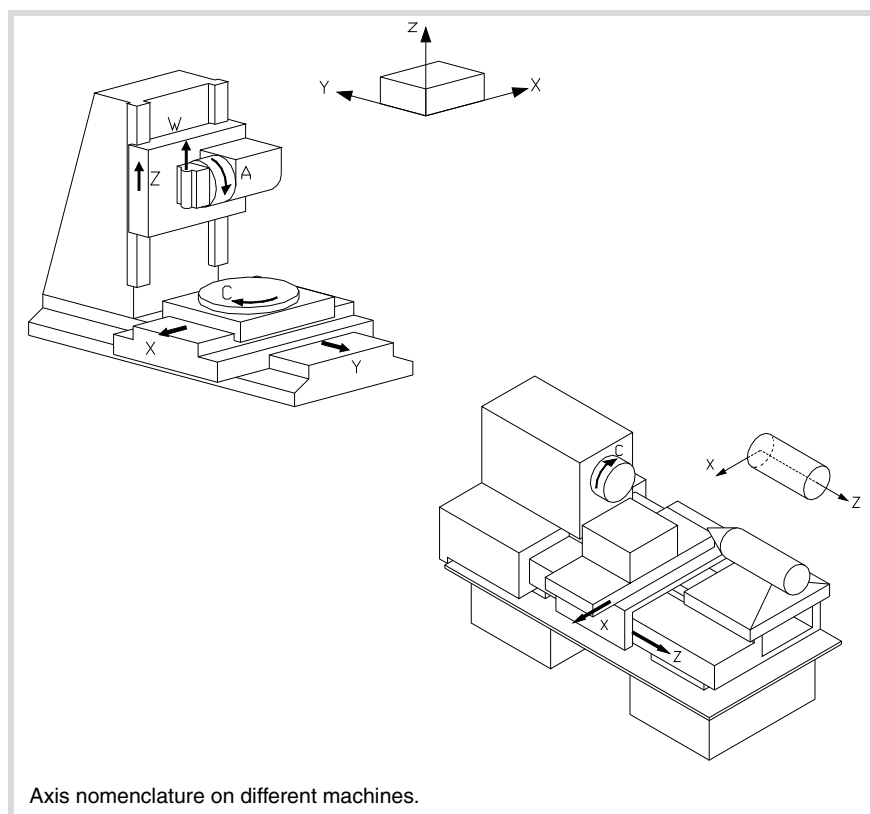
X-Y-Z Main axes of the machine. The X-Y axes form the main work plane whereas the Z axis is parallel to the main axis of the machine and perpendicular to the XY plane.

U-V-W Auxiliary axes, parallel to X-Y-Z respectively.

A-B-C Rotary axes, on X-Y-Z respectively.

However, the machine manufacturer may call the axes differently.

As an option, the name of the axes may be followed by a number between 1 and 9 (X1, X3, Y5, A8...).



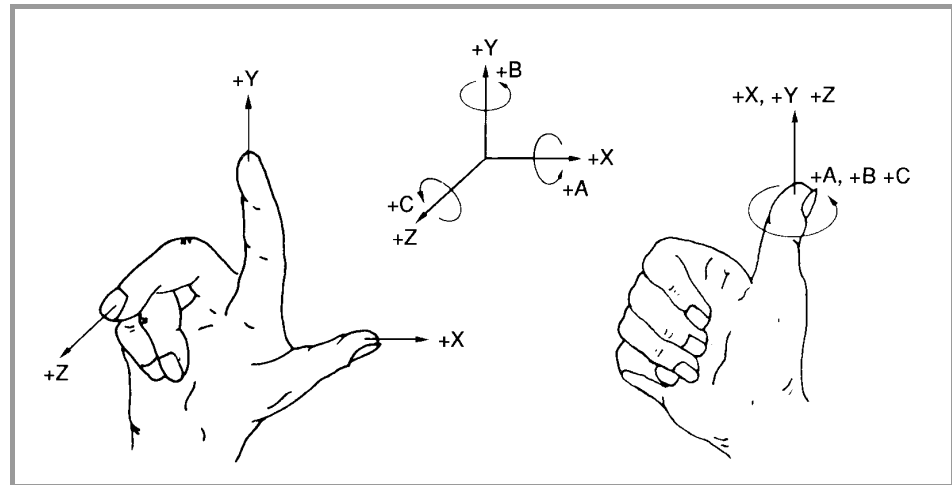
CNC 8060
CNC 8065

(REF. 1402)

Right-hand rule

The direction of the X-Y-Z axes can easily be remembered using the right-hand rule (see the drawing below).

On rotary axes, the positive turning direction is determined by the direction pointed by your fingers when holding the rotary axis with your hand while your thumb points in the positive direction of the linear axis.

**2.****MACHINE OVERVIEW**
Axis nomenclature

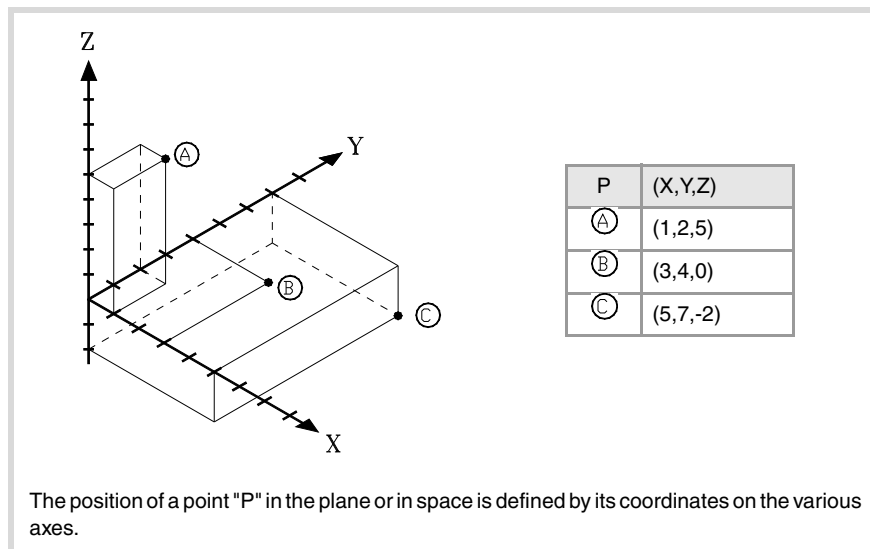
CNC 8060
CNC 8065

(REF. 1402)

2.2 Coordinate system

Since one of the CNC's purposes is to control the movement and positioning of the axes, a coordinate system is required that permits defining the position of the various target (destination) points in the plane (2D) or in space (3D).

The main coordinate system is formed by the X-Y-Z axes. These axes are perpendicular to each other and they meet at the origin point used as reference for the various points.



Other types of axes such as auxiliary and rotary axes may also be part of the coordinate system.

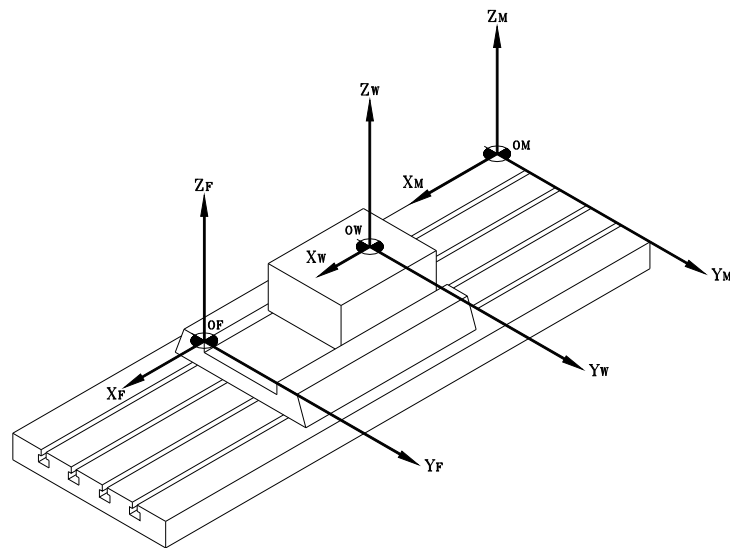
2.

2.3 Reference systems

A machine may use the following reference systems.

- Machine reference system.
It is the coordinate system of the machine and it is set by the manufacturer of the machine.
- Fixture reference system.
It establishes a coordinate system associated with the fixtures being used. It is activated by program and may be set by the operator in any position of the machine.
When the machine has several fixtures, each one may have its own reference system associated with it.
- Part reference system (datum point).
It establishes a coordinate system associated with the part being machined. It is activated by program and may be set by the operator anywhere on the part.

Example of the various coordinate systems on a milling machines.



$X_M Y_M Z_M$ Machine reference system.
 $X_F Y_F Z_F$ Fixture reference system.
 $X_W Y_W Z_W$ Part reference system (datum point).

2.

2.3.1 Origins of the reference systems

The position of the different reference systems is determined by their respective origin points.

O_M Machine zero

It is the origin point of the machine reference system, set by the machine manufacturer.

O_F Fixture zero

It is the origin point of the fixture reference system being used. Its position is defined by the operator by using the "fixture offset" and is referred to machine zero.

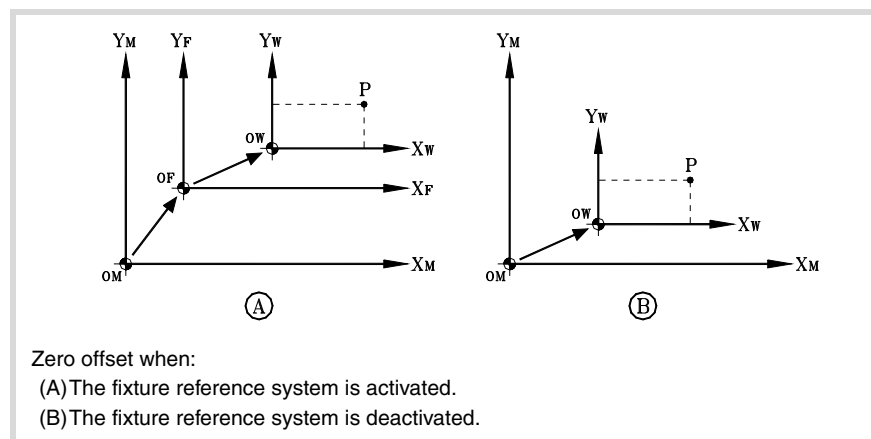
The "fixture offset" may be set by program or from the CNC's front panel, as described in the Operating Manual.

O_W Part zero

It is the origin point of the reference system of the part (workpiece). Its position is set by the operator using the "zero offset" and is referred:

- To the fixture offset, if the fixture reference system is active. When changing the fixture reference system, the CNC updates the part zero position by referring to the new fixture zero point.
- To the machine zero point (home), if the fixture reference system is NOT active. When activating the fixture reference system, the CNC updates the part zero position by referring it to the fixture zero point.

The "zero offset" may be set from the program or from the CNC front panel as described in the Operating Manual.



2.

2.4 Home search

2.4.1 Definition of "Home search"

It is the operation used to synchronize the system. This operation must be carried out when the CNC loses the position of the origin point (e.g. by turning the machine off).

In order to perform the "Home search", the machine manufacturer has set particular points of the machine; the machine zero and the machine reference point.

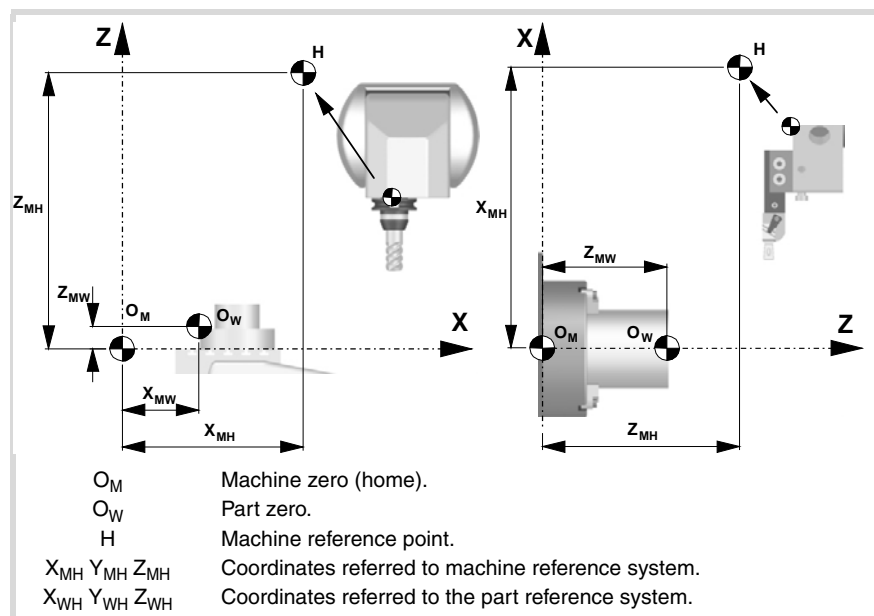
- Machine zero (home).

It is the origin point of the machine reference system.

- Machine reference point.

It is the physical point where the system is synchronized (except when the machine uses I_0 distance-coded reference marks or absolute feedback). It may be located anywhere on the machine.

When "searching home", the axes move to the machine reference point and the CNC assumes the coordinate values assigned to that point by the machine manufacturer, referred to machine zero. When using I_0 distance-coded reference marks or absolute feedback, the axes will only move the distance necessary to verify their position.



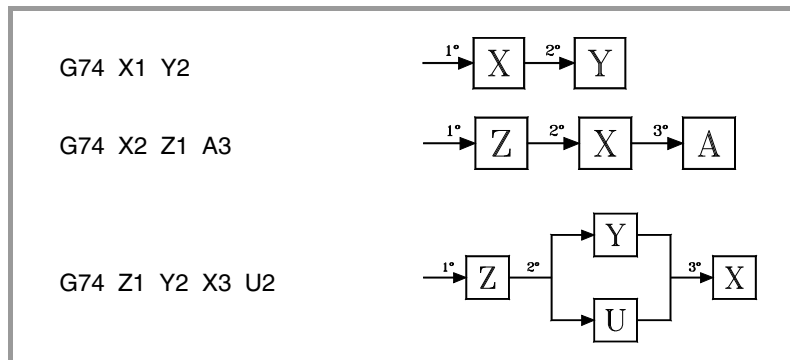
When programming a "Home search", neither the fixture offsets nor the zero offsets are canceled; therefore, the coordinates are displayed in the active reference system.

On the other hand, if "Home search" is carried out one axis at a time in JOG mode (not in MDI), the active offsets are canceled and the coordinates being displayed are referred to machine zero.

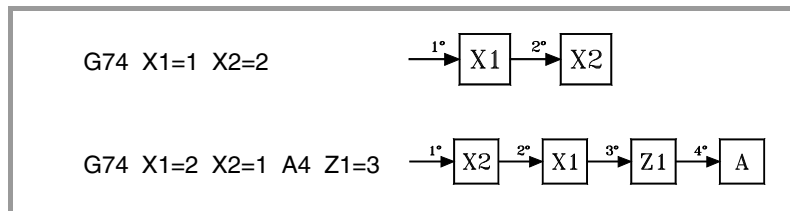
2.4.2 "Home search" programming

When programming a "Home search", the axes are homed sequentially in the order set by the operator. All the axes need not be included in the "Home search", only those being homed.

The "Home search" is programmed using the G74 function followed by the axes to be homed and the number indicating their homing order. If the same order number is assigned to several axes, those axes start homing at the same time and the CNC waits for all of them to end before homing the next one.



When having numbered axes, they may be defined together with the other ones by assigning them the order number as follows.



Spindle home search

The spindle home search is always carried out together with the first axis regardless of the order in which it has been defined.

Home search and loop status.

Axes usually work in closed loop, although rotary axes can also work in open loop so they can be controlled as if they were spindles.

The home search is carried out with the axes and spindles controlled in position; i.e. in closed position loop. The CNC will close the position loop automatically on all axes and spindles for which a home search has been programmed using function G74.

Using an associated subroutine

If the machine manufacturer has associated a home-search subroutine to the G74 function, this function may be programmed alone in the block and the CNC will automatically execute the associated subroutine [G.M.P. "REFPSUB (G74)"].

When using a subroutine, the "Home search" is carried out exactly as described earlier.

2.

MACHINE OVERVIEW
Home search

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

2.

MACHINE OVERVIEW

Home search



CNC 8060
CNC 8065

(REF. 1402)

3.1 Programming in millimeters (G71) or in inches (G70)

The displacements and feedrates of the axes may be defined in millimeters or in inches. The unit system may be selected by program using the following functions:

G70	Programming in inches.
G71	Programming in millimeters.

Both functions may be programmed anywhere in the program; they do not have to go alone in the block.

Operation

After executing one of these functions, the CNC assumes that unit system for the following blocks. If none of these functions is programmed, the CNC uses the unit system set by machine manufacturer [G.M.P. "INCHES"].

When changing the unit system, the CNC converts the currently active feedrate into the new unit system.

```
...  
G01 G71 X100 Y100 F508      (Programming in millimeters.)  
                             (Feedrate: 508 mm/minute)  
...  
G70                          (It changes the units.)  
                             (Feedrate: 20 inches/minute)  
...
```

Properties of the functions

Functions G70 and G71 are modal and incompatible with each other.

On power-up, after an M02 or M30 and after an EMERGENCY or a RESET, the CNC assumes function G70 or G71 as set by the machine manufacturer [G.M.P. "INCHES"].

3.2 Absolute (G90) or incremental (G91) coordinates

The coordinates of the various points may be defined in absolute coordinates (referred to the active origin point) or incremental coordinates (referred to the current position). The type of coordinates may be selected by program using the following functions:

- G90 Programming in absolute coordinates.
- G91 Programming in incremental coordinates.

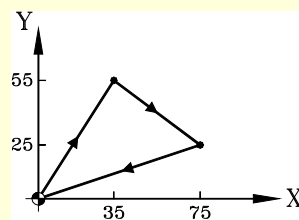
Both functions may be programmed anywhere in the program; they do not have to go alone in the block.

Operation

After executing one of these functions, the CNC assumes that programming mode for the following blocks. If none of these functions is programmed, the CNC uses the work mode selected by machine manufacturer [G.M.P. "ISYSTEM"].

Depending on the active work mode (G90/G91), the coordinates of the points are defined as follows:

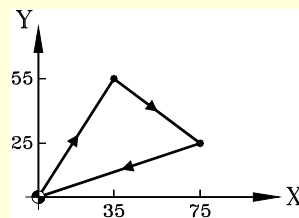
- When programming in absolute coordinates (G90), the coordinates of the point are referred to the current origin of the coordinate system, usually the part zero.



```
N10 G00 G71 G90 X0 Y0
N20 G01 X35 Y55 F450
N30 X75 Y25
N40 X0 Y0
N50 M30
```

Programming in absolute coordinates.

- When programming in incremental coordinates (G91), the coordinates of the point are referred to the current tool position. The preceding sign indicates the direction of the movement.



```
N10 G00 G71 G90 X0 Y0
N20 G01 G91 X35 Y55 F450
N30 X40 Y-30
N40 X-75 Y-25
N50 M30
```

Programming in incremental coordinates.

Properties of the function

Functions G90 and G91 are modal and incompatible with each other.

On power-up, after an M02 or M30 and after an EMERGENCY or a RESET, the CNC assumes function G90 or G91 as set by the machine manufacturer [G.M.P. "ISYSTEM"].

3.

COORDINATE SYSTEM
Absolute (G90) or incremental (G91) coordinates



CNC 8060
CNC 8065

(REF. 1402)

3.2.1 Rotary axes.

The CNC admits different ways to configure a rotary axis depending on how it is going to move. Hence, the CNC can have rotary axes with travel limits, for example between 0° and 180° (linearlike rotary axis); axes that always move in the same direction (unidirectional rotary axis); axes that choose the shortest path (positioning-only rotary axis).

All rotary axes must be programmed in degrees; therefore, they will not be affected by the mm-inch conversion. The number of revolutions the axis will turn when programming a distance greater than the module depends on the type of axis. The limits to display the position values (coordinates) also depend on the type of axis.

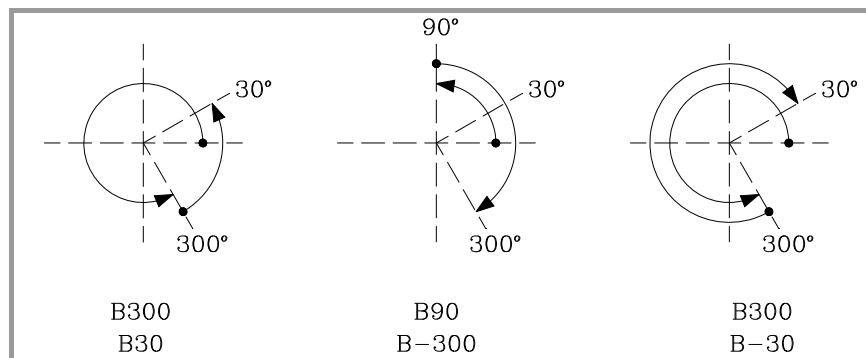
Linearlike rotary axis.

The axis behaves like a linear axis, but it is programmed in degrees. The CNC displays the position values between the travel limits.

Normal rotary axis.

This type of rotary axis can turn in both directions. The CNC displays the position values between the limits of the module.

G90 movements.	G91 movements.
The sign of the position value indicates the moving direction; the absolute position value indicates the target position.	Normal incremental movement. The sign of the position value indicates the moving direction; the absolute position value indicates the position increment.
Even if the programmed distance is greater than the module, the axis never turns more than one revolution.	If the programmed distance is greater than the module, the axis turns more than one revolution.



Unidirectional rotary axis.

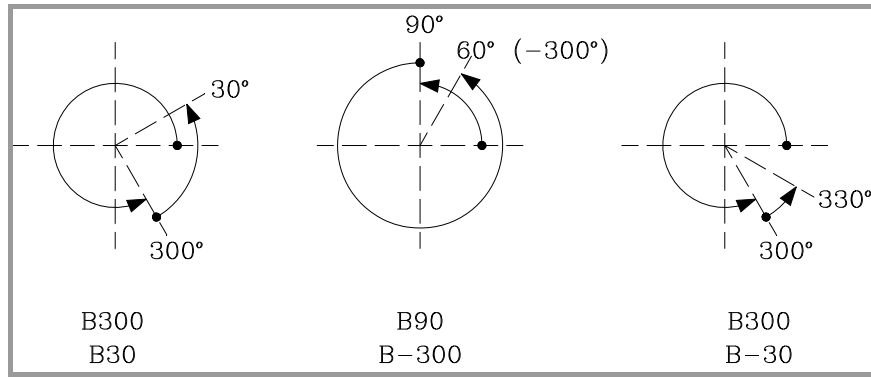
This type of rotary axis only moves in one direction, the one that has been preset for it. The CNC displays the position values between the limits of the module.

G90 movements.	G91 movements.
The axis moves in the preset direction up to the programmed position.	The axis only admits movements in the preset direction. The sign of the position value indicates the moving direction; the absolute position value indicates the position increment.
Even if the programmed distance is greater than the module, the axis never turns more than one revolution.	If the programmed distance is greater than the module, the axis turns more than one revolution.

3.

COORDINATE SYSTEM

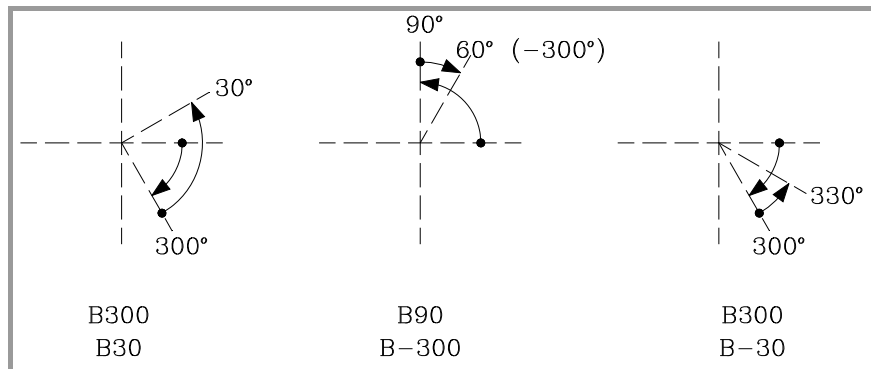
Absolute (G90) or incremental (G91) coordinates



Positioning-only rotary axis.

This type of rotary axis can move in both directions; but in absolute movements, it only moves via the shortest path. The CNC displays the position values between the limits of the module.

G90 movements.	G91 movements.
The axis moves via the shortest path up to the programmed position.	Normal incremental movement. The sign of the position value indicates the moving direction; the absolute position value indicates the position increment.
Even if the programmed distance is greater than the module, the axis never turns more than one revolution.	If the programmed distance is greater than the module, the axis turns more than one revolution.



CNC 8060
CNC 8065

(REF. 1402)

3.3 Programming in radius (G152) or in diameters (G151)



The following functions are oriented to lathe type machines. Programming in diameters is only available on the axes allowed by the machine manufacturer (DIAMPROG=YES).

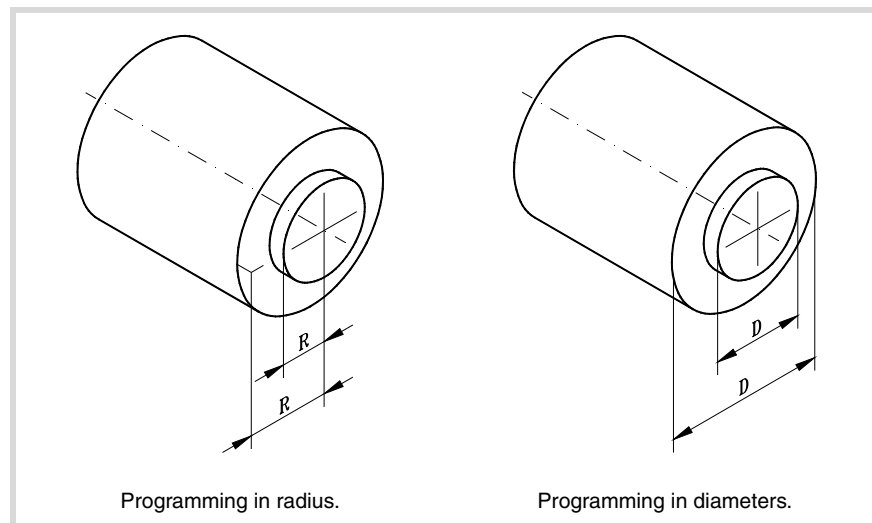
Programming in radius or diameters may be selected by program with these functions:

- | | |
|------|---------------------------|
| G151 | Programming in diameters. |
| G152 | Programming in radius. |

These functions may be programmed anywhere in the program and they don't have to go alone in the block.

Operation

After executing one of these functions, the CNC assumes that programming mode for the following blocks.



When switching programming modes, the CNC changes the way it displays the coordinates of the corresponding axes.

Properties of the function

Functions G151 and G152 are modal and incompatible with each other.

On power-up, after executing an M02 or M30, and after an EMERGENCY or RESET, the CNC assumes function G151 if machine parameter DIAMPROG of any of the axes is set to YES.

3.

COORDINATE SYSTEM
Programming in radius (G152) or in diameters (G151)

FAGOR

CNC 8060
CNC 8065

(REF. 1402)

3.4 Coordinate programming

3.4.1 Cartesian coordinates

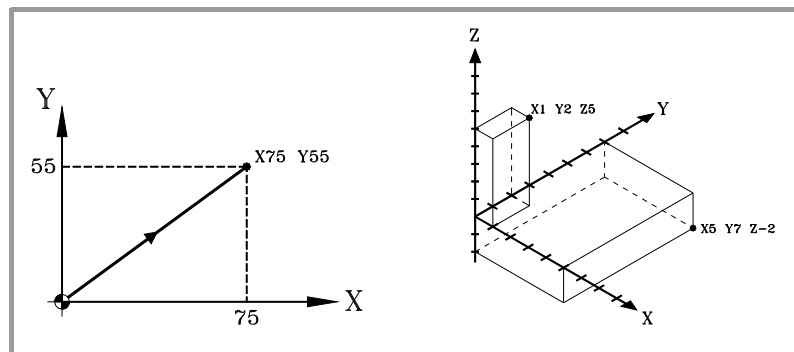
Coordinates are programmed according to a Cartesian coordinate system. This system consists of two axes in the plane and three or more in space.

Definition of position values

The position of a point in this system is given by its coordinates in the different axes. The coordinates are programmed in absolute or incremental coordinates and in millimeters or inches.

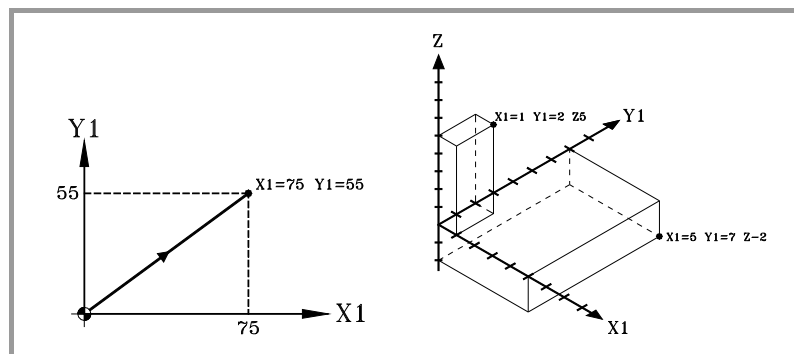
Standard axes (X...C)

The coordinates are programmed with the axis name followed by the coordinate value.



Numbered axes (X1...C9)

If the axis name is like X1, Y2... the "=" sign must be included between the axis name and the coordinate.



3.4.2 Polar coordinates

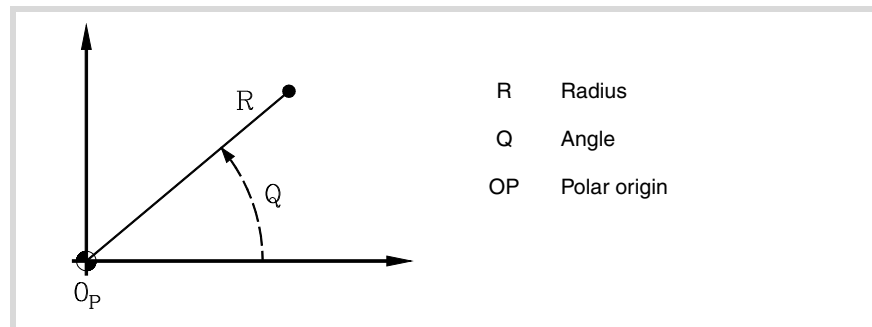
When having circular elements or angular dimensions, polar coordinates may be more convenient to express the coordinates of the various points in the plane.

This type of coordinates requires a reference point referred to as "polar origin" that will be the origin of the polar coordinate system.

Definition of position values

The position of the various points is given by defining the radius "R" and the angle "Q" as follows:

Radius	It will be the distance between the polar origin and the point.
Angle	It will be the one formed by the abscissa axis and the line joining the polar origin with the point.



The radius may be given in mm or in inches whereas the angle is given in degrees.

Both values may be given in either absolute (G90) or incremental (G91) coordinates.

- When working in G90, the "R" and "Q" values will be absolute. The value assigned to the radius must always be positive or zero.
- When working in G91, the "R" and "Q" values will be incremental. Although negative "R" values may be programmed, when programming in incremental coordinates, the resulting value assigned to the radius must always be positive or zero.

When programming a "Q" value greater than 360°, the module will be assumed after dividing it by 360. Thus, Q420 is the same as Q60 and Q-420 is the same as Q-60.

Polar origin preset

The "polar origin" may be selected from the program using function G30. If not selected, it assumes as "polar origin" the origin of the active reference system (part zero). See chapter ["5 Origin selection"](#).

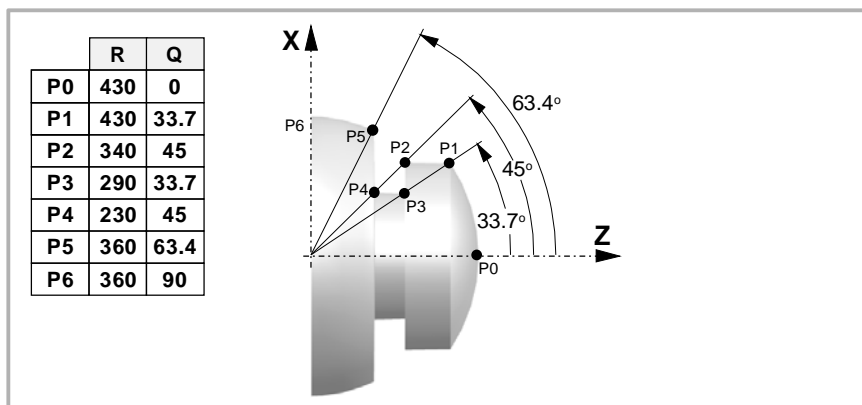
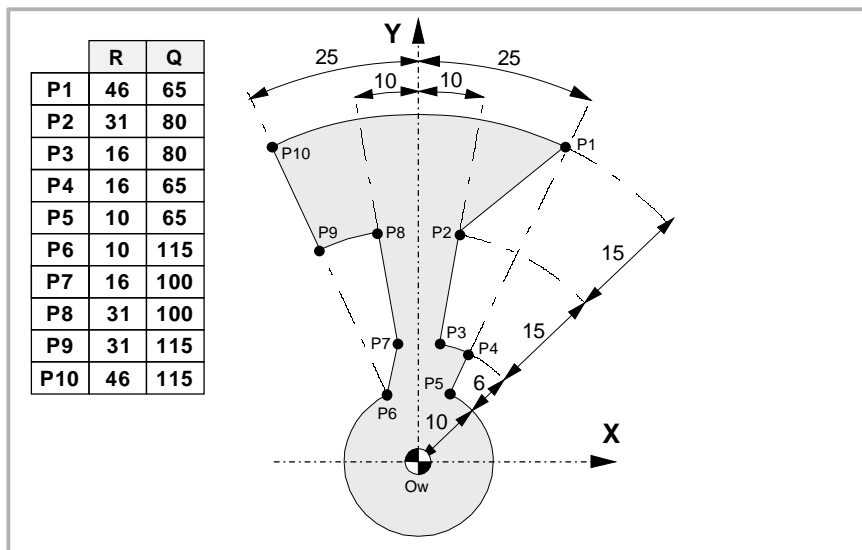
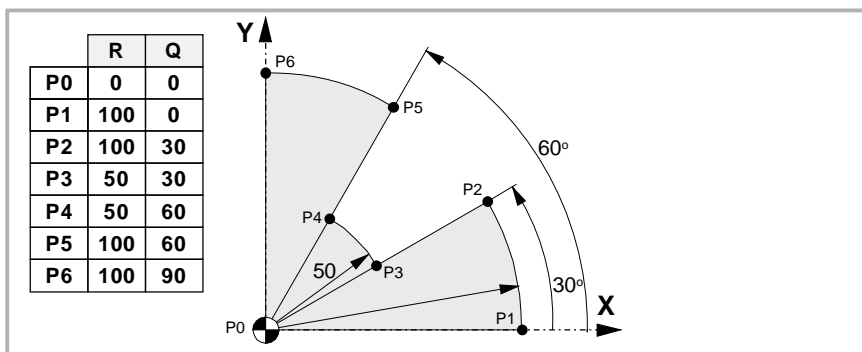
The selected "polar origin" is modified in the following instances:

- When changing the work plane, the CNC assumes the part zero as the new "polar origin".
- On power-up, after an M02 or M30 and after an EMERGENCY or a RESET, the CNC assumes the part zero as the new polar origin.

Examples. Point definition in polar coordinates.

3.

COORDINATE SYSTEM
Coordinate programming



CNC 8060
CNC 8065

(REF. 1402)

The work planes determine which axes define the work plane/trihedron and which axis corresponds to the longitudinal axis of the tool. Plane selection is required to execute operations like:

- Circular and helical interpolations.
- Corner chamfering and rounding.
- Tangential entries and exits.
- Machining canned cycles.
- Tool radius and length compensation.

These operations, except tool length compensation, can only be executed in the active work plane. Tool length compensation, on the other hand, can only be applied on the longitudinal axis.

Commands for changing the work planes.

Mill model or lathe model with "trihedron" type axis configuration.

Function.	Meaning.
G17	Main plane formed by the first axis (abscissa), second (ordinate) and third axis (perpendicular) of the channel.
G18	Main plane formed by the third axis (abscissa), first axis (ordinate) and second axis (perpendicular) of the channel.
G19	Main plane formed by the second axis (abscissa), third axis (ordinate) and first axis (perpendicular) of the channel.
G20	Select any work plane formed by the first three axes of the channel.

Instruction.	Meaning.
#TOOL AX	Select the longitudinal axis of the tool.

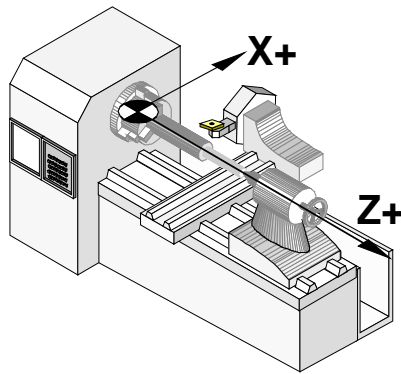
Lathe model with "plane" type axis configuration.

Function.	Meaning.
G18	Main plane formed by the second axis (abscissa) and first axis (ordinate) of the channel.
G20	Select the longitudinal axis of the tool.

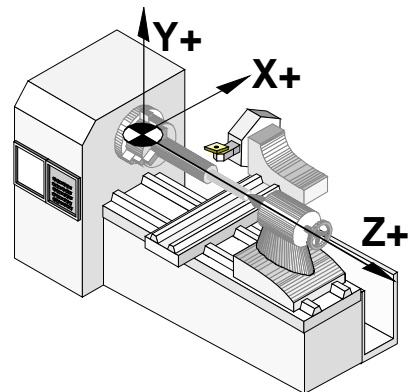
Instruction.	Meaning.
#TOOL AX	Select the longitudinal axis of the tool.

4.1 About work planes on lathe and mill models.

The operation of the work planes depends on the geometric configuration of the axes. At a mill model, the geometric configuration of the axes is always of the "trihedron" type whereas at a lathe model, the geometric configuration of the axes may be either a "trihedron" type or a "plane" type (parameter GEOCONFIG).



Configuration of "plane" type axes.



Configuration of "Trihedron" type axes.

Configuration of "Trihedron" type axes (lathe or mill model).

This configuration has three axes forming a trihedron Cartesian XYZ type. There may be more axes, besides those forming the trihedron; that may be part of the trihedron or be auxiliary axes, rotary axes, etc.

The order of the axes in the channel sets the main work planes, those selected with functions G17, G18 and G19. Function G20 may be used to form any work plane with the first three axes of the channel. The work plane by default is set by the manufacturer (parameter IPLANE), the usual plane being G17 at a mill model and G18 at a lathe model.

The CNC displays the ·G· functions associated with the work planes.

Configuration of "plane" type axes (lathe model).

This configuration has two axes forming the usual work plane on a lathe. There may be more axes, but they cannot be part of the trihedron; there must be auxiliary, rotary, etc.

With this configuration, the work plane is always G18 and will be formed by the first two axes defined in the channel, the second axis as abscissa and the first axis as ordinate. The ·G· functions associated with the work planes have the following effects.

Function.	Meaning.
G17	It does not change planes and shows a warning about it.
G18	It has no effect (except when function G20 is active).
G19	It does not change planes and shows a warning about it.
G20	It is permitted if it does not change the main plane; i.e. it can only be used to change the longitudinal axis.

The CNC does not display the ·G· functions associated with the work planes because it is always the same plane.

4.

WORK PLANES.
About work planes on lathe and mill models.



CNC 8060
CNC 8065

(REF. 1402)

4.2 Select the main new work planes.

4.2.1 Mill model or lathe model with "trihedron" type axis configuration.

The main planes may be selected by program using functions G17, G18 and G19 and are formed by two of the first three axes of the channel. The third axis corresponds to the axis perpendicular to the plane, which coincides with the longitudinal axis of the tool, the one on which tool length compensation is applied.

- G17 Main plane formed by the first axis (abscissa), second (ordinate) and third axis (perpendicular) of the channel.
- G18 Main plane formed by the third axis (abscissa), first axis (ordinate) and second axis (perpendicular) of the channel.
- G19 Main plane formed by the second axis (abscissa), third axis (ordinate) and first axis (perpendicular) of the channel.

The OEM, can use machine parameter LCOMPTYP to change the behavior of the longitudinal axis when changing planes so the CNC keeps the longitudinal axis that was active before changing planes.

Function G20 may select any plane with the first three axes of the channel. Function G20 and the instruction #TOOL AX can change the longitudinal axis of the tool.

Programming.

These functions may be programmed anywhere in the program and they don't have to go alone in the block.

Programming format.

The programming format is:

G17
G18
G19

G17
G18
G19

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

Functions G17, G18, G19 and G20 are modal and incompatible with each other. On power-up, after an M02 or M30 and after an emergency or a reset, the CNC assumes function G17 or G18 as set by the machine manufacturer (parameter "IPLANE").

4.

WORK PLANES.

Select the main new work planes.

FAGOR 

**CNC 8060
CNC 8065**

(REF. 1402)

4.2.2 Lathe model with "plane" type axis configuration.

The work plane is always G18 and will be formed by the first two axes defined in the channel. Functions G17 and G19 have no meaning for the CNC.

G18 Main plane formed by the second axis (abscissa) and first axis (ordinate) of the channel.

In the case of lathe tools, tool length compensation is applied on all the axes where a tool offset has been defined.

On milling tools, tool length compensation is applied on the second axis of the channel. If the X (first axis of the channel) and Z (second axis of the channel) axes have been defined, the work plane will be the ZX and Z will be the longitudinal axis. Function G20 and the instruction #TOOL AX can change the longitudinal axis of the tool.

Programming.

These functions may be programmed anywhere in the program and they don't have to go alone in the block.

Programming format.

The programming format is:

G18

G18

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

Functions G18 and G20 are modal and incompatible with each other. On power-up, after executing an M02 or M30, and after an emergency or reset, the CNC assumes function G18.

4.

WORK PLANES.
Select the main new work planes.



CNC 8060
CNC 8065

(REF. 1402)

4.3 Select any work plane and longitudinal axis.

The meaning of function G20 depends on the type of configuration of the machines axes; "plane" type for lathe or "trihedron" type for lathe or mill.

- When the axis configuration is of trihedron type, function G20 allows defining any work plane formed by the first three axes of the channel. To build a plane with other axes, first include them in the main trihedron (instruction #SET AX).
- When the axis configuration is of plane type, the work plane is always G18 and function G20 allows changing the longitudinal axis of the tool.

Programming.

When programming this instruction, you must define the new abscissa and ordinate axes of the plane and the longitudinal axis of the tool. If the longitudinal axis coincides with one of the axes of the plane, you must also define which axis is perpendicular to the plane.

Programming format.

The programming format is the following; the list of arguments appears between curly brackets and the optional ones between angle brackets.

G20 X~C{axistype} X~C{axistype} X~C{axistype} <X~C{axistype}> {axistype} Value that sets the location of the axis in the plane.

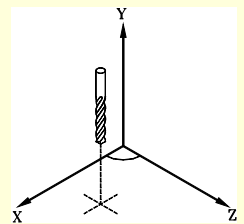
Values for setting the location of the axis in the plane.

The work plane is defined by selecting the abscissa and ordinate axes, the perpendicular axis and the longitudinal axis of the tool. It is selected by assigning one of the following values to the axes programmed with G20.

Value.	Type of axis within the work plane.
1	Abscissa axis.
2	Ordinate axis.
±3	Longitudinal axis of the tool. The sign indicates tool orientation.
4	Reserved.
5	Axis perpendicular to the work plane, only required when the longitudinal axis of the tool is the same as the abscissa or ordinate axis. Otherwise, the longitudinal axis of the tool will be the perpendicular axis.

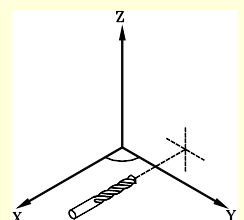
G20 X1 Z2 Y3

The X axis is the abscissa axis.
The Z axis is the ordinate axis.
The Y axis is the longitudinal axis of the tool and the axis perpendicular to the plane.



G20 X1 Y2 X3 Z5

The X axis is the abscissa axis and the longitudinal axis of the tool.
The Y axis is the ordinate axis.
The Z axis is the axis perpendicular to the plane.



4.

WORK PLANES.
Select any work plane and longitudinal axis.



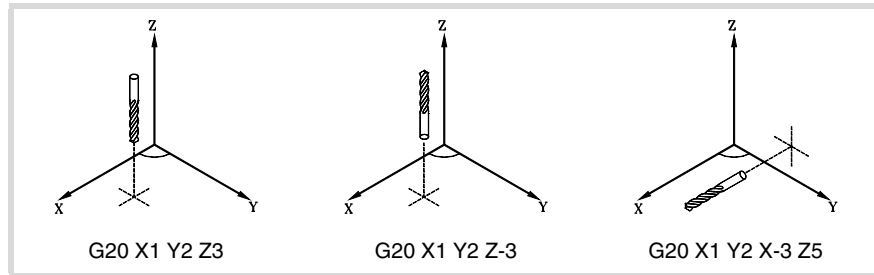
CNC 8060
CNC 8065

(REF. 1402)

Select the longitudinal axis of the tool.

When selecting the longitudinal axis with G20, tool orientation may be established according to the programmed sign.

- If the parameter to select the longitudinal axis is positive, the tool is positioned in the positive direction of the axis.
- If the parameter to select the longitudinal axis is negative, the tool is positioned in the negative direction of the axis.

**Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.**

Function G20 is modal and incompatible with G17, G18 and G19. On power-up, after an M02 or M30 and after an emergency or a reset, the CNC assumes function G17 or G18 as set by the machine manufacturer (parameter "IPLANE").

4.**WORK PLANES.**

Select any work plane and longitudinal axis.



CNC 8060
CNC 8065

(REF. 1402)

4.4 Select the longitudinal axis of the tool.

The instruction #TOOL AX allows changing the longitudinal axis of the tool except on those for turning. This instruction allows to select any machine axis as the new longitudinal axis.

Programming.

When programming this instruction, you must define the new axis and the orientation of the tool.

Programming format.

The programming format is the following; the list of arguments appears between curly brackets.

#TOOL AX [X~C{+|-}]

{+|-} Tool orientation.

#TOOL AX [Z+]

#TOOL AX [V2-]

Define the orientation of the tool.

Tool orientation is set as follows.

+ sign Positive tool orientation.

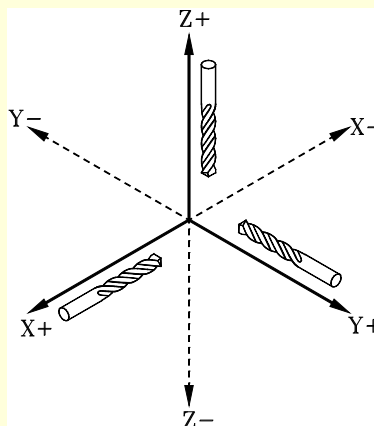
- sign Negative tool orientation.

Positive tool orientation.

#TOOL AX [X+]

#TOOL AX [Y+]

#TOOL AX [Z+]

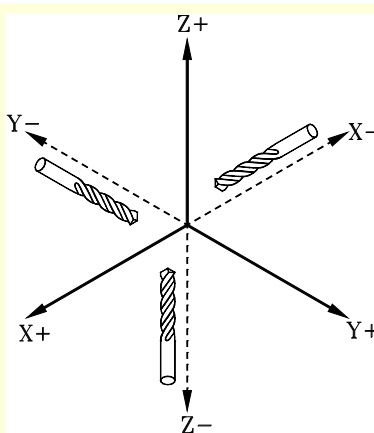


Negative tool orientation.

#TOOL AX [X-]

#TOOL AX [Y-]

#TOOL AX [Z-]



4.

WORK PLANES.
Select the longitudinal axis of the tool.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

4.

WORK PLANES.

Select the longitudinal axis of the tool.



CNC 8060
CNC 8065

(REF. 1402)

With this CNC, it is possible to program movements in the machine reference system or apply offsets in order to use reference systems referred to the fixtures or the part without having to change the coordinates of the different points of the part in the program.

There are three different offset types; fixture offset, zero offsets and PLC offsets. The CNC may have several of these offsets active at the same time, in that case, the system coordinate origin being used will be defined by the sum of the active offsets.

Type of offset.	Description.
Fixture offset.	Distance between the machine reference zero and the fixture's zero point. On machines using several fixtures, this offsets allows selecting the particular fixture to be used.
Zero offset.	Distance between the fixture's zero point and the part zero. If the fixture zero is not active (no fixture offset), the zero offset is measured from machine zero. The zero offset may be set by presetting a coordinate or a zero offset.
PLC offset.	Special offset handled by the PLC that is used to correct the deviations due to dilatations, etc. The PLC always applies this offset, even when programming with respect to machine zero.

5.1 Programming with respect to machine zero

Machine zero is the origin of the machine reference system. Movements referred to machine zero are programmed using the instructions #MCS and #MCS ON/OFF.

Program a movement referred to machine zero.

This instruction may be added to any block containing a movement so it is executed in the machine reference system.

```
G00 X30 Y30
G92 X0 Y0           (Coordinate preset)
G01 X20 Y20
#MCS X30 Y30       (Movement referred to machine zero. Offsets canceled)
G01 X40 Y40       (Offsets restored)
G01 X60 Y60
M30
```

Machine coordinate system.

The #MCS ON and #MCS OFF instructions activate and deactivate the machine reference system; therefore, the movements programmed between them are executed in the machine reference system. Both instructions must be programmed alone in the block.

```
G92 X0 Y0           (Coordinate preset)
G01 X50 Y50
#MCS ON             (Beginning of programming referred to machine zero)
G01 ...
G02 ...
G00 ...
#MCS OFF           (End of programming referred to machine zero. Offsets restored)
```

Considerations for movements referred to machine zero.

Zero offsets and coordinate transformations

When executing a movement referred to machine zero, the CNC ignores the active offsets (except the PLC offset), the kinematics and cartesian transformations; therefore, the movement is carried out in the machine reference system. Once the movement has ended, the CNC restores the offsets, kinematics and cartesian transformations that were active.

The programmed movements do not admit polar coordinates, nor other kinds of transformations such as mirror image, coordinate (pattern) rotation or scaling factor. While the #MCS function is active, functions for setting a new origin such as G92, G54-G59, G158, G30, etc. are not admitted either.

Tool radius and length compensation

Tool radius and length compensation is also canceled during the movements referred to machine zero. The CNC assumes that the coordinates have been programmed with respect to the tool base, not to the tool tip.

5.

ORIGIN SELECTION

Programming with respect to machine zero



CNC 8060
CNC 8065

(REF. 1402)

System units; millimeters or inches

When moving with respect to machine reference zero, the G70 or G71 units (inches/millimeters) selected by the user are ignored. It assumes the units predefined at the CNC (INCHES parameter); assumed by the CNC on power-up. These units are assumed for defining the coordinates, for the feedrate and for the speed.

5.

ORIGIN SELECTION

Programming with respect to machine zero



CNC 8060
CNC 8065

(REF. 1402)

5.2 Set the machine coordinate (G174).



Use this function with caution. Changing the machine coordinate can cause the axes to exceed the travel limits during the movement.

Function G174 may be used to set the machine coordinate of an axis or spindle; in other words, temporarily set a new machine zero. The new machine coordinate stays active until the axis or spindle is homed; then, the CNC restores the original machine reference zero (set in the machine parameters).

After executing function G174, the CNC assumes that the programmed coordinate defines the current position referred to machine reference zero (home). The zero offsets, movements with respect to machine zero, etc. will be referred to the coordinate programmed in G174.

Programming the function.

Program function G174, and then the machine coordinate of a single axis or spindle. With this function only the machine coordinate of an axis or spindle may be set; to set the machine coordinates of several, program one G174 for each one of them.

When setting the machine coordinate, the CNC ignores the G70/G71 units (inches/millimeters) selected by the user and uses the unit system pre-defined at the CNC (parameter INCHES). The CNC also ignores all the other options, radius/diameter, mirror image, scaling factor, etc.

Programming format.

The programming format is:

G174 X..C

G174 S

X..C Machine coordinate at the axes.

S Machine coordinate at the spindles.

G174 X100

G174 S180

Considerations and limitations.

Function G174, by itself, does not cause any axis or spindle movement. After executing function G174, the CNC considers that the axis or spindle is homed and verifies that it is within the software travel limits.

The CNC does not allow setting the machine coordinate on slaved axes, gantry, tandem or on axes that are part of the active kinematics or active transform. The CNC permits setting the machine coordinate for tandem axes. Before setting the new machine coordinate, the CNC checks that the axis or spindle is in position and it is not synchronized, if that's not the case, it issues an error message.



On Sercos axes, function G174 also resets the coordinate of the drive. Setting the machine coordinates on position-Sercos axes requires drive version V6.20 or newer.

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

Function G174 is modal. This function is neither affected by functions M02 and M30 nor by a reset, by an emergency or by turning the CNC off. On power-up, the CNC assumes the machine coordinates that were active when the CNC was turned off.

5.

ORIGIN SELECTION
Set the machine coordinate (G174).



CNC 8060
CNC 8065

(REF. 1402)

5.3 Fixture offset

With fixture offsets, it is possible to select the fixture system to be used (when having more than one fixture). When applying a new fixture offset, the CNC assumes the point set by the new selected fixture as the new fixture zero.

Defintion

In order to apply a fixture offset, it must have been previously set. To do that, the CNC has a table where the operator may define up to 10 different fixture offsets. The table data may be defined:

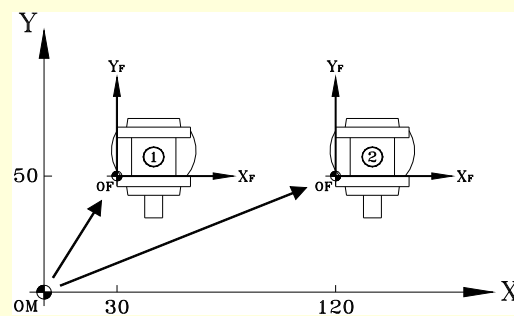
- Manually from the CNC's front panel (as described in the Operating Manual).
- By program, assigning the corresponding value (of the "n" offset and of the "Xn" axis) to the "V.A.FIXT[n].Xn" variable.

Activation

Once the fixture offsets have been defined in the table, they may be activated via program by assigning to the "V.G.FIX" variable, the offset number to be applied.

Only one fixture offset may be active at a time; therefore, when applying a fixture offset, it will cancel the previous one. Assigning a value of "V.G.FIX=0" will cancel the active fixture offset.

Fixture offset value on milling machine.



	X	Y
V.G.FIX=1	30	50
V.G.FIX=2	120	50

```
N100 V.A.FIXT[1].X=30 V.A.FIXT[1].Y=50
```

```
N110 V.A.FIXT[2].X=120 V.A.FIXT[2].Y=50
```

...

```
N200 V.G.FIX=1 (It applies the first fixture offset)
```

```
N210 ... (Programming at fixture 1)
```

```
N300 V.G.FIX=2 (It applies the first fixture offset)
```

```
N310 ... (Programming at fixture 2)
```

```
N400 V.G.FIX=0 (Cancel fixture offset. No fixture system is active)
```

Considerations

A fixture offset, by itself, does not cause any axis movement.

Properties

On power-up, the CNC assumes the fixture offset that was active when the CNC was turned off. On the other hand, the fixture offset is neither affected by functions M02 and M30 nor by RESETTING the CNC.

5.

ORIGIN SELECTION
Fixture offset

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

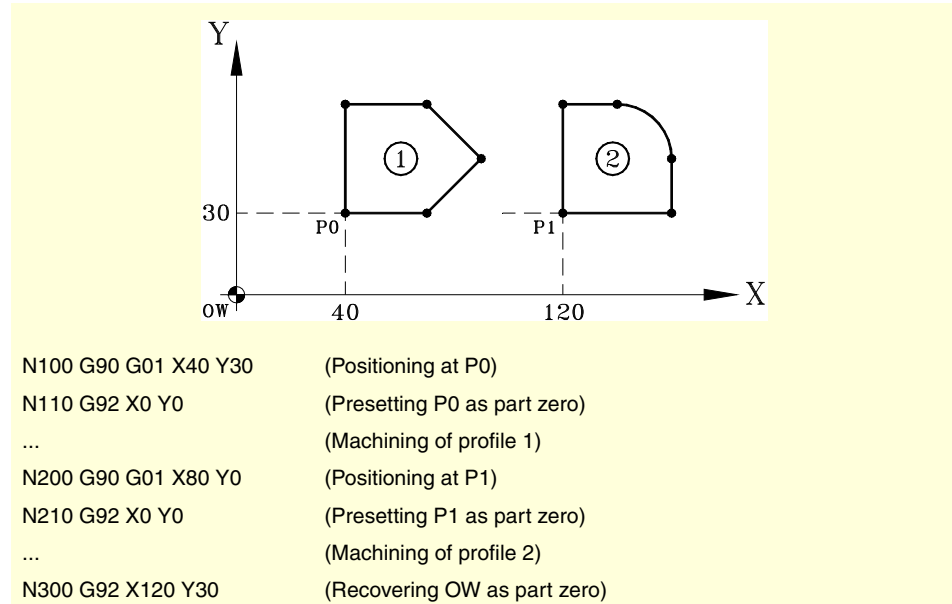
5.4 Coordinate preset (G92)

Coordinate presetting is done with function G92 and it may be applied onto any axis of the machine.

When presetting a coordinate, the CNC interprets that the axis coordinates programmed after the G92 set the current position of the axes. The rest of the axes that have not been defined with G92 are not affected by the preset.

5.

ORIGIN SELECTION
Coordinate preset (G92)



Considerations

A coordinate preset, by itself, does not cause any axis movement.

When homing an axis in JOG mode, the preset for that axis is canceled.

Properties of the function

G92 is modal, the preset values remain active until the preset is canceled (with another preset, a zero offset or with G53).

On power-up, the CNC assumes the coordinate preset that was active when the CNC was turned off. On the other hand, the coordinate preset is neither affected by functions M02 and M30 nor by RESETTING the CNC.



CNC 8060
CNC 8065

(REF. 1402)

5.5 Zero offsets (G54-G59/G159)

Using zero offsets, it is possible to place the part zero in different positions of the machine. When applying a zero offset, the CNC assumes as the new part zero the point defined by the selected zero offset.

Defining zero offsets.

In order to apply a zero offset, it must have been previously defined. To do that, the CNC has a table where the operator may define up to 99 different zero offsets. The table data may be defined manually (as described in the operating manual) or via program (using variables).

The OEM may have configured the zero offset table in one of the following ways (machine parameter FINEORG).

- Each zero offset has a single value. When executing function G159, the CNC assumes this value as the new zero offset.
- Each zero offset has a coarse (or absolute) value and a fine (or incremental) value. When executing function G159, the CNC assumes as new zero offset the sum of both parts.

Activation

Once the zero offsets have been defined in the table, they may be activated via program by programming function G59 followed by the offset number to be activated.

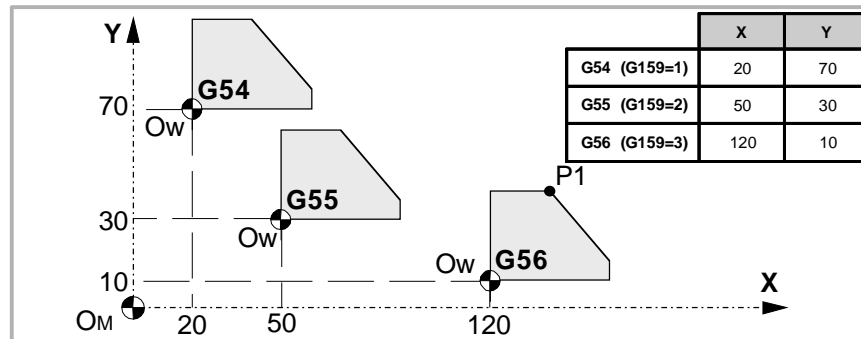
G159=2 The CNC applies the second zero offset.

G159=11 The CNC applies the 11th zero offset.

The first six zero offsets of the table can also be applied using functions G54 through G59; G54 for the first one (same as G159=1), G55 for the second one (same as G159=2) and so on.

G54 The CNC applies the first zero offset (G159=1).

G59 The CNC applies the sixth zero offset (G159=6).



```
N100 V.A. ORGT[1].X=20 V.A. ORGT[1].Y=70
N110 V.A. ORGT[2].X=50 V.A. ORGT[2].Y=30
N100 V.A. ORGT[3].X=120 V.A. ORGT[3].Y=10
```

...

```
N100 G54
      (It applies the first zero offset)
```

```
N200 G159=2
      (It applies the second zero offset)
```

```
N300 G56 X20 Y30
      (It applies the third zero offset)
      (The axes move to point X20 Y30 (point P1) referred to the third origin)
```

5.

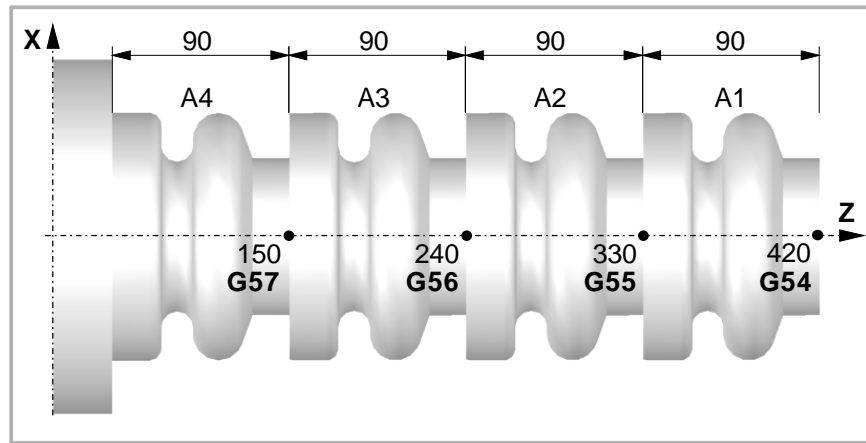
ORIGIN SELECTION
Zero offsets (G54-G59/G159)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

5.

ORIGIN SELECTION
 Zero offsets (G54-G59/G159)


	X	Z
G54 (G159=1)	0	420
G55 (G159=2)	0	330
G56 (G159=3)	0	240
G57 (G159=4)	0	150

```

N100 V.A. ORGT[1].X=0 V.A. ORGT[1].Z=420
N110 V.A. ORGT[2].X=0 V.A. ORGT[2].Z=330
N100 V.A. ORGT[3].X=0 V.A. ORGT[3].Z=240
N100 V.A. ORGT[4].X=0 V.A. ORGT[3].Z=150

N100 G54 (It applies the first absolute zero offset)
... (Machining of profile A1)
N200 G55 (It applies the second absolute zero offset)
... (Machining of profile A2)
N300 G56 (It applies the third absolute zero offset)
... (Machining of profile A3)
N200 G56 (It applies the fourth absolute zero offset)
... (Machining of profile A4)

```

Only one zero offset may be active at a time; therefore, when applying a zero offset, the previous one will be canceled. When programming G53, the zero offset currently active will be canceled.

The function corresponding to the selected zero offset may be programmed in any block of the program. When added to a block with path information, the zero offset will be applied before executing the programmed movement.

Considerations

A zero offset, by itself, does not cause any axis movement.

When homing an axis in JOG mode, the absolute zero offset for that axis is canceled.

Properties of the functions

Functions G54, G55, G56, G57, G58, G59 and G159 are modal and incompatible with each other and with G53 and G92.

On power-up, the CNC assumes the zero offset that was active when the CNC was turned off. On the other hand, the zero offset is neither affected by functions M02 and M30 nor by RESETTING the CNC.

(REF. 1402)

5.5.1 Variables for setting zero offsets

Zero offset table (without fine setting of the absolute zero offset).

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W).

Variable.	R/W	Meaning.
(V.)[ch].A.ORG.xn	R	Value of the active zero offset (absolute G159 + incremental G158).
(V.)[ch].A.ADDORG.xn	R	Value of the active incremental zero offset (G158).
(V.)[ch].A.ORG[nb].xn	R/W	Offset set in the zero offset [nb].

Zero offset table (with fine setting of the absolute zero offset).

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W).

Variable.	R/W	Meaning.
(V.)[ch].A.ORG.xn	R	Value of the active zero offset (coarse absolute G159 + fine absolute G159 + incremental G158).
(V.)[ch].A.ADDORG.xn	R	Value of the active incremental zero offset (G158).
(V.)[ch].A.COARSEORG.xn	R	Value of the active absolute zero offset (G159), coarse part.
(V.)[ch].A.FINEORG.xn	R	Value of the active absolute zero offset (G159), fine part.
(V.)[ch].A.ORG[nb].xn	R/W	Offset set in the zero offset [nb]; coarse part plus fine part. When writing this variable, the value is assigned to the coarse part deleting the fine part.
(V.)[ch].A.COARSEORG[nb].xn	R/W	Offset set in the zero offset [nb]; coarse part.
(V.)[ch].A.COARSEORG[nb].xn	R/W	Offset set in the zero offset [nb]; fine part.

Syntax of the variables.

- ch· Channel number.
- nb· Zero offset number.
- xn· Name, logic number or index of the axis.

V.A.ORG.Z	Z axis.
V.A.ADDORG.3	Axis with logic number -3.
V.[2].A.COARSEORG.3	Axis with index -3 in the channel -2.
V.[2].A.FINEORG.3	Axis with index -3 in the channel -2.
V.A.ORG[1].Z	Zero offset G54 (G159=1). Z axis.
V.A.ORG[1].Z	Zero offset G54 (G159=1). Z axis.
V.A.COARSEORG[4].3	Zero offset G57 (G159=4). Axis with logic number -3.
V.[2].A.FINEORG[9].3	Zero offset G159=9. Axis with index -3 in the channel -2.

5.

ORIGIN SELECTION
Zero offsets (G54-G59/G159)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

5.5.2 Incremental zero offset (G158)

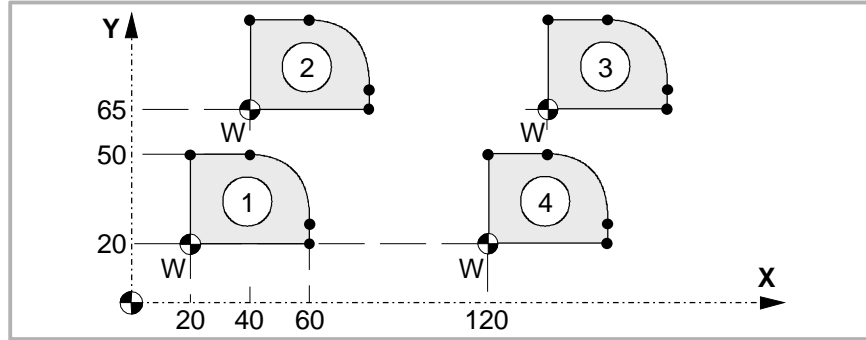
When applying an incremental zero offset, the CNC adds it to the absolute zero offset active at a time.

Programming

Incremental zero offset are defined by program using function G158 followed by the values of the zero offset to be applied on each axis. To cancel the incremental zero offset, program function G158 without axes in the block. To only cancel the incremental zero offset on particular axes, program an incremental offset of "0" on each of those axes.

5.

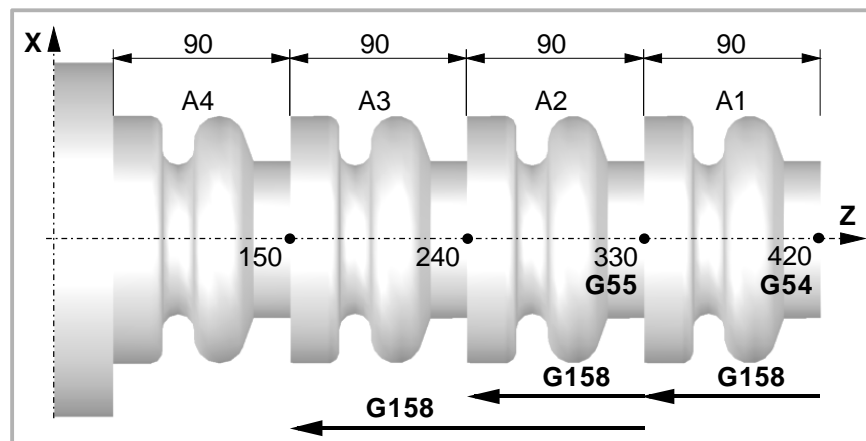
ORIGIN SELECTION
Zero offsets (G54-G59/G159)



	X	Y
G54 (G159=1)	30	20
G55 (G159=2)	120	20

```

N100 G54          (It applies the first zero offset)
...              (Machining of profile 1)
N200 G158 X20 Y45 (Apply incremental zero offset)
...              (Machining of profile 2)
N300 G55          (It applies the second zero offset. G158 stays active)
...              (Machining of profile 3)
N400 G158        (Cancel incremental zero offset. G55 stays active)
...              (Machining of profile 4)
    
```



	X	Z
G54 (G159=1)	0	420
G55 (G159=2)	0	330

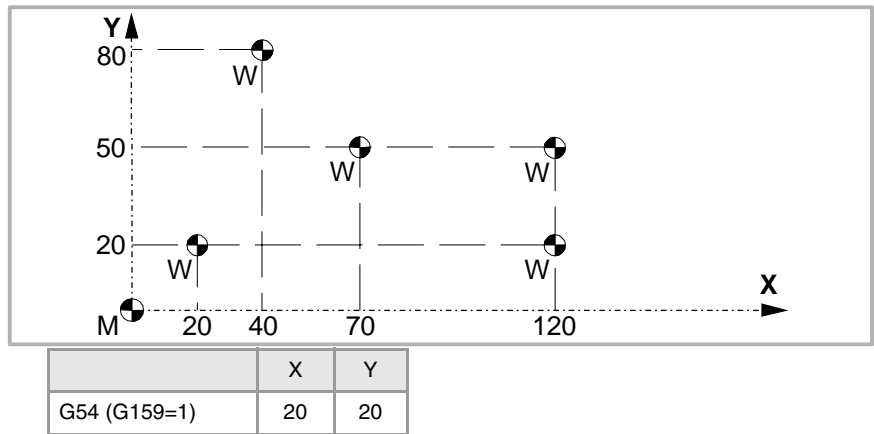


CNC 8060
CNC 8065

(REF. 1402)

N100 G54	(It applies the first absolute zero offset)
...	(Machining of profile A1)
N200 G158 Z-90	(Apply incremental zero offset)
...	(Machining of profile A2)
N300 G55	(It applies the second absolute zero offset)
...	(The incremental zero offset is still active)
...	(Machining of profile A3)
N200 G158 Z-180	(It applies the second incremental zero offset)
...	(Machining of profile A4)

Only one incremental zero may be active at a time for each axis; therefore, applying an incremental zero offset on an axis cancels the one that was active on that axis. The offsets on the rest of the axes are not affected.



N100 G54	(Apply absolute zero offset)
N200 G158 X20 Y60	(It applies the first incremental zero offset)
N300 G158 X50 Y30	(It applies the second incremental zero offset)
N400 G158 X100	(It applies the third incremental zero offset)
N500 G158 Y0	(It applies the fourth incremental zero offset)
N600 G158 X0	(Cancel incremental zero offset)

The incremental zero offset is not canceled after applying a new absolute zero offset (G54-G59 or G159).

Considerations

An incremental zero offset, by itself, does not cause any axis movement.

When homing an axis in JOG mode, the incremental zero offset for that axis is canceled.

Properties of the function

Function G158 is modal.

On power-up, the CNC assumes the incremental zero offset that was active when the CNC was turned off. On the other hand, the incremental zero offset is neither affected by functions M02 and M30 nor by **RESETTING** the CNC.

5.

ORIGIN SELECTION
Zero offsets (G54-G59/G159)



CNC 8060
CNC 8065

(REF. 1402)

5.5.3 Excluding axes in the zero offset (G157)

Excluding axes allows to select on to which axes the next absolute zero offset will not be applied. After applying the zero offset, the programmed axis exclusion is canceled and it has to be programmed again in order to apply it again.

Activation

Axis exclusion must be programmed using function G157 followed by the axes and the value indicating whether that axis is excluded (<axis>=1) or not (<axis>=0).

The exclusion may also be activated by programming only the axes affected by the exclusion after function G157.

The exclusion and the zero offset may be programmed in the same block. In that case, the exclusion will be activated before applying the zero offset.

```
G55
  (It applies the second zero offset on all the axes)
```

```
G157 X Z
  (Activation of the exclusion on the X-Z axes)
G57
  (It applies the fourth zero offset, except on the X-Z axes. These axes keep the previous zero offset)
...
G159=8
  (It applies the eighth zero offset on all the axes)
```

```
G59 G157 Y
  (It applies the sixth zero offset, except on the Y axis. That axis keeps the previous zero offset)
...
G54
  (It applies the first zero offset on all the axes)
```

Excluding axes does not affect the active zero offsets. If an axis is excluded, when applying a new zero offset, the CNC maintains the one that was active for that axis.

Considerations

Excluding axes does not affect the coordinate preset or the incremental zero offsets which are always applied on to all the axes. Likewise, neither fixture offsets nor PLC offsets are affected.

Properties of the function

Function G157 is modal and it remains active until an absolute zero offset is applied.

On power-up or after an EMERGENCY, the CNC does not assume any axis exclusion.

5.

ORIGIN SELECTION
Zero offsets (G54-G59/G159)



CNC 8060
CNC 8065

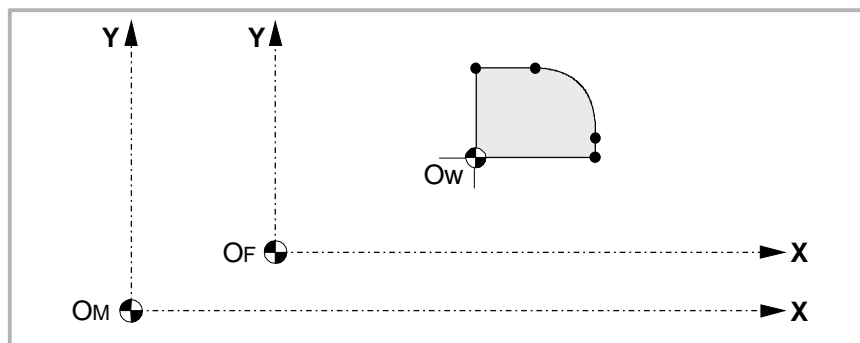
(REF. 1402)

5.6 Zero offset cancellation (G53)

Executing function G53 cancels the active zero offset resulting either from a preset (G92) or from a zero offset, including the incremental offset and the defined axis exclusion. It also cancels the zero offset due to a probing operation.

Fixture offsets and PLC offsets are not affected by this function.

Contrary to the #MCS and #MCS ON/OFF instructions that always execute movements referred to machine zero, function G53 allows to execute movements referred to the fixture zero (if it is active).



N10 V.G.FIX=1	(Activate fixture offset. Program with respect to OF)
N20 G54	(Apply the zero offset. Program with respect to OW)
N30 #MCS X20 Y20	(Activate machine coordinate system. Program with respect to OM)
N40 G01 X60 Y0	(Program with respect to OW)
N50 G53	(Cancel zero offset G54. Program with respect to OF)

Function G53 may be programmed in any block of the program. When added to a block with path information, the offset or preset is canceled before executing the programmed movement.

Considerations

Function G53, by itself, does not cause any axis movement.

Properties of the function

Function G53 is modal and incompatible with function G92, zero offsets and probing.

5.

ORIGIN SELECTION
Zero offset cancellation (G53)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

5.7 Polar origin preset (G30)

Function G30 may be used to preset any point of the work plane as the new polar origin. If not selected, it assumes as polar origin the origin of the active reference system (part zero).

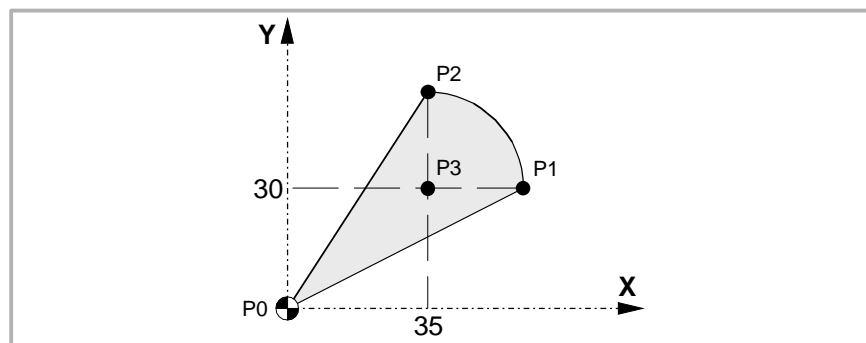
Programming

The polar origin preset must be programmed alone in the block. The programming format is "G30 I J", where:

- I, J They define the abscissa and ordinate of the new polar origin. They must be defined in absolute coordinates referred to part zero.
 When programmed, both parameters must be programmed.
 If not programmed, it will assume the current tool position as the polar origin.

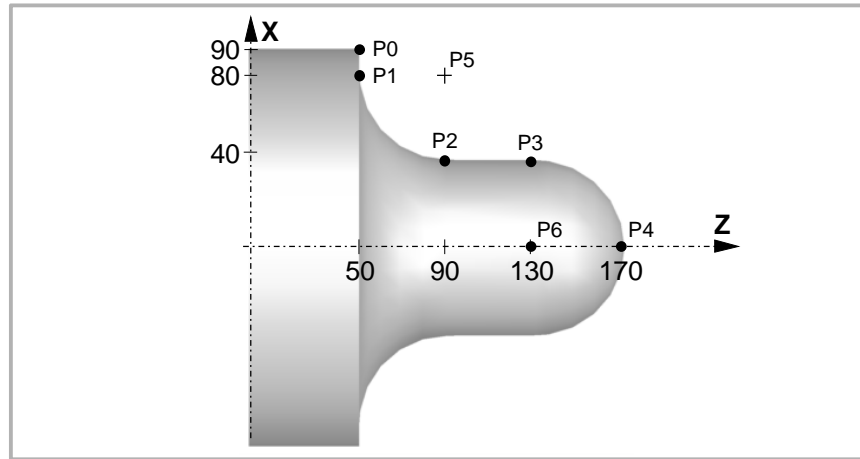
Therefore, function G30 may be programmed as follows:

- G30 I J It assumes as the new polar origin the point whose abscissa is "I" and ordinate "J" referred to part zero.
 G30 The current tool position is assumed as the new polar origin.



Assuming the initial point is X0 Y0:

```
G30 I35 J30      (Preset P3 as the polar origin)
G90 G01 R25 Q0  (Point P1)
G03 Q90         (Point P2)
G01 X0 Y0      (Point P0)
M30
```



G18 G151 ; Main plane Z-X, and programming in diameters.
 G90 X180 Z50 ; Point P0, programming in diameters.
 G01 X160 ; Point P1, in a straight line (G01).
 G30 I90 J160 ; Presets P5 as polar origin.
 G03 Q270 ; Point P2, in arc (G03).
 G01 Z130 ; Point P3, in a straight line (G01).
 G30 I130 J0 ; Presets P6 as polar origin.
 G02 Q0 ; Point P4, in arc (G02).

Properties of the function

Function G30 is modal. The polar origin stays active until another value is preset or the work plane is changed. When changing the work plane, it assumes the part zero of that plane as the new polar origin.

On power-up, after an M02 or M30 and after an EMERGENCY or a RESET, the CNC assumes the currently selected part zero as the new polar origin.

5.

ORIGIN SELECTION
 Polar origin preset (G30)

FAGOR 

CNC 8060
 CNC 8065

(REF. 1402)

5.

ORIGIN SELECTION

Polar origin preset (G30)



CNC 8060
CNC 8065

(REF. 1402)

6.1 Machining feedrate (F)

The machining feedrate may be selected by programmed using the "F" code which remains active until another value is programmed. The programming units depend on the active work mode (G93, G94 or G95) and the type of axis being moved (linear or rotary).

G94 - Feedrate in millimeters/minute (inches/minute).

G95 - Feedrate in millimeters/revolution (inches/revolution).

G93 - Machining time in seconds.

The programmed "F" is effective in linear (G01) or circular (G02, G03) interpolations. Movements in G00 (rapid traverse) are executed in rapid regardless of the programmed "F" value.

Movement without programmed feedrate.

In principle, the CNC will show the corresponding error when programming a movement in G01/G02/G03 without having set a feedrate.

Optionally, the manufacturer may have configured the CNC to make the movements at the maximum machining feedrate, set by machine parameter MAXFEED.

Feedrate limitation.

The manufacturer may have limited the maximum feedrate with machine parameter MAXFEED. When trying to exceed the maximum feedrate via part-program, via PLC or from the operator panel, the CNC limits the feedrate to the maximum value set without showing any error message or warning.

If this parameter is set to zero, the machining feedrate is not limited and the CNC assumes the one set for G00 as the maximum feedrate.

Variable to limit the feedrate via PLC.

(V.) [n] . PLC . G00FEED is a variable that may be written from the PLC to set, at a particular moment and in real time, the maximum feedrate of the channel for any type of movement.

Feedrate regulation.

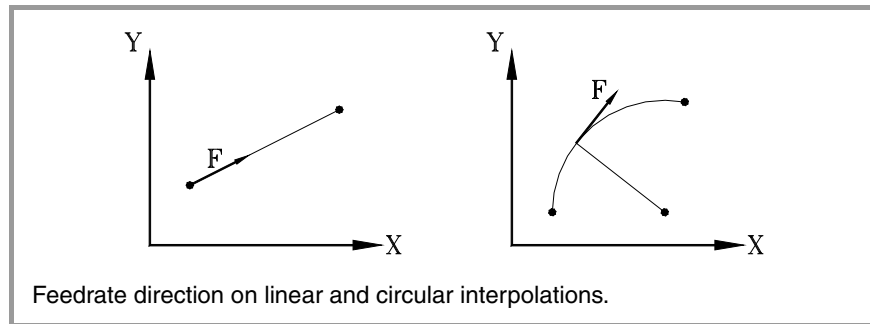
The programmed feedrate "F" may be varied between 0% and 200% using the selector switch on the CNC's operator panel or it may be selected by program or by PLC. However, the maximum override is limited by the machine manufacturer [G.M.P. "MAXOVR"].

When making movements in G00 (rapid traverse), the feedrate override percentage will be fixed at 100% or it may be varied between 0% and 100% depending on how the machine manufacturer has set [G.M.P. "RAPIDOVR"].

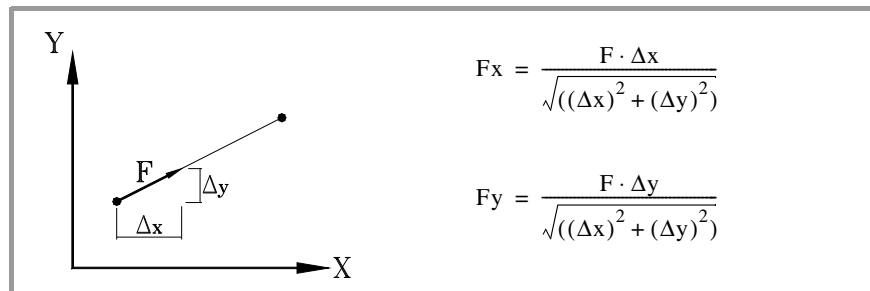
When carrying out threading operations, the feedrate percentage will be fixed at 100% of the programmed feedrate.

Understanding how the CNC calculates the feedrate.

The feedrate is measured along the tool path, either along the straight line (linear interpolations) or along the tangent of the indicated arc (circular interpolations).



When only the main axes are involved in the interpolation, the relationship between the components of the feedrate on each axis and the programmed "F" is the same as between the displacement of each axis and the resulting programmed displacement.



When rotary axes are involved in the interpolations, the feedrate of these axes is calculated so the beginning and the end of their movement coincides with the beginning and the end of the main axes. If the feedrate calculated for the rotary axis is greater than the maximum allowed, the CNC will adapt the programmed "F" so the rotary axis can turn at its maximum speed.

6.2 Feedrate related functions

6.2.1 Feedrate programming units (G93/G94/G95)

The functions related to programming units permit selecting whether mm/minute (inches/minute) or mm/revolution (inches/rev.) are programmed or, instead, the time the axes will take to reach their target position.

Programming

The functions related to programming units are:

G94	Feedrate in millimeters/minute (inches/minute).
G95	Feedrate in millimeters/revolution (inches/revolution).
G93	Machining time in seconds.

These functions may be programmed anywhere in the program and they don't have to go alone in the block. If the moving axis is rotary, the programming units will be in degrees instead of millimeters or inches as follows:

	Linear axes	Rotary axes
G94	millimeters (inches)/minute	degrees/minute
G95	millimeters (inches)/revolution	degrees/revolution
G93	seconds	seconds

G94

Feedrate in millimeters/minute (inches/minute).

After executing G94, the CNC interprets that the feedrates programmed with the "F" code are in millimeters/minute (inches/minute). If the moving axis is rotary, the CNC interprets that the programmed feedrate is in degrees/minute.

G95

Feedrate in millimeters/revolution (inches/revolution)

After executing G95, the CNC interprets that the feedrates programmed with the "F" code are in mm/rev (inches/rev) of the master spindle of the channel. If the moving axis is rotary, the CNC interprets that the programmed feedrate is in degrees/revolution.

If the spindle does not have an encoder, the CNC will use the programmed theoretical rpm to calculate the feedrate. This function does not affect the movements in G00 which are always executed in millimeters/minute (inches/minute).

G93

Machining time in seconds

After executing G93, the CNC interprets that the movements must be carried out in the time period (seconds) indicated by the "F" code.

This function does not affect the movements in G00 which are always executed in millimeters/minute (inches/minute).

Properties of the functions

Functions G93,G94 and G95 are modal and incompatible with each other.

On power-up, after an M02 or M30 and after an EMERGENCY or a RESET, the CNC assumes function G94 or G95 as set by the machine manufacturer [G.M.P. "IFEED"].

6.

TECHNOLOGICAL FUNCTIONS
Feedrate related functions

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

6.2.2 Feedrate blend (G108/G109/G193)

With these functions, it is possible to blend the feedrate between consecutive blocks programmed with different feedrates.

Programming

The functions related to feedrate blending are:

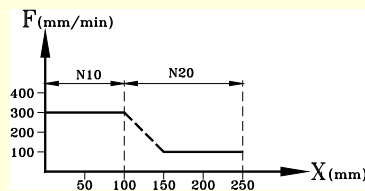
- G108 Feedrate blending at the beginning of the block.
- G109 Feedrate blending at the end of the block.
- G193 Interpolating the feedrate.

These functions may be programmed anywhere in the program and they don't have to go alone in the block.

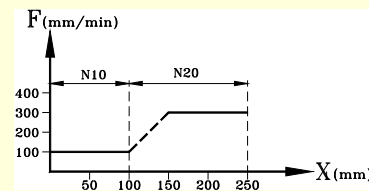
G108

Feedrate blending at the beginning of the block

When G108 is active, the adaptation to the new feedrate (by accelerating or decelerating) takes place at the beginning of the next block and the current block ends at the programmed feedrate "F".



N10 G01 G108 X100 F300
N20 X250 F100

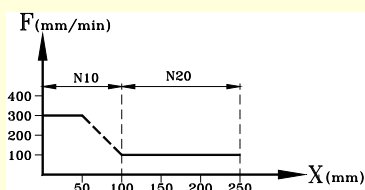


N10 G01 G108 X100 F100
N20 X250 F300

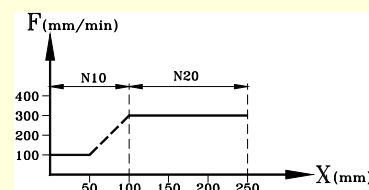
G109

Feedrate blending at the end of the block

When programming G109 the adaptation to the new feedrate (by accelerating or decelerating) takes place at the end of the current block so the next block starts executing at its programmed feedrate "F".



N10 G01 G109 X100 F300
N20 X250 F100

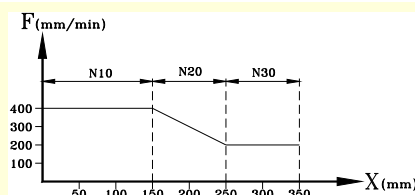


N10 G01 G109 X100 F100
N20 X250 F300

G193

Interpolating the feedrate

When programming G193, the adaptation to the new feedrate is interpolated linearly during the movement programmed in the block.



N10 G01 X150 F400
N20 G193 X250 F200
N30 X350

6.

TECHNOLOGICAL FUNCTIONS
Feedrate related functions



CNC 8060
CNC 8065

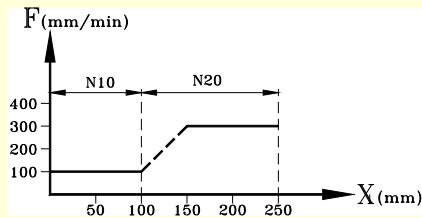
(REF. 1402)

Considerations

Adapting the feedrate (G108 and G109) is only available when the manufacturer has set the CNC to operate with either trapezoidal or square-sine (bell shaped) acceleration. Feedrate interpolation (G193) is only available when the manufacturer has set the CNC to operate with linear acceleration. The type of acceleration active at the CNC may be consulted in general machine parameter SLOPE TYPE.

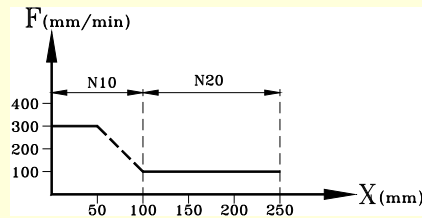
By default, the CNC applies the most restrictive feedrate adaptation in each situation without exceed the feedrate defined for each block. In other words, the CNC applies G108 to raise the feedrate and G109 to lower it.

Raise the feedrate, G108.



N10 G01 X100 F100
N20 X250 F300

Lower the feedrate, G109.



N10 G01 X100 F300
N20 X250 F100

Properties of the functions

Functions G108, G109 and G193 are not modal.

On power-up, after executing an M02 or M30, and after an EMERGENCY or RESET, the CNC applies function G108 to accelerate and G109 to decelerate.

6.

TECHNOLOGICAL FUNCTIONS
Feedrate related functions

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

6.2.3 Constant feedrate mode (G197/G196)

With these functions, it is possible to choose whether the feedrate at the tool center is maintained constant while machining or the feedrate at the cutting edge so when working with tool radius compensation, the programmed "F" corresponds to the contact point between the part and the tool.

Programming

The functions related to the feedrate type are:

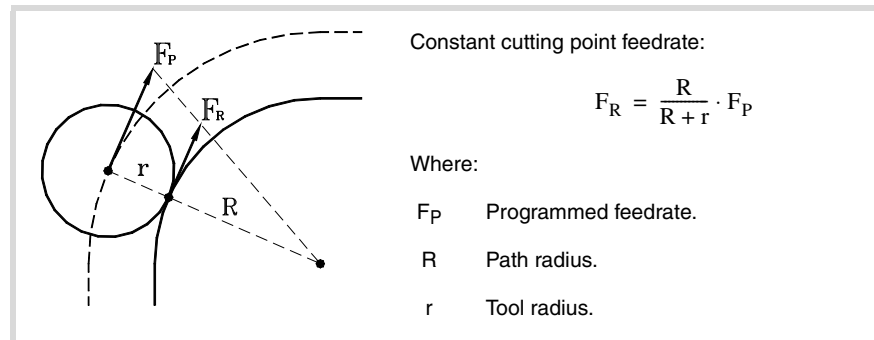
- G197 Constant tool center feedrate.
- G196 Constant cutting point feedrate.

These functions may be programmed anywhere in the program and they don't have to go alone in the block.

G197

Constant tool center feedrate

After executing G197, the CNC interprets that the programmed "F" corresponds to the tool center. This means that the feedrate at the cutting point increases on inside arcs and decreases on outside arcs.



G196

Constant cutting point feedrate

After executing G196, the CNC interprets that the programmed "F" corresponds to the contact point between the tool and the part. This results in a uniform part surface even on arcs.

Minimum radius for applying constant feedrate

Using the instruction "#TANGFEED RMIN [<radius>]" a minimum radius may be set so that constant tangential feedrate is only applied on those arcs whose radius is bigger than the minimum set. If it is not programmed or it is set to zero, the CNC will apply constant tangential feedrate on all the arcs.

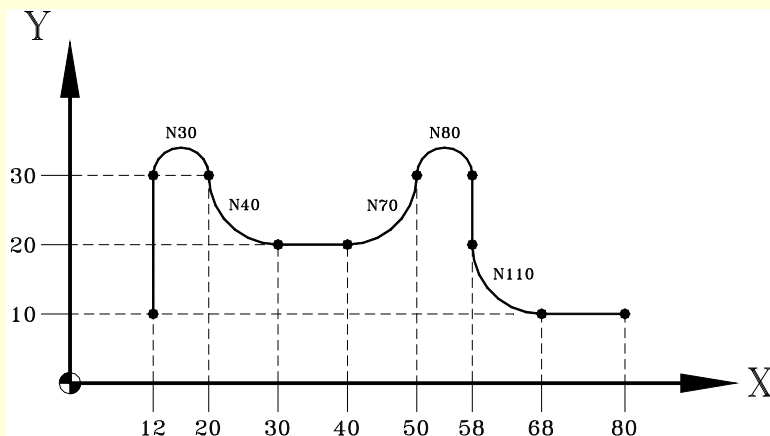
The minimum radius is applied from the next motion block on and it keeps its value after executing G197.

Properties of the functions

Functions G197 and G196 are modal and incompatible with each other.

On power-up, after executing an M02 or M30, and after an EMERGENCY or RESET, the CNC assumes function G197.

6.



N10 G01 G196 G41 X12 Y10 F600	(Tool radius compensation and constant tangential feedrate)
N20 G01 X12 Y30	
N30 G02 X20 Y30 R4	(Constant tangential feedrate)
N40 G03 X30 Y20 R10	(Constant tangential feedrate)
N50 #TANGFEED RMIN [5]	(Minimum radius = 5)
N60 G01 X40 Y20	
N70 G03 X50 Y30 R10	(Constant tangential feedrate)
N80 G02 X58 Y30 R4	(No constant tangential feedrate. $R_{PROGRAMMED} < R_{MINIMUM}$)
N90 G01 X58 Y20	
N100 #TANGFEED RMIN [15]	(Minimum radius = 15)
N110 G03 X68 Y10 R10	(No constant tangential feedrate. $R_{PROGRAMMED} < R_{MINIMUM}$)
N120 G01 X80 Y10	
N130 G01 G40 X100	
N140 M30	

6.

TECHNOLOGICAL FUNCTIONS
Feedrate related functions



CNC 8060
CNC 8065

(REF. 1402)

6.2.4 Cancellation of the % of feedrate override (G266)

G266 Feedrate override at 100%

This function sets the feedrate override at 100%, which can neither be changed by selector switch on the operator panel nor via PLC.

Function G266 only affects the block where it has been programmed, therefore, it only makes sense to add it to a block that defines a movement (motion block).

6.

TECHNOLOGICAL FUNCTIONS

Feedrate related functions



CNC 8060
CNC 8065

(REF. 1402)

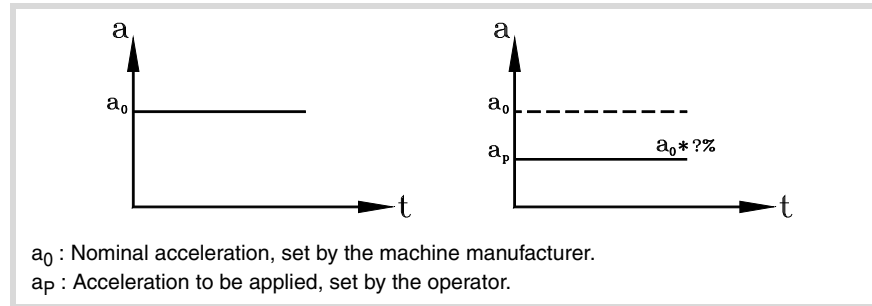
6.2.5 Acceleration control (G130/G131)

These functions allow to change the acceleration and deceleration of the axes and spindles.

Programming

The functions related to acceleration control are:

- G130 Percentage of acceleration to be applied per axis or spindle.
 G131 Percentage of acceleration to be applied, global.



G130

Percentage of acceleration to be applied per axis or spindle.

The percentage of acceleration to be applied per axis or spindle is set by G130 followed by the axes and spindles together with the percentage to be applied to each of them.

The acceleration values to be applied must be integers (not decimals).

```
...
G00 X0 Y0
G01 X100 Y100 F600
G130 X50 Y20          (Acceleration on the X axis = 50%)
                      (Acceleration on the Y axis = 20%)

G01 X0
G01 Y0
G131 100 X50 Y80     (Restore 100% of acceleration on all the axes)
                      (Movement to point X=50 Y=80)
...
```

G131

Percentage of acceleration to be applied, global

The percentage of acceleration to be applied to all the axes and spindles is set by G131 followed by the new acceleration value to be applied.

The acceleration values to be applied must be integers (not decimals).

When added to a motion block, the new values will be assumed before executing the move.

Considerations

The #SLOPE instruction determines the influence of the values defined with these values.

- In rapid positioning (G00)
- In the acceleration or deceleration stage.
- In the jerk of the acceleration or deceleration stages.

The programmed percentages are absolute, in other words, programming a 50% twice means that 50% will be applied, not 25%.

6.

TECHNOLOGICAL FUNCTIONS
 Feedrate related functions

FAGOR 

CNC 8060
 CNC 8065

(REF. 1402)

Properties of the functions

Functions G130 and G131 are modal and incompatible with each other.

On power-up, after an M02, M30, EMERGENCY or a RESET, the CNC restores 100% of acceleration for all the axes and spindles.

6.**TECHNOLOGICAL FUNCTIONS**
Feedrate related functions

CNC 8060
CNC 8065

(REF. 1402)

6.2.6 Jerk control (G132/G133)

The jerk of axes and spindles may be modified with these functions.

Programming

The functions associated with jerk control are:

G132	Percentage of jerk to be applied per axis or spindle.
G133	Percentage of jerk to be applied, global.

G132

Percentage of jerk to be applied per axis or spindle.

The percentage of jerk to be applied per axis or spindle is set by G132 followed by the axes and spindles together with the new jerk to be applied to each of them.

The jerk values to be applied must be integers (not decimals).

```
G00 X0 Y0
G01 X100 Y100 F600
G132 X20 Y50          (Jerk on the X axis = 20%)
                      (Jerk on the Y axis = 50%)

G01 X0
G01 Y0
G133 100 X50 Y80     (Restore 100% of jerk on all the axes. Movement to point X=50 Y=80)
```

G133

Percentage of jerk to be applied, global

The percentage of jerk to be applied to all the axes and spindles is set by G133 followed by the new jerk value to be applied.

The jerk values to be applied must be integers (not decimals).

When added to a motion block, the new jerk values will be assumed before executing the move.

Considerations

The #SLOPE instruction determines whether the new percentages are to be applied or not on to rapid traverse movements (G00).

The programmed percentages are absolute, in other words, programming a 50% twice means that 50% will be applied, not 25%.

Properties of the functions

Functions G132 and G133 are modal and incompatible with each other.

On power-up, after an M02, M30, EMERGENCY or a RESET, the CNC restores 100% of jerk for all the axes and spindles.

6.

6.2.7 Feed-Forward control (G134)

Feed-Forward control may be used to reduce the amount of following error (axis lag).

Feed-forward may be applied via machine parameters and via PLC as well as by program. The value defined by PLC will be the one with the highest priority and the one defined by the machine parameters will have the lowest priority.

Programming

G134

Percentage of Feed-Forward to be applied

The Feed-Forward percentage is set by G134 followed by the axes and the new percentage of Feed-Forward to be applied to each axis.

The Feed-forward values to be applied may be defined with up to two decimals.

```
G134 X50.75 Y80 Z10    (Percentage of Feed-Forward to be applied:)
                        (On the X axis = 50.75%)
                        (On the Y axis = 80%)
                        (On the Z axis = 10%)
```

Considerations

The maximum Feed-Forward value to be applied is 120%.

The programmed percentages are absolute, in other words, programming a 50% twice means that 50% will be applied, not 25%.

The value defined with G134 prevails over those defined in the machine parameters, but not over the one defined by PLC.

Properties of the functions

Function G134 is modal.

On power-up, after an M02 or M30, EMERGENCY or a RESET, the CNC restores the Feed-Forward set by the machine manufacturer for each axis.

Variable to define the feed-forward from the PLC

The write variable (V.) `PLCFFGAIN.Xn` may be used to set the feed-forward for each axis from the PLC. The value defined by this variable prevails over the ones defined by machine parameters or by program.

Setting this variable with a negative value cancels its effect (a zero value is also valid). This variable is initialized neither by a reset nor when validating the parameters.

6.

6.2.8 AC-Forward control (G135)

AC-Forward control may be used to improve system response in acceleration changes and reduce the amount of following error (axis lag) on the acceleration and deceleration stages.

AC-forward may be applied via machine parameters and via PLC as well as by program. The value defined by PLC will be the one with the highest priority and the one defined by the machine parameters will have the lowest priority.

Programming

G135

Percentage of AC-Forward to be applied

The AC-Forward percentage is set by G135 followed by the axes and the new percentage of AC-Forward to be applied to each axis.

The AC-forward values to be applied may be defined with up to one decimal.

G135 X55.8 Y75 Z110	(Percentage of AC-Forward to be applied:)
	(On the X axis = 55.8%)
	(On the Y axis = 75%)
	(On the Z axis = 110%)

Considerations

The maximum AC-Forward value to be applied is 120%.

The programmed percentages are absolute, in other words, programming a 50% twice means that 50% will be applied, not 25%.

The value defined with G135 prevails over those defined in the machine parameters, but not over the one defined by PLC.

Properties of the functions

Function G135 is modal.

On power-up, after an M02 or M30, EMERGENCY or a RESET, the CNC restores the AC-Forward set by the machine manufacturer for each axis.

Variable to define the AC-forward from the PLC

The write variable (V.) A . PLCACGAIN . X_n may be used to set the AC-forward for each axis from the PLC. The value defined by this variable prevails over the ones defined by machine parameters or by program.

Setting this variable with a negative value cancels its effect (a zero value is also valid). This variable is initialized neither by a reset nor when validating the parameters.

6.

TECHNOLOGICAL FUNCTIONS
Feedrate related functions

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

6.3 Spindle speed (S)

The spindle speed is selected by program using the spindle name followed by the desired speed. The speeds of all the spindles of the channel may be programmed in the same block. See chapter "7 *The spindle. Basic control.*".

```
S1000
S1=500
S1100 S1=2000 S4=2345
```

The programmed speed stays active until another value is programmed. On power-up, after executing an M02 or M30, and after an emergency or reset, the spindles assume a ·0· speed.

The speed may be programmed in rpm or in m/min (feet/min), depending on whether G197 or G196 is active. The default units are rpm.

Spindle start and stop

Defining a speed does not imply starting the spindle. The startup is defined using the following auxiliary functions.

- M03 - Starts the spindle clockwise.
- M04 - Starts the spindle counterclockwise.
- M05 - Stops the spindle.

Maximum speed

The maximum turning speed in each range (gear) is limited by the machine manufacturer. When programming a higher turning speed, the CNC limits its value to the maximum allowed by the active range (gear). The same thing occurs when trying to exceed the maximum limits using the "+" and "-" keys of the operator panel or doing it via PLC or by program.

Speed override

The programmed "S" speed may be varied between 50% and 120% using the "+" and "-" of the operator panel or via PLC. However the maximum and minimum variation may be different depending on how the machine manufacturer has set [A.M.P. "MINOVR" and "MAXOVR"].

Likewise, the incremental step associated with the "+" and "-" keys of the Operator Panel to change the programmed spindle speed "S" will be 10; but this value may be different depending on the setting of axis machine parameter ["STEPOVR"]

During threading operations, the programmed speed cannot be overridden and it will be set at 100% of the programmed "S" speed.

6.

6.4 Tool number (T)

The "T" code identifies the tool to be selected. The tools may be in a magazine managed by the CNC or in a manual magazine (referred to as ground tools).

The programming format is T<0-4294967294> and it can be programmed using parameters and arithmetic expressions. In these cases, by default, the value calculated is rounded up to an integer. If the result is negative, the CNC will issue the pertinent error message.

Definition

To select a tool, it must be previously defined. To do that, the CNC offers a table where the user may define the data for each tool. On the other hand, when having a magazine managed by the CNC, one must define the magazine position occupied by each tool. To do that, the CNC offers a table where the user may define the position of each tool. The table data may be defined:

- Manually from the CNC's front panel (as described in the Operating Manual).
- Via program, using the associated variables (as described in the relevant chapter of this manual).

Select a tool

The tool required for machining may be selected by program using the "T<n>" code where <n> is the tool number.

- On a lathe, the "T" code selects the tool in the tool holder.

Example on a lathe model.

```
N10 ...
N20 T1      (The CNC selects tool T1 in the turret)
N30 ...    (The CNC loads tool T1 into the spindle)
N40 ...
N50 T2      (The CNC selects tool T2 in the turret)
```

- On a milling machine, The "T" code only selects the tool. After selecting a tool, function M06 must be programmed to load it into the spindle. Loading/unloading is carried out depending on the subroutine associated with the M06 (if so defined by the machine manufacturer).

Example on a mill model.

```
N10 ...
N20 T1      (The CNC selects tool T1 in the tool magazine)
N30 M06    (The CNC loads tool T1 into the spindle)
N40 ...
N50 T2      (The CNC selects tool T2)
N60 ...
N70 ...
N80 ...
N90 M06    (The CNC loads tool T2 into the spindle)
N100 ...
N110 M30
```

6.

TECHNOLOGICAL FUNCTIONS
Tool number (T)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

Loading and unloading a tool in the magazine

To load the tools into the magazine, the magazine must be in load mode. To unload the tools from the magazine, the magazine must be in unload mode. The tools are loaded from ground to the magazine going through the spindle and are unloaded to ground going through the spindle.

The magazine's work mode is set by variable `V.[n].TM.MZMODE` where `n` is the channel number. Depending on the value of the variable, the manager will assume one of the following work modes.

Value	Meaning
0	Normal mode (by default and after Reset).
1	Magazine loading mode.
2	Magazine unloading mode.

Once the magazine is load or unload mode, the operation is carried out from the program using the code `Tn` where `n` is the tool number. Once the tools have been loaded or unloaded, the magazine must be set to normal mode (value of `0`).

```
V.[1].TM.MZMODE = 1
T1 M6
T2 M6
...
V.[1].TM.MZMODE = 0
```

Loading a tool in a specific magazine position

Some tools, due to their characteristics (size, weight, etc.) must be placed in a specific magazine position.

The command `POSn` defines the magazine position for the tool. It must always be programmed in the same block as `Tn`.

```
V.[1].TM.MZMODE = 1
T3 M6 POS24
    (Places tool 3 in magazine position 24)
...
V.[1].TM.MZMODE = 0
```

The magazine position can only be selected when the magazine is in load mode. Otherwise, it issues the relevant error message.

Loading a tool in a system with several magazines

When using more than one tool magazine, one must indicate in which one of them the tool is to be loaded using the code `MZn`, where `n` indicates the magazine number. It must always be programmed in the same block as `Tn`.

```
T1 MZ1 M6
    (Place tool 1 in the first magazine)
T8 MZ2 POS17 M6
    (Place tool 8 in position 17 of the second magazine)
```

Considerations. The tool and the M06 function.

The machine manufacturer may have associated a subroutine with the "T" code, that will be executed automatically when selecting the tool. If the M06 has been included in this subroutine, the tool will be loaded into the spindle when executing the "T" code.

6.

Positioning a turret magazine.

The CNC allows positioning the turret in a particular position whether there is a tool in the indicated position or not. If the selected position contains a tool, the CNC assumes it as programmed tool; otherwise, the CNC assumes T0.

Programming.

When programming this instruction, you must define the magazine number and the position to be selected on the turret. The new position of the turret may be defined either incrementally, defining the number of positions to rotate and the rotating direction, or absolutely, defining the position to be reached.

The programming format is the following; the list of arguments appears between curly brackets and the optional ones between angle brackets.

```
#ROTATEMZ{mz} P{pos}
#ROTATEMZ{mz} {±n}
```

{mz}	Magazine number.
{pos}	Absolute turret position.
{±n}	Number of positions to rotate; the sign indicates the rotating direction, positive or negative. If only the sign is programmed, the turret rotates one position.

```
#ROTATEMZ1 P5
  (Absolute position; select position 5).
#ROTATEMZ2 +3
  (Incremental positioning; rotate the turret 3 positions in positive direction).
#ROTATEMZ1 -7
  (Incremental positioning; rotate the turret 7 positions in negative direction).
#ROTATEMZ2 +
  (Incremental positioning; rotate the turret 1 position in positive direction).
#ROTATEMZ1 -
  (Incremental positioning; rotate the turret 1 position in negative direction).
```

6.

TECHNOLOGICAL FUNCTIONS
Tool number (T)

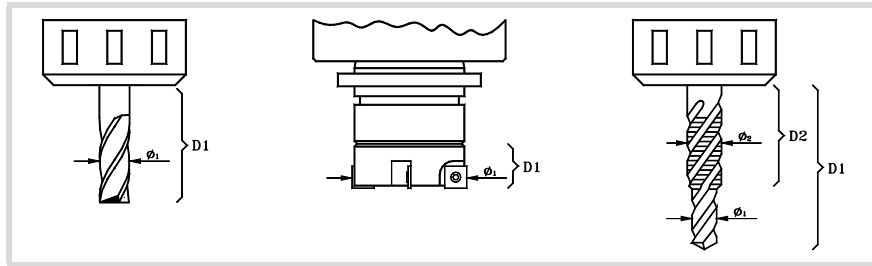
FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

6.5 Tool offset number (D)

The tool offset contains the tool dimensions. Each tool may have several offsets associated with it in such a way that when using combined tools having parts with different dimensions, a different offset number will be used for each of those parts.



When activating a tool offset, the CNC assumes the tool dimensions defined for that offset, therefore when working with tool radius or length compensation, the CNC will apply those dimensions for compensating the path.

Defintion

To activate an offset, it must be previously defined. To do that, the CNC offers a portion of the tool table where the user may define several offsets. The table data may be defined:

- Manually from the CNC's front panel (as described in the Operating Manual).
- Via program, using the associated variables (as described in the relevant chapter of this manual).

The offsets are only associated with the tool for which they have been defined. This means that when activating a tool offset, the offset of the corresponding active tool will be activated.

Activation

Once the tool offsets have been defined in the table, they may be selected by program using the "D<n>" code where <n> is the number of the offset to be applied. The offset number may also be defined using a parameter or an arithmetic expression.

If no tool offset is programmed, the CNC assumes tool offset D1.

	N10 ...	
	N20 T7 D1	(Select tool T7 and tool offset D1)
	N30 M06	(Load tool T7 into the spindle)
	N40 F500 S1000 M03	
	N50 ...	(Operation 1)
	N60 D2	(Select tool offset D2 of T7)
	N70 F300 S800	
	N80 ...	(Operation 2)
	N90 ...	

Only one tool offset may be active at a time; therefore, activating a tool offset will cancel the previous one. Programming "D0" will cancel the active offset.

6.

TECHNOLOGICAL FUNCTIONS
Tool offset number (D)



CNC 8060
CNC 8065

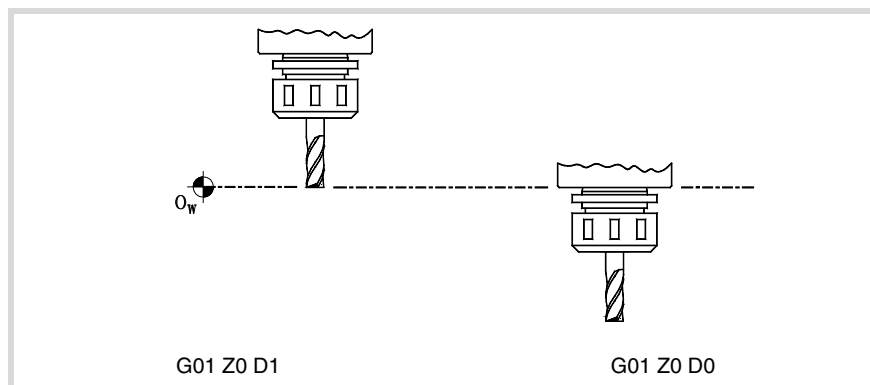
(REF. 1402)

N10 ...	
N20 T1 M06	(Select and load tool T1. Offset D1 is activated by default)
N30 F500 S1000 M03	
N40 ...	(Operation 1)
N50 T2	(Prepare tool T2)
N60 D2	(Select tool offset D2 for tool T1)
N70 F300 S800	
N80 ...	(Operation 2)
N90 M6	(loading tool T2 with its offset D1)
N100 F800 S1200 M03	
N110 ...	(Operation 3)
N120 ...	

Considerations

Activating the tool offset also activates tool length compensation. This compensation is also activated after a tool change because it "D1" is assumed after the change (if another one has not been programmed).

Canceling the tool offset with "D0" also cancels tool length and radius compensation.



6.6 Auxiliary (miscellaneous) functions (M)

Auxiliary "M" functions are related to the overall CNC program execution and the control of the various devices of the machine such as spindle gear change, coolant, tool changes and so on.

Programming

Up to 7 "M" functions may be programmed in a block. The programming format is M<0 - 65535>, and it can be programmed using parameters and arithmetic expressions. In these cases, by default, the value calculated is rounded up to an integer. If the result is negative, the CNC will issue the pertinent error message.

Execution

Depending on how they have been set by the machine manufacturer ("M" function table):

- The "M" functions will be executed before or after the movement of the block where they were programmed.

When setting an "M" to be executed after the move of the block, depending on the active function G05 or G07:

- | | |
|-----|--|
| G05 | The "M" function is executed with the theoretical end of the movement (when the axes have not reached position). |
| G07 | The "M" function is executed with the real end of the movement (when the axes are already in position). |

- The CNC will wait or not for the confirmation that the "M" function has been executed before resuming program execution. If it has to wait for confirmation, it will have to be received before or after executing the movement of the block where it has been programmed.
- The "M" functions that have not been set in the table will be executed before the movement of the block where they have been programmed and the CNC will wait for the "M-done" confirmation before executing the movement of the block.

Certain "M" functions have a particular internal meaning in the CNC. The section on ["6.6.1 List of "M" functions"](#) in this chapter shows a list of these functions with their internal meaning for the CNC.

Associated subroutine

The "M" functions may have an associated subroutine that will be executed instead of the function.

If, within a subroutine associated with an "M" function, the same "M" function is programmed, this function will be executed, but not its associated subroutine.

6.

6.6.1 List of "M" functions

Program Interruption (M00/M01)

M00

Program stop.

Function M00 interrupts the execution of the program. It does not stop the spindle or initialize the cutting conditions.

The [CYCLE START] key of the operator panel must be pressed again in order to resume program execution.

This function should be set in the "M" function table so it is executed at the end of the block where it is programmed.

M01

Conditional program stop.

When the external conditional stop switch is active (PLC signal "M01 STOP"), it interrupts program execution. It does not stop the spindle or initialize the cutting conditions.

The [CYCLE START] key of the operator panel must be pressed again in order to resume program execution.

This function should be set in the "M" function table so it is executed at the end of the block where it is programmed.

Tool change (M06)

M06

Tool change.

The M06 function executes the tool change. The CNC will manage the tool change and update the table for the tool magazine.

This function should be set in the "M" table so it executes the subroutine corresponding to the tool changer installed on the machine.

6.

TECHNOLOGICAL FUNCTIONS
Auxiliary (miscellaneous) functions (M)



CNC 8060
CNC 8065

(REF. 1402)

6.7 Auxiliary functions (H)

Auxiliary "H" functions are used to send information out to the PLC. They differ from the "M" functions in that the "H" functions do not wait for confirmation that the function has been executed in order to go on executing the program.

Programming

Up to 7 "H" functions may be programmed in the same block. The programming format is H<0 - 65535>, and it can be programmed using parameters and arithmetic expressions. In these cases, by default, the value calculated is rounded up to an integer. If the result is negative, the CNC will issue the pertinent error message.

Execution

The auxiliary "H" functions are executed at the beginning of the block where they have been programmed.

6.**TECHNOLOGICAL FUNCTIONS**
Auxiliary functions (H)

CNC 8060
CNC 8065

(REF. 1402)

The CNC can have up to four spindles distributed between the various channels of the system. A channel may have one, several or no spindles associated with it.

Each channel can only control its spindles; it is not possible to start up or stop the spindles of another channel directly. Indirectly, the CNC can control the spindles of another channel using the instruction #EXBLK.

Multi-spindle channel

A multi-spindle channel is one that has two or more spindles. Via part-program or MDI, it is possible to indicate which spindle the commands are directed to; when not indicated, the commands are directed to the master spindle of the channel.

All the spindles of the channel may be running at the same time. Also, each of them may be in a different mode; turn in different directions, be in positioning mode, etc.

Master spindle of the channel

The master spindle is the main spindle of the channel. In general, whenever a channel has a single spindle, it will be its master spindle. When a channel has several spindles, the CNC will choose the master spindle according to the established criteria. See "[7.1 The master spindle of the channel](#)" on page 116.

7.1 The master spindle of the channel

The master spindle is the main spindle of the channel. It is the spindle that receives the commands when no specific spindle is mentioned. In general, whenever a channel has a single spindle, it will be its master spindle.

CNC criterion for selecting the master spindle after executing M02, M30 or after an emergency or a reset or restarting the CNC.

The selection of the master spindle in the channel depends on machine parameter MASTERSPDL. This parameter indicates whether the channel keeps the current master spindle or recovers its original master spindle after executing M02, M30 or after an emergency or reset or restarting the CNC.

MASTERSPDL	Meaning.
Temporary.	The channel recovers its original master spindle if it is free; otherwise, it selects as master the first spindle available in the original configuration.
Maintained.	The channel keeps the active master spindle.

When a channel does not keep its master spindle, on CNC power-up and after a reset, the channel assumes as master spindle the first spindle defined by the machine parameters of the channel (original master). If this spindle is parked or "handed out" to another channel, the channel assumes as master spindle the next spindle defined by the machine parameters and so on. If the channel does not have spindles of the original configuration (the one defined by the machine parameters) because they are parked or "handed out", it assumes as master spindle the first one of the current configuration that is not parked.

Exchanging spindles between channels.

When spindles are being exchanged between channels, the behavior of this parameter also depends on parameter AXISEXCH that sets whether the channel change of a spindle is temporary or permanent. If the current master spindle of the channel is a spindle "loaned" by another channel and its permission to change channels is temporary (AXISEXCH = Temporary), the spindle returns to its original channel.

Which one is the master spindle after executing M30?

When executing an M30, it follows the same criteria, but considering that after executing this function, the temporary spindle swaps are not undone; they are undone at the beginning of the next program. This means that the original master might not be available after executing an M30, but it will be at the beginning of the next program. In this situation, after an M30, the channel will temporarily assume a master spindle that will change at the beginning of the next program.

Which one is the master spindle after modifying the configuration of the channel?.

If no master spindle is indicated, after parking or swapping spindles, it assumes one according to the following criteria. In general, whenever a channel has a single spindle, it will be its master spindle.

- If the whole system only has one spindle, it will be the master spindle of the current channel.
- If a spindle is added to a channel that does not have one, it will be the master spindle.
- If a channel releases its master spindle and it has only one spindle left, this one will be its new master spindle.
- If a channel having two spindles but no master spindle releases one of them, the remaining one will be its master spindle.
- At first, in a channel with several spindles, the master spindle will be the one configured by machine parameters.

7.

THE SPINDLE. BASIC CONTROL.
The master spindle of the channel



CNC 8060
CNC 8065

(REF. 1402)

- If two or more spindles remain in a channel and none of the previous rules can be applied, it applies the following criteria.

If any of the spindles is the original master, it is assumed as master spindle. If it is parked, it selects the next spindle from the original configuration (those defined by the machine parameters) and so on.

If no spindles from the original configuration are available in the channel, it assumes as master spindle the first one of its current configuration. If it is parked, it selects the next spindle and so on.

Which one is the master spindle after parking or unparking spindles?

The same treatment is applied as when modifying the configuration of the channel.

7.

THE SPINDLE. BASIC CONTROL.

The master spindle of the channel

FAGOR 

**CNC 8060
CNC 8065**

(REF. 1402)

7.1.1 Manual selection of a master spindle

Selecting a new master spindle

Whenever a channel has a single spindle, it will be its master spindle. When a channel has several spindles, the CNC will choose the master spindle according to the criteria described earlier. However, a different master spindle may be selected via MDI or via part-program using the #MASTER instruction.

Programming format.

```
#MASTER sp
```

sp Spindle name.

```
#MASTER S  
#MASTER S2
```

Canceling the master spindle

The master spindle may be selected at any time. If the master spindle changes channels, the channel will select a new master spindle according to the criterion described earlier.

On power-up, after executing an M02 or M30, and after an emergency or reset, the CNC acts as defined by the OEM (parameter MASTERSPDL).

7.

THE SPINDLE. BASIC CONTROL.
The master spindle of the channel

7.2 Spindle speed

The spindle speed is selected by program using the spindle name followed by the desired speed. The speeds of all the spindles of the channel may be programmed in the same block. It is not possible to program the speed of a spindle that is not in the channel.

The programmed speed stays active until another value is programmed. On power-up, after executing an M02 or M30, and after an emergency or reset, the spindles assume a ·0· speed.

Programming format

The spindle may have any name in the S, S1...S9 range. There is no need to program the "=" sign for the "S" spindle.

Sn={vel}

S{vel}

Sn Spindle name.

S Spindle "S".

{vel} Turning speed.

```
S1000
S1=500
S1100 S1=2000 S4=2345
```

The speed may be programmed in rpm or in m/min (feet/min), depending on whether G197 or G196 is active. The default units are rpm.

Spindle start and stop

Defining a speed does not imply starting the spindle. The startup is defined using the following auxiliary functions. See "[7.3 Spindle start and stop](#)" on page 122.

M03 - Starts the spindle clockwise.

M04 - Starts the spindle counterclockwise.

M05 - Stops the spindle.

Speed ranges (gears)

Each spindle may have up to 4 different ranges (gears). Each gear means a speed range for the CNC work in. The programmed speed must be within the active gear; otherwise, a gear change will be required. The CNC does not admit speeds higher than the one defined for the last gear.

The gear change may be automatic or manual. When the change is manual, the gear is selected with auxiliary functions M41 through M44. When the change is automatic, the CNC itself generates these functions according to the programmed speed. See "[7.4 Gear change.](#)" on page 124.

7.

THE SPINDLE. BASIC CONTROL.
Spindle speed

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

7.2.1 G192. Turning speed limit

Function G192 limits the spindle turning speed in both work modes; G96 and G97. This function is especially useful when working at constant cutting speed while machining large parts or when doing spindle maintenance work.

If function G192 is not programmed, the turning speed is limited by machine parameter G00FEED of the gear.

G192. Programming the turning speed limit

The turning speed limit is set by programming function G192 and then the maximum speed for each spindle. This function may be programmed while the spindle is running; in this case, the CNC will limit the speed to the new programmed value.

Programming format

The spindle may have any name in the S, S1...S9 range. There is no need to program the "=" sign for the "S" spindle.

```
G192 Sn={vel}
```

```
G192 S{vel}
```

{vel} Maximum spindle speed.

```
G192 S1000
G192 S1=500
```

The maximum turning speed is always set in RPM. It is possible to program using arithmetic parameters, variables or expressions.

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

Function G192 is modal.

The CNC cancels function G192 on power-up or after an emergency. After executing an M02 or M30, and after a reset, the CNC cancels maintains function G192.

7.

THE SPINDLE. BASIC CONTROL.
Spindle speed



CNC 8060
CNC 8065

(REF. 1402)

7.2.2 Constant surface speed



The following functions are oriented to lathe type machines. In order for Constant Surface Speed mode to be available, the machine manufacturer must have set one of the axis -face axis- (usually axis perpendicular to the shaft of the part).

The functions related to spindle speed programming may be used to select either Constant Surface Speed mode or Constant turning speed mode. Constant Surface Speed is only available at the master spindle of the channel.

G96 - Constant surface speed.

G97 - Constant turning speed.

At constant surface speed, the CNC changes the spindle speed as the perpendicular axis moves in order to maintain the cutting speed constant between the tool and the part, thus optimizing the machining conditions. When working at constant surface speed, it is recommended to limit by program the maximum turning speed that the spindle can reach. See "7.2.1 G192. Turning speed limit" on page 120.

G96. Constant surface speed.

The G96 function only affects the master spindle of the channel.

After executing G96, the CNC interprets that the spindle speeds programmed for the master spindle of the channel are in meters/minute (feet/minute). This work mode is activated when programming a new speed while G96 is active.

This function may be programmed anywhere in the program and it doesn't have to go alone in the block. It is recommended to program the speed in the same block as the G96 function. The spindle gear (range) must be selected in the same block or in a previous one.

G97. Constant turning speed

The G97 function affects all the spindles of the channel.

After executing G97, the CNC interprets that the spindle speeds programmed are in rpm and starts working at constant turning speed.

This function may be programmed anywhere in the program and it doesn't have to go alone in the block. It is recommended to program the speed in the same block as the G97 function; if not programmed, the CNC assumes as programmed speed the one the spindle is currently turning at. The gear may be selected at any time.

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

Functions G96 and G97 are modal and incompatible with each other.

The CNC assumes function G97 on power-up or after an emergency. After executing an M02 or M30, and after a reset, the CNC cancels maintains function G96 or G97 that was active before.

7.

THE SPINDLE. BASIC CONTROL.
Spindle speed

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

7.3 Spindle start and stop

A speed must be set in order to start up a spindle. The spindle start-up and stop are defined using the following auxiliary functions.

M03 - Start the spindle clockwise.

M04 - Start the spindle counterclockwise.

M05 - Stop the spindle.

These functions are modal and incompatible with each other and with function M19.

M03/M04. Start the spindle clockwise/counterclockwise.

Function M03 starts the spindle clockwise and function M04 counterclockwise. These functions should be set in the "M functions" table so they are executed at the end of the block where it is programmed.

These functions may be defined together with the programmed speed or in a separate block. If the block where they are programmed does not mention any spindle, they will be applied to the master spindle of the channel.

```
S1000 M3
  (The spindle "S" starts clockwise at 1000 rpm)
S1=500 M4
  (The spindle "S1" starts counterclockwise at 500 rpm)
M4
  (The master spindle starts counterclockwise)
```

If several spindles are programmed in a single block, functions M3 and M4 apply to all of them. To start the spindles in different directions, define next to each M function the spindle it is associated with, as follows.

M3.S / M4.S M3 or M4 associated with the spindle S.

```
S1000 S2=456 M3
  (Spindle "S" turning at 1000 rpm and S2 at 456 rpm, both clockwise)
M3.S S1000 S2=456 M4.S2
  (The spindle "S" turns clockwise at 1000 rpm)
  (The spindle "S2" turns counterclockwise at 456 rpm)
```

M05. Stop the spindle.

Function M05 stops the spindle.

To stop a spindle, define next to the M5 the spindle it is associated with, as follows. If it does not mention any spindle, it applies to the master spindle.

M5.S Function M5 associated with the spindle S.

```
S1000 S2=456 M5
  (Stops the master spindle)
M5.S M5.S2 S1=1000 M3.S1
  (Stops the spindles "S" and "S2")
  (Spindle "S1" turns clockwise)
```

Turning direction preset in the tool table.

It is possible to set a preset turning direction for each tool. The value is defined in tool table.

When assigning a turning direction in the table, the CNC will verify, during execution, that the turning direction in the table is the same as the one programmed (M03/M04). If the two directions are not the same, the CNC will display the corresponding error message. The CNC verifies this every time an M03, M04 or M06 is programmed.

7.

Knowing which is the preset turning direction.

The turning direction preset for each tool may be consulted in the tool table; the one for the active tool can also be consulted with a variable.

(V.)G.SPDLTURDIR

This variable returns the preset turning speed of the active tool. ·0· value if there is no preset turning direction, ·1· if it is M03, and ·2· if it is M04.

Canceling the preset turning direction temporarily.

The preset turning speed of the active tool may be temporarily canceled from the part-program. This is done by setting the variable V.G.SPDLTURDIR = 0.

When changing a tool, this variable will take the corresponding value according to what has been set in the tool table.

7.

THE SPINDLE. BASIC CONTROL.

Spindle start and stop

FAGOR 

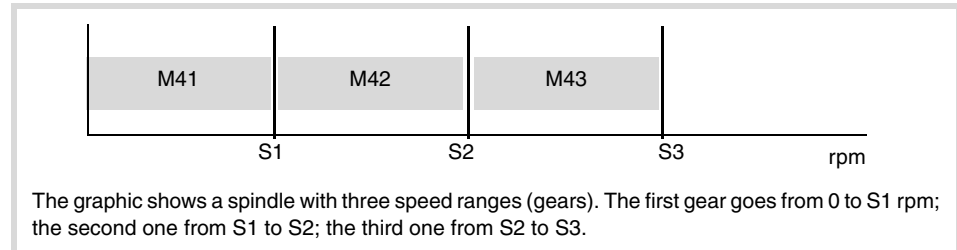
CNC 8060
CNC 8065

(REF. 1402)

7.4 Gear change.

Each spindle may have up to 4 different ranges (gears). Each gear means a speed range for the CNC work in. The programmed speed must be within the active gear; otherwise, a gear change will be required.

The gear change may be automatic or manual. When the change is manual, the gear is selected with auxiliary functions M41 (gear 1) through M44 (gear 4). When the change is automatic, the CNC itself generates these functions according to the programmed speed.



The configuration of the speed ranges or gears (automatic or manual change, maximum speed for each gear, etc.) is defined by the machine manufacturer. See ["How to know the gear configuration of a spindle."](#) on page 125.

Manual gear change

When the change is manual, the gear is selected with auxiliary functions M41 through M44.

- M41 - Selects gear ·1·.
- M42 - Selects gear ·2·.
- M43 - Selects gear ·3·.
- M44 - Selects gear ·4·.

These functions may be defined together with the programmed spindles or in a separate block. If the block where they are programmed does not mention any spindle, they will be applied to the master spindle of the channel.

```
S1000 M41
S1=500 M42
M44
```

If several spindles are programmed in a single block, the functions apply to all of them. To apply different gears to the spindles, define next to each M function the spindle it is associated with, as follows.

M41.S Function M41 associated with the spindle S.

```
S1000 S2=456 M41
(Gear 1 for spindle "S" and S2)
M41.S M42.S3
(Gear 1 for spindle "S")
(Gear 2 for spindle "S3")
```

Influence of the reset, turning the CNC off and of the M30.

Gears (speed ranges) are modal. On power-up, the CNC assumes the gear defined by the machine manufacturer. After executing an M02 or M30 and after an emergency or a reset, the CNC maintains the active gear.

Knowing which is the active gear.

The window for the M functions in the jog and automatic modes shows which is the active gear; if none is shown, it means that gear ·1· is the active one.

7.

THE SPINDLE. BASIC CONTROL.
Gear change.



CNC 8060
CNC 8065

(REF. 1402)

The active gear may be consulted using the following variable.

(V.)[n].G.MS[i]

Variable that can only be read from the PRG and PLC.

The variable indicates the status of the auxiliary Mi function This variable returns a ·1· if it is active and a ·0· if not.

Gear change on Sercos spindles.

When using Sercos spindles, functions M41-M44 also involve changing the drive's velocity set.

How to know the gear configuration of a spindle.

Both the type of gear change (automatic or manual) and the maximum speed for each gear are defined by the machine manufacturer. The configuration may be checked directly in the machine parameter table or using the following variables.

How to know whether the spindle has an automatic tool changer or not.

(V.)SP.AUTOGEAR.Sn

Variable that can only be read from the PRG and PLC.

This variable indicates whether the spindle Sn has an automatic gear change or not. This variable returns a ·1· if the gear changer is automatic and a ·0· if it is manual.

Number of gears available

(V.)SP.NPARSETS.Sn

Variable that can only be read from the PRG and PLC.

This variable indicates the number of gears defined for the spindle Sn.

Maximum speed for each gear

(V.)SP.G00FEED[g].Sn

Variable that can only be read from the PRG and PLC.

This variable indicates the maximum speed of the spindle Sn in the gear g.

Gear active by default.

(V.)SP.DEFAULTSET.Sn

Variable that can only be read from the PRG and PLC.

This variable indicates which is the gear assumed by the CNC on power-up for the spindle Sn.

7.

THE SPINDLE. BASIC CONTROL.
Gear change.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

7.5 Spindle orientation.



This work mode is only available on machines having a rotary encoder installed on the spindle.

The spindle orientation is defined with function M19. This function stops the spindle and it positions it at an angle defined by parameter "S". See *"How is positioning carried out"* on page 127.

After executing function M19, the spindle no longer works in speed mode and it switches to positioning mode. This mode stays active until the spindle is started up again in speed mode with M3/M4.

Programming a spindle orientation

Every spindle positioning move requires an M19 and the positioning angle. If no angle is defined, the CNC orients the master spindle at 0°.

Even if function M19 is active, if an "S" value is defined without an M19, the CNC assumes it as the new turning speed for the next time the spindle is turned on in speed mode using functions M03/M04.

Programming format (1).

When executing function M19, the CNC interprets that the value entered with the "Sn" code indicates the angular position of the spindle. If several spindles are programmed in a single block, function M19 applies to all of them.

M19 S{pos}

S{pos} Spindle to be oriented and positioning angle.
The angle is defined in degrees.

```
M19 S0
  (Positioning of spindle S at 0°)
M19 S2=120.78
  (Positioning of spindle S2 at 120.78°)
M19 S1=10 S2=34
  (Positioning of spindle S1 at 10° and S2 at 34°)
```

This angular position is programmed in degrees and it is always assumed in absolute coordinates, thus not being affected by functions G90/G91. To do the positioning, the CNC calculates the module (between 0 and 360°) of the programmed value.

Programming format (2). Positioning of the spindle at 0°

To orient the spindle to the ·0· position, it may also be programmed by defining, next to the M19, the spindle to be oriented. If no spindle is defined, the CNC assumes that the master spindle is the one to be oriented.

M19.S

S Spindle to be oriented at 0°.

```
M19.S4
  (Positioning of spindle S4 at 0°)
M19
  (Positioning of the master spindle at 0°)
```

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

M19 is modal and incompatible with M03, M04 and M05.

On power-up, after executing an M02 or M30, and after an emergency or reset, the CNC sets the spindle in speed mode with function M05.

7.

How is positioning carried out

When executing function M19, the CNC behaves as follows.

- 1 The CNC stops the spindle (if it was turning).
- 2 The spindle no longer works in speed mode and it switches to positioning mode.
- 3 If it is the first time the M19 is executed, the CNC homes the spindle (home search).
- 4 It positions the spindle at 0° or at the angular position defined by the "S" code (if it has been programmed). To do that, it will calculate the module (between 0 and 360°) of the programmed value and the spindle will reach that position.

```
N10 G97 S2500 M03
    (The spindle turns at 2500 RPM)
N20 M19 S50
    (Spindle in positioning mode. The spindle orients at 50°)
N30 M19 S150
    (Positioning at 150°)
N40 S1000
    (New spindle speed. The spindle stays in positioning mode)
N50 M19 S-100
    (Positioning at -100°)
N60 M03
    (Spindle controlled in speed. The spindle turns at 1000 RPM)
N70 M30
```

It executes the M19 for the first time

When executing the M19 function for the first time, it homes the spindle. The M19 functions programmed afterwards only orient the spindle. To home the spindle again, use function G74.

7.

THE SPINDLE. BASIC CONTROL.
Spindle orientation.

FAGOR 

**CNC 8060
CNC 8065**

(REF. 1402)

7.5.1 The turning direction for spindle orientation

The turning direction for positioning may be set with function M19; if not defined, the CNC applies a turning direction by default. Each spindle may have a different turning direction by default.

Turning direction by default

If no turning direction has been defined, the CNC behaves as follows. If when executing function M19, there was an M3 or M4 active, even if the speed is zero, this function will set the spindle orienting direction. If no M3 or M4 is active, the turning direction is set depending on machine parameter `SHORTESTWAY`.

- If it is a `SHORTESTWAY` spindle, it positions via the shortest way.
- If it is not a `SHORTESTWAY` type spindle, it positions in the same direction as the last spindle movement.

Turning direction defined set by the user.

The positioning direction programmed next to the M19 is applied to all the spindles programmed in the block. If the turning direction has not been programmed, each spindle will turn in the direction set earlier; if none has been defined, it will assume the turning direction by default.

The programmed turning direction is maintained until another one is programmed.

Programming format (1). Turning direction for all the spindles programmed.

M19.POS S{pos}

M19.NEG S{pos}

POS Positioning in the positive direction

NEG Positioning in the negative direction

S{pos} Spindle to be oriented and positioning angle.

```
M19.NEG S120 S1=50
```

(The negative direction is applied to spindle "S" and "S1")

```
M19.POS S120 S1=50
```

(The positive direction is applied to spindle "S" and "S1")

If no spindle is defined, the CNC orients the master spindle at 0° in the indicated direction.

When programming the orienting direction for a `SHORTESTWAY` type spindle, the programmed direction will be ignored.

Programming format (2). Turning direction for one spindle only.

Since several spindles may be programmed in a block, it is possible to apply the turning direction just to one of them. The rest of the spindles will turn in their active direction.

M19.POS.S S{pos} S{pos}

M19.NEG.S S{pos} S{pos}

POS.S Spindle that orients in the positive direction.

NEG.S Spindle that orients in the negative direction.

S{pos} Spindle to be oriented and positioning angle.

```
M19.NEG.S1 S1=100 S34.75
```

(The negative direction is applied to spindle "S1")

7.

How to know the type of spindle.

The type of spindle may be checked directly in the machine parameter table or using the following variables.

(V.)SP.SHORTESTWAY.Sn

Variable that can only be read from the PRG and PLC.

This variable indicates whether the spindle Sn orients via the shortest way. This variable returns a -1- if so.

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

On power-up, after executing an M02 or M30, and after an emergency or reset, the CNC cancels the turning direction set by the user.

7.

THE SPINDLE. BASIC CONTROL.

Spindle orientation.



CNC 8060
CNC 8065

(REF. 1402)

7.5.2 M19 function with an associated subroutine.

Function M19 may have a subroutine associated with them that the CNC executes instead of the function. If, within a subroutine associated with an "M" function, the same "M" function is programmed, The CNC will executed it; but not its associated subroutine.

Although the function may affect more than one spindle in the same block, the CNC only executes the subroutine once. The following behavior is applicable to all positioning movements programmed in the block.

When programming function M19 and a positioning (M19 S), the CNC executes the subroutine associated with the function and ignores the positioning movement. The CNC executes the positioning when executing the M19 function from the subroutine.

- If inside the subroutine, the M19 function is not accompanied by a positioning move (S), the CNC executes the positioning programmed in the calling block.
- If inside the subroutine, the M19 function is accompanied by a positioning move (S), the CNC executes this positioning.

The same criterion applies to the moving direction. If the turning direction is programmed with the M19 function that calls the subroutine, the turning direction is applied in the M19 programmed inside the subroutine if it another one has not been specified.

7.

THE SPINDLE. BASIC CONTROL.
Spindle orientation.



CNC 8060
CNC 8065

(REF. 1402)

7.5.3 Positioning speed

It is possible to set the spindle positioning (orienting) speed; if it is not set, the CNC assumes the one set by machine parameter `REFEED1` as the positioning speed. Each spindle may have a different positioning speed.

Programming format.

The positioning speed is set as follows.

S.POS={vel}

S Spindle name.

{vel} Positioning speed.

```
M19 S.POS=120 S1.POS=50
      (Positioning of spindle "S" at 120 rpm and S1 at 50 rpm)
```

The positioning speed is given in rpm.

Knowing the active positioning speed.

The active positioning speed for the CNC may be consulted using the following variable.

(V.)SP.SPOS.Sn

Variable that can only be read from the PRG and PLC.

This variable indicates the active positioning speed for the spindle Sn.

7.

THE SPINDLE. BASIC CONTROL.
Spindle orientation.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

7.6 M functions with an associated subroutine.

M3, M4, M5, M19 and M41 to M44 may have a subroutine associated with them that the CNC executes instead of the function. Although a function may affect more than one spindle in the same block, the CNC only executes the subroutine once per block.

If, within a subroutine associated with an "M" function, the same "M" function is programmed, The CNC will executed it; but not its associated subroutine. When a spindle M function is programmed inside the subroutine, it will be applied to the spindles programmed in the subroutine block itself. If the subroutine block does not indicate which spindles the function is for, the CNC will assume that it is for the spindles programmed in the subroutine calling block.

The CNC relates the functions with the spindles according to the following criterion, whether it is in the calling block or inside the subroutine.

- If the M function is assigned to a spindle (for example, M3, S), the CNC only applies the function to the indicated spindle.
- If functions M3 and M4 are not assigned to any spindle, the CNC applies them to all the spindles whose speed has been programmed in the block and have not been assigned to another M function. If there are no spindles with programmed speed, the CNC applies them to the master spindle.
- If function M19 is not assigned to any spindle, the CNC applies them to all the spindles whose speed has been programmed in the block and have not been assigned to another M function.
- If functions M5 and M41 to M44 are not assigned to any spindle, the CNC applies them to the master spindle.

Inside the subroutine, the CNC applies this criterion to all the programmed spindle M functions, not only to the M functions that correspond to the calling block.

7.

THE SPINDLE. BASIC CONTROL.
M functions with an associated subroutine.

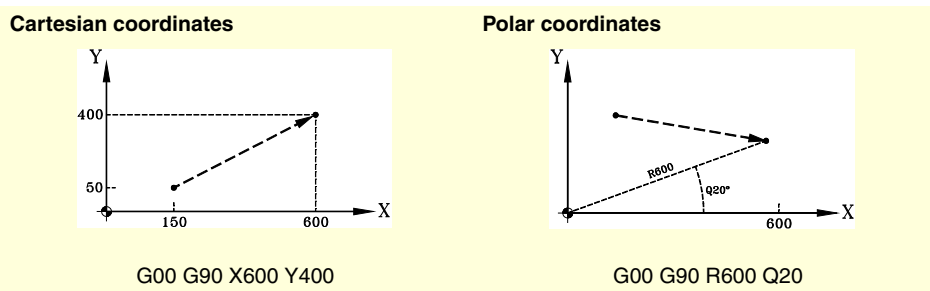


CNC 8060
CNC 8065

(REF. 1402)

8.1 Rapid traverse (G00)

Movements programmed after G00 are executed in a straight line and at the rapid feedrate set by the machine manufacturer from the current position to the destination or target point. Regardless of the number of axes involved, the resulting path is always a straight line.



When auxiliary or rotary axes are involved in rapid positioning, the movement is carried out so it begins and ends when the main axes begin and end their movement.

Programming

The movements may be defined as follows:

- In Cartesian coordinates ("X", "X1"..."C9")
 - Defining the coordinates of the end point on the various axes.
 - All the axes need not be programmed, only the ones to move.
- In polar coordinates ("R", "Q")
 - Defining the radius and the angle of the end point referred to the polar origin.
 - The "R" radius will be the distance between the polar origin and the point. The "Q" angle will be formed by the abscissa axis and the line joining the polar origin with the point.
 - If the angle or the radius is not programmed, it keeps the value programmed for the last move.

Feedrate behavior

A G00 movement temporarily cancels the programmed "F" and the rapid traverse movement is carried out at the value set by the machine manufacturer [A.M.P. "G00FEED"]. The "F" value is restored when programming a G01, G02 or G03 type function.

When several axes are involved, the resulting feedrate is calculated so at least one of the axis moves at its maximum speed.

When defining an "F" value and G00 in the same block, the CNC will store the value assigned to "F" and it will apply it next time a G01, G02 or G03 type function is programmed.

The override percentage is set at 100% or may be varied between 0% and 100% with the switch at the operator panel depending on how the machine manufacturer has set [G.M.P. "RAPIDOVR"].

Properties of the function

Function G00 is modal and incompatible with G01, G02, G03, G33 and G63.

Function G00 may be programmed as G0.

On power-up, after an M02 or M30 and after an EMERGENCY or a RESET, the CNC assumes function G00 or G01 as set by the machine manufacturer [G.M.P. "IMOVE"].

8.**TOOL PATH CONTROL**
Rapid traverse (G00)

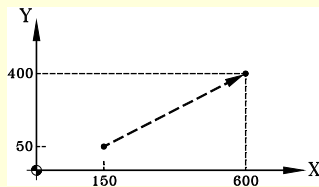
CNC 8060
CNC 8065

(REF. 1402)

8.2 Linear interpolation (G01)

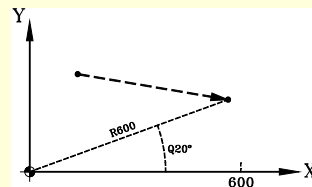
Movements programmed after G01 are executed in a straight line and at the programmed feedrate "F" from the current position to the indicated target point. Regardless of the number of axes involved, the resulting path is always a straight line.

Cartesian coordinates



G01 G90 X600 Y400 F150

Polar coordinates

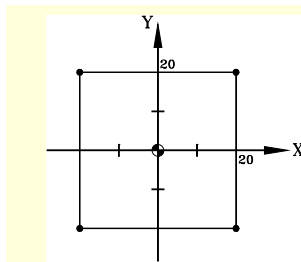


G01 G90 R600 Q20 F185

Auxiliary and rotary axes may also be programmed in the linear interpolation block. In those cases, the CNC will calculate the feedrate for those axes so their movement begins and ends simultaneously with the main axes.

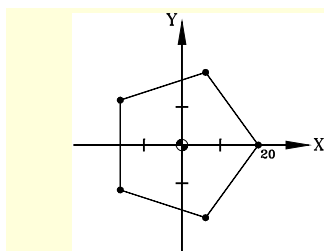
Programming

- In Cartesian coordinates ("X", "X1"..."C9")
 Defining the coordinates of the end point on the various axes.
 All the axes need not be programmed, only the ones to move.



```
G00 X20 Y0
G01 Y20 F350
G01 X-20
G01 Y-20
G01 X20
G01 Y0
M30
```

- In polar coordinates ("R", "Q")
 Defining the radius and the angle of the end point referred to the polar origin.
 The "R" radius will be the distance between the polar origin and the point. The "Q" angle will be formed by the abscissa axis and the line joining the polar origin with the point.
 If the angle or the radius is not programmed, it keeps the value programmed for the last move.



```
G00 X20 Y0
G01 R20 Q72 F350
G01 Q144
G01 Q216
G01 Q288
G01 Q360
M30
```

8.

TOOL PATH CONTROL
 Linear interpolation (G01)



CNC 8060
 CNC 8065

(REF. 1402)

Feedrate behavior

The programmed feedrate "F" stays active until a new value is programmed, thus not being necessary to program it in every block.

When several axes are involved, the CNC calculates the feedrate for each axis so the resulting path is executed at the programmed feedrate "F".

The programmed feedrate "F" may be varied between 0% and 200% using the selector switch on the CNC's operator panel or it may be selected by program or by PLC. However, the maximum override is limited by the machine manufacturer [G.M.P. "MAXOVR"].

The feedrate of the auxiliary axes

The behavior of the auxiliary axes is determined by general machine parameter FEEDND.

- If its value is TRUE, none of the axes will exceed the programmed feedrate.
- If its value is FALSE, the feedrate is applied to the main axes whereas the auxiliary axes may exceed it, but without ever exceeding their MAXFEED. If an axis were to exceed the MAXFEED, the programmed feedrate of the main axes would be limited.

Properties of the function

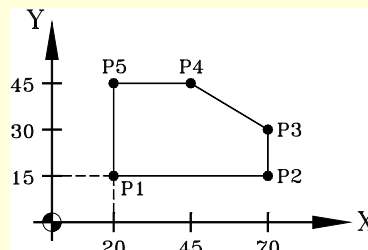
Function G01 is modal and incompatible with G00, G02, G03, G33 and G63.

Function G01 may also be programmed as G1.

On power-up, after an M02 or M30 and after an EMERGENCY or a RESET, the CNC assumes function G00 or G01 as set by the machine manufacturer [G.M.P. "IMOVE"].

Programming examples

Programming in Cartesian coordinates.



	X	Y
P1	20	15
P2	70	15
P3	70	30
P4	45	45
P5	20	45

Absolute coordinates

```
N10 G00 G90 X20 Y15
N20 G01 X70 Y15 F450
N30 Y30
N40 X45 Y45
N50 X20
N60 Y15
N70 G00 X0 Y0
N80 M30
```

Incremental coordinates

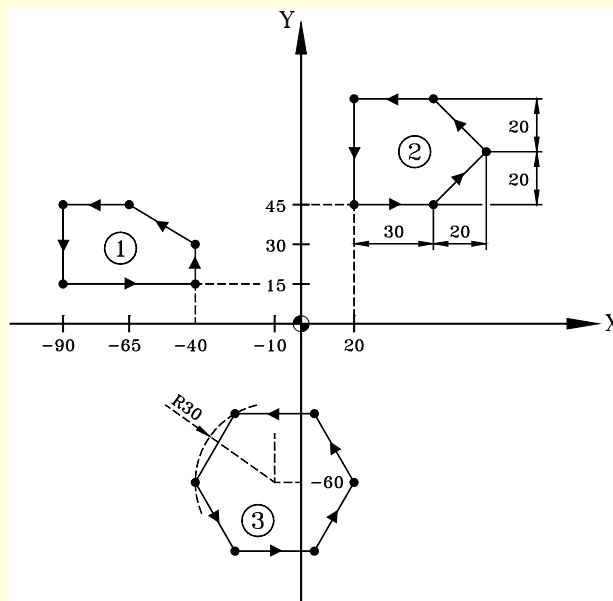
```
N10 G00 G90 X20 Y15
N20 G01 G91 X50 Y0 F450
N30 Y15
N40 X-25 Y15
N50 X-25
N60 Y-30
N70 G00 G90 X0 Y0
N80 M30
```



CNC 8060
CNC 8065

(REF. 1402)

Cartesian and Polar coordinate programming.



```

N10 T1 D1
N20 M06
N30 G71 G90 F450 S1500 M03      (Initial conditions)
N40 G00 G90 X-40 Y15 Z10      (Approach to profile 1)
N50 G01 Z-5
N60 X-40 Y30                    (Machining of profile 1)
N70 X-65 Y45
N80 X-90
N90 Y15
N100 X-40                        (End of profile 1)
N110 Z10
N120 G00 X20 Y45 F300 S1200     (Approach to profile 2)
N130 G92 X0 Y0                  (Preset new part zero)
N140 G01 Z-5
N150 G91 X30                    (Machining of profile 2)
N160 X20 Y20
N170 X-20 Y20
N180 X-30
N190 Y-40                        (End of profile 2)
N200 G90 Z10
N210 G92 X20 Y45                (Restore previous part zero)
N220 G30 I-10 J-60              (Polar origin preset)
N230 G00 R30 Q60 F350 S1200     (Approach to profile 3)
N240 G01 Z-5
N250 Q120                        (Machining of profile 3)
N260 Q180
N270 Q240
N280 Q300
N290 Q360
N300 Q60                        (End of profile 3)
N310 Z10
N320 G00 X0 Y0
N330 M30
    
```



TOOL PATH CONTROL
Linear interpolation (G01)



CNC 8060
CNC 8065

(REF. 1402)

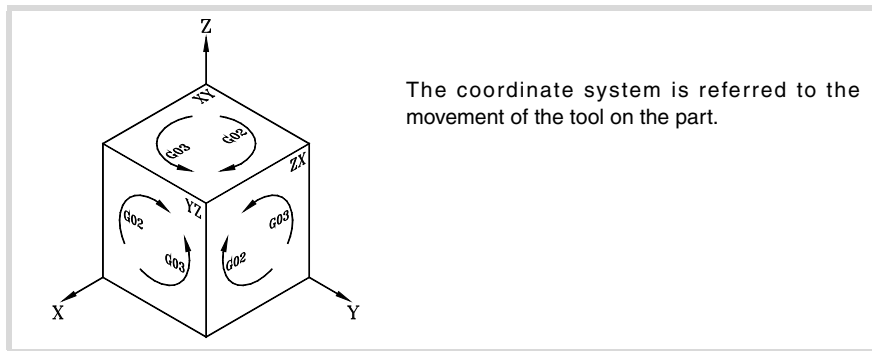
8.3 Circular interpolation (G02/G03)

Movements programmed after G02 and G03 are executed along a circular path at the programmed feedrate "F" from the current position to the indicated target point.

A circular interpolation can only be executed in the active plane. There are two types of circular interpolations:

- G02 Clockwise circular interpolation.
- G03 Counterclockwise circular interpolation.

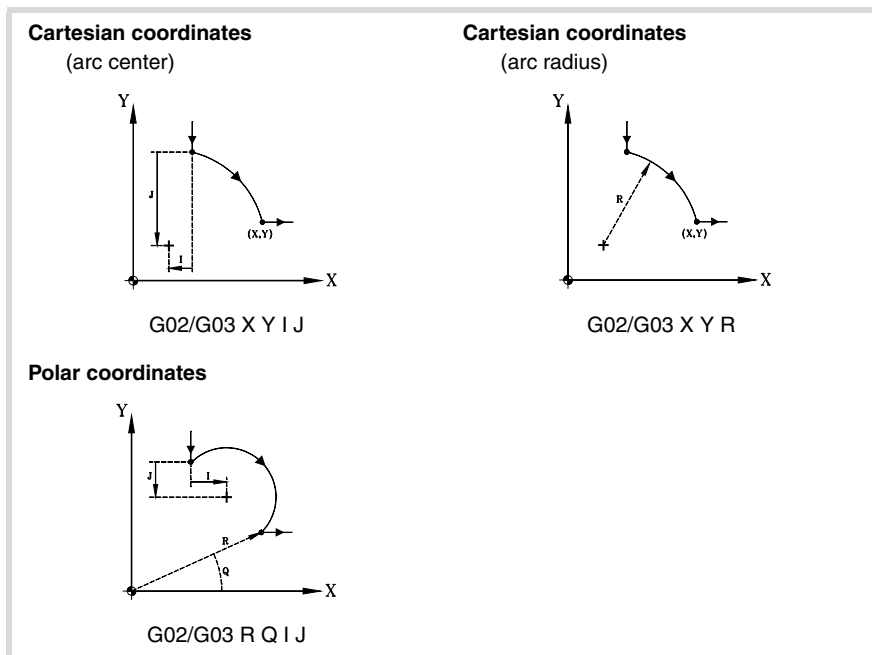
Clockwise (G02) and counterclockwise (G03) moving directions have been established according to the following coordinate system.



Programming

A circular interpolation may be defined as follows:

- In cartesian coordinates, by defining the coordinates of the target point and the center of the arc.
- In cartesian coordinates, by defining the coordinates of the target point and the arc radius.
- In polar coordinates, defining the radius and the angle of the end point as well as the arc center coordinates.



8.
TOOL PATH CONTROL
Circular interpolation (G02/G03)



CNC 8060
CNC 8065

(REF. 1402)

Feedrate behavior

The programmed feedrate "F" stays active until a new value is programmed, thus not being necessary to program it in every block.

The programmed feedrate "F" may be varied between 0% and 200% using the selector switch on the CNC's operator panel or it may be selected by program or by PLC. However, the maximum override is limited by the machine manufacturer [G.M.P. "MAXOVR"].

Properties of the function

Functions G02 and G03 are modal and incompatible with each other and with G00, G01, G33 and G63.

Function G74 (Home search) also cancels functions G02 and G03.

Functions G02 and G03 may also be programmed as G2 and G3.

On power-up, after an M02 or M30 and after an EMERGENCY or a RESET, the CNC assumes function G00 or G01 as set by the machine manufacturer [G.M.P. "IMOVE"].

8.

TOOL PATH CONTROL
Circular interpolation (G02/G03)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

8.3.1 Cartesian coordinates (Arc center programming)

The arc is defined by programming function G02 or G03 followed by the coordinates of the arc's end point and those of its center (referred to the starting point of the arc) according to the axes of the active work plane.

Coordinates of the arc's final point

It is defined with its coordinates along the axes of the active work plane and may be given in either absolute or incremental coordinates.

If they are not programmed or are the same as the starting point, a full circle will be executed.

Arc center coordinates

The arc center coordinates are defined by the letters "I", "J" or "K" depending on the active plane.

G17 G18 G19 Letters "I", "J" and "K" are associated with the first, second and third axis of the channel respectively.

G20 Letters "I", "J" and "K" are associated with the abscissa, ordinate and perpendicular axes of the defined plane.

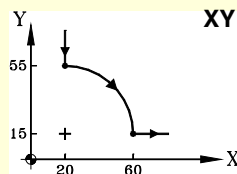
#FACE [X, C, Z] The active trihedron is formed by the axes defined in the C axis activation instruction. The "I", "J", and "K" centers are associated with the axes in the same order that they were defined when activating the C axis.

When the center coordinate on an axis is "0", it does not have to be programmed. These coordinates are not affected by functions G90 and G91.

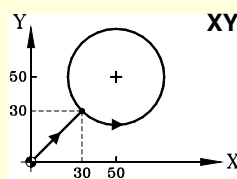
Depending on the active work plane, the programming format is:

XY plane (G17)	G02/G03	X...	Y...	I...	J...
ZX plane (G18)	G02/G03	X...	Z...	I...	K...
YZ plane (G19)	G02/G03	Y...	Z...	J...	K...

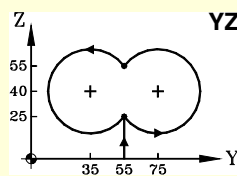
Circular interpolation programming by defining the center.



```
...
G02 X60 Y15 I0 J-40
...
```



```
N10 G17 G71 G94
N20 G01 X30 Y30 F400
N30 G03 X30 Y30 I20 J20
N40 M30
```



```
N10 G19 G71 G94
N20 G00 Y55 Z0
N30 G01 Y55 Z25 F400
N40 G03 Z55 J20 K15
N50 Z25 J-20 K-15
N60 M30
```



CNC 8060
CNC 8065

(REF. 1402)

8.3.2 Cartesian coordinates (Radius programming)

The arc is defined by programming function G02 or G03 followed by the coordinates of the arc's end point and its radius.

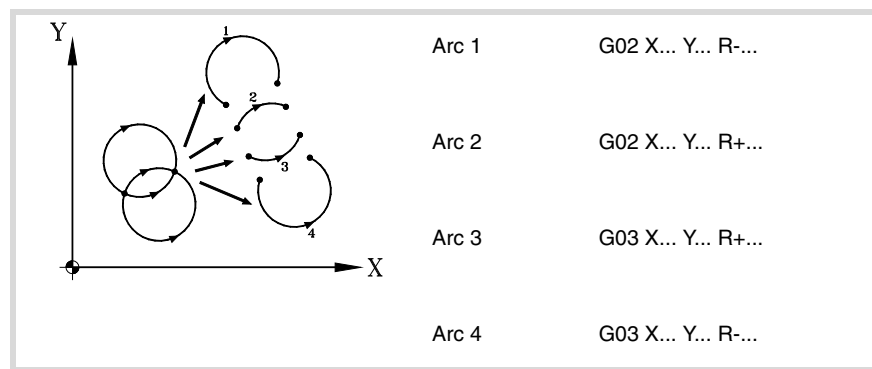
Coordinates of the arc's final point

It is defined with its coordinates along the axes of the active work plane and may be given in either absolute or incremental coordinates.

Arc radius

The arc radius is defined with the letter "R" or using assignments "R1=<radius>" or "G263=<radius>". The radius value stays active until a new value is assigned or an arc is programmed using the center coordinates or a movement is programmed in polar coordinates.

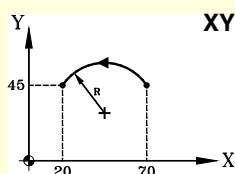
If the arc is smaller than 180°, the radius will be programmed with a positive sign and with a negative sign if it is greater than 180°. This way and depending on the selected circular interpolation (G02 or G03), the desired arc will be defined.



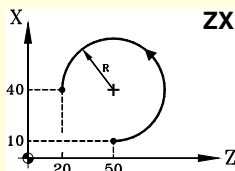
Depending on the active work plane, the programming format is:

XY plane (G17)	G02/G03	X...	Y...	R+/-
ZX plane (G18)	G02/G03	X...	Z...	R+/-
YZ plane (G19)	G02/G03	Y...	Z...	R+/-

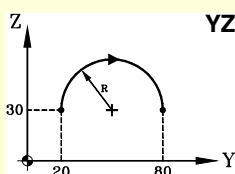
Different formats to define the same arc.



```
G03 G17 X20 Y45 R30
G03 G17 X20 Y45 G263=30
G03 G17 X20 Y45 R1=30
```



```
G03 G18 Z20 X40 R-30
G03 G18 Z20 X40 G263=-30
G03 G18 Z20 X40 R1=-30
```



```
G02 G19 Y80 Z30 R30
G02 G19 Y80 Z30 G263=30
G02 G19 Y80 Z30 R1=30
```

8.

TOOL PATH CONTROL
Circular interpolation (G02/G03)



CNC 8060
CNC 8065

(REF. 1402)

The radius may also be programmed in a block prior to the one defining the circular interpolation. In this case, the radius is defined using the assignments "R1=<radius>" or "G263=<radius>".

N10 G01 G90 X0 Y0 F500	N10 G01 G90 X0 Y0 F450
N20 G263=50	N20 G01 R1=50
N30 G02 X100	N30 G02 X100

N10 G01 G90 X0 Y0	N10 G01 G90 X0 Y0
N20 G02 G263=50	N20 G02 R1=50
N30 X100	N30 X100

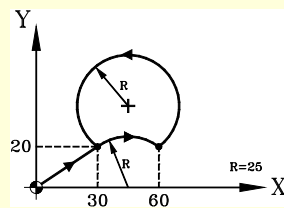
The previous examples will make semicircles with a radius of 50.

The CNC keeps the radius value until a circular interpolation is programmed by defining the center coordinates or a movement is programmed in polar coordinates.

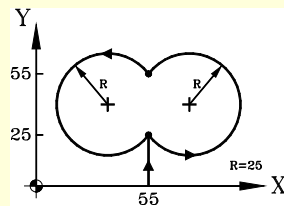


When programming an arc using the radius, it is not possible to program full circles because there are infinite solutions.

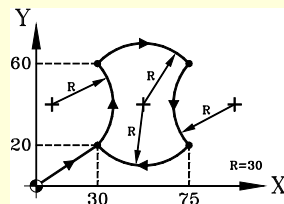
Circular interpolation programming by defining the radius.



```
N10 G01 G90 G94 X30 Y20 F350
N20 G263=25
N30 G02 X60
N40 G263=-25
N50 G03 X30
N60 M30
```



```
N10 G17 G71 G94
N20 G00 X55 Y0
N30 G01 X55 Y25 F400
N40 G263=-25
N50 G03 Y55
N60 Y25
N70 M30
```



```
N10 G17 G71 G94
N20 G01 X30 Y20 F400
N30 R1=30
N40 G03 Y60
N50 G02 X75
N60 G03 Y20
N70 G02 X30
N80 M30
```

8.

TOOL PATH CONTROL Circular interpolation (G02/G03)



CNC 8060
CNC 8065

(REF. 1402)

8.3.3 Polar coordinates

The arc is defined by programming function G02 or G03 followed by the coordinates of the arc's end point and those of its center (referred to the starting point of the arc) according to the axes of the active work plane.

Coordinates of the end point

The position of the end point is given by defining the radius "R" and the angle "Q" as follows:

- Radius Distance between the polar origin and the point.
- Angle Angled formed by the line joining the polar origin with the point and the horizontal going through the polar origin.

If the angle or the radius is not programmed, it keeps the value programmed for the last move. The radius and the angle may be defined both in absolute (G90) and incremental coordinates (G91).

When programming the angle in G91, it is incremented with respect to the polar origin of the previous point; if programmed in G90, It indicates the angle formed with the horizontal going through the polar origin.

Programming a 360° angle in G91 means programming a whole circle. Programming a 360° angle in G90 means programming an arc where the target point forms a 360° angle with the horizontal going through the polar origin.

Center coordinates

The arc center coordinates are defined by the letters "I", "J" or "K" depending on the active plane.

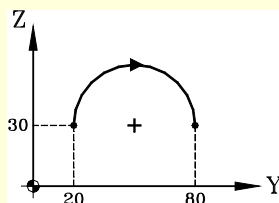
- G17 G18 G19 Letters "I", "J" and "K" are associated with the first, second and third axis of the channel respectively.
- G20 Letters "I", "J" and "K" are associated with the abscissa, ordinate and perpendicular axes of the defined plane.
- #FACE [X, C, Z] The active trihedron is formed by the axes defined in the C axis activation
- #CYL [Z, C, X, R] instruction. The "I", "J", and "K" centers are associated with the axes in the same order that they were defined when activating the C axis.

When the center coordinate on an axis is zero, it does not have to be programmed; if neither of them are programmed, it will assume the polar origin as the arc center. These coordinates are not affected by functions G90 and G91.

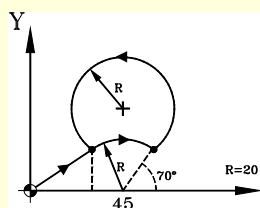
Depending on the active work plane, the programming format is:

- XY plane (G17) G02/G03 R... Q... I... J...
- ZX plane (G18) G02/G03 R... Q... I... K...
- YZ plane (G19) G02/G03 R... Q... J... K...

Circular interpolation programming in polar coordinates.



```
N10 G0 G90 X20 Y30 F350
N20 G30
N30 G02 R60 Q0 I30
N40 M30
```



```
N10 G0 G90 X0 Y0 F350
N20 G30 I45 J0
N30 G01 R20 Q110
N40 G02 Q70
N50 G03 Q110 I-6.8404 J18.7938
N60 M30
```



TOOL PATH CONTROL
Circular interpolation (G02/G03)



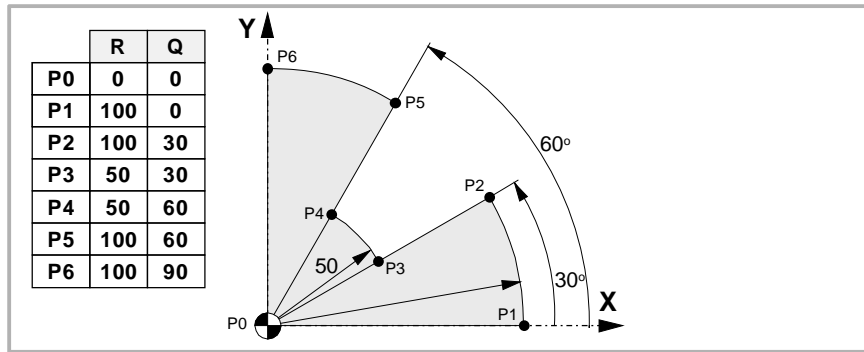
CNC 8060
CNC 8065

(REF. 1402)

Programming examples.

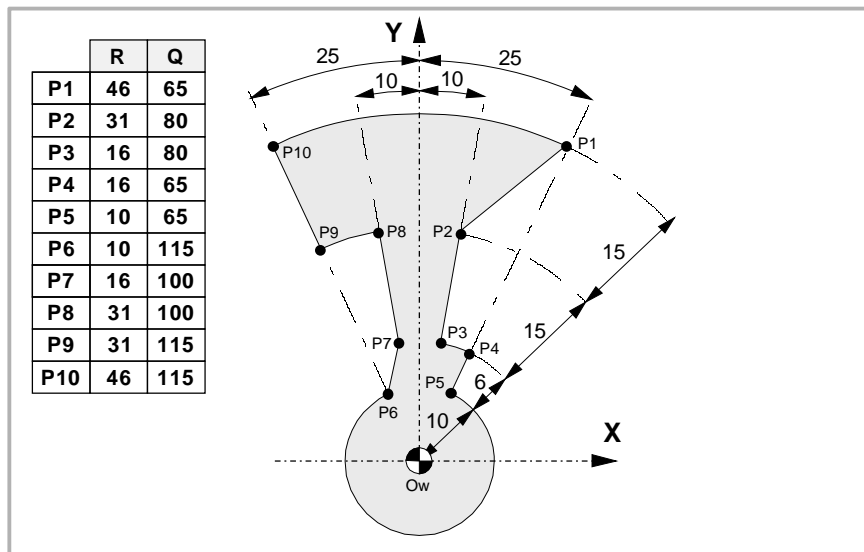
8.

TOOL PATH CONTROL
Circular interpolation (G02/G03)



	R	Q
P0	0	0
P1	100	0
P2	100	30
P3	50	30
P4	50	60
P5	100	60
P6	100	90

Absolute coordinates.	Incremental coordinates.	
G00 G90 X0 Y0 F350	G00 G90 X0 Y0 F350	; Point P0.
G01 R100 Q0	G91 G01 R100 Q0	; Point P1. Straight line.
G03 Q30	G03 Q30	; Point P2. Counterclockwise arc.
G01 R50 Q30	G01 R-50	; Point P3. Straight line.
G03 Q60	G03 Q30	; Point P2. Counterclockwise arc.
G01 R100 Q60	G01 R50	; Point P5. Straight line.
G03 Q90	G03 Q30	; Point P6. Counterclockwise arc.
G01 R0 Q90	G01 R-100	; Point P0, in a straight line.
M30	M30	



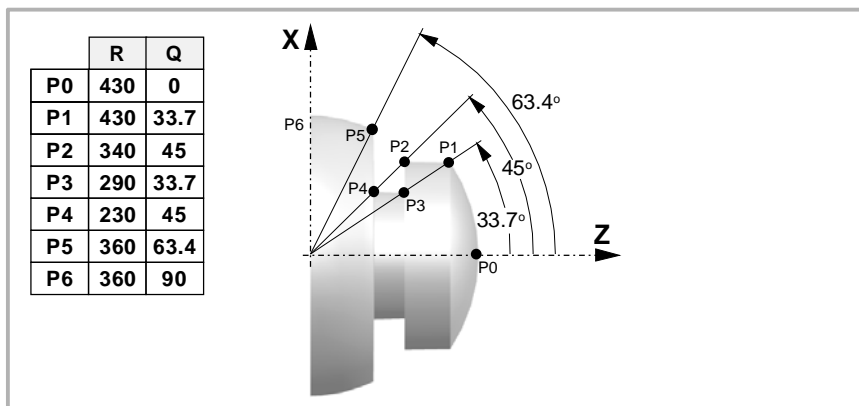
	R	Q
P1	46	65
P2	31	80
P3	16	80
P4	16	65
P5	10	65
P6	10	115
P7	16	100
P8	31	100
P9	31	115
P10	46	115

Absolute coordinates	Incremental coordinates	
G90 R46 Q65 F350	G90 R46 Q65 F350	; Point P1.
G01 R31 Q80	G91 G01 R-15 Q15	; Point P2. Straight line.
G01 R16	G01 R-15	; Point P3. Straight line.
G02 Q65	G02 Q-15	; Point P4. Clockwise arc.
G01 R10	G01 R-6	; Point P5. Straight line.
G02 Q115	G02 Q-310	; Point P6. Clockwise arc.
G01 R16 Q100	G01 R6 Q-15	; Point P7. Straight line.
G01 R31	G01 R15	; Point P8. Straight line.
G03 Q115	G03 Q15	; Point P9. Counterclockwise arc.
G01 R46	G01 R15	; Point P10. Straight line.
G02 Q65	G02 Q-50	; Point P1. Clockwise arc.
M30	M30	



CNC 8060
CNC 8065

(REF. 1402)



Absolute coordinates	Incremental coordinates	
G18	G18	; Z-X plane,
G152	G152	; Programming in radius.
G90 R430 Q0 F350	G90 R430 Q0 F350	; Point P0.
G03 Q33.7	G91 G03 Q33.7	; Point P1. Counterclockwise arc.
G01 R340 Q45	G01 R-90 Q11.3	; Point P2. Straight line.
G01 R290 Q33.7	G01 R-50 Q-11.3	; Point P3. Straight line.
G01 R230 Q45	G01 R-60 Q11.3	; Point P4. Straight line.
G01 R360 Q63.4	G01 R130 Q18.4	; Point P5. Straight line.
G03 Q90	G03 Q26.6	; Point P6. Counterclockwise arc.
M30	M30	

8.

TOOL PATH CONTROL
Circular interpolation (G02/G03)



CNC 8060
CNC 8065

(REF. 1402)

8.3.4 Temporary polar origin shift to the center of arc (G31)

When defining an arc in polar coordinates, the polar origin may be shifted temporarily to the center of the arc.

G31

Temporary polar origin shift to the center of arc

Function G31 shifts temporarily the polar origin to the center of the programmed arc. This function only acts in the block that contains it; once the block has been executed, it restores the previous polar.

This function is added to the programmed circular interpolation G2/G3. In this case, at least one of the center coordinates must be programmed.

8.

TOOL PATH CONTROL
Circular interpolation (G02/G03)



CNC 8060
CNC 8065

(REF. 1402)

8.3.5 Arc center in absolute coordinates (G06/G261/G262)

When defining an arc, one may select whether the center position is referred to the starting point of the arc or it is defined in absolute coordinates.

Programming

This selection is made using the following functions:

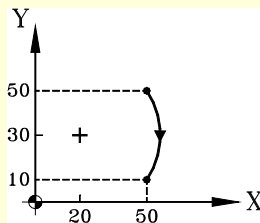
G06	Arc center in absolute coordinates (not modal).
G261	Arc center in absolute coordinates (modal).
G262	Arc center referred to starting point.

G06-G261

Arc center in absolute coordinates

While one of these functions is active, the CNC interprets that the arc center coordinates are referred to the active reference system origin (part zero, polar origin, etc).

Function G261 stays active throughout the program whereas G06 only acts in the block where it has been programmed, therefore it can only be added to a block where a circular interpolation has been defined.



```
G261
G90 G02 X50 Y10 I20 J30

G261
G91 G02 X0 Y-40 I20 J30

G90 G06 G02 X50 Y10 I20 J30

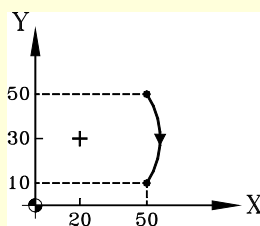
G91 G06 G02 X0 Y-40 I20 J30
```

The example shows 4 different ways to define an arc using absolute center coordinates.

G262

Arc center referred to starting point

When this function is active, the CNC interprets that the coordinates of the arc center are referred to the starting point of the arc.



```
G262
G90 G02 X50 Y10 I-30 J-20

G262
G91 G02 X0 Y-40 I-30 J-20
```

The example shows two different ways to define an arc by indicating its center with respect to the starting point of the arc.

Properties of the functions

Functions G261 and G262 are modal and incompatible with each other.

On power-up, after executing an M02 or M30, and after an EMERGENCY or RESET, the CNC assumes function G262.

8.

TOOL PATH CONTROL
Circular interpolation (G02/G03)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

8.3.6 Arc center correction (G264/G265)

In order to execute the programmed arc, the CNC calculates the radii of the initial and final points which must be the same. When this is not the case, using center correction it is possible to execute the programmed arc by correcting its center.

The tolerance allowed for the difference between both radii or for locating the corrected arc center is set by the machine manufacturer [G.M.P. "CIRINERR" and "CIRINFACT"].

Programming

Arc center correction may be turned on and off using the following functions:

G264	Cancellation of arc center correction.
G265	Activation of arc center correction.

G264

Cancellation of arc center correction

When the difference between the initial and final radii is within the allowed tolerance, it executes the arc with the radius calculated using the initial point. The center position stays the same.

If the difference between both radii exceeds the allowed tolerance, the relevant error message will be issued.

G265

Activation of arc center correction.

If the initial and final arc radii are not the same, the CNC tries to calculate a new center within the set tolerance so as to be able to execute an arc between the programmed points as close as possible to the defined arc.

To calculate whether the error margin is within tolerance or not, the CNC considers two values:

- The absolute error (radius difference).
- The relative error (% over the radius).

If any of these values is within the tolerance set by the OEM, the CNC corrects the center position.

If the CNC cannot find the center within those limits, it will issue the pertinent error message.

Properties of the functions

Functions G264 and G265 are modal and incompatible with each other.

On power-up, after executing an M02 or M30, and after an EMERGENCY or RESET, the CNC assumes function G265.

8.

TOOL PATH CONTROL
Circular interpolation (G02/G03)



CNC 8060
CNC 8065

(REF. 1402)

8.4 Arc tangent to previous path (G08)

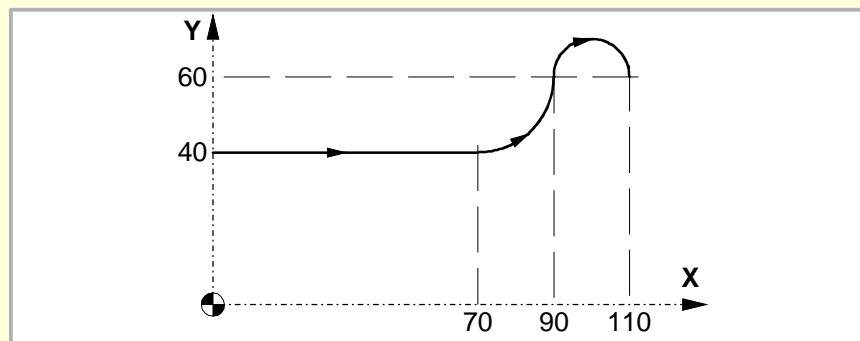
Function G08 may be used to program a circular path tangent to the previous path without having to program the center coordinates (I, J or K).



Function G08 cannot be used to program full circles because there are infinite solutions.

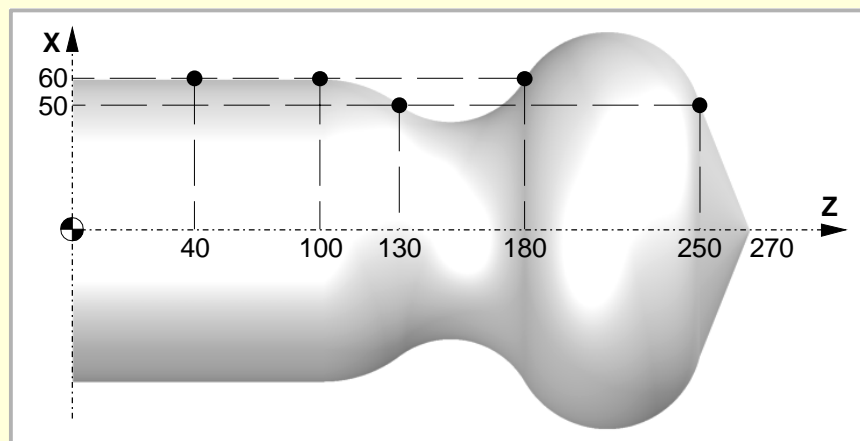
Programming

Only the coordinates of the arc's final (end) point must be programmed either in polar or Cartesian coordinates along the axes of the work plane. The previous path may be either linear or circular.



Assuming the starting point is X0 Y40, we would like to program a straight line, then an arc tangent to it and finally an arc tangent to the previous one.

```
G90 G01 X70
G08 X90 Y60      (Arc tangent to previous path)
G08 X110         (Arc tangent to previous path)
```



```
G18 G152      ; Main plane Z-X, and programming in radius.
G90 G01 X0 Z270
X50 Z250
G08 X60 Z180  ; Arc tangent to previous path.
G08 X50 Z130  ; Arc tangent to previous path.
G08 X60 Z100  ; Arc tangent to previous path.
G01 X60 Z40
```

8.

TOOL PATH CONTROL
Arc tangent to previous path (G08)

FAGOR

CNC 8060
CNC 8065

(REF. 1402)

Properties of the function

Function G08 is not modal, consequently, it must be programmed every time when programming an arc tangent to the previous path. After executing it, the CNC restores the G01, G02 or G03 function that was active before.

Function G08 may also be programmed as G8.

8.**TOOL PATH CONTROL**

Arc tangent to previous path (G08)



CNC 8060
CNC 8065

(REF. 1402)

8.5 Arc defined by three points (G09)

G09 may be used to define an arc by programming the end point and an intermediate point (the initial point of the arc is the starting point of the move). In other words, instead of programming the center coordinates, any intermediate point is programmed.



Function G09 may not be used to programmed a full circle because all three points must be different.

Coordinates of the end point

It may defined in cartesian or polar coordinates both absolute and incremental.

Coordinates of the intermediate point

It must be defined in cartesian coordinates by the letters "I", "J" or "K" depending on the active plane.

G17 G18 G19 Letters "I", "J" and "K" are associated with the X, Y and Z axes respectively.

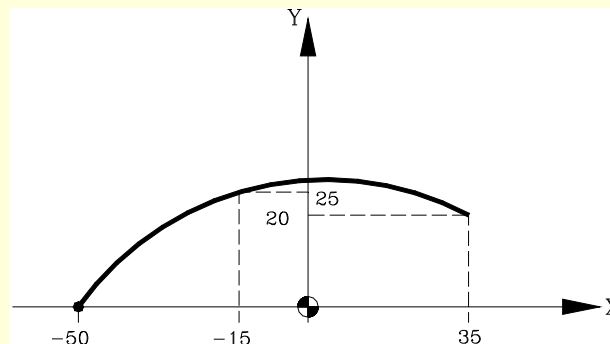
G20 Letters "I" and "J" are associated with the abscissa and ordinate axes of the defined plane.

These coordinates are affected by functions G90 and G91.

The programming format depends on the active work plane. In the XY plane is:

XY plane (G17)	G02/G03	X...	Y...	I...	J...
	G02/G03	R...	Q...	I...	J...

Where X-50 Y0 is the starting point.



G09 X35 Y20 I-15 J25

Programming G09 does not require programming the direction of the movement (G02 or G03).

Properties of the function

Function G09 is not modal, consequently, it must be programmed every time when programming an arc defined by three points. After executing it, the CNC restores the G01, G02 or G03 function that was active before.

Function G09 may be programmed as G9.

8.

TOOL PATH CONTROL
Arc defined by three points (G09)

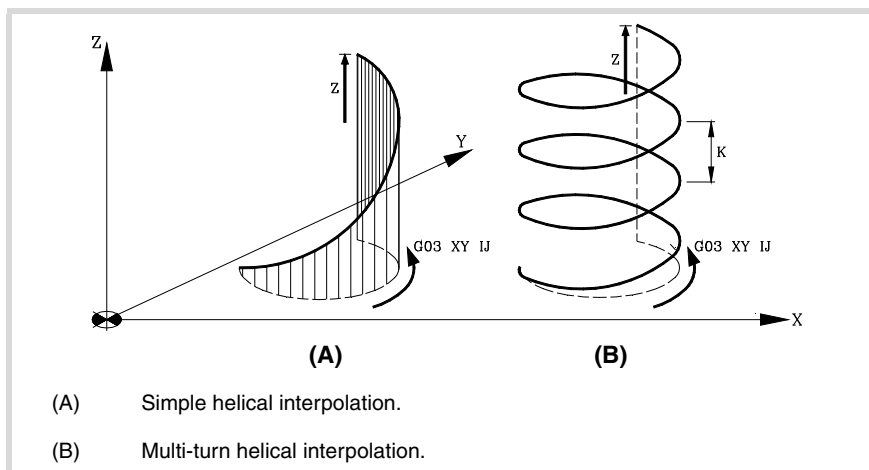
FAGOR

CNC 8060
CNC 8065

(REF. 1402)

8.6 Helical interpolation (G02/G03)

Helical interpolation consists of a circular interpolation in the work plane and a linear movement of the rest of the axes programmed.



Helical interpolation is programmed in a block whose circular interpolation must be programmed using function G02, G03, G08 or G09.

Programming

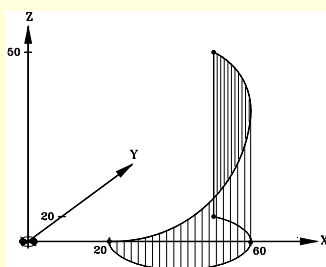
Simple helical interpolation

The helical interpolation is defined by programming the circular interpolation in the active plane and then the linear movement of the other axes.

The programming format depends on the active work plane. In the XY plane is:

XY plane (G17)	G02/G03	X...	Y...	I...	J...	<axes>
	G02/G03	X...	Y...	R...		<axes>
	G02/G03	R...	Q...	I...	J...	<axes>
	G08	X...	Y...			<axes>
	G09	X...	Y...	I...	J...	<axes>

Different ways to program a helical interpolation.



G03 X40 Y20 I20 J0 Z50

G03 X40 Y20 R-20 Z50

G03 R44.7213 Q26.565 I20 J0 Z50

G09 X40 Y20 I60 J0 Z50

Starting point: X20 Y0 Z0

End point: X40 Y20 Z50

Programming

Multi-turn helical interpolation

If the helical interpolation is to make several turns, besides programming the circular interpolation in the active work plane and the linear movement of the other axes, the helical pitch must also be programmed.

When defining the center of the circular interpolation, it is not necessary to define the coordinates for the end point in the work plane. This point will be calculated by the CNC depending on the height and pitch of the helix.

Pass definition

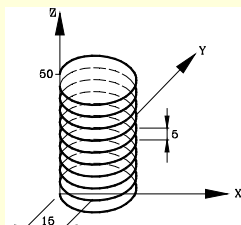
The helical pitch is defined using the letter "I", "J" or "K" associated with 3rd axis of the active work plane.

- G17 G18 G19 The pitch is defined with the letter "K" (G17), "J" (G18) or "I" (G19).
- G20 The pitch is defined with the letter "K".

The programming format depends on the active work plane. In the XY plane is:

XY plane (G17)	G02/G03	X... Y...	I... J...	<axes>	K...
	G02/G03	I... J...	<axes>	K...	
	G02/G03	R... Q...	I... J...	<axes>	K...
	G08	X... Y...	<axes>	K...	
	G09	X... Y...	I... J...	<axes>	K...

Programming a helical interpolation where the starting point is X0 Y0 Z0.



```
G03 X0 Y0 I15 J0 Z50 K5
G03 R0 Q0 I15 J0 Z50 K5
```



TOOL PATH CONTROL
Helical interpolation (G02/G03)



CNC 8060
CNC 8065

(REF. 1402)

8.7 Electronic threading with constant pitch (G33)



For electronic threading, the machine must have a rotary encoder installed on the spindle.

When doing an electronic threading, the CNC does NOT interpolate the movement of the axes with the spindle. A rigid tapping may be carried out to make a thread by interpolating the spindle with the axes. See "8.8 Rigid tapping (G63)" on page 159.

Although, this type of threads are carried out along an axis, the CNC permits interpolating several axes. Besides, with electronic threading, it is possible to make threads with several entries (starts) and to blend threads.

Electronic threading may be carried out with any spindle, but when not using the master spindle, the spindle being used must be synchronized with it. It is synchronized via PLC (mark SYNC).

Programming

An electronic threading is programmed with G33 followed by the coordinates of the end point of the thread and the thread pitch. Optionally, the entry angle may be defined which allows multi-entry threads and thread blending.

G33 X··Z I··J <Q1>

X··Z Coordinates of the end point.
 I··K Thread pitch.
 Q1 Optional. Entry angle.
 If not programmed, the thread is synchronized when going through 0°.

Coordinates of the end point

The coordinates of the end point may be defined either in Cartesian or Polar coordinates. They may be given either in absolute or incremental coordinates.

Thread pitch

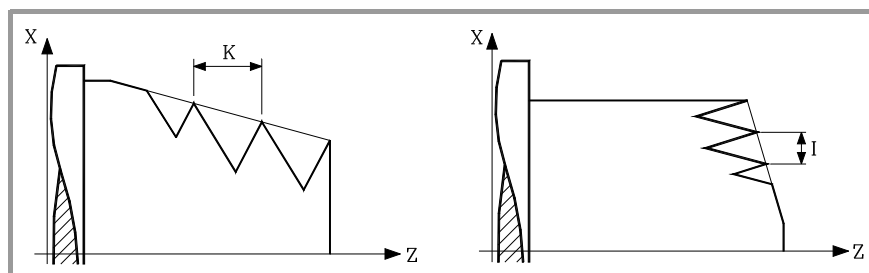
The pitch is defined by the letters "I", "J" or "K" depending on the active plane.

G17 G18 G19 Letters "I", "J" and "K" are associated with the first, second and third axis of the channel respectively.
 G20 Letters "I", "J" and "K" are associated with the abscissa, ordinate and perpendicular axes of the defined plane.

Example of electronic threading with the Z axis in different planes. Let us assume an X-Y-Z axis configuration in the channel.

G17	G18	G19
G33 Z40 K2	G33 Z40 K2	G33 Z40 K2
G20 Z1 Y2 X3	G20 Y1 Z2 X3	G20 Y1 Z3 X2
G33 Z40 I2	G33 Z40 J2	G33 Z40 K2

When interpolating several axes in the electronic threading, the pitch is not defined on the path, it is defined on one of the axes.



8.

TOOL PATH CONTROL
Electronic threading with constant pitch (G33)



CNC 8060
CNC 8065

(REF. 1402)

Entry angle

It indicates the spindle angular position (± 359.9999) of the starting point of the thread. With this parameter, it is possible to make multi-entry threads.

Programming it is an option. If not programmed, the thread is synchronized when going through 0° (same as programming $Q1=0$).

Thread blending

When blending threads, only the entry angle of the first one is taken into account. Q1 is taken into account only in the first thread after activating G33. Parameter Q1 is ignored until this function is canceled and activated back and it is synchronized when going through that angle.

Considerations for the execution**Spindle home search**

If the spindle has not been homed (referenced), the first G33 will be done automatically when using the master spindle. If the spindle is the master and it has not been homed, it will issue a warning.

Feedrate behavior.

The threading feedrate depends on the programmed spindle speed and thread pitch (Feedrate = Spindle speed x Pitch). The electronic threading is carried out at 100% of the calculated feedrate and these values cannot be modified from the CNC's operator panel or via PLC.

Behavior of the spindle speed and of the spindle speed override

If the OEM allows it, (parameter THREADOVR), the user can modify the speed override from the operator panel and, in that case, the CNC will adapt the feedrate automatically respecting the thread pitch. In order to be able to modify the override, the feed forward active on the axes involved in threading must be higher than 90%.

If more than one G33 have been programmed for the same thread, all the threading operations must start at the same speed; otherwise, the entry point (start) to the thread will not be the same in all the threads. The CNC permits changing the spindle override during the thread cutting pass.

If more than one G33 have been programmed for a multi-start (multi-entry) thread, all the threading operations must start at the same speed; otherwise, the angle between the starts (entry points) to the thread will not be the same as the one programmed. The CNC permits changing the spindle override during the thread cutting pass.

Properties of the functions

Function G33 is modal and incompatible with G00, G01, G02, G03, G63 and G100.

On power-up, after an M02 or M30 and after an EMERGENCY or a RESET, the CNC assumes function G00 or G01 as set by the machine manufacturer [G.M.P. "IMOVE"].

8.

TOOL PATH CONTROL
Electronic threading with constant pitch (G33)

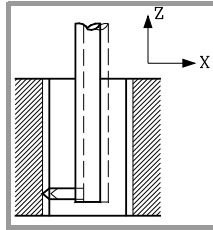
FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

8.7.1 Programming examples for a mill

Single-entry electronic threading



To make the following electronic thread in a single pass.

Position : X30 Y30 Z0
 Depth : 30mm
 Pitch : 1.5mm

```
S100 M03
G01 G90 X30 Y30 Z0
G33 Z-30 K1.5
M19 S0                (Spindle orientation)
G91 G00 X3            (Tool withdrawal)
G90 Z10               (Withdrawal and exit the hole)
```

Since a spindle speed of 100 rpm and a pitch of 1.5 mm have been programmed, the resulting feedrate will be 150 mm/min (pitch feedrate).

Multi-entry electronic threading

To make a thread similar to the previous one but with three entries (starts), the first one being at 20°.

```
S100 M03
G01 G90 X30 Y30 Z0
G33 Z-30 K1.5 Q1=20    (First thread)
M19 S0
G91 G00 X3
G90 Z10
S100 M03
G33 Z-30 K1.5 Q1=140  (Second thread)
M19 S0
G91 G00 X3
G90 Z10
S100 M03
G33 Z-30 K1.5 Q1=260  (Third thread)
M19 S0
G91 G00 X3
G90 Z10
S100 M03
M30
```

8.

TOOL PATH CONTROL
 Electronic threading with constant pitch (G33)



CNC 8060
 CNC 8065

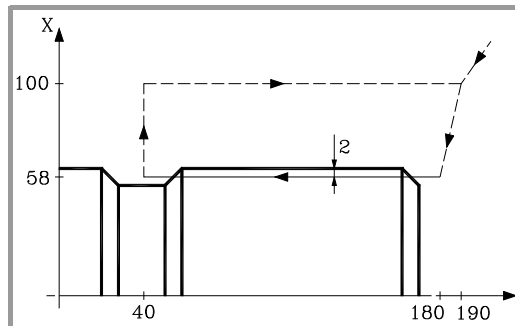
(REF. 1402)

8.7.2 Programming examples for a lathe

Example programming the X axis in radius.

Longitudinal electronic threading

To make a cylindrical thread in a single pass, 2 mm deep and with a 5 mm pitch.

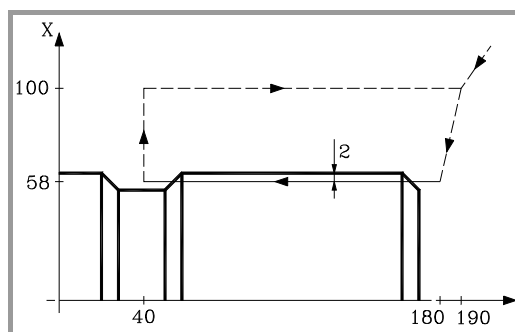


```
S100 M03
G00 G90 X200 Z190
X116 Z180
G33 Z40 K5
G00 X200
Z190
```

Since a spindle speed of 100 rpm and a pitch of 5 mm have been programmed, the resulting feedrate will be 500 mm/min (pitch feedrate).

Multi-entry longitudinal electronic threading

To make a thread similar to the previous one, but with two entries shifted 180° from each other.



```
S100 M03
G00 G90 X200 Z190
X116 Z180
G33 Z40 K5 Q1=0
G00 X200
Z190
X116 Z180
G33 Z40 K5 Q1=180
G00 X200
Z190
```

8.

TOOL PATH CONTROL
Electronic threading with constant pitch (G33)

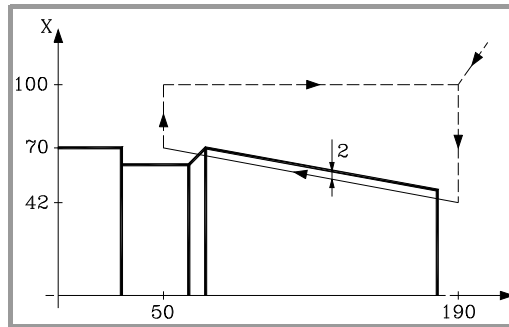
FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

Electronic taper threading

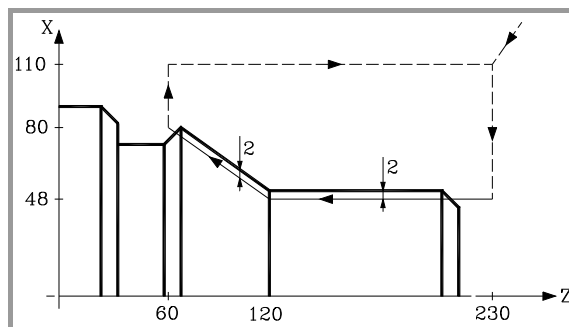
To make a taper thread in a single pass, 2 mm deep and with a 5 mm pitch.



```
S100 M03
G00 G90 X200 Z190
X84
G33 X140 Z50 K5
G00 X200
Z190
```

Thread blending

To blend a longitudinal thread and a taper thread, 2 mm deep and 5 mm pitch.



```
S100 M03
G00 G90 G05 X220 Z230
X96
G33 Z120 Z50 K5
G33 X160 Z60 K5
G00 X220
Z230
```

8.8 Rigid tapping (G63)



For rigid tapping, the machine must have a rotary encoder installed on the spindle.

When rigid tapping, the CNC interpolates the movement of the longitudinal axis with the spindle.

Programming

To define a rigid tapping, program function G63 and then the coordinates of the end point of the thread which may be defined in Cartesian or polar coordinates. The thread pitch will be calculated by the CNC depending on the active feedrate "F" and spindle speed "S" (Pitch = Feedrate / Spindle speed).

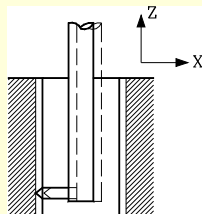
Function G63 starts the spindle in the direction indicated by the programmed speed "S" ignoring the active M3, M4, M5 or M19 functions. A negative turning speed can only be programmed if function G63 is active.

```
...
G94 F300
G01 G90 X30 Y30 Z50
G63 Z20 S200
...
```

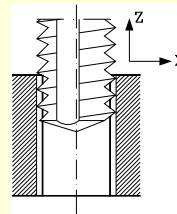
The thread pitch will be: $\frac{F}{S} = \frac{300}{200} = 1,5\text{mm}$

Since G63 does not withdraw the tool automatically after the tap, an inverted tap must be programmed in order to withdraw the tool by inverting the turning direction of the spindle (by changing the sign of the "S" speed). If the thread is made with a cutter tip, the tool may be also be withdrawn by orienting the spindle (M19) and separating the tool tip away from the thread.

To make a 4 mm pitch thread in X30 Y30 Z0 in a single pass with a depth of 30mm.



```
G94 F400
G01 G90 X30 Y30 Z0
G63 Z-30 S100
M19 S0
G91 G01 X3
G90 Z10
```



```
G94 F400
G01 G90 X30 Y30 Z0
G63 Z-30 S-100
G01 Z10
```

Multiple-entry threads

With this type of threading, it is possible to make threads with several entry points. The positioning for each entry must be defined before each threading operation.

```
...
G90 G01 X0 Y0 Z0 F150
M19 S0 (First entry at 0°)
G63 Z-50 S150 (Tapping)
G63 Z0 S-150 (Withdrawal)
M19 S120 (Second entry at 120°)
G63 Z-50 S150
```

8.

TOOL PATH CONTROL
Rigid tapping (G63)

FAGOR

CNC 8060
CNC 8065

(REF. 1402)

```
G63 Z0 S-150
M19 S240                (Third entry at 240°)
G63 Z-50 S150
G63 Z0 S-150
...
3-entry thread, 50mm deep and 1mm pitch.
```

8.

TOOL PATH CONTROL Rigid tapping (G63)

Considerations for the execution

Spindle speed behavior

Threading is carried out at the speed defined with function G63. If no particular threading speed is defined, threading will be executed at the speed active at the time. If a speed is programmed with function G63, it will be the active spindle speed at the end of the threading operation.

The spindle turning direction is determined by the sign of the programmed "S" speed ignoring the active M3, M4, M5 or M19 functions. Programming any of these functions will cancel G63.

Feedrate behavior

While rigid tapping, the feedrate may be varied between 0% and 200% using the feedrate override switch on the CNC's operator panel or via PLC. The CNC will adapt the spindle speed in order to keep the interpolation between the axis and the spindle.

Rigid tapping and tool inspection mode

When interrupting the rigid tapping and accessing the tool inspection mode, it is possible to jog the axes (only in jog mode) that are involved in threading. When moving the axis, the interpolated spindle will also move; the spindle used to make the thread. If rigid tapping involves several axes, when moving one of them, all the other ones involved in the thread will also move.

This allows moving the axis into or out of the thread as often as desired until pressing the repositioning softkey. The axes move at the programmed F except when an axis or spindle exceeds its maximum feedrate allowed (parameter `MAXMANFEED`), in which case, the feedrate will be limited to that value.

The spindle jogging keys are disabled during tool inspection. It is only possible to get out of the thread by jogging one of the axes involved in rigid tapping. Functions M3, M4, M5 and M9 cannot be programmed at the spindle; they are ignored.

While repositioning, when selecting one of the axes of the thread on the softkey menu, it will move all the axes and spindle involved in the thread.

Properties of the functions

Function G63 is modal and incompatible with G00, G01, G02, G03 and G33.

On power-up, after an M02 or M30 and after an EMERGENCY or a RESET, the CNC assumes function G00 or G01 as set by the machine manufacturer [G.M.P. "IMOVE"].



CNC 8060
CNC 8065

(REF. 1402)

8.9 Manual intervention (G200/G201/G202)

With these functions, it is possible to activate the JOG mode by program; in other words, the axes may be jogged even while executing a program. The movement may be made using handwheels or the JOG keys (incremental or continuous JOG).

Programming

The functions related to manual intervention are:

G200	Exclusive manual intervention.
G201	Activation of additive manual intervention.
G202	Cancellation of additive manual intervention.

The difference between exclusive and additive interventions is that the exclusive one (G200) interrupts the execution of the program to activate the jog mode whereas the additive one (G201) lets you jog an axis while executing the programmed movements.

Feedrate behavior

The feedrate of the jogging movements during manual intervention is independent from the active "F" and may be defined by the operator using instructions in high-level language; a different feedrate may be set for each work mode (incremental or continuous JOG). If not defined, the movements are carried out at the feedrate set by the machine manufacturer.

The feedrate may be varied between 0% and 200% using the feedrate override switch on the CNC's operator panel which affects the same way the programmed "F" and the feedrate of manual intervention.

Properties of the functions

Functions G201, G202 (modal) and G200 (not modal) are incompatible with each other.

On power-up, after executing an M02 or M30, and after an EMERGENCY or RESET, the CNC assumes function G202.

8.**TOOL PATH CONTROL**
Manual intervention (G200/G201/G202)**FAGOR** **CNC 8060**
CNC 8065

(REF. 1402)

8.9.1 Additive manual intervention (G201/G202)

With additive manual intervention, it is possible to jog the axes using handwheels or the JOG keys (continuous or incremental) while executing the program.

It may be applied on any axis of the machine. It cannot be applied on the spindle even if it can work in positioning mode.

G201

Activation of additive manual intervention.

To activate the additive manual intervention, program G201 followed by the axes affected by it using the instruction "#AXIS[<axes>]".

Function G201 must always be followed by the "#AXIS" instruction defining at least one axis.

G202

Cancellation of additive manual intervention

To cancel the additive manual intervention, program G202 followed by the axes to be canceled using the instruction AXIS[<axes>].

Programming G202 alone cancels manual intervention on all the axes.

```

...
N100 G71 G90 X0 Y0 F400
N110 G201 #AXIS [X, Z]      (Activate additive manual intervention on the X-Z axes)
N120 G01 X100 Y50          (The X-Z axes may be jogged)
N130 G202 #AXIS [X]        (Cancel manual intervention on X)
N140 G01 X50 Y150          (The Z axis may be jogged)
N150 G202 #AXIS [Z]        (Cancel manual intervention on Z)
...
N200 G201 #AXIS [X, Y, Z]  (Activate additive manual intervention on the X-Y-Z axes)
N220 G01 X100 Y50          (The X-Y-Z axes may be jogged)
N230 G202                  (Cancel intervention on all axes)
...

```

Considerations

Axis machine parameters MANFEEDP, IPOFEEDP, MANACCP, IPOACCP determine the feedrate and maximum acceleration permitted for each type of movement (jog or automatic). If the addition of the two exceeds 100%, it will be up to the user to ensure that the two movements are not simultaneous on the same axis because it may cause the dynamics to overshoot.

8.

8.9.2 Exclusive manual intervention (G200)

With exclusive manual intervention, the axes may be jogged using handwheels or JOG keys (continuous or incremental) by interrupting the execution of the program.

To cancel manual intervention and resume program execution, press the [CYCLE START] key.

It may be applied on any axis of the machine. It cannot be applied on the spindle even if it can work in positioning mode.



G200

Exclusive manual intervention

To activate exclusive manual intervention, program G200 followed by the axes affected by it using the instruction "#AXIS[<axes>]".

Programming G200 alone selects manual intervention on all the axes.

```

...
N100 G71 G90 X0 Y0 F400
N110 G200 #AXIS [X, Z]           (Interrupts program execution. Activates manual intervention on
                                  the X-Z axes)
                                  (Press the [CYCLE-START])

N120 G01 X100 Y100
N130 G200                       (Interrupts program execution. Activates manual intervention on
                                  all axes)
                                  (Press the [CYCLE-START])

N140 G01 X50 Y150
N150 G01 X0 Y0
...

```

Considerations

If a manual intervention is executed before a circular interpolation and one of the axes involved in the circular interpolation is jogged, it could issue an error message indicating that a circle has been programmed wrong or it may execute a circle other than the one programmed.

8.

TOOL PATH CONTROL
Manual intervention (G200/G201/G202)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

8.9.3 Jogging feedrate.

With these instructions, it is possible to set the feedrate and the movements in jog mode when manual intervention is active. The following may be defined with these instructions:

- The axis feedrate for manual intervention in each work mode (continuous or incremental JOG) and handwheel resolution. These values may be defined before or after activating manual intervention and stay active until the end of the program or a reset.
- The limits for the movements made with additive manual intervention. These limits are ignored when executing the movements by program. The limits may be defined after activating manual intervention and stay active until it is deactivated.

#CONTJOG Continuous JOG

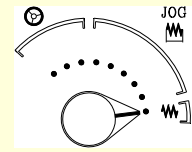
This instruction defines the indicated axis feedrate for continuous JOG.

The programming format is:

#CONTJOG [<F>] <Xn>

Parameter	Meaning
<F>	Feedrate.
<Xn>	Axis.

The feedrate will be programmed in mm/min. or inches/min. according to the active units.



...

N100 #CONTJOG [400] X Feedrate in continuous JOG. X axis.

N110 #CONTJOG [600] Y Feedrate in continuous JOG. Eje Y.

N120 G201 #AXIS [X,Y]

...

#INCJOG Incremental JOG

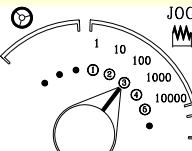
This instruction defines the indicated incremental movement and axis feedrate for each incremental JOG position of the selector switch.

The programming format is:

#INCJOG [<inc1>, <F>]... [<inc10000>, <F>] <Xn>

Parameter	Meaning
<inc>	Increment in each position of the incremental jog.
<F>	Feedrate in each position of the incremental jog.
<Xn>	Axis.

The feedrate will be programmed in mm/min. or inches/min. and the movement in mm. or inches according to the active units.



...

N100 #INCJOG [[0.1,100][0.5,200][1,300][5,400][10,500]] X

N110 G201 #AXIS [X]

...

The movements and feedrates of the X axis in each position are:

- (1) 0.1mm a 100mm/min.
- (2) 0.5mm a 200mm/min.
- (3) 1mm a 300mm/min.
- (4) 5mm a 400mm/min.
- (5) 10mm a 500mm/min.

8.

TOOL PATH CONTROL
Manual intervention (G200/G201/G202)



CNC 8060
CNC 8065

(REF. 1402)

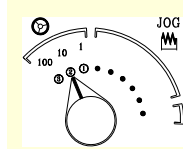
#MPG Handwheels

This instruction defines the distance per handwheel pulse for the indicated axis at each position of the selector switch.

The programming format is:

#MPG [<pos1>, <pos2>, <pos3>] <Xn>

Parameter	Meaning
<pos>	Resolution in each handwheel position.
<Xn>	Axis.



...
 N100 #MPG [0.1,1,10] X
 N110 G201 #AXIS [X]
 N120 #MPG [0.5] Y
 ...

The distance per X axis handwheel pulse in each position is:

- (1) 0.1 mm/turn of the handwheel.
- (2) 1mm/turn of the handwheel.
- (3) 10mm/turn of the handwheel.



This instruction sets the distance per handwheel pulse in a time period equal to the CNC's cycle time. If the feedrate required for this movement exceeds the maximum set by the machine manufacturer, the feedrate will be limited to this value and the axis moving distance will be less than what has been programmed in the instruction.

Example: If a 5 mm move is programmed and the cycle time is 4 msec, the resulting feedrate is 1250 mm/sec. If the maximum feedrate is limited to 1000 mm/s, the actual distance moved will be 4 mm.

#SET OFFSET Limits

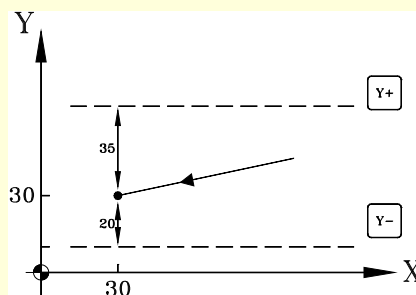
This instruction defines the upper and lower limits of the indicated axis, within which the axes can be jogged during additive manual intervention.

The programming format is:

#SET OFFSET [<lower>, <upper>] <Xn>

Parameter	Meaning
<lower>	Lower limit.
<upper>	Upper limit
<axis>	Axis.

The limits are referred to the axis position. The lower limit must be less than or equal to zero and the upper limit must be zero or greater than zero.



...
 G01 X30 Y30 F550
 N100 G201 #AXIS [Y] (Y axis manual interpolation)
 N110 #SET OFFSET [-20,35] Y (Y axis limits)
 N120 G01 X100 Y45 F400
 ...



TOOL PATH CONTROL
Manual intervention (G200/G201/G202)



CNC 8060
CNC 8065

(REF. 1402)

#SYNC POS Synchronization

This instruction synchronizes the preparation coordinate with the execution one and assumes the additive manual offset.

The programming format is:

```
#SYNC POS
```

8.**TOOL PATH CONTROL**

Manual intervention (G200/G201/G202)



CNC 8060
CNC 8065

(REF. 1402)

9.1 Square corner (G07/G60)

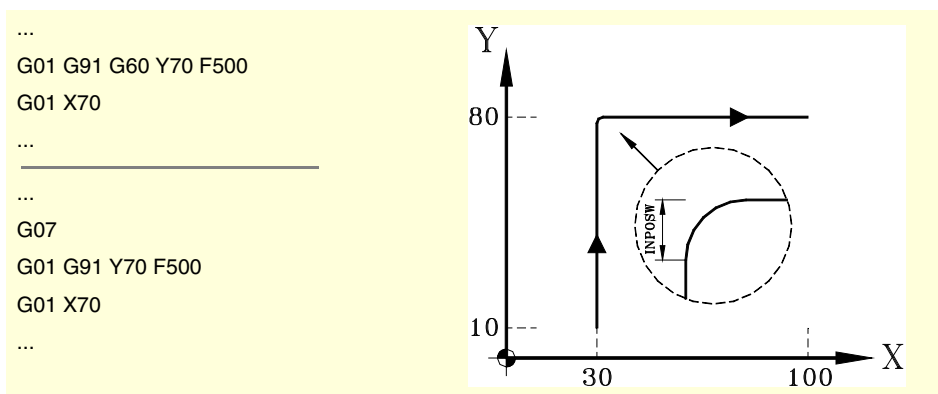
When working in square corner mode, the CNC does not begin executing the next movement until the axis reaches the programmed position. The CNC considers that the programmed position has been reached when the axis is located within the "in position" zone set by the machine manufacturer (OEM) [A.M.P. "INPOSW"].

Programming

The square corner machining mode may be activated by program using two different functions:

- G07 Square corner (modal).
- G60 Square corner (not modal).

Function G07 remains active throughout the program whereas function G60 only affects the block that contains it; therefore, it can only be added to a block containing a movement.



The theoretical and real profiles are the same, thus resulting in square corners as shown in the figure.

Properties of the functions

Function G07 is modal and incompatible with G05, G50, G60, G61 and the HSC mode.

Function G60 is not modal. After it is executed, the CNC restores the function G05, G07, G50 or HSC that was previously active.

On power-up, after executing M02 or M30 and after an EMERGENCY or a RESET, the CNC assumes function G05, G07 or G50 as set by the OEM [G.M.P. "ICORNER"].

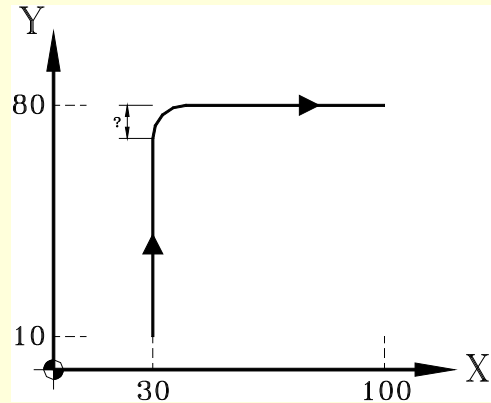
9.2 Semi-rounded corner (G50)

When working in semi-rounded corner, the CNC starts executing the next movement once the theoretical interpolation of the current move is completed without waiting for the axes to be in position. The distance from the programmed position to the position where the next move begins depends on the feedrate of the axis.

Programming

The semi-rounded corner machining mode may be activated by program using function G50.

```
...
G50
G01 G91 Y70 F500
G01 X70
...
```



This function provides rounded corners as shown in the figure.

Properties of the function

Function G50 is modal and incompatible with G05, G07, G60, G61 and the HSC mode.

On power-up, after executing M02 or M30 and after an EMERGENCY or a RESET, the CNC assumes function G05, G07, G50 or HSC as set by the OEM [G.M.P. "ICORNER"].

9.

GEOMETRY ASSISTANCE
Semi-rounded corner (G50)



CNC 8060
CNC 8065

(REF. 1402)

9.3 Controlled corner rounding, radius blend, (G05/G61)

When working in round corner, it is possible to control the corners of the programmed profile. How this machining is carried out depends on the type of corner rounding selected.

Programming

The type of corner rounding is selected with the "#ROUNDPAR" instruction and stays active until a different one is selected. Section "9.3.1 Types of corner rounding" of this chapter shows a description of the different types of corner rounding available.

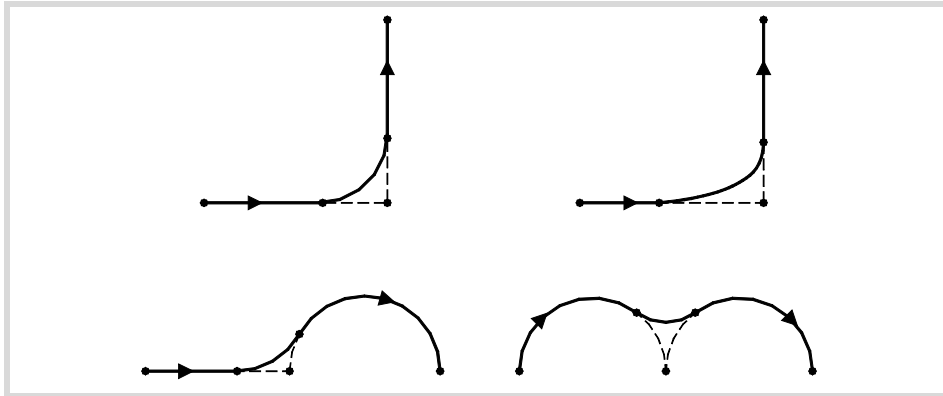
After selecting the type of corner rounding, it may be activated by program using functions:

- | | |
|-----|--|
| G05 | Control corner rounding, radius blend (modal). |
| G61 | Control corner rounding, radius blend (not modal). |

Function G05 remains active throughout the program whereas function G61 only affects the block that contains it; therefore, it can only be added to a block containing a movement.

Considerations

This operation may be applied to any corner, regardless of whether it is defined between straight and/or circular paths.



The corner is machined along a curved path, not with arcs. The shape of the curve depends on the type of corner rounding selected and on the dynamic conditions (feedrate and acceleration) of the axes involved.

Properties of the functions

Function G05 is modal and incompatible with G07, G50, G60, G61 and the HSC mode.

Function G61 is not modal. After it is executed, the CNC restores the function G05, G07, G50 or HSC that was previously active.

On power-up, after executing M02 or M30 and after an EMERGENCY or a RESET, the CNC assumes function G05, G07 or G50 as set by the OEM [G.M.P. "ICORNER"].

9.

GEOMETRY ASSISTANCE

Controlled corner rounding, radius blend, (G05/G61)

FAGOR 
**CNC 8060
CNC 8065**

(REF. 1402)

9.3.1 Types of corner rounding

There are 5 different corner contouring types. The first 4 execute the different corner rounding types whereas the last one executes a square corner. The last one is aimed at special machines (Laser, water jet, etc.), that use it to avoid "burning" the corner, thus not being recommended for a milling machine.

Corner rounding is selected and defined through the parameters associated with the "#ROUNDPAR" instruction. This instruction may have associated up to 6 parameters whose meaning will depend on the type of corner rounding selected.

Type 1 #ROUNDPAR [1,e]

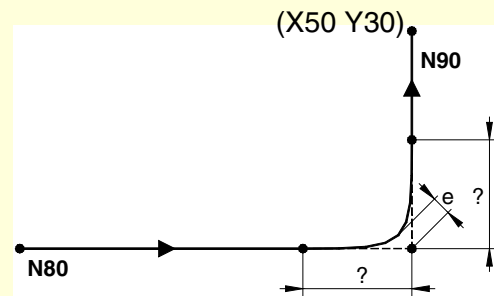
Set the maximum deviation allowed between the programmed point and the profile resulting from rounding the corner.

The corner is rounded by giving priority to the machining dynamic conditions (feedrate and acceleration). It executes the machining operation that is closer to the programmed point without exceeding the programmed deviation and that does not require decreasing the programmed feedrate "F".

```
...
N70 #ROUNDPAR [1,3]
N80 G01 G91 G61 X50 F850
N90 G01 Y30
...


---


...
N70 #ROUNDPAR [1,3]
N75 G05
N80 G01 G91 X50 F850
N90 G01 Y30
...
```



#ROUNDPAR [1,e]

e : Distance between programmed point and real profile.

The distances from the programmed point to the points where the corner rounding begins and ends are calculated automatically and they cannot be greater than half the path programmed in the block. Both distances will be the same, except when one of them is limited to half the programmed path.

For this type of corner rounding, only the values of the first two parameters of the "#ROUNDPAR" instructions are used, therefore, all parameters need not be included.

Type 2 #ROUNDPAR [2,f]

Set the percentage of the active feedrate "F" to be used to carry out the corner rounding.

It executes the corner rounding closer to the programmed point and that may be machined at the set feedrate percentage.

9.

GEOMETRY ASSISTANCE

Controlled corner rounding, radius blend, (G05/G61)



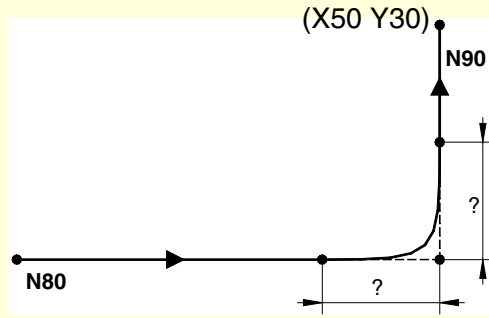
CNC 8060
CNC 8065

(REF. 1402)

```

...
N70 #ROUNDPAR [2,40]
N80 G01 G91 G61 X50 F850
N90 G01 Y30
...

```



#ROUNDPAR [2,f]
 f : Percentage of feedrate "F" for corner rounding.

The distances from the programmed point to the points where the corner rounding begins and ends are calculated automatically and they cannot be greater than half the path programmed in the block. Both distances will be the same, except when one of them is limited to half the programmed path.

For this type of corner rounding, only the values of the first two parameters of the "#ROUNDPAR" instructions are used, therefore, all parameters need not be included.

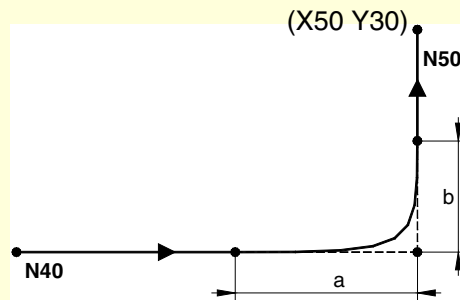
Type 3
#ROUNDPAR [3,a,b]

It defines the distance from the programmed point to the points where corner rounding begins and ends.

```

...
N20 #ROUNDPAR [3,10,3]
N30 G00 G90 X0 Y0
N40 G01 X50 F850
N50 Y30
...

```



#ROUNDPAR [3,a,b]
 a : Distance to the starting point of corner rounding.
 b : Distance to the end point of the corner rounding.
 Depending on parameters "a" and "b", a deviation may occur at the programmed profile (as shown in the example).

For this type of corner rounding, only the values of the first three parameters of the "#ROUNDPAR" instructions are used, therefore, all parameters need not be included.

Type 4
#ROUNDPAR [4,e]

Set the maximum deviation allowed between the programmed point and the profile resulting from rounding the corner.

The corner is rounded by giving priority to the machining geometrical conditions. The programmed machining operation is executed by decreasing the programmed feedrate "F" if necessary.

9.

GEOMETRY ASSISTANCE
 Controlled corner rounding, radius blend, (G05/G61)



CNC 8060
 CNC 8065

(REF. 1402)

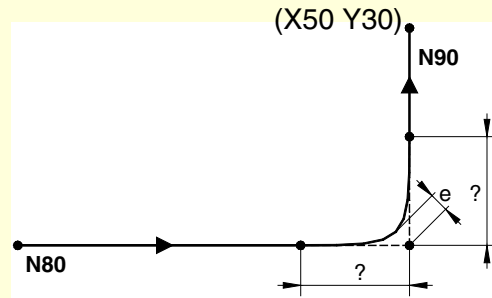
9.

GEOMETRY ASSISTANCE

Controlled corner rounding, radius blend, (G05/G61)

```
...
N70 #ROUNDPAR [4,3]
N80 G01 G91 G61 X50 F850
N90 G01 Y30
...
```

```
...
N70 #ROUNDPAR [4,3]
N75 G05
N80 G01 G91 X50 F850
N90 G01 Y30
...
```



```
#ROUNDPAR [4,e]
```

e : Distance between programmed point and real profile.

The distances from the programmed point to the points where the corner rounding begins and ends are calculated automatically and they cannot be greater than half the path programmed in the block. Both distances will be the same, except when one of them is limited to half the programmed path.

For this type of corner rounding, only the values of the first two parameters of the "#ROUNDPAR" instructions are used, therefore, all parameters need not be included.

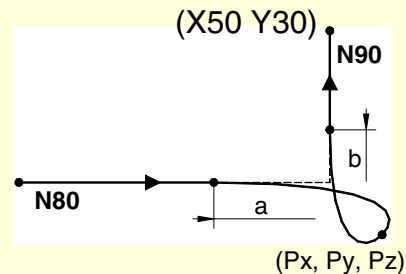
Type 5

```
#ROUNDPAR [5,a,b,Px,Py,Pz]
```

It defines the distance from the programmed point to the points where corner rounding begins and ends. Also set the coordinates of an intermediate point of the corner rounding.

```
...
N70 #ROUNDPAR [5,7,4,55,-15,0]
N80 G01 G91 G61 X40 F850
N90 G01 Y20
...
```

```
...
N70 #ROUNDPAR [5,7,4,55,-15,0]
N75 G05
N80 G01 G91 X40 F850
N90 G01 Y20
...
```



```
#ROUNDPAR [5,a,b,Px,Py,Pz]
```

a : Distance to the starting point of corner rounding.

b : Distance to the end point of the corner rounding.

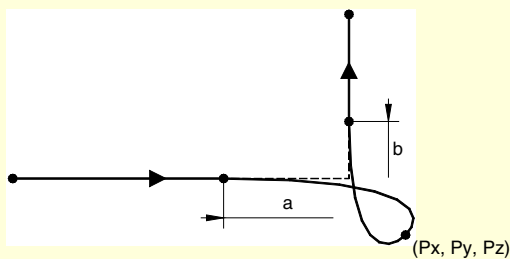
Px : X coordinate of the intermediate point.

Py : Y coordinate of the intermediate point.

Pz : Z coordinate of the intermediate point.

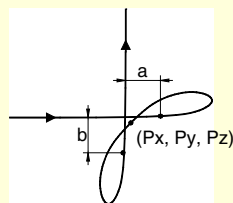
This type of corner rounding only uses the values of the first six parameters of the "#ROUNDPAR" instruction.

In this type of corner rounding, the shape of the curve depends on the position of the intermediate point and on the distance from the programmed point to the starting and ending points of the corner rounding.



```

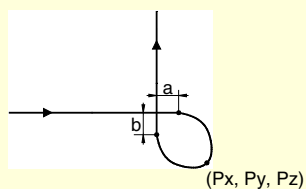
...
G92 X0 Y0
G71 G90
#ROUNDPAR [5,-30,-30,55,-5,0]
G01 G61 X50 F850
N90 G01 Y40
    
```



"a" and "b" distances negative and greater (in absolute value) than the distance from the programmed point to the intermediate point on each axis (about 4 times).

```

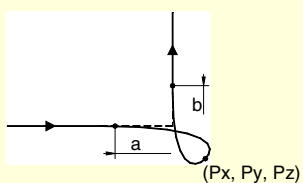
...
G92 X0 Y0
G71 G90
#ROUNDPAR [5,-5,-5,65,-15,0]
G01 G61 X50 F850
G01 Y40
    
```



"a" and "b" distances, negative and smaller (in absolute value) than the distance from the programmed point to the intermediate point on each axis.

```

...
G92 X0 Y0
G71 G90
#ROUNDPAR [5,5,5,65,-15,0]
G01 G61 X50 F850
G01 Y40
    
```



Positive "a" and "b" distances.

9.

GEOMETRY ASSISTANCE

Controlled corner rounding, radius blend, (G05/G61)



CNC 8060
CNC 8065

(REF. 1402)

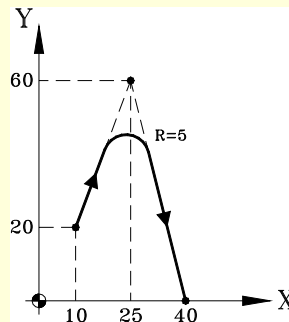
9.4 Corner rounding, radius blend, (G36)

G36 may be used to round a corner with a particular radius without having to calculate either the center or the starting and ending points of the arc.

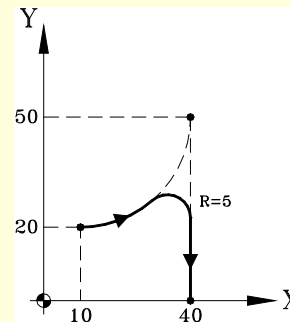
Programming

The rounding definition must be programmed between the two paths that define the corner to be rounded. These paths may be linear and/or circular.

The programming format is "G36 I<radius>", where the radius value is programmed in millimeters or in inches, depending on which are the active units.



```
G01 G90 X25 Y60
G36 I5
G01 X40 Y0
```



```
G03 G90 X40 Y50 I0 J30
G36 I5
G01 X40 Y0
```

Considerations

The "I" value of the rounding radius remains active until another value is programmed, therefore, it won't be necessary to program it in successive rounding operations with the same radius.

The "I" value of the rounding radius is also used by functions:

G37 (Tangential entry) as entry radius.

G38 (Tangential exit) as exit radius.

G39 (corner chamfering) as size of the chamfer.

This means that the rounding radius set in G36 will be the new value of the entry radius, exit radius or chamfer size when programming one of these functions or vice versa.

```
N10 G01 X10 Y10 F600
N20 G01 X10 Y50
N30 G36 I5 (Rounding. Radius=5)
N40 G01 X50 Y50
N50 G36 (Rounding. Radius=5)
N60 G01 X50 Y10
N70 G39 (Chamfer. Size=5)
N80 G01 X90 Y10
N90 G39 I10 (Chamfer. Size=10)
N100 G01 X90 Y50
N110 G36 (Rounding. Radius=10)
N120 G01 X70 Y50
N130 M30
```

9.

GEOMETRY ASSISTANCE
Corner rounding, radius blend, (G36)



CNC 8060
CNC 8065

(REF. 1402)

The programmed rounding feedrate depends on the type of movement programmed afterwards:

- If the next movement is in G00, the rounding will be carried out in G00.
- If the next movement is in G01, G02 or G03, the rounding will be carried out at the feedrate programmed in rounding definition block. If no feedrate has been programmed, the rounding will be carried out at the active feedrate.

```
N10 G01 G94 X10 Y10 F600
N20 G01 X10 Y50
N30 G36 I5 (Chamfering in G00)
N40 G00 X50 Y50
N50 G36 (Chamfer. F=600mm/min.)
N60 G01 X50 Y10
N70 G36 F300 (Chamfer. F=300mm/min.)
N80 G01 X90 Y10 F600
N90 M30
```

When defining a plane change between the two paths that define a rounding, it is carried out in the plane where the second path is defined.

```
N10 G01 G17 X10 Y10 Z-10 F600
N20 X10 Y50 Z0 (X-Y plane)
N30 G36 I10
N40 G18 (Z-X plane. The rounding is carried out in this plane)
N50 X10 Z30
N60 M30
```

Properties of the function

Function G36 is not modal, therefore, it must be programmed every time a corner is to be rounded.

9.

GEOMETRY ASSISTANCE
Corner rounding, radius blend, (G36)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

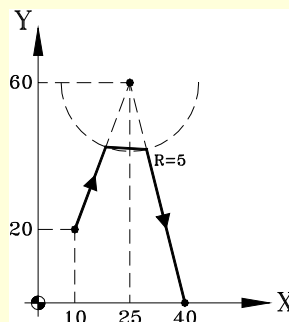
9.5 Corner chamfering, (G39)

Function G39 may be used to insert a chamfer of a particular size without having to calculate the intersection points.

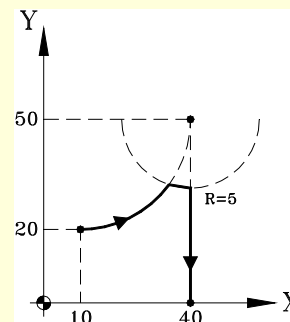
Programming

The chamfer definition must be programmed between the two paths that define the corner to be chamfered. These paths may be linear and/or circular.

The programming format is "G39 I<size>", where the size value is programmed in millimeters or in inches, depending on which are the active units.



```
G01 G90 X25 Y60
G39 I5
G01 X40 Y0
```



```
G03 G90 X40 Y50 I0 J30
G39 I5
G01 X40 Y0
```

Considerations

The "I" value of the chamfer size remains active until another value is programmed, therefore, it won't be necessary to program it in successive chamfering operations of the same size.

The "I" value of the chamfer size is also used by functions:

G36 (Corner rounding) as rounding radius.

G37 (Tangential entry) as entry radius.

G38 (Tangential exit) as exit radius.

This means that the chamfer size set in G39 will be the new value of the entry radius, exit radius or rounding radius when programming one of these functions or vice versa.

```
N10 G01 X10 Y10 F600
N20 G01 X10 Y50
N30 G36 I5 (Rounding. Radius=5)
N40 G01 X50 Y50
N50 G36 (Rounding. Radius=5)
N60 G01 X50 Y10
N70 G39 (Chamfer. Size=5)
N80 G01 X90 Y10
N90 G39 I10 (Chamfer. Size=10)
N100 G01 X90 Y50
N110 G36 (Rounding. Radius=10)
N120 G01 X70 Y50
N130 M30
```



CNC 8060
CNC 8065

(REF. 1402)

The programmed chamfering feedrate depends on the type of movement programmed afterwards:

- If the next movement is in G00, the chamfer will be carried out in G00.
- If the next movement is in G01, G02 or G03, the chamfer will be carried out at the feedrate programmed in chamfer definition block. If no feedrate has been programmed, the chamfer will be carried out at the active feedrate.

```
N10 G01 G94 X10 Y10 F600
N20 G01 X10 Y50
N30 G39 I5 (Chamfering in G00)
N40 G00 X50 Y50
N50 G39 (Chamfer. F=600mm/min.)
N60 G01 X50 Y10
N70 G39 F300 (Chamfer. F=300mm/min.)
N80 G01 X90 Y10 F600
N90 M30
```

When defining a plane change between the two paths that define a chamfer, it is carried out in the plane where the second path is defined.

```
N10 G01 G17 X10 Y10 Z-10 F600
N20 X10 Y50 Z0 (X-Y plane)
N30 G39 I10
N40 G18 (Z-X plane. The chamfer is carried out in this plane)
N50 X10 Z30
N60 M30
```

Properties of the function

Function G39 is not modal, therefore, it must be programmed every time a corner is to be chamfered.

9.

GEOMETRY ASSISTANCE
Corner chamfering, (G39)

9.6 Tangential entry (G37)

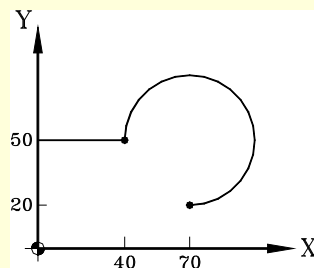
Function G37 may be used to begin machining with a tangential entry of the tool without having to calculate the intersection points.

Programming

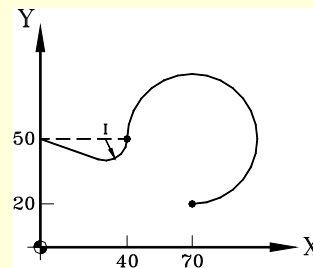
Tangential entry must be programmed alone in the block and after the block whose path is to be modified; this path must be a straight line (G00 or G01).

The programming format is "G37 I<radius>", where the radius value is programmed in millimeters or in inches, depending on which are the active units.

The linear path before the tangential entry must have a length equal to or greater than twice the entry radius. Likewise, the radius must be positive and when working with tool radius compensation, it must be greater than the tool radius.



```
G01 G90 X40 Y50 F800
G02 X70 Y20 I30 J0
```



```
G01 G90 X40 Y50 F800
G37 I10
G02 X70 Y20 I30 J0
```

Considerations

The "I" value of the tangential entry radius remains active until another value is programmed, therefore, it won't be necessary to program it in successive tangential entries with the same radius.

The "I" value of the entry radius is also used by functions:

G36 (Corner rounding) as rounding radius.

G38 (Tangential exit) as exit radius.

G39 (corner chamfering) as size of the chamfer.

This means that the entry radius set in G37 will be the new value of the exit radius, rounding radius or chamfer size when programming these functions or vice versa.

Properties of the function

Function G37 is not modal, therefore, it must be programmed every time a tangential entry is to be carried out.

9.

GEOMETRY ASSISTANCE
Tangential entry (G37)



CNC 8060
CNC 8065

(REF. 1402)

9.7 Tangential exit (G38)

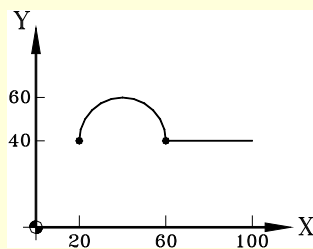
Function G38 may be used to end machining with a tangential exit of the tool without having to calculate the intersection points.

Programming

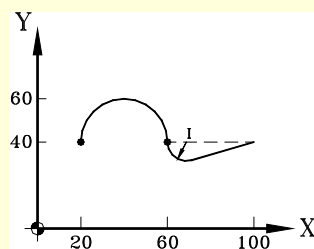
Tangential exit must be programmed alone in the block and before the block whose path is to be modified; this path must be a straight line (G00 or G01).

The programming format is "G38 I<radius>", where the radius value is programmed in millimeters or in inches, depending on which are the active units.

The linear path after the tangential exit must have a length equal to or greater than twice the exit radius. Likewise, the radius must be positive and when working with tool radius compensation, it must be greater than the tool radius.



```
G02 X60 Y40 I20 J0 F800
G01 X100
```



```
G02 X60 Y40 I20 J0 F800
G38 I10
G01 X100
```

Considerations

The "I" value of the tangential exit radius remains active until another value is programmed, therefore, it won't be necessary to program it in successive tangential exits with the same radius.

The "I" value of the exit radius is also used by functions:

G36 (Corner rounding) as rounding radius.

G37 (Tangential entry) as entry radius.

G39 (corner chamfering) as size of the chamfer.

This means that the exit radius set in G38 will be the new value of the entry radius, rounding radius or chamfer size when programming these functions or vice versa.

Properties of the function

Function G38 is not modal, therefore, it must be programmed every time a tangential exit is to be carried out.

9.

GEOMETRY ASSISTANCE
Tangential exit (G38)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

9.8 Mirror image (G11, G12, G13, G10, G14)

Mirror image may be used to repeat the programmed machining operation in a symmetrical position with respect one or more axes. When using with mirror image, the movements of the axes where mirror image is applied are executed with the opposite sign.

Programming

Mirror image may be applied by program using these functions:

G10	Mirror image cancellation.
G11	Mirror image in X.
G12	Mirror image in Y.
G13	Mirror image in Z.
G14	Mirror image in the programmed directions.

G10

Mirror image cancellation

It cancels mirror image on all axes, including the mirror image activated with G14.

If it is added to a path defining block, the mirror image will be canceled before the movement.

G11 to G13

Mirror image on X, on Y or on Z

Functions G11, G12 and G13 activate mirror image on the X, Y and Z axis respectively. These functions do not cancel each other, thus being possible to keep mirror image active on several axes at the same time.

If they are added to a path defining block, the mirror image will be activated before the movement.

```
G11
  (Mirror image on the X axis)
G12
  (Mirror image on the Y axis. The one on the X axis remains active)
...
G10
  (Mirror image cancellation on all the axes)
```

G14

Mirror image in the programmed directions

It may be used to activate or cancel mirror image on any axis. The activation or cancellation is defined by programming function G14 and then, the axes next to the value that determines whether to activate (<axis>=-1) or to cancel (<axis>=1) mirror image on that axis.

```
G14 X-1 V-1
  (Mirror image on the X and V axes)
G14 X1
  (Mirror image cancellation on the X axis. The one on the V axis remains active)
...
G14 V1
  (Mirror image cancellation on the V axis)
```

9.

GEOMETRY ASSISTANCE
Mirror image (G11, G12, G13, G10, G14)

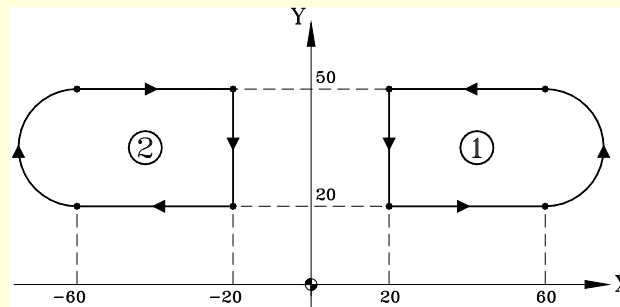


CNC 8060
CNC 8065

(REF. 1402)

Considerations

When machining a profile with a mirror image, the machining direction is opposite to that of the programmed profile. If this profile has been defined with tool radius compensation, when activating the mirror image, the CNC will change the type of compensation (G41 or G42) to obtain the programmed profile.



```

%PROGRAM          (Main program)
G00 G90 X0 Y0 Z20
...
G11               (Machining of profile 1)
...               (Mirror image on X).
...               (Machining of profile 2)
G10               (Mirror image cancellation on all the axes)
M30
    
```

Properties of the functions

Functions G11, G12, G13 and G14 are modal. Once mirror image is active on an axis, it stays active until canceled with G10 or G14.

Functions G10 and G14 are incompatible with each other as well as with G11, G12 and G13.

On power-up and after an emergency, the CNC cancels mirror image (it assumes function G10). The behavior of the mirror image function after executing an M02 or M30 and after a reset depends on the setting of machine parameter MIRRORCANCEL.

MIRRORCANCEL	Behavior of the mirror image function.
Yes	Functions M02, M30 and reset cancel the mirror image function.
No	Functions M02, M30 and reset do not affect the mirror image function.

9.

GEOMETRY ASSISTANCE
Mirror image (G11, G12, G13, G10, G14)

FAGOR 

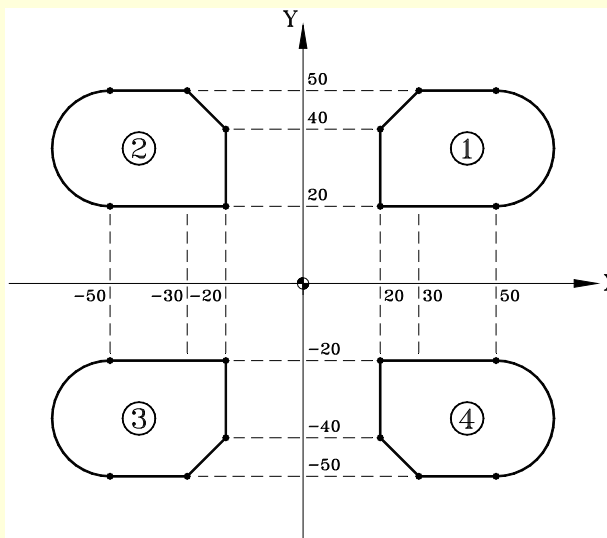
CNC 8060
CNC 8065

(REF. 1402)

Programming examples.

9.

GEOMETRY ASSISTANCE
Mirror image (G11, G12, G13, G14)



```

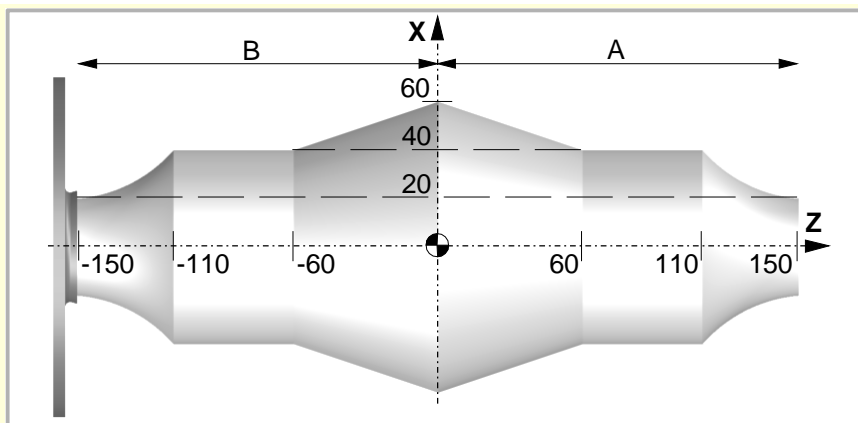
%L PROFILE                                ("PROFILE" subroutine definition)
N10 G00 X10 Y10
N20 G01 Z0 F400
N30 G01 X20 Y20 F850
N40 X50
N50 G03 X50 Y50 R15
N60 G01 X30
N70 X20 Y40
N80 Y20
N90 X10 Y10
N100 Z10 F400
M29                                        (End of subroutine)

%PROGRAM                                  (Main program)
N10 G0 X0 Y0 Z10
N20 LL PROFILE                            (Call to a subroutine. Profile 1)
N30 G11                                    (Mirror image on X).
N40 LL PROFILE                            (Call to a subroutine. Profile 2)
N50 G12                                    (Mirror image on X and Y).
N60 LL PROFILE                            (Call to a subroutine. Profile 3)
N70 G14 X1                                (Mirror image cancellation on the X axis)
N80 LL PROFILE                            (Call to a subroutine. Profile 4)
N90 G10                                    (Mirror image cancellation on all the axes)
N100 G00 X0 Y0 Z50
M30
    
```



CNC 8060
CNC 8065

(REF. 1402)



```

%L PROFILE (Subroutine that defines the "A" zone of the part)
G90 G00 X40 Z150
G02 X80 Z110 R60
G01 Z60
G01 X124 Z-6
M17

%PROGRAM (Main program)
G18 G151 (Main plane ZX and programming in diameters)
V.A.ORG1[1].Z=160 (Definition of the first zero offset, G54)
G54 (Application of the zero offset)
LL PROFILE (Call to a subroutine. Machining of the "A" zone)
G0 Z-150 (Movement to avoid colliding with the part)
G13 (Mirror image on Z)
LL PROFILE (Call to a subroutine. Machining of the "B" zone)
G0 Z-200 (Return to the starting point)
G10 (Cancel mirror image on all the axes)
M30
    
```

9.

GEOMETRY ASSISTANCE
 Mirror image (G11, G12, G13, G10, G14)



CNC 8060
 CNC 8065

(REF. 1402)

9.9 Coordinate system rotation, pattern rotation, (G73)

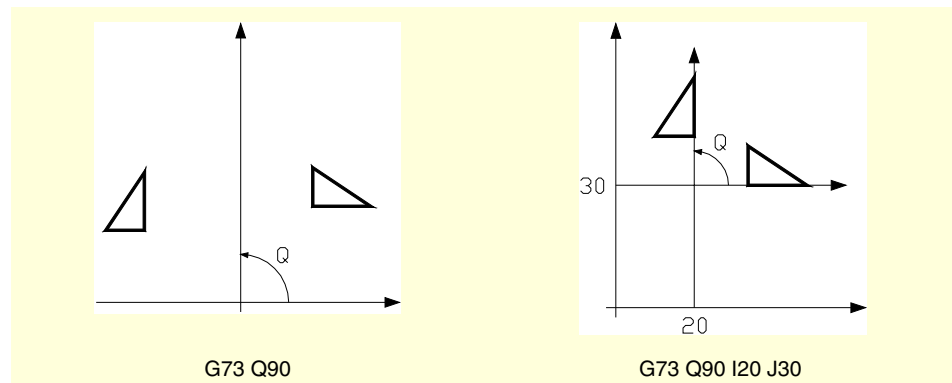
Function G73 may be used to rotate the coordinate system taking as rotation center the active reference system (part zero) or the programmed rotation center.

Programming

The coordinate system rotation must be programmed alone in the block. The programming format is "G73 Q I J", where:

- Q Indicates the rotation angle in degrees.
- I, J They define the abscissa and ordinate of the rotation center. They must be defined in absolute coordinates referred to part zero.
When programmed, both parameters must be programmed.
If not programmed, the part zero will be assume as the rotation center.

To cancel the coordinate (pattern) rotation, program function G73 alone, with no additional data.

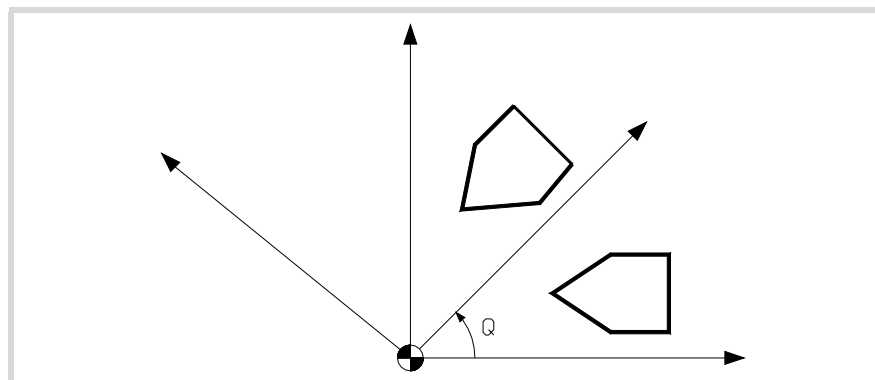


Therefore, function G73 may be programmed as follows:

- G73 Q I J Rotate "Q" degrees with the center at abscissa "I" and ordinate "J" referred to part zero.
- G73 Q Rotate "Q" degrees with the center at part zero.
- G73 Cancellation of coordinate (pattern) rotation.

Considerations

Function G73 is incremental; i.e. the various "Q" values programmed are added up.



The "I" and "J" values are affected by the active mirror images. If any mirror image function is active, the CNC applies first the mirror image and then the coordinate system rotation.

Properties of the function

Function G73 is modal. The coordinate rotation stays active until it is canceled by function G73 or until the work plane is changed.

9.

GEOMETRY ASSISTANCE
Coordinate system rotation, pattern rotation, (G73)



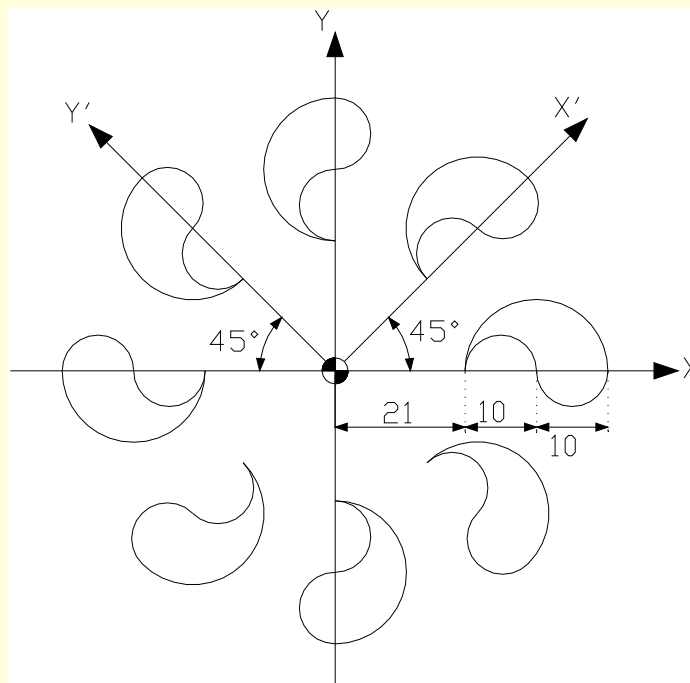
CNC 8060
CNC 8065

(REF. 1402)

On power-up, after executing an M02 or M30, and after an EMERGENCY or RESET, the CNC cancels the active coordinate system (pattern) rotation.

Programming example

Assuming the initial point is X0 Y0:



```

%L PROFILE                (Subroutine with the profile)
G01 X21 Y0 F300
G02 G31 Q0 I5 J0
G03 G31 Q0 I5 J0
G03 G31 Q180 I-10 J0
M29                        (End of subroutine)

%PROGRAM                  (Program)
$FOR P0=1, 8, 1           (Repeats the profile and the pattern rotation 8 times)
  LL PROFILE              (Machining of the profile)
  G73 Q45                 (Coordinate rotation)
$ENDFOR
M30
    
```

9.

GEOMETRY ASSISTANCE

Coordinate system rotation, pattern rotation, (G73)



CNC 8060
CNC 8065

(REF. 1402)

9.10 General scaling factor

It may be used to enlarge or reduce the scale of the programmed paths and contours. This permits using a single program to make sets of similar profiles of different dimensions.

The general scaling factor is applied to all the axes of the channel. After activating the scaling factor, all the programmed coordinates will be multiplied by the defined scaling factor until it is canceled or a new scaling factor is programmed.

Activate the scaling factor

The general scaling factor may be activated using the commands `G72` or `#SCALE`. Either command may be used.

Although there are two different commands, the scaling factor is the same; i.e. the scaling factor programmed with `G72` modifies the one programmed with `#SCALE` and vice versa.

Programming with `G72`.

Program function `G72` and then the scaling factor set by parameter `S` as follows.

```
G72 S<scale>
```

Programming function `G72` alone or a scaling factor of `.0` or `.1` cancels the active scaling factor.

Parameter "S" that sets the scaling factor must be programmed after function `G72`. If programmed before, it will be interpreted as spindle speed.

Programming with `#SCALE`.

Program the instruction `#SCALE` and then the scaling factor as follows. The brackets must be programmed.

```
#SCALE [<scale>]
```

Programming a scaling factor of `.0` or `.1` cancels the active scaling factor.

```
G72 S2
#SCALE [3]
G72
#SCALE [1]
```

Cancel the scaling factor

The general scaling factor is canceled using the same commands `G72` or `#SCALE`, setting a scaling factor of `.0` or `.1`. When using function `G72`, the scaling factor is also canceled by programming it alone in the block.

Considerations

Activating the machine coordinate system (`#MCS ON`) cancels the scaling factor temporarily until the machine coordinate system is canceled (`#MCS OFF`).

While the machine coordinate system is active, the scaling factor can neither be activated nor modified.

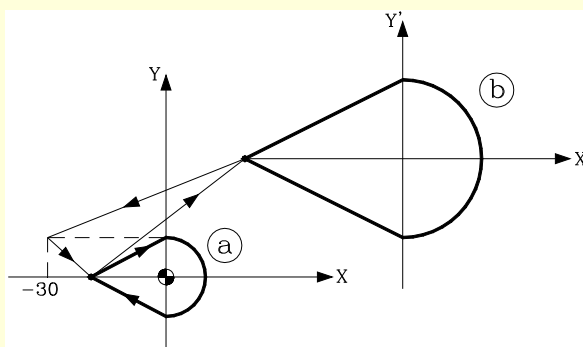
Properties

The scaling factor stays active until canceled with another scaling factor.

On power-up, after executing an `M02` or `M30`, and after an `EMERGENCY` or `RESET`, the CNC cancels the active scaling factor.

9.

Programming example



%L PROFILE (Profile to be machined)

G90 X-19 Y0

G01 X0 Y10 F150

G02 X0 Y-10 I0 J-10

G01 X-19 Y0

M29

%PROGRAM

G00 X-30 Y10

#CALL PROFILE (Machining of profile "a")

G92 X-79 Y-30 (Coordinate preset)

#SCALE [2] (Applies a scaling factor of 2)

#CALL PROFILE (Machining of profile "b")

#SCALE [1] (Cancels the scaling factor)

M30

9.

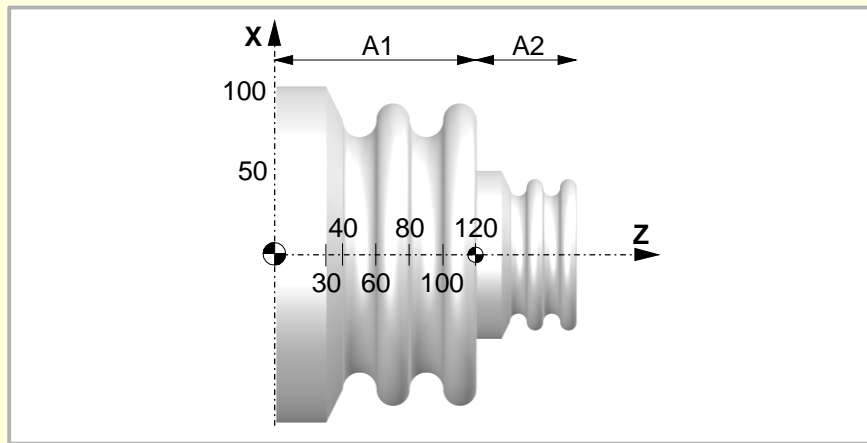
GEOMETRY ASSISTANCE
General scaling factor

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

9.

GEOMETRY ASSISTANCE
 General scaling factor


%L PROFILE	(Subroutine that defines the "A1" zone of the part)
G90 G01 X200 Z0	
G01 X200 Z30 F150	
G01 X160 Z40	
G03 X160 Z60 R10	
G02 X160 Z80 R10	
G03 X160 Z100 R10	
G02 X160 Z120 R10	
M29	
%PROGRAM	(Main program)
G18 G151	(Main plane ZX and programming in diameters)
G00 X206 Z0	(Approach)
LL PROFILE	(Call to a subroutine. Machining of the "A1" zone)
G92 Z0	(Coordinate preset)
G72 S0.5	(Application of the scaling factor)
LL PROFILE	(Call to a subroutine. Machining of the "A2" zone)
G72 S1	(Cancellation of the scaling factor)
G01 X0	
G0 X250 Z200	(Return to the starting point)
G53	(Cancellation of coordinate preset)
M30	

ADDITIONAL PREPARATORY FUNCTIONS

10

10.1 Dwell (G04)

The dwell may be used to interrupt the execution of the program for the specified period of time.

Programming

The dwell may be activated using the commands G04 or #TIME. Either command may be used.

Programming with G04

Program function G04 and then the time in seconds set by parameter K as follows.

G04 K<time>

G04 K0.5	(0.5 second dwell)
P1=3	
G04 KP1	(3 second dwell)
G04 K[P1+7]	(10 second dwell)

This syntax also admits the following format when the time is programmed using a constant.

G04<time>

G04 5	(5 second dwell)
-------	------------------

Programming with #TIME

Program the instruction #TIME and then the time in seconds as follows. The brackets may be omitted when the time is programmed with a constant or a parameter.

#TIME [<time>]

#TIME [5]	(5 second dwell)
#TIME 5	
P1=2	
#TIME P1	(2 second dwell)
#TIME [P1+3]	(5 second dwell)

Properties of the function

Function G04 is not modal, therefore, it must be programmed every time a dwell is desired.

Function G04 may also be programmed as G4.



CNC 8060
CNC 8065

(REF. 1402)

10.2 Software limits by program (G198-G199)

The software limits for each axis may be changed by program using the following functions:

- G198 Setting of lower software travel limits.
- G199 Setting of upper software travel limits.

When programming G198 or G199, the CNC interprets that the coordinates programmed next set the new software limits.

```
G198 X-1000 Y-1000
      (New lower limits X=-1000 Y=-1000)
G199 X1000 Y1000
      (New upper limits X=1000 Y=1000)
```

Depending on the active work mode G90 or G91, the position of the new limits will be defined in absolute coordinates (G90) in the machine reference system or in incremental coordinates (G91) referred to the current active limits.

```
G90
G198 X-800
      (New lower limit X=-800)
G199 X500
      (New upper limit X=500)
G90 X-800
G91
G198 X-700
      (New incremental lower limit X=-1500)
```

Considerations

Defining both limits with a ·0· value cancels the axis limits, including the ones set by machine parameters. To restore the limits, they must be programmed again.

Both limits may be positive or negative; but the lower limits must be smaller than the upper ones. Otherwise, the might not move in any direction.

If after setting the new limits, an axis positions beyond them, it will be possible to move that axis towards the work zone (between those limits).

The software limits are always defined in radius, regardless of the setting of parameter DIAMPROG and the active function G151/G152.

Properties of the functions

On power-up or after validating the axis machine parameters the CNC assumes the software limits set by the machine parameters.

After an M02 or M30 and after an EMERGENCY or a RESET, the CNC maintains the software limits set by G198 and G199.

10.

ADDITIONAL PREPARATORY FUNCTIONS
Software limits by program (G198-G199)



CNC 8060
CNC 8065

(REF. 1402)

10.3 Hirth axes (G170-G171)

Hirth axes may be canceled and activated by program. When a Hirth axis is active, it can only reach concrete positions whereas when deactivated, it behaves like a normal rotary or linear axis and can reach any position.

Programming

Hirth axes may be canceled and activated using:

```
G170      Hirth axes OFF.
G171      Hirth axes ON.
```

To activate or cancel a Hirth axis, program its relevant function and then the axes to be activated or canceled and the number indicating the order in which those axes will be activated.

Assuming that the B and C axes have been set as rotary Hirth axes with a 10^3 pitch.

```
G171 B1 C2      (Activate B and C axes as Hirth axes)
G01 B50 C20     (Interpolate both axes)
...
G170 B1         (B axis deactivation)
G01 X100 B33
```

If when activating a Hirth axis, it is located in the wrong position, the CNC will issue a warning so the operator can turn it to a correct position.

Considerations

A Hirth axis must always be positioned at specific positions. When positioning, the active zero offset (preset or zero offset) is taken into consideration.

Both linear and rotary axes may be Hirth. Only those axes defined as Hirth axes by the machine manufacturer [A.M.P. "HIRTH"] may be activated as Hirth axes.

Properties of the functions

Functions G170 and G171 are modal and incompatible with each other.

On power-up, after an M02 or M30 and after an EMERGENCY or a RESET, the CNC activates all the Hirth axes.

10.

ADDITIONAL PREPARATORY FUNCTIONS
Hirth axes (G170-G171)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

10.4 Changing of parameter range of an axis (G112)

The CNC may have up to 4 sets of parameters for each axis to define different dynamic characteristics (acceleration, gains, etc.) for each of them.

The parameter set may be selected by program using function G112. This function does not carry out any physical change on the machine (gear change), it only assumes the parameters of the active set.

When using Sercos axes, function G112 also involves changing the drive's velocity gear.

Programming

Changing the parameter range of the axes.

To assume a different set of parameter, program G112 and then the axes and the new parameter set to be selected for each one of them.

```
...
G112 X2 Y3      (Selects the second set of parameters for the X axis, and the third one
                for the Y axis)
...
```

Changing the parameter set for the spindle.

In this case, changing the parameter set will be used when working in positioning mode (M19). When working in speed mode (M03/M04), function G112 will only change the parameter set; it is NOT the same as functions M41 through M44 because it does not make a physical gear change.

```
...
G112 S2        (Selects the second set of spindle parameters)
...
```

When making a gear change using M41 through M44, it is not necessary to program G112.

Properties of the function

Function G112 is modal.

After validating the machine parameters, every time a program is executed from the automatic mode, on power-up, after executing an M02 or M30, after an EMERGENCY or a RESET, the CNC acts as follows depending on the value assigned to machine parameter "DEFAULTSET".

If DEFAULTSET is 0, it maintains the range defined by G112. Otherwise, it assumes the range defined by machine parameter DEFAULTSET.



CNC 8060
CNC 8065

(REF. 1402)

Tool compensation allows programming the machining contour based on the dimensions of the part without taking into account the dimensions of the tool that will be used later on. This way, there is no need to calculate and redefine the tool path depending on the radius and length of each tool.

Types of compensation

Tool radius compensation (milling).

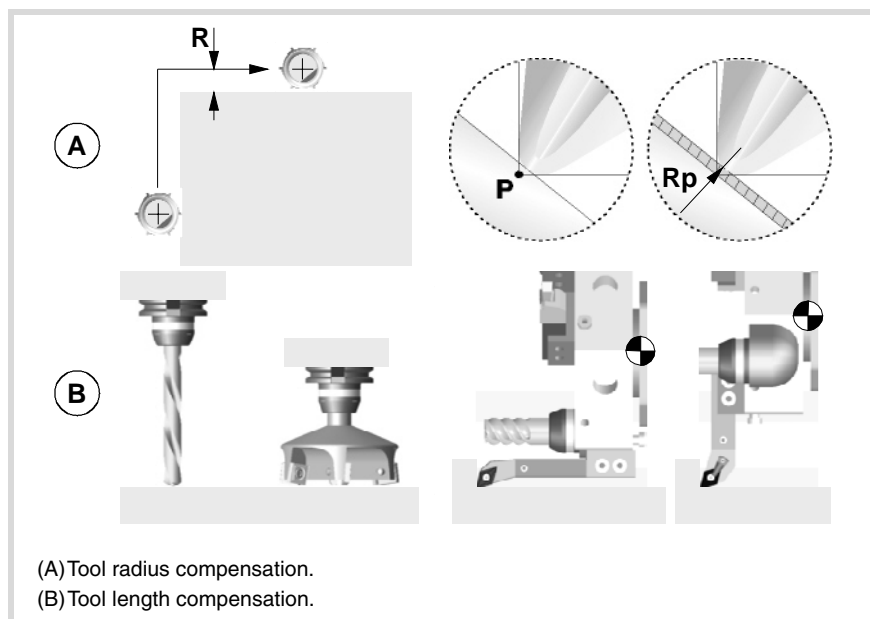
When working with tool radius compensation, the tool center follows the programmed path at a distance equal to the tool radius. Thus obtaining the right dimensions of the programmed part.

Tool radius compensation (lathe).

The CNC assumes as theoretical tool tip (P) the result of the sides used when calibrating the tool. Without tool radius compensation, the theoretical tool tip (P) travels the programmed path leaving machining ridges in incline and curved sections. With tool radius compensation, it takes into account the tool tip radius and the location code (shape) or type of tool obtaining the dimensions of the programmed part.

Tool length compensation.

When working with tool length compensation, the CNC compensates for the length difference between the different programmed tools.



Compensation values

The compensation values applied in each case is calculated from the tool dimensions.

- In tool radius compensation, the applied value is the sum of the radius and radius wear of the selected tool.
- In tool length compensation, the applied value is the sum of the length and length wear of the selected tool.

The tool "T" and the tool offset "D", containing the tool dimensions, may be selected anywhere in the program, even while tool compensation is active. If no tool offset is selected, the CNC assumes tool offset "D1".

11.

TOOL COMPENSATION



CNC 8060
CNC 8065

(REF. 1402)

11.1 Tool radius compensation

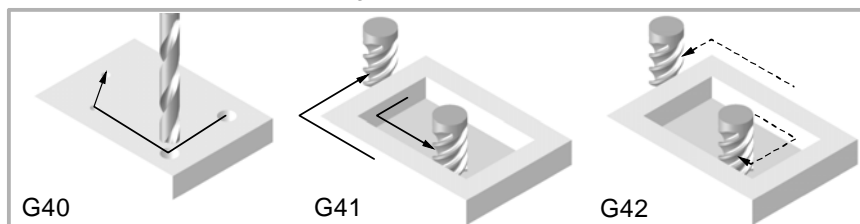
Radius compensation is applied in the active work plane, previously selected using functions G17 (XY plane), G18 (ZX plane), G19 (YZ plane) or G20 (user defined plane).

Programming

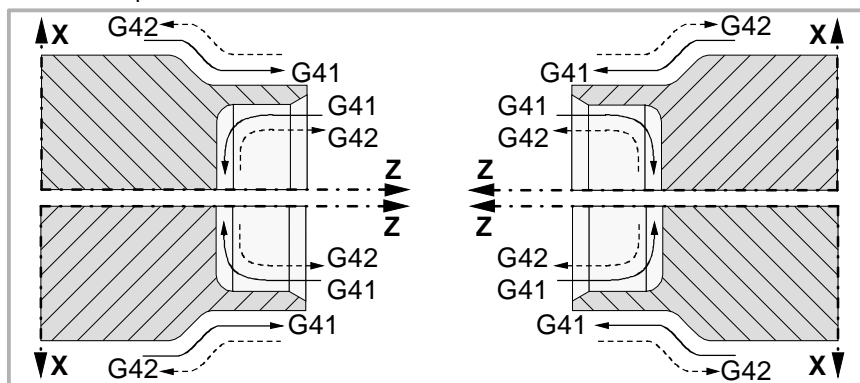
The functions for selecting tool radius compensation are:

- G41 Left-hand tool radius compensation.
- G42 Right-hand tool radius compensation.
- G40 Cancellation of tool radius compensation.

Tool radius compensation on a milling machine.



Tool radius compensation on a horizontal lathe.



Tool radius compensation on a vertical lathe.



Depending on the type of compensation selected (G41/G42), the CNC will place the tool will be placed to the left or to the right of the programmed path along the machining direction and will apply the compensation value. If no tool compensation is selected (G40), on a milling machine the CNC will place the tool center right on the programmed tool path; on a lathe, the CNC will place the theoretical tool tip on the programmed path.

Being tool radius compensation active, the CNC analyzes in advance the blocks to be executed in order to detect compensation errors related to steps, null arcs, etc. When detected, the CNC will not execute the blocks that cause them and the screen will display a warning to let the operator know that the programmed profile has been modified. A warning will come up for every profile correction made.

Properties of the functions

Functions G40, G41 and G42 are modal and incompatible with each other. On power-up, after executing an M02 or M30, and after an EMERGENCY or RESET, the CNC assumes function G40.

11.

TOOL COMPENSATION
Tool radius compensation



CNC 8060
CNC 8065

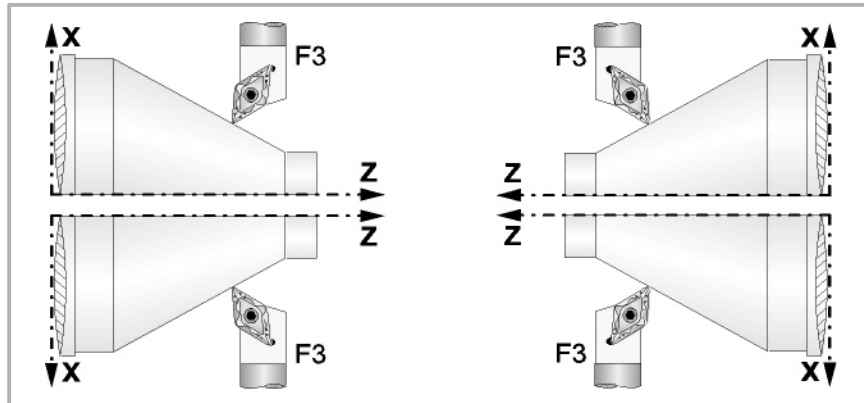
(REF. 1402)

11.1.1 Location code (shape or type) of the turning tools

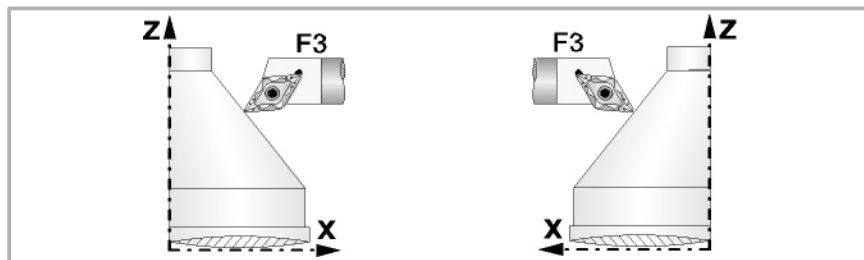
The location code indicates the type of tool and the sides used to calibrate it. The location code depends on the position of the tools and on the orientation of the machine axes.

The next example shows the location code F3 on different machines. Observe how the CNC keeps the relative position of the tool with respect to the axes.

Location code F3 on a horizontal lathe.



Location code F3 on a vertical lathe.



Here are the location codes available on most common horizontal lathes.

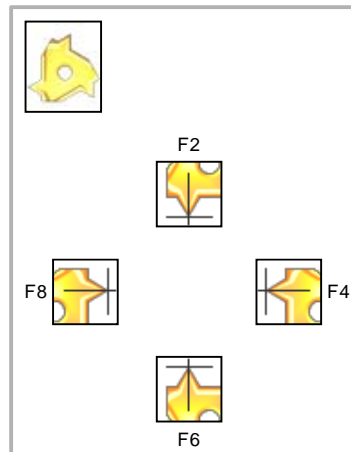
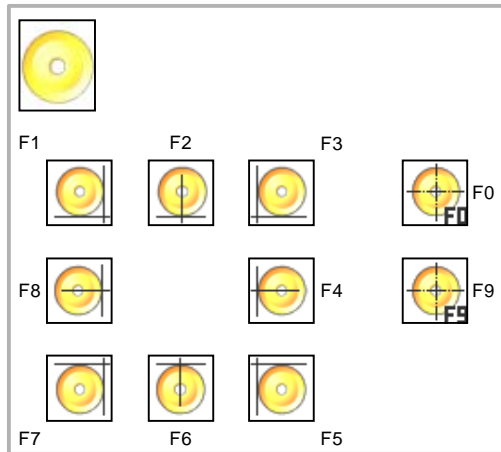
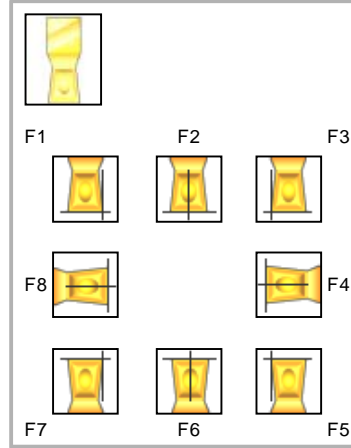
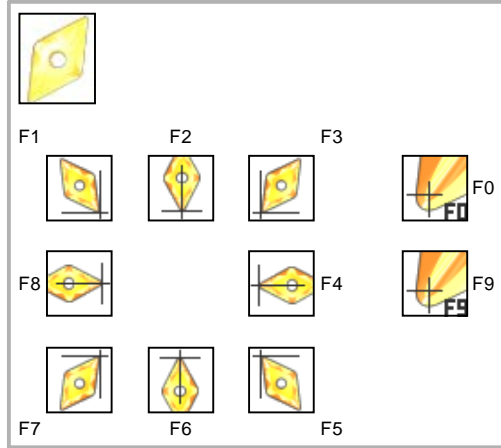
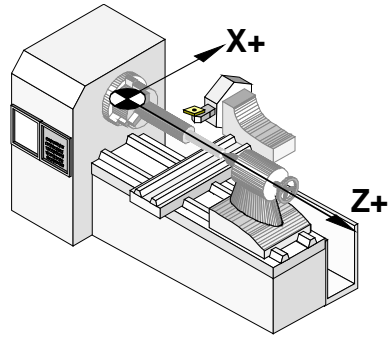
11.

TOOL COMPENSATION
Tool radius compensation



CNC 8060
CNC 8065

(REF. 1402)



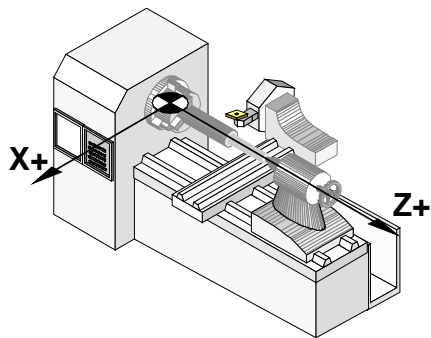
11.

TOOL COMPENSATION
Tool radius compensation



CNC 8060
CNC 8065

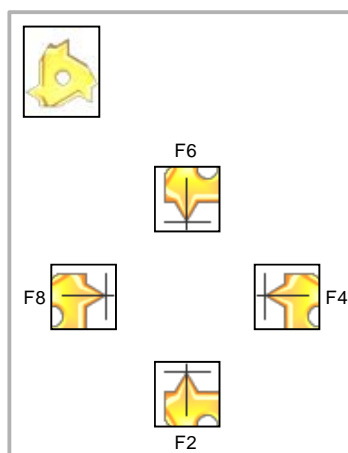
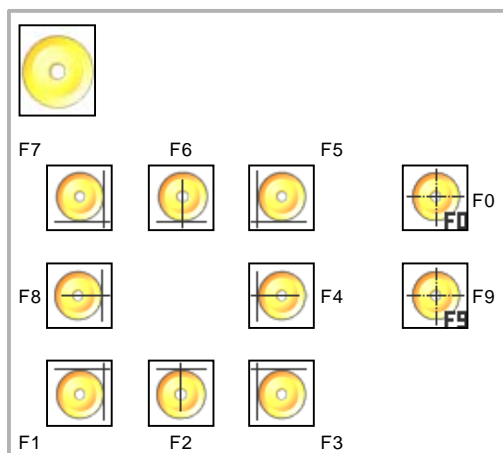
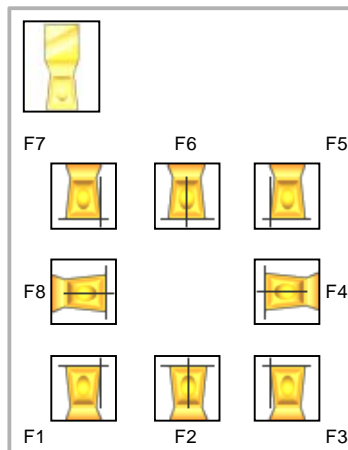
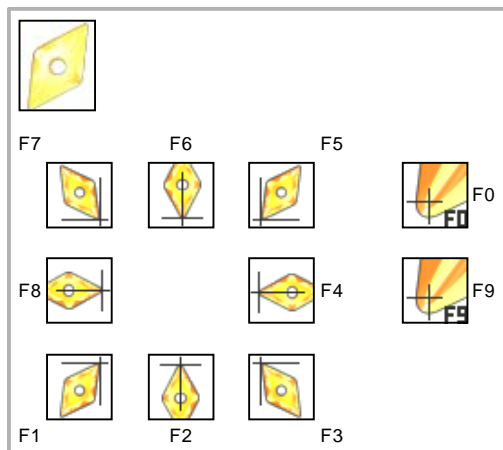
(REF. 1402)



11.

TOOL COMPENSATION

Tool radius compensation



CNC 8060
CNC 8065

(REF. 1402)

11.1.2 Functions associates with radius compensation

The functions associated with tool compensation may be programmed anywhere in the program, even while tool radius compensation is active.

SELECTING THE TYPE OF TRANSITION BETWEEN BLOCKS

The transition between blocks determines how the compensated paths are joined together.

Programming

The type of transition may be selected from the program by means of the following functions:

- G136 Circular transition between blocks.
- G137 Linear transition between blocks.

G136

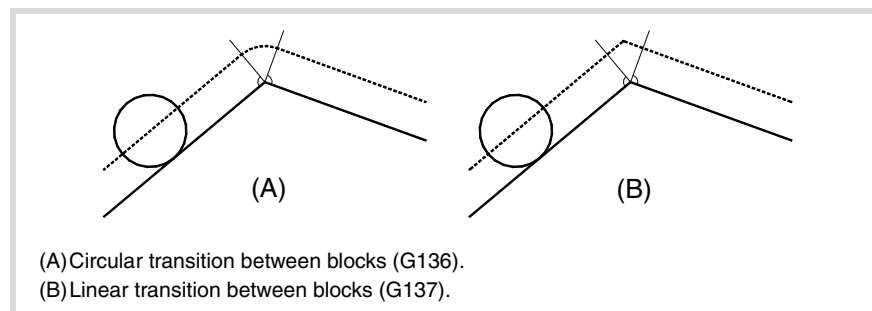
Circular transition between blocks.

Being function G136 active, the CNC joins the compensated paths using circular paths.

G137

Linear transition between blocks.

Being function G137 active, the CNC joins the compensated paths using linear paths.



Remarks

Later sections of this chapter offer graphic descriptions of how different paths are joined, depending on the type of transition selected (G136/G137).

Properties of the functions

Functions G136 and G137 are modal and incompatible with each other.

On power-up, after executing an M02 or M30, and after an EMERGENCY or RESET, the CNC assumes function G136 or G137 depending on the value of machine parameter IRCOMP.

11.

TOOL COMPENSATION
Tool radius compensation

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

HOW TOOL RADIUS IS ACTIVATED AND CANCELED

The functions associated with the strategy for activating and canceling establish how tool radius compensation starts and ends.

Programming

The type of strategy may be selected from the program by means of the following functions:

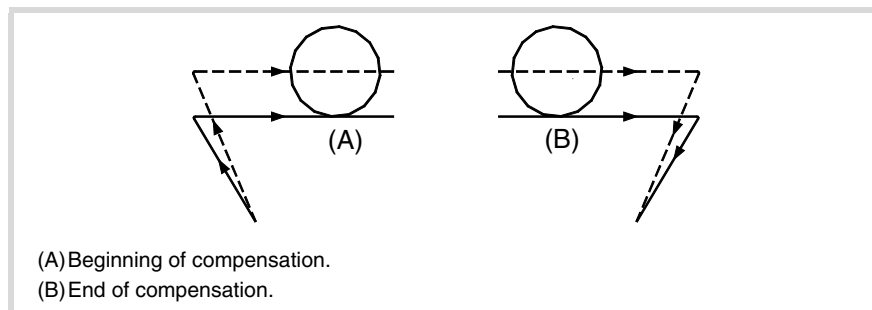
- | | |
|------|--|
| G138 | Direct activation/cancellation of tool compensation. |
| G139 | Indirect activation/cancellation of tool compensation. |

G138

Direct activation/cancellation of tool compensation.

When compensation is turned on, the tool moves directly to the perpendicular of the next path (without contouring the corner).

When compensation is turned off, the tool moves directly to the programmed end point (without counting the corner).

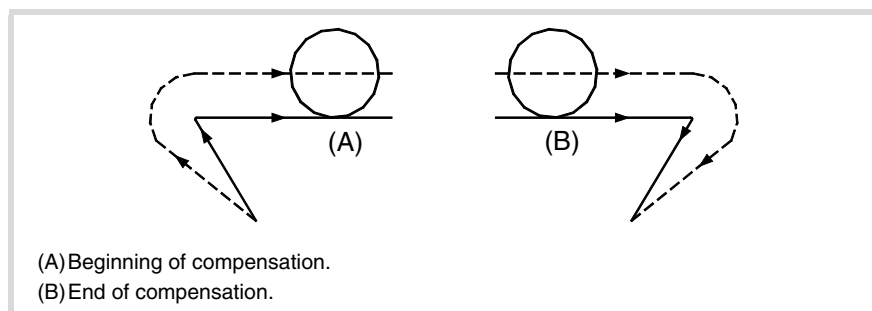


G139

Indirect activation/cancellation of tool compensation.

When compensation is turned on, the tool moves to the perpendicular of the next path contouring the corner.

When compensation is turned off, the tool moves to the end point contouring the corner.



The way the tool goes around the corner depends on the type of transition selected (G136/G37).

Remarks

Later sections of this chapter offer a graphic description of how tool radius compensation begins and ends depending on the selected type of compensation ON/OFF (G138/G139).

11.

TOOL COMPENSATION
Tool radius compensation

Properties of the functions

Functions G138 and G139 are modal and incompatible with each other.

On power-up, after executing an M02 or M30, and after an EMERGENCY or RESET, the CNC assumes function G139.

11.

TOOL COMPENSATION
Tool radius compensation

FAGOR 

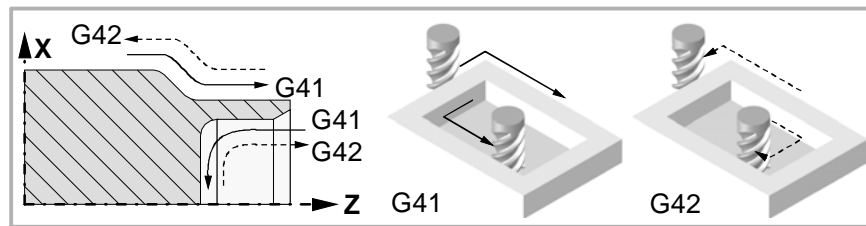
**CNC 8060
CNC 8065**

(REF. 1402)

11.1.3 Beginning of tool radius compensation

Tool radius compensation is selected with these functions:

- G41 Left-hand tool radius compensation.
- G42 Right-hand tool radius compensation.



After executing one of these functions, radius compensation will be active for the next movement in the work plane, that must be a linear movement.

The way radius compensation will begin depends on how it is activated G138/G139 and on the type of transition G136/G137 selected:

- G139/G136
The tool moves to the perpendicular of the next path contouring the corner along a circular path.
- G139/G137
The tool moves to the perpendicular of the next path contouring the corner along linear paths.
- G138
The tool moves directly to the perpendicular of the next path. Regardless of the type of transition (G136/G137) programmed.

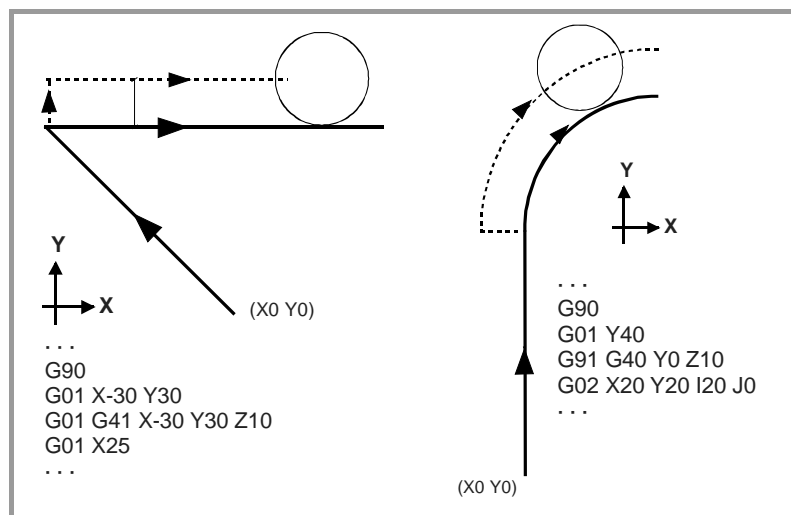
The following tables show the different ways tool compensation may begin, depending on the selected functions. The programmed path is shown with solid line and the compensated path with dashed line.

Beginning of the compensation without programmed movement

After activating the compensation, it may occur that the axes of the plane will not be involved in the first motion block. For example, because they have not been programmed, or the current tool position has been programmed or an incremental movement has been programmed.

In this case, the compensation is applied at the same point where the tool is, as follows. Depending on the first movement programmed in the plane, the tool moves perpendicular to the path on its starting point.

The first movement programmed in the plane may be either linear or circular.



CNC 8060
CNC 8065

(REF. 1402)

STRAIGHT-TO-STRAIGHT PATH

When the angle between paths is smaller than or equal to 180° , the way radius compensation is activated is independent from the functions G136/G137 or G138/G139 selected.

$0^\circ < \alpha < 90^\circ$	$\alpha = 90^\circ$
$90^\circ < \alpha < 180^\circ$	$\alpha = 180^\circ$

When the angle between paths is greater than 180° , the way radius compensation is activated depends on the functions selected for type of beginning (G138/G139) and type of transition (G136/G137).

G139/G136	G139/G137	G138
$180^\circ < \alpha < 270^\circ$	$180^\circ < \alpha < 270^\circ$	$180^\circ < \alpha < 270^\circ$
$\alpha = 270^\circ$	$\alpha = 270^\circ$	$\alpha = 270^\circ$
$270^\circ < \alpha < 360^\circ$	$270^\circ < \alpha < 360^\circ$	$270^\circ < \alpha < 360^\circ$

11.

TOOL COMPENSATION
Tool radius compensation

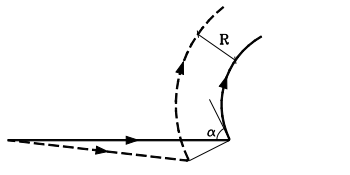
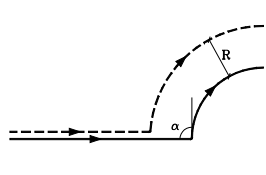
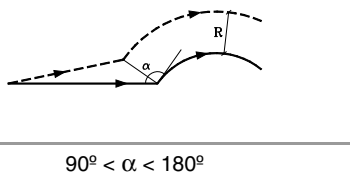
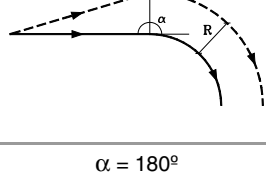


CNC 8060
CNC 8065

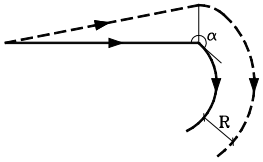
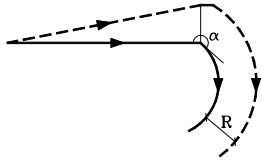
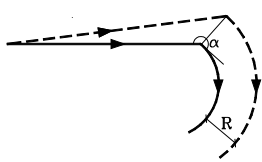
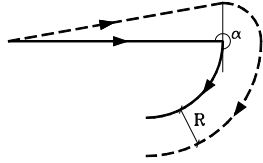
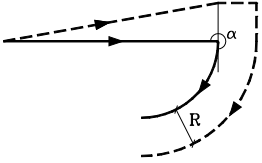
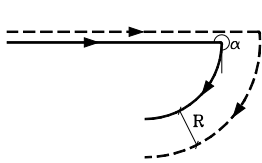
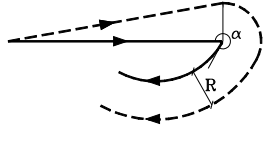
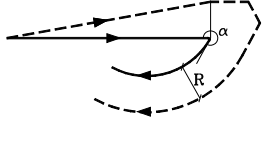
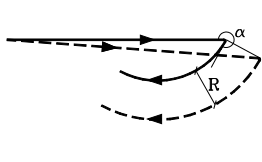
(REF. 1402)

STRAIGHT-TO-ARC PATH

When the angle between the straight path and the tangent of the arc is smaller than or equal to 180° , the way radius compensation is activated is independent from the functions G136/G137 and G138/G139 selected.

	
$0^\circ < \alpha < 90^\circ$	$\alpha = 90^\circ$
	
$90^\circ < \alpha < 180^\circ$	$\alpha = 180^\circ$

When the angle between the straight path and the tangent of the arc is greater than 180° the way radius compensation is activated depends on the type of beginning (G138/G139) and type of transition (G136/G137) selected.

G139/G136	G139/G137	G138
		
$180^\circ < \alpha < 270^\circ$	$180^\circ < \alpha < 270^\circ$	$180^\circ < \alpha < 270^\circ$
		
$\alpha = 270^\circ$	$\alpha = 270^\circ$	$\alpha = 270^\circ$
		
$270^\circ < \alpha < 360^\circ$	$270^\circ < \alpha < 360^\circ$	$270^\circ < \alpha < 360^\circ$

11.

TOOL COMPENSATION

Tool radius compensation



CNC 8060
CNC 8065

(REF. 1402)

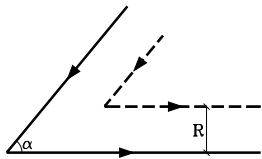
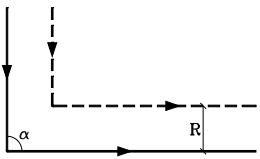
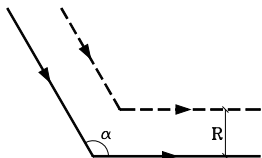
11.1.4 Sections of tool radius compensation

The way the compensated paths are joined depends on the type of transition selected (G136/G137).

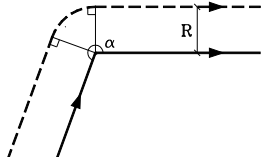
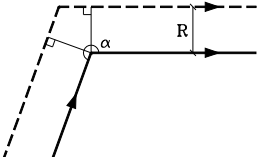
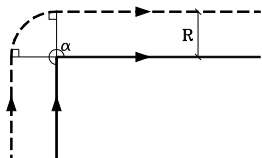
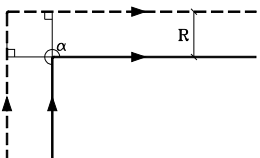
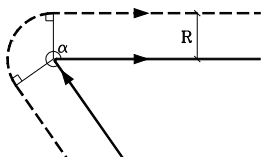
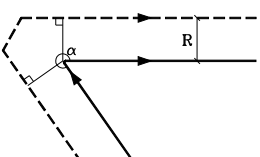
The following tables show the different transition possibilities between various paths depending on the selected function G136 or G137. The programmed path is shown with solid line and the compensated path with dashed line.

STRAIGHT-TO-STRAIGHT PATH

When the angle between paths is smaller than or equal to 180° , the transition between paths is independent from the G136/G137 function selected.

	
$0^\circ < \alpha < 90^\circ$	$\alpha = 90^\circ$
	
$90^\circ < \alpha < 180^\circ$	

When the angle between paths is greater than 180° , the way the compensated paths are joined depends on the type of transition selected (G136/G137).

G136	G137
	
$180^\circ < \alpha < 270^\circ$	$180^\circ < \alpha < 270^\circ$
	
$\alpha = 270^\circ$	$\alpha = 270^\circ$
	
$270^\circ < \alpha < 360^\circ$	$270^\circ < \alpha < 360^\circ$

11.

TOOL COMPENSATION
Tool radius compensation

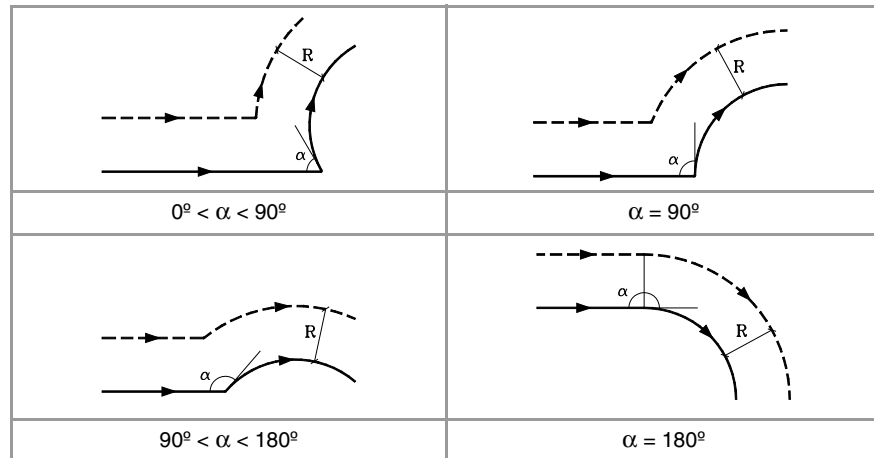


CNC 8060
CNC 8065

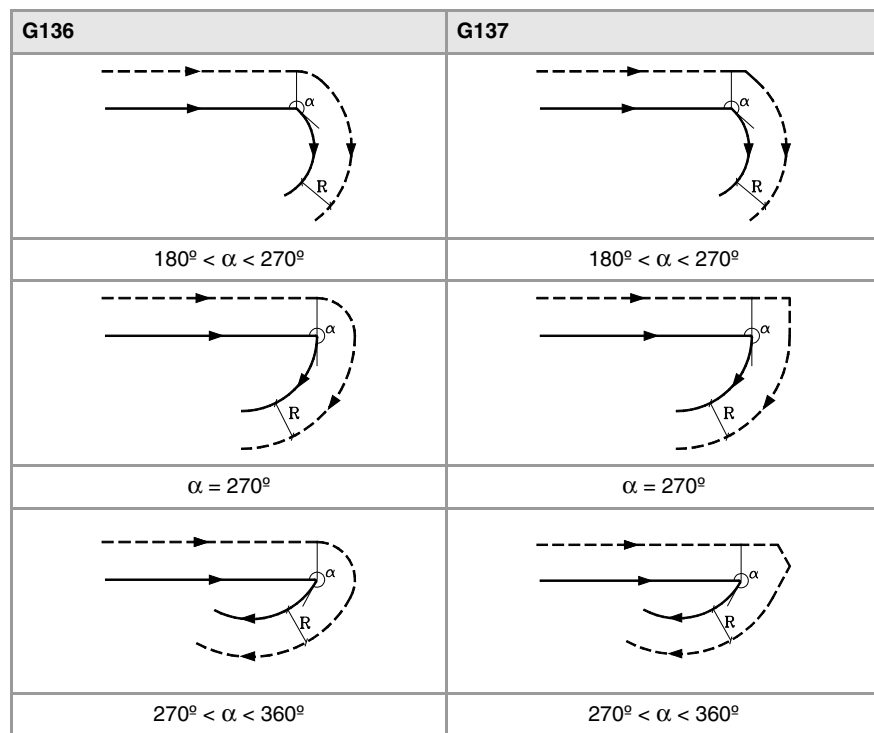
(REF. 1402)

STRAIGHT-TO-ARC PATH

When the angle between the straight line and the tangent of the arc is smaller than or equal to 180° , the transition between the paths is independent from the selected G136/G137 function.



When the angle between the straight path and the tangent of the arc is greater than 180° , the way the compensated paths are joined depends on the type of transition selected (G136/G137).



11.

TOOL COMPENSATION
Tool radius compensation

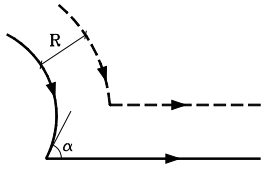
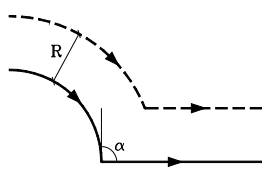
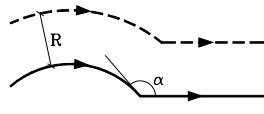
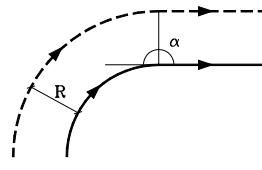


CNC 8060
CNC 8065

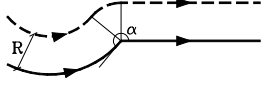
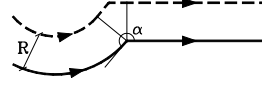
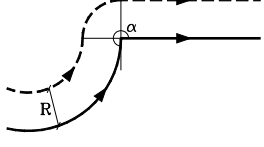
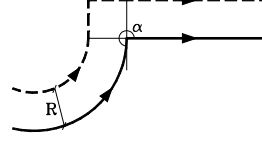
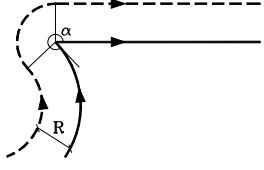
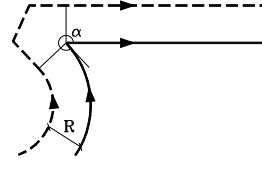
(REF. 1402)

ARC-TO-STRAIGHT PATH

When the angle between the tangent of the arc and the straight line is smaller than or equal to 180° , the transition between the paths is independent from the selected G136/G137 function.

	
$0^\circ < \alpha < 90^\circ$	$\alpha = 90^\circ$
	
$90^\circ < \alpha < 180^\circ$	$\alpha = 180^\circ$

When the angle between the tangent of the arc and the straight line is greater than 180° , the way the compensated paths are joined depends on the type of transition selected (G136/G137).

G136	G137
	
$180^\circ < \alpha < 270^\circ$	$180^\circ < \alpha < 270^\circ$
	
$\alpha = 270^\circ$	$\alpha = 270^\circ$
	
$270^\circ < \alpha < 360^\circ$	$270^\circ < \alpha < 360^\circ$

11.

TOOL COMPENSATION
Tool radius compensation

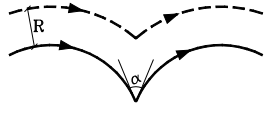
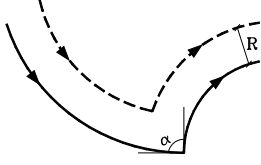
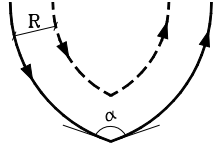
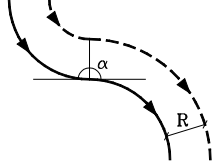
FAGOR 

CNC 8060
CNC 8065

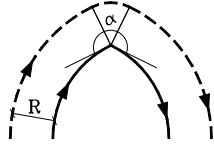
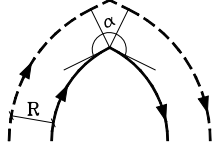
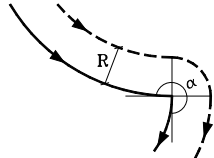
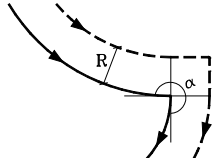
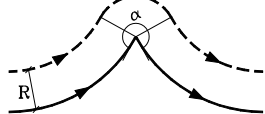
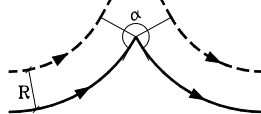
(REF. 1402)

ARC-TO-ARC PATH

When the angle between the tangents of the arcs is smaller than or equal to 180° , the transition between the paths is independent from the selected G136/G137 function.

	
$0^\circ < \alpha < 90^\circ$	$\alpha = 90^\circ$
	
$90^\circ < \alpha < 180^\circ$	$\alpha = 180^\circ$

When the angle between the tangents of the arcs is greater than 180° , the way the compensated paths are joined depends on the type of transition selected (G136/G137).

G136	G137
	
$180^\circ < \alpha < 270^\circ$	$180^\circ < \alpha < 270^\circ$
	
$\alpha = 270^\circ$	$\alpha = 270^\circ$
	
$270^\circ < \alpha < 360^\circ$	$270^\circ < \alpha < 360^\circ$

11.

TOOL COMPENSATION

Tool radius compensation



CNC 8060
CNC 8065

(REF. 1402)

11.1.5 Change of type of radius compensation while machining

The compensation may be changed from G41 to G42 or vice versa without having to cancel it with a G40. It may be changed in any motion block or even in a motionless one; i.e. without moving the axis of the plane or by programming the same point twice.

The last movement before the change and the first movement after the change are compensated independently. To change the type of compensation, the different cases are solved according to the following criteria:

A The compensated paths intersect each other.

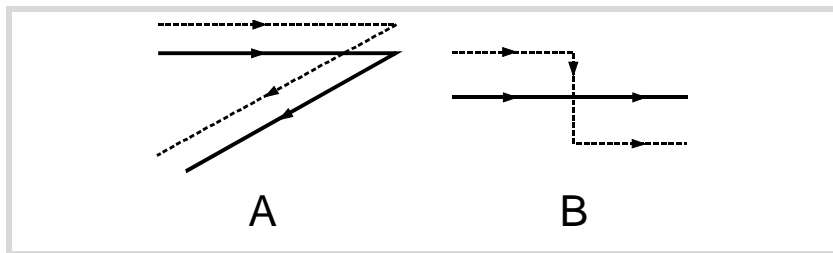
The programmed paths are compensated each on its corresponding side. The side change takes place in the intersection point between both paths.

B The compensated paths do not intersect each other.

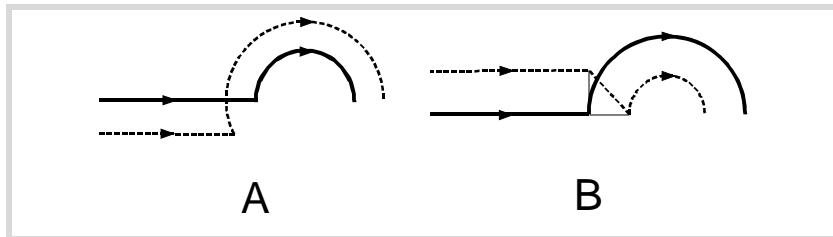
An additional section is inserted between the two paths. From the point perpendicular to the first path at the end point up to the point perpendicular to the second path at the starting point. Both points are located at a distance R from the programmed path.

Here is a summary of the different cases:

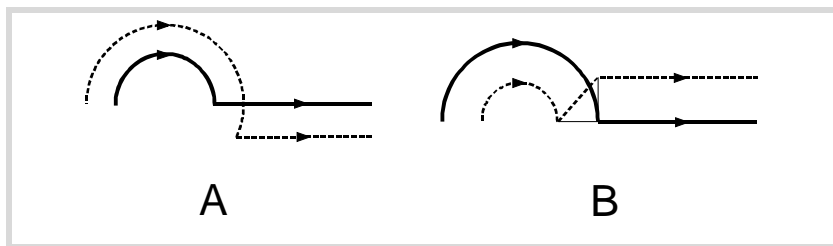
- Straight - straight path:



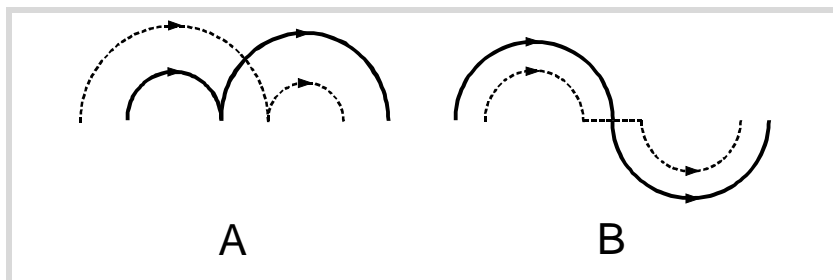
- Straight - circle path:



- Circle - straight path:



- Circle - circle path:



11.

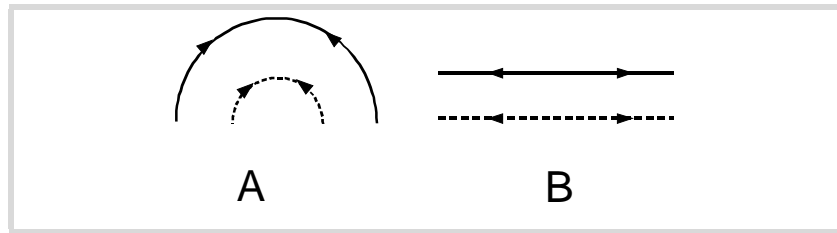
TOOL COMPENSATION
Tool radius compensation



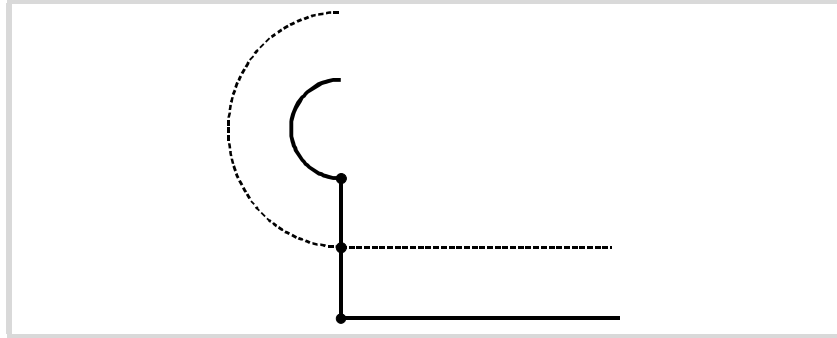
CNC 8060
CNC 8065

(REF. 1402)

- Back-and-forth path along the same way.



- Intermediate path as long as the tool radius:



11.

TOOL COMPENSATION

Tool radius compensation

11.1.6 Cancellation of tool radius compensation

Tool radius compensation is canceled with function G40.

After executing one of this function, radius compensation will be canceled during the next movement in the work plane, that must be a linear movement.

The way this compensation is canceled depends on the type of cancellation end (G138/G139) and the type of transition G136/G137 selected:

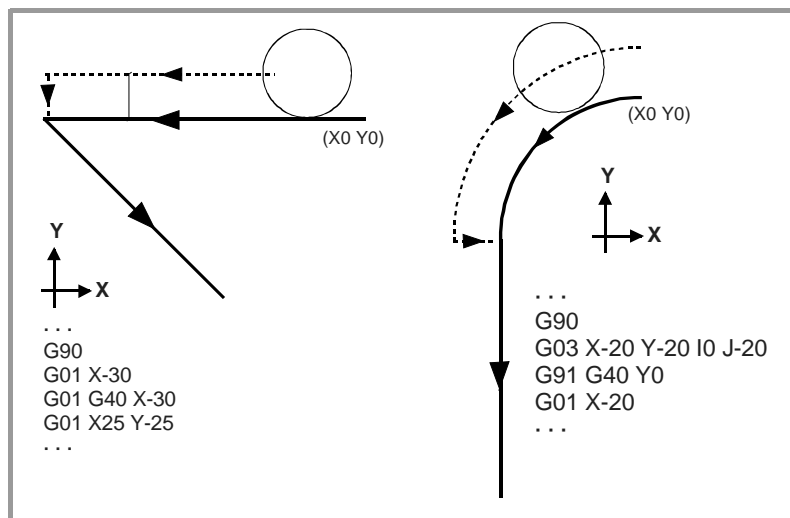
- G139/G136
 - The tool goes to the endpoint, contouring the corner along a circular path.
- G139/G137
 - The tool goes to the endpoint, contouring the corner along linear paths.
- G138
 - The tool goes straight to the endpoint. Regardless of the type of transition (G136/G137) programmed.

The following tables show the different possibilities of canceling tool radius compensation depending on the selected functions. The programmed path is shown with solid line and the compensated path with dashed line.

End of the compensation without programmed movement

After canceling the compensation, it may occur that the axes of the plane will not be involved in the first motion block. For example, because they have not been programmed, or the current tool position has been programmed or an incremental movement has been programmed.

In this case, the compensation is canceled at the same point where the tool is, as follows. Depending on the last movement made in the plane, the tool moves to the end point (uncompensated) of the programmed path.



11.

TOOL COMPENSATION
Tool radius compensation

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

STRAIGHT-TO-STRAIGHT PATH

When the angle between the paths is smaller or equal to 180° , the way radius compensation is canceled is independent from the G136/G137 and G138/G139 functions selected.

$0^\circ < \alpha < 90^\circ$	$\alpha = 90^\circ$
$90^\circ < \alpha < 180^\circ$	$\alpha = 180^\circ$

When the angle between paths is greater than 180° , the way radius compensation is canceled depends on the functions selected for type of ending (G138/G139) and type of transition (G136/G137).

G139/G136	G139/G137	G138
$180^\circ < \alpha < 270^\circ$	$180^\circ < \alpha < 270^\circ$	$180^\circ < \alpha < 270^\circ$
$\alpha = 270^\circ$	$\alpha = 270^\circ$	$\alpha = 270^\circ$
$270^\circ < \alpha < 360^\circ$	$270^\circ < \alpha < 360^\circ$	$270^\circ < \alpha < 360^\circ$

11.

TOOL COMPENSATION

Tool radius compensation



CNC 8060
CNC 8065

(REF. 1402)

ARC-TO-STRAIGHT PATH

When the angle between the tangent of the arc and the straight path is smaller or equal to 180° , the way radius compensation is canceled is independent from the G136/G137 and G138/G139 functions selected.

$0^\circ < \alpha < 90^\circ$	$\alpha = 90^\circ$
$90^\circ < \alpha < 180^\circ$	$\alpha = 180^\circ$

When the angle between the tangent of the arc and the straight line is greater than 180° , the way radius compensation is canceled depends on the type of ending (G138/G139) and type of transition selected (G136/G137).

G139/G136	G139/G137	G138
$180^\circ < \alpha < 270^\circ$	$180^\circ < \alpha < 270^\circ$	$180^\circ < \alpha < 270^\circ$
$\alpha = 270^\circ$	$\alpha = 270^\circ$	$\alpha = 270^\circ$
$270^\circ < \alpha < 360^\circ$	$270^\circ < \alpha < 360^\circ$	$270^\circ < \alpha < 360^\circ$

11.

TOOL COMPENSATION
Tool radius compensation



CNC 8060
CNC 8065

(REF. 1402)

11.2 Tool length compensation

Tool length compensation on a milling machine.

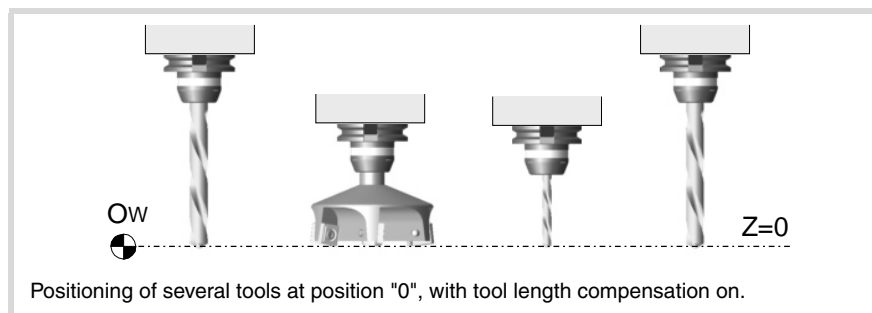
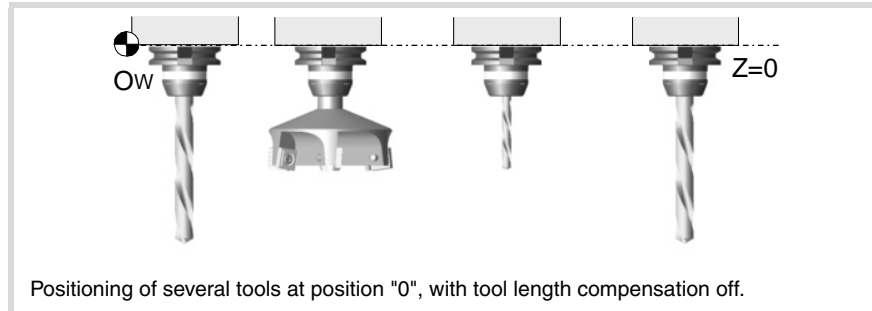
On a milling machine, tool length compensation is applied to the longitudinal axis; i.e. on the axis indicated by the instruction "#TOOL AX", or when missing, to the longitudinal axis designated by selecting the plane.

If G17, tool length compensation is applied to Z axis.

If G18, tool length compensation is applied to Y axis.

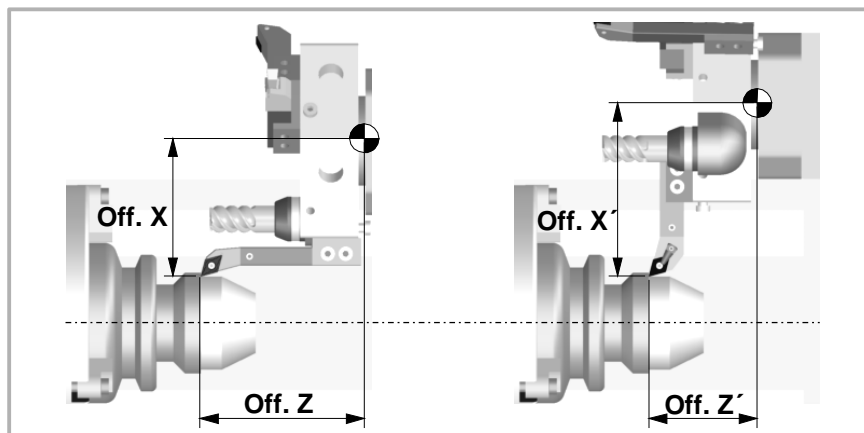
If G19, tool length compensation is applied to X axis.

Whenever any of functions G17, G18 or G19 is executed, the CNC assumes the axis perpendicular to the selected plane as the new longitudinal axis. If, then, "#TOOL AX" is executed, the new selected longitudinal axis replaces the previous one.



Tool length compensation on a lathe.

In a turning operation, the CNC takes into account the dimensions of the new tool, defined in the corresponding tool offset, and moves the tool holding turret so the tip of the new tool occupies the same position as the previous one.



11.

TOOL COMPENSATION
Tool length compensation

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

Programming

Tool length compensation is activated when selecting a tool offset.

- To activate this compensation, program "D<n>", where <n> is the tool offset number that contains the tool dimensions that will be used as compensation values.
- To cancel this compensation, program "D0".

Once one of these codes has been executed, tool length compensation will be activated or cancel during the next movement of the longitudinal axis.

11.

TOOL COMPENSATION
Tool length compensation

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

11.

TOOL COMPENSATION

Tool length compensation



CNC 8060
CNC 8065

(REF. 1402)

A subroutine is a set of blocks that, once properly identified, may be called upon several times from another subroutine or from the program. Subroutines are normally used for defining a bunch of operations or movements that are repeated several times throughout the program.

Types of subroutines.

The CNC has two types of subroutines, namely local and global. There is also a third type available, OEM subroutines, that are a special case of a global subroutine defined by the OEM. See "[12.5 OEM subroutine execution.](#)" on page 228.

Global subroutines.

The global subroutine is stored in CNC memory as an independent program. This subroutine may be called upon from any program or subroutine being executed.

Local subroutines.

The local subroutine is defined as part of a program. This subroutine may only be called upon from the program where it has been defined.

A program can have several local subroutines; but they all must be defined before the body of the program. A local subroutine can call a second local subroutine with the condition that the calling subroutine be defined after the one being called.

Subroutine nesting levels and parameters.

The defined subroutines may be called upon from the main program or from another subroutine; they may in turn call a second one, the second one may call a third one, and so on. The CNC limits this calls to a maximum of 20 nesting levels.

Arithmetic parameters in the subroutines

Local parameters.

The CNC has global parameters (accessible from the program or any subroutine) and local parameters (accessible only from the program or subroutine where they have been programmed).

Local parameters may be assigned to more than one subroutine up to 7 parameter nesting levels within the 20 subroutine nesting levels. Not all the subroutine call types change the nesting level; only the #CALL, #PCALL, #MCALL calls and functions G180 to G189 and G380 to G399..

Global parameters.

Global parameters will be shared by the program and the subroutines of the channel. They may be used in any block of the program and of the subroutine regardless of the nesting level they may be at.

Common parameters.

Common parameters will be shared by the program and the subroutines of any channel. They may be used in any block of the program and of the subroutine regardless of the nesting level they may be at.

12.1 Executing subroutines from RAM memory.

If the same subroutines are executed repeatedly during execution, it is more efficient to load them into the RAM memory of the CNC because this way, they may be accessed faster and execution time may be consequently optimized. This option is valid for OEM as well as for user subroutines. To load a subroutine into RAM, it must have the extension .fst.

User global variables.

User routines having a .fst extension are loaded into RAM memory during block preparation. The CNC checks whether or not it is loaded into RAM memory; if it is not and there is room for it (5 Mb), it loads it.

At the end of the program (M30), if no other channel is executing subroutines, the CNC deletes them from its RAM memory. This way, if a user subroutine having a .fst extension is edited or modified, the CNC assumes the changes the next time it executes it.

OEM subroutines.

- Being the CNC in USER mode, the OEM routines having a .fst extension are loaded into RAM memory when starting up the CNC application.

When the OEM is debugging his subroutines, they must have another extension so the changes are assumed without having to restart the application. Once they are debugged, the OEM must change the extension of the subroutines to .fst so they are loaded into RAM memory.

- Being the CNC in SETUP mode, the OEM subroutines whose extension is .fst will be loaded into RAM memory the first time they are executed in the program. This way, the changes made in the subroutine will be taken into account the next time the program is executed.

12.

SUBROUTINES.
Executing subroutines from RAM memory.



CNC 8060
CNC 8065

(REF. 1402)

12.2 Definition of the subroutines

Like the body of the program, a subroutine has a header, a body and an end-of-subroutine function.

Header of a local subroutine.

The header of the subroutine is a block consisting of the "%L" characters followed by a blank space and the name of the subroutine. The name of the subroutine may be up to 14 characters long and may consist of uppercase and lowercase characters as well as numbers (no blank spaces are allowed).

```
%L O123456789
%L SUBROUTINE
%L SUB234S
```

The header must be programmed. The name of the header is used when calling a subroutine.

Header of a global subroutine.

The header of a global subroutine is the same as that of a program; in other words, it is a block consisting of the "%" character followed by the name of the subroutine. The name may be up to 14 characters long and may consist of uppercase and lowercase characters as well as numbers (no blank spaces are allowed).

```
%O123
%GLOBSUBROUTINE
%PART923R
```

It is optional to program the header. The header name must not be used when calling a global subroutine, use the name of the file as it is stored at the CNC, instead.

The name defined in the header has nothing to do with the name of the file. The two may be different.

End of a global or local subroutine.

The end of the subroutine is defined with M17, M29 or the #RET instruction and they all are equivalent. One of them must be programmed to end the subroutine.

```
M17
M29
#RET
```

12.

12.3 Subroutine execution.

The CNC offers the following types of commands to call the subroutines.

Command.	Call type.
L	Call to a global subroutine. Parameters cannot be initialized with this command.
LL	Call to a local subroutine. Parameters cannot be initialized with this command.
#CALL	Call to a global or local subroutine. Parameters cannot be initialized with this command.
#PCALL	Call to a global or local subroutine. Local parameters can be initialized with this command.
#MCALL	Modal call to a local or global subroutine. Local parameters can be initialized with this command.
#MDOFF	It makes a function non-modal.

When executing one of these commands, the CNC executes the selected subroutine. When the subroutine has ended, the program execution resumes at the calling statement.

Location (path) of the global subroutines.

When calling a global subroutine its path (location) may be defined. When indicating the whole path, the CNC only looks for the subroutine in the indicated directory. If the path has not been indicated, the CNC looks for the subroutine in the following directories and in this order.

- 1 Directory selected with the #PATH instruction.
- 2 Directory of the program being executed.
- 3 Directory defined by machine parameter SUBPATH.

12.

SUBROUTINES.
Subroutine execution.



CNC 8060
CNC 8065

(REF. 1402)

12.3.1 LL. Call to a local subroutine.

The command LL calls a local subroutine. This type of call allows initializing local parameters of the subroutine.

Programming format.

The programming format is:

```
LL sub
sub      Name of the subroutine
```

```
LL sub2.nc
```

12.3.2 L Call to a global subroutine.

The command L calls a global subroutine. This type of call allows initializing local parameters of the subroutine. When it is a global subroutine, its whole path may be defined.

Programming format.

The programming format is:

```
L <path> sub
path      Optional. Subroutine location.
sub      Name of the subroutine
```

```
L C:\Cnc8070\Users\Prg\sub1.nc
L C:\Cnc8070\Users\sub2.nc
L Sub3.nc
```

12.3.3 #CALL. Call to a global or local subroutine.

The #CALL instruction calls a local or global subroutine. This type of call allows initializing local parameters of the subroutine. When it is a global subroutine, its whole path may be defined.

When there are two subroutines, one local and the other one global, with the same name, the following criteria is applied. If the path has been defined in the call, the CNC will execute the global subroutine, otherwise, it will execute the local one.

Programming format.

The programming format is:

```
#CALL <path> sub
path      Optional. Subroutine location.
sub      Name of the subroutine
```

```
#CALL C:\Cnc8070\Users\Prg\sub1.nc
#CALL C:\Cnc8070\Users\sub2.nc
#CALL Sub3.nc
```

Path definition.

Defining the path is optional. If defined, the CNC only looks for the subroutine in that folder; if not defined, the CNC looks for the subroutine in the default folders. See "[Location \(path\) of the global subroutines.](#)" on page 220.

12.

SUBROUTINES.
Subroutine execution.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

12.3.4 #PCALL. Call to a global or local subroutine initializing parameters.

The #PCALL instruction calls a local or global subroutine. This type of call allows initializing local parameters of the subroutine. When it is a global subroutine, its whole path may be defined.

When there are two subroutines, one local and the other one global, with the same name, the following criteria is applied. If the path has been defined in the call, the CNC will execute the global subroutine, otherwise, it will execute the local one.

Programming format.

The programming format is:

```
#PCALL <path> sub <P0..Pn>
```

path Optional. Subroutine location.

sub Name of the subroutine

P0..Pn Optional. Initializing parameters.

```
#PCALL C:\Cnc8070\Users\Prg\sub1.nc
#PCALL C:\Cnc8070\Users\sub2.nc A12.3 P10=6
#PCALL Sub3.nc A12.3 F45.3 P10=6
```

How to set local parameters.

The parameter values may be defined after the calling instruction and may be defined in two ways; using the parameter numbers P0-P25 or their letters A-Z (except "Ñ" and "Ç"), "A" for P0 and "Z" for P25.

Both ways of setting local parameters are equivalent and may be combined in the same block.

Path definition.

Defining the path is optional. If defined, the CNC only looks for the subroutine in that folder; if not defined, the CNC looks for the subroutine in the default folders. See "[Location \(path\) of the global subroutines.](#)" on page 220.

Local parameter nesting levels.

If local parameters are initialized in the #PCALL instruction, this instruction generates a new nesting level for the local parameters. Remember that up to 7 parameter nesting levels are possible within 20 subroutine nesting levels.

12.

SUBROUTINES.
Subroutine execution.



CNC 8060
CNC 8065

(REF. 1402)

12.3.5 #MCALL. Modal call to a local or global subroutine.

The #MCALL instruction calls a local or global subroutine. This type of call allows initializing local parameters of the subroutine. When it is a global subroutine, its whole path may be defined.

When there are two subroutines, one local and the other one global, with the same name, the following criteria is applied. If the path has been defined in the call, the CNC will execute the global subroutine, otherwise, it will execute the local one.

With this type of call, the subroutine becomes modal; i.e. the subroutine remains active in successive movements and it is repeated at the end of each move. See "[Considerations about the modal character of the subroutine.](#)" on page 224.

Programming format.

The programming format is:

```
#MCALL <path> sub <P0..Pn>
```

path	Optional. Subroutine location.
sub	Name of the subroutine
P0..Pn	Optional. Initializing parameters.

```
#MCALL C:\Cnc8070\Users\Prg\sub1.nc
#MCALL C:\Cnc8070\Users\sub2.nc A12.3 P10=6
#MCALL Sub3.nc A12.3 F45.3 P10=6
```

How to set local parameters.

The parameter values may be defined after the calling instruction and may be defined in two ways; using the parameter numbers P0-P25 or their letters A-Z (except "N" and "Ç"), "A" for P0 and "Z" for P25.

Both ways of setting local parameters are equivalent and may be combined in the same block.

Path definition.

Defining the path is optional. If defined, the CNC only looks for the subroutine in that folder; if not defined, the CNC looks for the subroutine in the default folders. See "[Location \(path\) of the global subroutines.](#)" on page 220.

Turning the subroutine into non-modal.

A subroutine stops being modal with the instruction #MDOFF and in the following cases: See "[12.4 #PATH. Define the location of the global subroutines.](#)" on page 227.

- After executing an M02 or an M30 and after a RESET.
- When changing the work plane.
- When programming a probing move (G100).
- When changing the configuration of the axes (#FREE AX, #CALL AX and #SET AX).
- When calling another subroutine (#PCALL, #CALL, L, LL, G180-G189).
- When activating a canned cycle.

Local parameter nesting levels.

If local parameters are initialized in the #MCALL instruction, this instruction generates a new nesting level for the local parameters. Remember that up to 7 parameter nesting levels are possible within 20 subroutine nesting levels.

12.

SUBROUTINES.
Subroutine execution.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

Considerations about the modal character of the subroutine.

The modal subroutine will not be executed in the motion blocks programmed inside the subroutine itself or in the subroutines associated with T or M6. It will not be executed either, when programming a number of block repetitions using a NR value of ·0·.

If a motion block contains a number of repetitions NR other than 0 while a modal subroutine is active, both the movement and the subroutine will be repeated NR times.

If while a subroutine is selected as modal, a block containing the instruction #MCALL is executed, the current subroutine will stop being modal and the new selected subroutine will become modal.

12.

SUBROUTINES.

Subroutine execution.



CNC 8060
CNC 8065

(REF. 1402)

12.3.6 #MDOFF. Turning the subroutine into non-modal.

The subroutine stops being modal with the instruction #MDOFF. .

Programming format.

The programming format is:

#MDOFF

```
#MDOFF
```

12.

SUBROUTINES.
Subroutine execution.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

12.3.7 #RETDSBLK. Execute subroutine as a single block.

The #RETDSBLK instruction ends the subroutine and cancels the single block treatment.

Programming format.

Program the instruction alone in the block and at the end of the subroutine.

```
#RETDSBLK
```

```
#RETDSBLK
```

How to create the subroutine.

For a subroutine to be executed as a single block, it usually has the following structure.

```
%Sub.nc
#ESBLK; Beginning of the single-block treatment.
.
.
#DSBLK; End of the single-block treatment.
#RET; End of subroutine.
```

When executing this subroutine in single-block mode, the [START] key must be pressed twice because the execution stops in the #RET block. To prevent this so the subroutine is executed with a single [START], the subroutine must begin with #ESBLK and end with #RETDSBLK.

```
%Sub.nc
#ESBLK; Beginning of the single-block treatment.
.
.
#RETDSBLK; End of subroutine and end of the single-block treatment.
```

12.

SUBROUTINES.
Subroutine execution.



CNC 8060
CNC 8065

(REF. 1402)

12.4 #PATH. Define the location of the global subroutines.

The instruction #PATH defines the pre-determined location of the global subroutines. If no path is defined in the global subroutine call, the CNC will look for the subroutine in the path defined using the instruction #PATH.

If the path is defined when calling a global subroutine, the CNC looks for the subroutine in that location, it ignores the location defined in the #PATH statement.

Programming format.

The programming format is:

```
#PATH ["path"]
```

path Pre-determined subroutine location.

```
#PATH ["C:\Cnc8070\Users\Prg\"]
```

```
#PATH ["C:\Cnc8070\Users\"]
```

12.

SUBROUTINES.

#PATH. Define the location of the global subroutines.



CNC 8060
CNC 8065

(REF. 1402)

12.5 OEM subroutine execution.

The machine manufacturer may define up to 30 subroutines per channel and associate them with functions G180 through G189 and G380 through G399 in such a way that when a channel executes one of these functions, it will execute the subroutine associated with that function for that channel. These OEM subroutines may be executed either in a non-modal or modal way and it also allows resetting the local parameters of the subroutine.

12.

SUBROUTINES.
OEM subroutine execution.

Programming format.

These functions may be programmed anywhere in the program and they don't have to go alone in the block and allow resetting the local parameters of the subroutine.

Programming format. Executing the subroutine in a non-modal way.

The programming format is the following; the list of arguments appears between curly brackets. To execute the subroutine in a non-modal way, call it using the G code (G180, G181, etc).

G180

G380

G180 {P0..Pn}

G380 {P0..Pn}

P0..Pn Optional. Local parameters of the subroutine.

```
G180
G183 P1=12.3 P2=6
G388 A12.3 B45.3 P10=6
```

Programming format. Executing the subroutine in a modal way.

The programming format is the following; the list of arguments appears between curly brackets. To execute the subroutine in a modal way, call it using the MG code (MG180, MG181, etc).

MG180

MG380

MG180 {P0..Pn}

MG380 {P0..Pn}

P0..Pn Optional. Local parameters of the subroutine.

```
G180
G183 P1=12.3 P2=6
G388 A12.3 B45.3 P10=6
```

How to set local parameters.

These functions allow initializing local parameters in the subroutine. The parameter values may be defined after the calling function and may be defined in two ways; using the parameter numbers (P0, P1, etc.) or their letters A-Z (except "Ñ" and "Ç") like "A" for P0 and "Z" for P25. Both ways of setting local parameters are equivalent and may be combined in the same block.

Additional data in the block.

Besides initializing the parameters, any other type of additional information may be added to these functions, even movements. This information must be programmed before the subroutine calling function; otherwise, the data will be considered as for initializing the parameters. The associated subroutine is executed once the execution of the rest of the information programmed in the block has ended.

```
G01 X50 F450 G180 P0=15 P1=20
```

It first executes the programmed movement to point X50 and then the subroutine associated with G180 initializing parameters P0 and P1.



CNC 8060
CNC 8065

(REF. 1402)

```
G180 P0=15 P1=20 G01 X50 F450
```

All the data is interpreted as parameter setting, where P6(G)=1, P23(X)=50 and P5(F)=450.

Cancelling a modal subroutine.

A subroutine stops being modal and in the following cases.

- When programming G80 or #MDOFF.
- When changing the work plane.
- When programming a probing move (G100).
- When executing another subroutine (#PCALL, #CALL, #MCALL, L, LL, G180-G189, G380-G399).
- When executing a canned cycle.
- After executing an M02 or an M30 and after a RESET.
- When changing the configuration of the axes (#FREE AX, #CALL AX and #SET AX).

Local parameter nesting levels.

If these functions initialize local parameters, this instruction generates a new nesting level for the local parameters. Remember that up to 7 parameter nesting levels are possible within 20 subroutine nesting levels.

Considerations about the modal character of the subroutine.

The modal subroutine will not be executed in the motion blocks programmed inside the subroutine itself or in the subroutines associated with T or M6. It will not be executed either, when programming a number of block repetitions using a NR value of -0.

If a motion block contains a number of repetitions NR other than 0 while a modal subroutine is active, both the movement and the subroutine will be repeated NR times.

If while a subroutine is selected as modal, another modal OEM subroutine is executed, the current subroutine will stop being modal and the new selected subroutine will become modal.

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

Functions G180-G189 and G380-G399 are not modal. Functions MG180-MG189 and MG380-MG399 are modal.

12.

SUBROUTINES.
OEM subroutine execution.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

12.6 Generic user subroutines (G500-G599).

The CNC lets the users define up to 100 subroutines, common to all channels and that will be associated with functions G500 through G599, so when the CNC executes one of these functions, it will execute its associated subroutine. These OEM subroutines may be executed either in a non-modal or modal way and it also allows resetting the local parameters of the subroutine.

These subroutines are loaded into RAM memory when they are executed for the first time. If there is no room in RAM, the CNC will issue a warning and it will execute the subroutine from the disk. At the end of the program (M30), if no other channel is executing subroutines, the CNC deletes them from its RAM memory. This way, if a user subroutine is edited or modified, the CNC assumes the changes the next time it executes it.

When updating the version, only the subroutines supplied by Fagor will be updated when choosing the third installation level "rename previous version and install completely".

Programming format.

These functions may be programmed anywhere in the program and they don't have to go alone in the block and allow resetting the local parameters of the subroutine.

Programming format. Executing the subroutine in a non-modal way.

The programming format is the following; the list of arguments appears between curly brackets. To execute the subroutine in a non-modal way, call it using the G code (G500, G501, etc).

G500

G500 {P0..Pn}

P0..Pn Optional. Local parameters of the subroutine.

```
G500
G583 P1=12.3 P2=6
G588 A12.3 B45.3 P10=6
```

Programming format. Executing the subroutine in a modal way.

The programming format is the following; the list of arguments appears between curly brackets. To execute the subroutine in a modal way, call it using the MG code (MG500, MG501, etc).

MG500

MG500 {P0..Pn}

P0..Pn Optional. Local parameters of the subroutine.

```
G500
G583 P1=12.3 P2=6
G588 A12.3 B45.3 P10=6
```

How to set local parameters.

These functions allow initializing local parameters in the subroutine. The parameter values may be defined after the calling function and may be defined in two ways; using the parameter numbers (P0, P1, etc.) or their letters A-Z (except "Ñ" and "Ç") like "A" for P0 and "Z" for P25. Both ways of setting local parameters are equivalent and may be combined in the same block.

12.

SUBROUTINES.
Generic user subroutines (G500-G599).



CNC 8060
CNC 8065

(REF. 1402)

Additional data in the block.

Besides initializing the parameters, any other type of additional information may be added to these functions, even movements. This information must be programmed before the subroutine calling function; otherwise, the data will be considered as for initializing the parameters. The associated subroutine is executed once the execution of the rest of the information programmed in the block has ended.

```
G01 X50 F450 G500 P0=15 P1=20
```

It first executes the programmed movement to point X50 and then the subroutine associated with G500 initializing parameters P0 and P1.

```
G500 P0=15 P1=20 G01 X50 F450
```

All the data is interpreted as parameter setting, where P6(G)=1, P23(X)=50 and P5(F)=450.

Cancelling a modal subroutine.

A subroutine stops being modal and in the following cases.

- When programming G80 or #MDOFF.
- After executing an M02 or an M30 and after a RESET.

Define the subroutines.

The subroutines associated with these functions will be global and will have the same name as the function, without extension. The subroutines must be defined in the folder ..\Users\Sub. If the CNC executes a function and there is no subroutine, the CNC will issue an error message.

G500 will have subroutine G500 associated with it.
 G501 will have subroutine G501 associated with it.
 . . .
 G599 will have subroutine G599 associated with it.

Subroutines supplied by Fagor.

Subroutine.	Meaning.
G500	HSC cancellation.
G501	HSC activation for roughing operations.

Execute the subroutines.

The CNC executes the subroutine after executing the function with which they are associated. To execute the subroutine as a single block, program the instructions #ESBLK and #RETDSBLK. After executing the instruction #ESBLK, the CNC executes the blocks programmed next as a single block until it reaches the end of the subroutine (#RETDSBLK).

If the file that contains the subroutine has the "hidden" attribute, the CNC does not display the contents of the subroutine while it is executed. The attributes of the files may be modified from the utilities mode (see the operating manual).

Local parameter nesting levels.

If these functions initialize local parameters, this instruction generates a new nesting level for the local parameters. Remember that up to 7 parameter nesting levels are possible within 20 subroutine nesting levels.

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

Functions G500 - G599 are not modal. Functions MG500 - MG599 are modal.

12.

SUBROUTINES.

Generic user subroutines (G:500-G:599).



CNC 8060
CNC 8065

(REF. 1402)

12.7 Assistance for subroutines.

12.7.1 Subroutine help files.

Help files may be associated with each OEM subroutine and each global subroutine called upon using #MCALL or #PCALL and they will be displayed while editing. Each subroutine may have two help files; a text file (txt) and an image file (bmp).

The help window is displayed while editing, after a blank space or tab following a G180-G189, G380-399 or following the name of the subroutine. The help window is only informative, it cannot be accessed with the cursor nor browse through it. When the help file is displayed, its text may be inserted into the part-program using the [INS] key. The help window closes with [ESC], deleting the key word or going on to another line of the program.

The help window of the subroutines is only available when the editor uses the CNC language; when the editor is enabled for the 8055 CNC, these helps are not available. The help window of the subroutines is available even when the contextual helps of the editor are disabled.

How to create help files.

Each subroutine may have two help files; a text file (txt) and an image file (bmp). There's no need to define both files; either one may be defined alone. The name of the files must follow the following rule:

Subroutine.	Name of the help files.
G180-G189 G380-G399	The name of the files must be the function it is associated with. For example <i>G180.txt</i> y <i>G180.bmp</i> .
#MCALL #PCALL	The name of the files must be the name of the subroutine. For example <i>subroutine.txt</i> and <i>subroutine.bmp</i> .

When the help window is only informative, it cannot be accessed with the cursor nor browse through it with the page-up-down keys. This is why it is recommended to use short help files; for example, that only contain the description of the parameters of the subroutine.

When the help file is displayed, its text may be inserted into the part-program using the [INS] key. Hence, we recommend the following.

- That the help file contains the line calling the subroutine. Since the user must have written part of the call to display the help window, the editor deletes the call before inserting the help text.
- That all the lines of the help file follow the format of a CNC comment, except the line containing the call to the subroutine.

The format of the text file may be the following.

```
G180 P0= P1= P2= P3= P4= P5=
#COMMENT BEGIN
----- G180 -----
P1 = Movement in X
P2 = Movement in Y
P3 = Movement in Z
P4 = Feedrate F
P5 = Speed S
-----
#COMMENT END
```

Where to save the help files.

The machine manufacturer can save the help files in the folder ..\MTB\SUB\HELP\language. Since the modifications to the MTB directory in the "User" work mode disappear when turning the unit off, the user must save his help files in the folder ..\USERS\HELP\language. The CNC looks for the help files in the language that is selected; if the files are not there, the CNC will not show any help.

The CNC first looks for the files in the OEM folder and then in the user folder, that's why the user must not define subroutines and/or help files with the same name as those of the OEM. If both files have the same name, the CNC will first show the OEM files.

12.

SUBROUTINES.
Assistance for subroutines.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

12.7.2 List of available subroutines.

The editor allows having in a text file (txt) a list that is displayed while editing the part-program, every time a #PCALL or an #MCALL instruction is edited.

The editor shows the list of subroutines while editing, after a blank space or tab following a G180-G189 or following a #PCALL or #MCALL instruction. This list works the same way as the lists of variables, it is possible to use the arrow keys to scroll through the various elements. Pressing [ENTER], the editor inserts the selected line in the current position of the cursor. The list of subroutines disappears with [ESC], deleting the key word or going on to another line of the program.

This help is always active, regardless of the editing help softkey, "Prog. help".

How to create the list of subroutines

The list of subroutines must be in a text (txt) file that must be called *pcall.txt*. The file must be edited so each line is the name of a possible subroutine to be called.

```
C:\CNC8070\USERS\SUB\FAGOR.NC
SUBROUTINE.NC
EXAMPLE.NC
POSITIONING.NC
```

Where to save the list of subroutines.

The machine manufacturer must save the *pcall.txt* file in the folder `..\MTB\SUB\HELP\language`. Since the modifications to the MTB directory in the "User" work mode disappear when turning the unit off, the user must save his *pcall.txt* file in the folder `..\USERS\HELP\language`. The CNC looks for the help files in the language that is selected; if the files are not there, the CNC will not show any help. If there is a *pcall.txt* file in both directories, the list will show the names of the subroutines contained in both of them.

12.

SUBROUTINES.
Assistance for subroutines.



CNC 8060
CNC 8065

(REF. 1402)

12.8 Interruption subroutines.

The interruption subroutines are defined by the machine manufacturer and will be executed from the PLC. When the PLC commands the execution of one of these subroutines, the channel interrupts the execution of the program and executes the corresponding interruption subroutine.

If the program is already interrupted (STOP) or no program is in execution (channel in READY state), the execution of the subroutine depends on parameter SUBINTSTOP. Also, in order to execute the subroutine when no program is in execution, the channel must be in automatic mode; the subroutine cannot be executed from jog mode.

The CNC executes the subroutine with the current history of the interrupted program (G functions, feedrate, etc.). Once the subroutine is executed, the CNC resumes the execution of the program from the interruption point and maintaining the changes made to the history by the subroutine (G functions, etc.).

The execution of an interruption subroutine may be interrupted with a STOP, but not by another interruption subroutine. When a subroutine is interrupted, it is not possible to get into tool inspection mode.

12.

SUBROUTINES.
Interruption subroutines.



CNC 8060
CNC 8065

(REF. 1402)

12.8.1 Repositioning axes and spindles from the subroutine (#REPOS).

The #REPOS can only be used inside the interruption subroutines and allows repositioning axes and spindles before ending that subroutine. The CNC does not reposition the axes when the instruction is executed, it does when returning from the subroutine to the program as the last action associated with the subroutine.

An interruption subroutine may contain several #REPOS instructions, but they all must be programmed at the end of the subroutine, in the blocks before the end of the subroutine (#RET, M17, M29). Blocks programmed between the last #REPOS instruction and the end-of-subroutine block will cause an error.

Programming.

This instruction must be programmed at the end of the subroutine, before the end-of-subroutine block. When programming this instruction, you must define the axes to be repositioned. Optionally, it is possible to indicate whether the reposition point for the axes is the point where the program was interrupted or the starting point of the interrupted block.

Programming format.

The programming format is the following; the list of arguments appears between curly brackets and the optional ones between angle brackets.

```
#REPOS <{point}> X~C <X~C>
```

{point} Optional. Reposition point. This parameter is set with one of the INT/INI values.

X~C Sequence of axes and spindles to be repositioned.

```
#REPOS A1 A2 S1
```

The reposition point is the point where the program was interrupted.

```
#REPOS INT X A1 U Z S
```

The reposition point is the point where the program was interrupted.

```
#REPOS INI X Y Z
```

The reposition point is the starting point of the interrupted block.

Sequence of axes and spindles to be repositioned.

The CNC repositions the axes in the program sequence, except the axes of the active plane that reposition simultaneously when the first of them repositions. Since there may be several #REPOS instructions in the same subroutine, axes or spindles repeated (duplicated) in the same instruction or in a previous one are ignored.

Reposition point.

This parameter is defined with one of the following commands; if not programmed, the instruction assumes the INT value.

Value.	Meaning.
INT	The reposition point for the axes is the point where the program was interrupted when activating the subroutine.
INI	The reposition point for the axes is the starting point of the interrupted block.

The same subroutine may have several #REPOS instructions, but all of them must have the same INT/INI reposition point.

12.

SUBROUTINES.
Interruption subroutines.



CNC 8060
CNC 8065

(REF. 1402)

12.9 Subroutine "program_start".

If there is a PROGRAM_START subroutine, the CNC executes at when pressing the [START] key to start the execution of a program from the automatic mode. If that subroutine is missing, the CNC executes the program directly. This subroutine may be used, for example, to have certain machining conditions defined to determine the execution of user programs.

The CNC only calls upon the PROGRAM_START subroutine if it is going to execute the whole main program; i.e. when no program starting point has been selected. The CNC does not call upon the subroutine either when executing a cycle from the conversational mode.

The CNC executes the subroutine as a single block. Even if it is not programmed inside the subroutine, the CNC executes it as if a #ESBLK were programmed at the beginning of the subroutine and the final #RET will be equivalent as #RETDSBLK. This way, when single block is active, the [START] key must be pressed less often to execute the program.

Name and location of the subroutine.

The name of the subroutine must be PROGRAM_START and it will be saved in the folder ..\user\sub. If there are several channels, there may be a different subroutine per each channel, whose name must be PROGRAM_START_Cn where n is the channel number (PROGRAM_START_C1, PROGRAM_START_C2, etc). For the first channel, the name may be either PROGRAM_START or PROGRAM_START_C1; when having both, the CNC executes PROGRAM_START.

12.

SUBROUTINES.
Subroutine "program_start".



CNC 8060
CNC 8065

(REF. 1402)

12.

SUBROUTINES.

Subroutine "program_start".



CNC 8060
CNC 8065

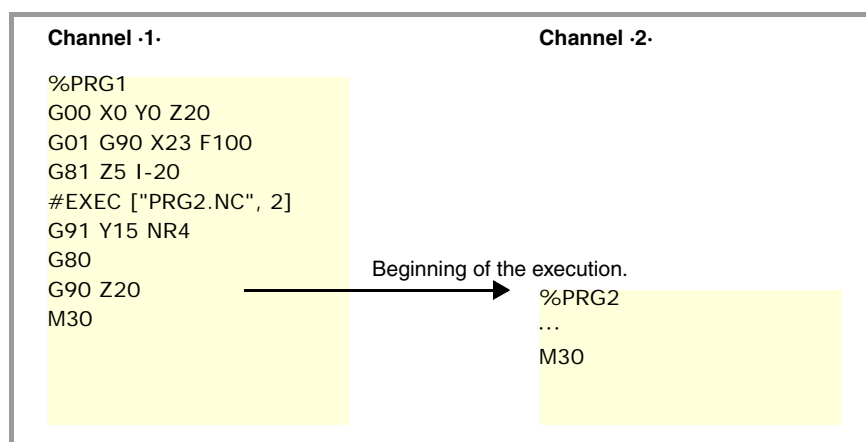
(REF. 1402)

EXECUTING BLOCKS AND PROGRAMS

13

13.1 Executing a program in the indicated channel.

With the #EXEC instruction, it is possible, from a program in execution, to begin the execution of a second program in another channel. The execution of the program starts in the indicated channel in parallel (at the same time) with the block following the #EXEC instruction. If the channel where the program to be executed is busy, the CNC waits for the operation in progress to end.



Programming format.

The programming format is: Optional parameters are indicated between angle brackets.

#EXEC [{"prg}"<,{channel}>]

{prg} Part-program location.

{channel} Optional. Channel where the block is to be executed.

```
#EXEC ["PRG1.NC",2]
    (It executes in channel 2 the indicated program)
#EXEC ["MYPRG.NC"]
    (It executes the program as a subroutine).
#EXEC ["C:\CNC8070\USERS\PRG\EXAMPLE.NC",3]
    (It executes in channel 3 the indicated program)
```

Program location (path).

The program to be executed may be defined by either writing the full path or without it. When indicating the whole path, the CNC only looks for the program in the indicated folder. If the path has not been indicated, the CNC looks for the program in the following folders and in this order.

- 1 Directory selected with the #PATH instruction.
- 2 Directory of the program that executes the #EXEC instruction.
- 3 Directory defined by machine parameter SUBPATH.



CNC 8060
CNC 8065

(REF. 1402)

Channel where the block is to be executed.

Programming the channel is optional. If the channel is not indicated or it coincides with the channel where the #EXEC instruction is executed, the second program will be executed as a subroutine. In this case, functions M02 and M30 will carry out all the associated actions (initialization, sending to the PLC, etc.) except the one for finishing the program. After executing function M02 or M30, it goes on executing the blocks programmed after the #EXEC instruction.

Considerations.

A program containing the #EXEC instruction may be executed, simulated, syntax checked or searched for a particular block. In all the cases, programs called upon using the #EXEC instruction are executed in the same conditions as the original program

13.**EXECUTING BLOCKS AND PROGRAMS**

Executing a program in the indicated channel.



CNC 8060
CNC 8065

(REF. 1402)

13.2 Executing a block in the indicated channel.

With the #EXBLK instruction, it is possible, from a program in execution or via MDI, to execute a block in another channel.

If the channel where the block to be executed is busy, the CNC waits for the operation in progress to end. After executing the block, the channel goes back to the previous work mode.

Programming format.

The programming format is: Optional parameters are indicated between angle brackets.

#EXBLK [{block} < , {channel} >]

{block} Block to be executed.

{channel} Optional. Channel where the block is to be executed.

```
#EXBLK [G01 X100 F550, 2]
    (The block is executed in channel ·2·)
#EXBLK [T1 M6]
    (The block is executed in the current channel)
```

Channel where the block is to be executed.

Programming the channel is optional. If the channel is not indicated and the instruction is executed from the program, the block is executed in its own channel. If the instruction is executed in MDI and the channel is not indicated, the block is executed in the active channel.

13.

EXECUTING BLOCKS AND PROGRAMS
Executing a block in the indicated channel.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

13.3 Abort the execution of the program and resume it in another block or program.

The CNC offers a special interruption mode, handled via PLC, that allows aborting the execution of the program and resuming it either from a particular previously defined block or in another program.

The point where the execution is resumed is defined with the #ABORT instruction. If no resume point has been defined, the execution of the program will not be interrupted.

13.

Aborting the execution of the program.

Usually, this feature is turned on and off using an external push button or key configured for that purpose. This interruption mode is not applied when pressing the [CYCLE STOP] key.

When interrupting the program from the PLC, the CNC channel aborts the execution of the program but without affecting the spindle, initializes the program history and resumes the execution in the point indicated by the active #ABORT instruction. If no #ABORT instruction is active in the part-program, the execution is not interrupted.

Threading and other machining operations that cannot be interrupted.

If the program is aborted during a threading operation that cannot be interrupted, the CNC behaves like when doing a reset in these cases.

When aborting the execution, the CNC interrupts the execution once the operation has been properly completed. Being the program interrupted, it will be necessary to repeat the command to abort the program in order for the CNC to do so.

Considerations when resuming the program.

When the program is interrupted, the history is initialized. Thus, it is recommended to define the minimum machining conditions such as feedrate, ·M· functions, etc in the block where the execution will be resumed.

Define the execution resuming point.

The program resuming point may be either a block of the same program or a different program. If the execution resumes in a different program, it will be executed from the beginning; it will not be possible to select the starting block.

Within the same program, it is possible to define several resume points; when interrupting the program, the CNC will use the one that is active at the time; in other words, the one executed last.

Programming format (1). The block resumes in a block of the same program.

The block where the execution resumes may be defined in two ways; using the block number or labels. The programming format is:

```
#ABORT {block}
```

{block} Block where the execution resumes.

```
#ABORT N120
#ABORT [LABEL]
```

Remember that when the block number is the target for a jump, it must be defined in the program with the ":" character. (semi-colon).

```
#ABORT N500
...
N500: T1 D1
```

Programming format (2). The execution resumes in a different program.

#ABORT [{"prg}"]

{ prg} Program where the execution resumes.

#ABORT ["PRG.NC"]

#ABORT ["C:\CNC8070\USERS\PRG\EXAMPLE.NC"]

The program to be executed may be defined by either writing the full path or without it. When indicating the whole path, the CNC only looks for the program in the indicated folder. If the path has not been indicated, the CNC looks for the program in the following folders and in this order.

- 1 Directory selected with the #PATH instruction.
- 2 Directory of the program that executes the #ABORT instruction.
- 3 Directory defined by machine parameter SUBPATH.

Canceling the execution resuming point.

When canceling the resume point, the execution of the program will not be interrupted.

Programming format.

#ABORT OFF

#ABORT OFF

Programming suggestions.

It is recommended to program the target labels at the beginning of the program, outside the main program. Otherwise, and depending on the length of the program, if the jump labels are defined at the end of it, the #ABORT instruction may take longer to find them.

13.

EXECUTING BLOCKS AND PROGRAMS
 Abort the execution of the program and resume it in another block or program.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

13.

EXECUTING BLOCKS AND PROGRAMS

About the execution of the program and resume it in another block or program.



CNC 8060
CNC 8065

(REF. 1402)

The CNC allows activating axes and spindles as C axis, that interpolated with a linear axis makes it possible to mill the cylindrical surface or the face of a turning part. Although the machine may have several axes or spindle defined as "C" axis, only one of them may be active.

·C· axis on a lathe.

On a lathe, it is most common to activate the spindle as ·C· axis and use a live tool to machine the part.

·C· axis on a milling machine.

On a milling machine, it is most common to activate a rotary axis as ·C· axis and use the spindle to machine the part.

Configuration of a ·C· axis.

To activate an axis or spindle as "C" axis, it must have been defined as such by the machine manufacturer. To know whether an axis or spindle may be activated as a ·C· axis, refer to parameter CAXIS in the machine parameter table or to its variable.

(V.)MPA.CAXIS.Xn

Variable that indicates whether the axis or spindle may be enabled as a ·C· axis or not.
Value of ·1· if so, value of ·0· if otherwise.

In the machine parameter table, parameter CAXNAME indicates the default name of the ·C· axis of the channel. This is the name that a spindle enabled as ·C· axis will take if not indicated otherwise via part-program.

The zero offsets on the C axis.

Once the zero offsets have been defined in the table, they may be activated by program using functions G54 to G59 and G159. The zero offsets on the C axis have the following peculiarities.

- If there is an active zero offset and, then, a C axis is activated, the zero offset for the C axis is not assumed.
- When the spindle works as C axis (#CAX instruction) the zero offset is applied in degrees.
- When face machining is active (#FACE instruction) or cylinder machining (#CYL instruction) is active, the zero offset is applied in the active units, millimeters or inches.



CNC 8060
CNC 8065

(REF. 1402)

14.1 Activating the spindle as "C" axis.

To use a spindle as ·C· axis, it must be enabled as such first. Once this is done, it will be possible to program machining operation on the face or on the side using the instructions #FACE or #CYL respectively.

Activating the spindle as ·C· axis.

The #CAX instruction activates a spindle as ·C· axis.

The programming format is: Optional parameters are indicated between angle brackets.

```
#CAX [<{spdl}>,<{name}>]
```

{spdl} Optional. Spindle to be activated as C axis.

{name} Optional. Name of the C axis.

```
#CAX
#CAX [S1]
#CAX [S,C]
```

The spindle needs only be indicated when a spindle other than the master is to be activated as a C axis. Otherwise, there is no need to program it.

The {name} parameter sets the name that will identify the C axis. This name will be used in the part program to define the movements. If the name is not defined, the CNC assigns a name by default. See "[Configuration of a ·C· axis.](#)" on page 245.

Programming	Spindle that is activated as ·C· axis.	Axis name.
#CAX	Master spindle.	By default.
#CAX [S1]	Spindle S1 (it may be the master).	By default.
#CAX [S,C]	Spindle S (it may be the master).	C
#CAX [S3,B2]	Spindle S3 (it may be the master).	B2

Considerations about working with the C axis

Activating a running spindle as C axis stops the spindle. While being a spindle active as "C" axis, no speed may be programmed for it.

When activating the spindle as "C" axis, the CNC carries out a home search of the "C" axis.

Accessing the variables of a spindle activated as C axis.

After activating a spindle as C axis, the new name of the spindle must be used to access its variables from the part-program or MDI. The access to the variables from the PLCA or from an interface does not change; the original name of the spindle remains unchanged.

Influence of the M3/M4/M5 functions.

If the spindle is working as C axis, executing an M3, M4 or M5 function will automatically set the spindle in open loop (same as programming #CAX OFF).

Deactivating the spindle as C axis.

The C axis is deactivated with the #CAX instruction and the spindle goes back to working as a normal spindle.

```
#CAX OFF
```

```
#CAX OFF
```

Programming the spindle as ·C· axis.

When the spindle works as ·C· axis, it will be programmed as if it were a rotary axis (in degrees).

14.

"C" AXIS
Activating the spindle as "C" axis.



CNC 8060
CNC 8065

(REF. 1402)

Programming the master spindle as ·C· axis.

```
#CAX  
G01 Z50 C100 F100  
G01 X20 C20 A50  
#CAX OFF
```

Programming any spindle as ·C· axis.

```
#CAX [S1,C1]  
  (The spindle "S1" is activated as "C" axis under the name of "C1")  
G01 Z50 C1=100 F100  
G01 X20 C1=20 A50 S1000  
#CAX OFF
```

14.

"C" AXIS
Activating the spindle as "C" axis.



CNC 8060
CNC 8065

(REF. 1402)

14.2 Machining of the face of the part

For this type of machining either a rotary axis or a spindle may be used as "C" axis. When using a spindle, it must be activated as "C" axis in advance using the #CAX instruction. See "[14.1 Activating the spindle as "C" axis.](#)" on page 246.

Activating the machining of the face of the part.

The #FACE instruction activates the machining of the face and it also defines the work plane. The axis to be activated as "C" axis will be determined by the work plane defined.

The programming format is: Optional parameters are indicated between angle brackets.

```
#FACE [{abs},{ord}<,{long}>]<[{{kin}}]>
```

{abs} Abscissa axis of the work plane.
 {ord} Ordinate axis of the work plane.
 {long} Optional. Longitudinal axis of the tool.
 {kin} Optional. Number of the kinematics.

```
#FACE [X,C]
#FACE [X,C][1]
#FACE [X,C,Z]
#FACE [X,C,Z][1]
```

Programming the kinematics is optional; if not programmed, the CNC applies the first kinematics that has been defined in the machine parameters and is valid for this type of machining.

Cancel the machining of the face of the part.

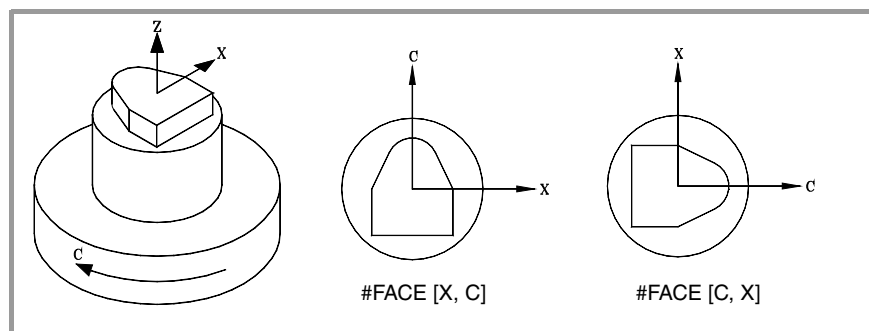
Machining is canceled with the #FACE instruction as follows.

```
#FACE OFF
```

```
#FACE OFF
```

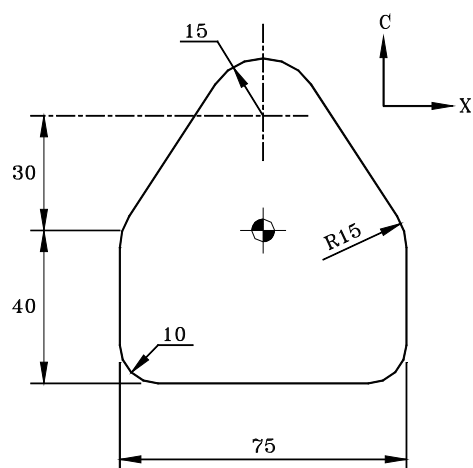
C axis programming.

The "C" axis will be programmed as if it were a linear axis (in millimeters or inches) and the CNC will calculate the corresponding angular movement depending on the selected radius. When the machining is activated, the CNC switches to operate in radius and G94 (mm/min).



CNC 8060
CNC 8065

(REF. 1402)



```
#FACE [X,C]
G90 X0 C-90
G01 G42 C-40 F600
G37 I10
X37.5
G36 I10
C0
G36 I15
X12.56 C38.2
G03 X-12.58 C38.2 R15
G01 X-37.5 C0
G36 I15
C-40
G36 I10
X0
G38 I10
G40 C-90
#FACE OFF
M30
```

14.

"C" AXIS
Machining of the face of the part



CNC 8060
CNC 8065

(REF. 1402)

14.3 Machining of the turning side of the part

For this type of machining either a rotary axis or a spindle may be used as "C" axis. When using a spindle, it must be activated as "C" axis in advance using the #CAX instruction. See "14.1 Activating the spindle as "C" axis." on page 246.

Activating the machining of the side of the part.

The #CYL instruction activates the machining of the side and it also defines the work plane. The axis to be activated as "C" axis will be determined by the work plane defined.

The programming format is: Optional parameters are indicated between angle brackets.

```
#CYL [{abs},{ord},{long}]{radius}<[{kin}]>
```

{abs}	Abscissa axis of the work plane.
{ord}	Ordinate axis of the work plane.
{long}	Longitudinal axis of the tool.
{radius}	Radius of the cylinder that will be machined.
{kin}	Optional. Number of the kinematics.

```
#CYL [X,C,Z45]
#CYL [C,Y,Z30]
#CYL [X,C,Z45][3]
```

If the radius is programmed with a value of -0., it assumes as the cylinder radius the distance between the rotation center and the tool tip. This makes it possible to develop the surface on cylinders with variable radius without having to indicate the radius.



In versions older than V3.10, programming the radius was optional. If the software is updated from an older version, the programs will have to be corrected.

Programming the kinematics is optional; if not programmed, the CNC applies the first kinematics that has been defined in the machine parameters and is valid for this type of machining.

Cancel the machining of the side of the part.

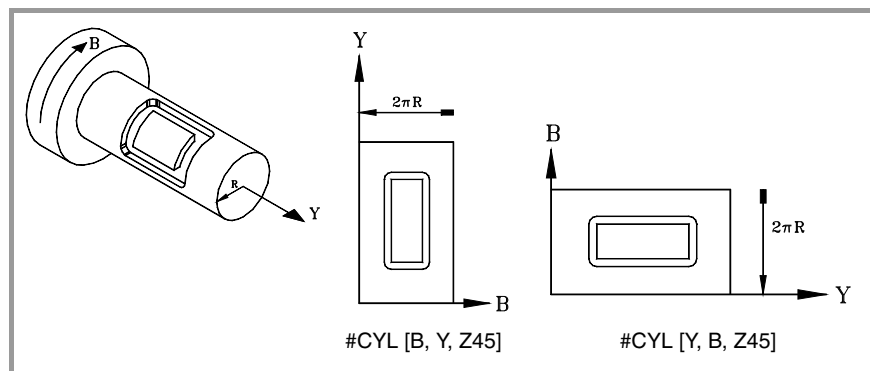
Machining is canceled with the #CYL instruction as follows.

```
#CYL OFF
```

```
#CYL OFF
```

C axis programming.

The "C" axis will be programmed as if it were a linear axis (in millimeters or inches) and the CNC will calculate the corresponding angular movement depending on the selected radius. When the machining is activated, the CNC switches to operate in radius and G94 (mm/min).



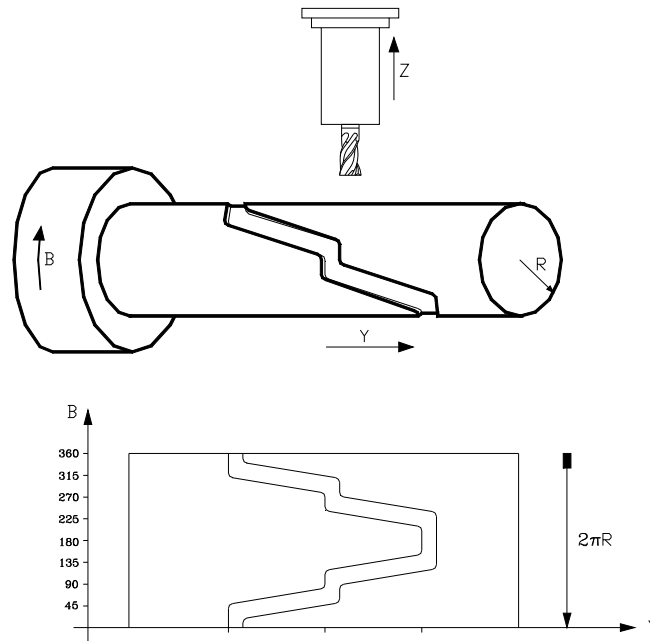
14.

"C" AXIS
Machining of the turning side of the part



CNC 8060
CNC 8065

(REF. 1402)



```
#CYL [Y,B,Z20]
G90 G42 G01 Y70 B0
G91 Z-4
G90 B15.708
G36 I3
Y130 B31.416
G36 I3
B39.270
G36 I3
Y190 B54.978
G36 I3
B70.686
G36 I3
Y130 B86.394
G36 I3
B94.248
G36 I3
Y70 B109.956
G36 I3
B125.664
G91 Z4
#CYL OFF
M30
```

14.

"C" AXIS
Machining of the turning side of the part



CNC 8060
CNC 8065

(REF. 1402)

14.

"C" AXIS

Machining of the turning side of the part



CNC 8060
CNC 8065

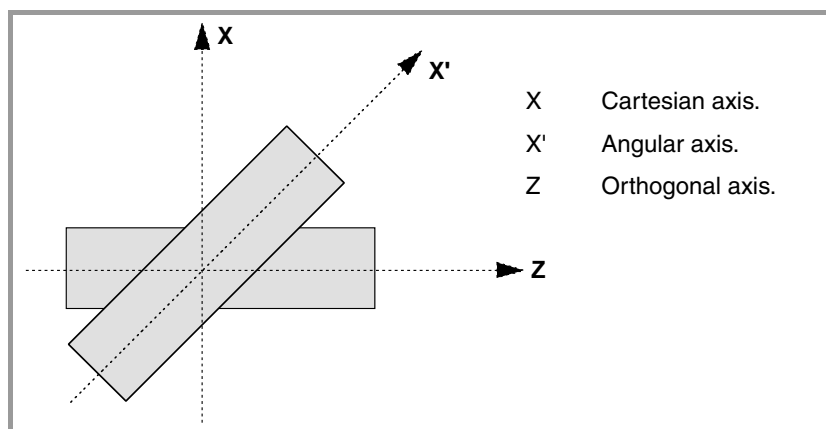
(REF. 1402)

ANGULAR TRANSFORMATION OF AN INCLINE AXIS.

15

With the angular transformation of an incline axis, it is possible to make movements along an axis that is not perpendicular to another. The movements are programmed in the Cartesian system and they are transformed into movements on the real axes.

On certain machines, the axes are configured in a Cartesian way, they are not perpendicular to each other. A typical case is the X axis of a lathe that due to sturdiness reasons is not perpendicular to the Z axis.



Programming in the Cartesian system (Z-X) requires activating an angular transformation of an inclined plane that converts the movements of the real (non-perpendicular) axes (Z-X'). This way, a movement programmed on the X axis is transformed into movements on the Z-X' axes; i.e. it then moves along the Z axis and the angular X' axis.

Turning angular transformation on and off.

The CNC assumes no transformation on power-up; the angular transformations are activated via part-program. Several angular transformations may be active at the same time.

The angular transformations are turned off via part-program. Optionally, a transformation may be "frozen" (suspended) to move angular axis by programming in Cartesian coordinates.

Influence of the reset, turning the CNC off and of the M30.

The angular transformation of an incline axis is kept active after a RESET or an M30. Turning the CNC off turns the active angular transformation off.



CNC 8060
CNC 8065

(REF. 1402)

Considerations for the angular transformation of an incline axis.

The axes that make up the angular transformation must meet the following requirements:

- Both axes must belong to the same channel.
- Both axes must be linear.
- Both axes may be masters in a pair of slaved (coupled) axes or gantry axes.

Home search is not possible when the angular transformation is active.

If the angular transformation is active, the coordinates displayed will be those of the Cartesian system. Otherwise, it will display the coordinates of the real axes.

15.

ANGULAR TRANSFORMATION OF AN INCLINE AXIS.



CNC 8060
CNC 8065

(REF. 1402)

15.1 Turning angular transformation on and off

Turn angular transformation on.

When the transformation is on, the movements are programmed in the Cartesian system and to make the movements, the CNC transforms them into movements on the real axes. The coordinates displayed on the screen will be those of the Cartesian system.

The angular transformation is turned on with the instruction #ANGAX. This instruction may be used to activate the transformation in one or several axes.

```
#ANGAX ON [1,...,n]
```

1,...,n Angular transformation to be activated.

At least one transformation must be programmed in the activating instruction, otherwise the relevant error message will be issued. The number of the angular transformation is determined by the order in which they have been defined in the machine parameter table.

```
#ANGAX ON [1]
```

```
#ANGAX ON [5,7]
```

Angular transformations may be activated either all of them at the same time or one by one. Activating one transformation does not cancel the previous ones.

This instruction turns a "frozen" (suspended) transformation on again. See "[15.2 Freezing \(suspending\) the angular transformation.](#)" on page 256.

Turn angular transformation off.

If the transformation is off, the movements are programmed and executed in the system of the real axes. The coordinates displayed on the screen will be those of the real axes.

The angular transformation is turned on with the instruction #ANGAX. The programming format is: Optional parameters are indicated between angle brackets.

```
#ANGAX OFF <[1,...,n]>
```

1,...,n Optional. Angular transformation to be activated.

If no transformation has been defined, all the transformations of the channel are turned off.

```
#ANGAX OFF
```

```
#ANGAX OFF [1]
```

```
#ANGAX OFF [5,7]
```

The angular transformation of an incline axis is kept active after a RESET or an M30. Turning the CNC off turns the active angular transformation off.

15.

ANGULAR TRANSFORMATION OF AN INCLINE AXIS.
Turning angular transformation on and off

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

15.2 Freezing (suspending) the angular transformation.

Freezing the angular transformation is a special way to make movements along the angular axis, but programming it in the Cartesian system. The angular transformation cannot be "frozen" (suspended) while jogging.

The angular transformation is "frozen" (suspended) using the instruction `#ANGAX SUSP` whose programming format is:

```
#ANGAX SUSP [1,...,n]
```

1,...,n Angular transformation to be activated.

If no angular transformation has been programmed, all the transformations of the channel are "frozen" (suspended). The number of the angular transformation is determined by the order in which they have been defined in the machine parameter table.

<code>#ANGAX SUSP</code>	Freezing all the transformations of the channel.
<code>#ANGAX SUSP [1]</code>	Freezing transformation ·1·.
<code>#ANGAX SUSP [5,7]</code>	Freezing transformations ·5· and ·7·.

Programming movements after "freezing" the angular transformation.

If an angular transformation is "frozen" (suspended), only the coordinate of the angular axis must be programmed in the motion block. If the coordinate of the orthogonal axis is programmed, the movement is carried out according to the normal angular transformation.

Canceling the freezing of a transformation.

The "freezing" of an angular transformation is canceled after a reset or an M30.

Programming `#ANGAX ON` on the frozen transformation turns it back on.

15.

ANGULAR TRANSFORMATION OF AN INCLINE AXIS.
Freezing (suspending) the angular transformation.



CNC 8060
CNC 8065

(REF. 1402)

15.3 Obtaining information on angular transformation.

Checking the configuration of the angular transformation.

The configuration data of the angular transformation may be checked directly in the machine parameter table or using the following variables.

Number of angular transformations defined.

(V.)MPK.NANG

Variable that can only be read from the PRG, PLC and INT.

It returns the number of angular transformations defined in the machine parameter table.

Axes that make up the angular transformation.

These variables refer to the angular transformation n. The brackets must be programmed.

(V.)MPK.ANGAXNA[n]

(V.)MPK.ORTGAXNA[n]

Variable that can only be read from the PRG, PLC and INT.

The first one returns the name of the angular axis. The second one returns the name of the orthogonal axis.

Geometry of the angular transformation.

These variables refer to the angular transformation n. The brackets must be programmed.

(V.)MPK.ANGANTR[n]

Variable that can only be read from the PRG, PLC and INT.

Angle between the Cartesian angle and the angular axis it is associated with. Positive angle when the angular axis has been rotated clockwise and negative if counterclockwise.

(V.)MPK.OFFANGAX[n]

Variable that can only be read from the PRG, PLC and INT.

Offset of the origin of the angular transformation. Distance between machine zero and the origin of the coordinate system of the incline axis.

Checking the status of the angular transformation.

Status of the angular transformation.

(V.)[n].G.ANGAXST

Variable that can only be read from the PRG, PLC and INT.

It returns the status of angular transformation defined in the channel.

(V.)[n].G.ANGIDST

Variable that can only be read from the PRG, PLC and INT.

It returns the status of angular transformation defined in the [i] position in the machine parameters.

Both variables return the following values:

Value	Meaning
0	The transformation is deactivated.
1	The transformation is activated.
2	The transformation is frozen (suspended).

15.

ANGULAR TRANSFORMATION OF AN INCLINE AXIS.
Obtaining information on angular transformation.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

15.

ANGULAR TRANSFORMATION OF AN INCLINE AXIS.

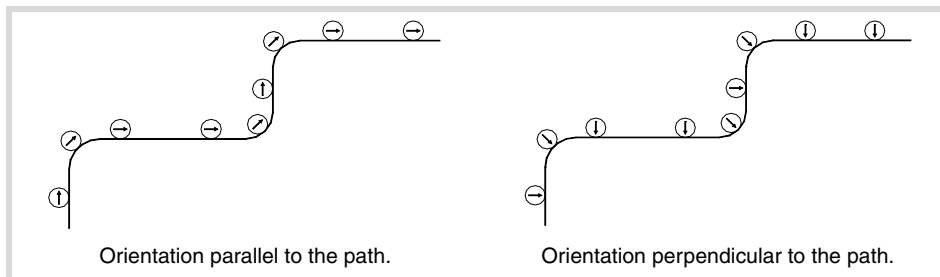
Obtaining information on angular transformation.



CNC 8060
CNC 8065

(REF. 1402)

Using tangential control, a rotary axis can keep the same orientation with respect to the programmed path all the time. The machining path is defined on the axes of the active plane and the CNC maintains the orientation of the rotary axis along the whole path.



Turning tangential control on and off.

The CNC does not activate tangential control on power-up; it is activated via part-program. Tangential control may be active on several axes. Once tangential control is active, the tangential axis cannot be moved in jog or by program; the CNC is in charge of orienting this axis.

Optionally, tangential control can also be "frozen" so it can be re-activated in the same conditions later on.

There are two ways to program tangential control; using either ISO-coded functions or high level commands. Both modes are equivalent and may be combined in the same part-program.

Influence of the reset, turning the CNC off and of the M30.

Tangential control is modal. Tangential control is canceled on power-up, after executing an M02 or M30 and after an emergency or reset.

Considerations about tangential control.

Tangential control is compatible with tool radius and length compensation. Mirror image may also be applied while tangential control is active.

Axes allowed in tangential control.

Tangential control can only be activated on module type rotary axes. An axis of the plane or the longitudinal axis cannot be set as tangential axis. Likewise, a tangential axis can also be a gantry axis, including the gantry axis associated with the rotary axis.

Tool inspection.

Tool inspection is possible while tangential control is active. When going into tool inspection, the CNC cancels tangential control so the axes can move. After quitting tool inspection, the CNC re-activates tangential control in the same conditions as before.

Jogging the axes.

The tangential axis cannot be moved while tangential control is active. The axes not affected by tangential control may be moved freely.

When jogging the axes, the CNC cancels tangential control. Once the movement has ended, the CNC re-activates tangential control in the same conditions as before.

MDI mode.

The MDI mode may be accessed from jog mode to activate tangential control and move the axes using blocks programmed in MDI. The tangential axis cannot be moved while tangential control is active.

16.**TANGENTIAL CONTROL.**

CNC 8060
CNC 8065

(REF. 1402)

16.1 Turning tangential control on and off.

There are two ways to manage tangential control; using either ISO-coded functions or high level commands. Both modes are equivalent and may be combined in the same part-program.

Activating tangential control.

While tangential control is active, the movements are programmed for the axes of the active work plane. It is not possible to program movements of the tangential axis; it is up to the CNC to orient this axis.

Tangential control is activated with function G45 or with the instruction #TANGCTRL. These commands also restore a "frozen" tangential control, but the angle must be programmed again. See "[16.2 Freezing tangential control.](#)" on page 264.

Programming format (1).

This function may be used to activate tangential control on one or several axes; it does not allow to set the positioning feedrate of the tangential axis. At least one tangential axis must be defined in this function.

G45 X-C

X-C Axis upon which tangential control is applied and angular position with respect to the path. The angle is defined in degrees (± 359.9999).

```
G45 A90
G45 B45 W15.123 B2=-34.5
```

Programming format (2).

This statement may be used to activate tangential control on one or several axes and to set the positioning feedrate of the tangential axis. There is no need to activate any axis in order to set the feedrate.

The programming format is: Optional parameters are indicated between angle brackets.

#TANGCTRL ON [<X-C>, <F>]

X-C Optional. Axis upon which tangential control is applied and angular position with respect to the path. The angle is defined in degrees (± 359.9999).

F Optional. Tangential axis orienting feedrate.

Although both parameters are optional, at least one of them must be programmed.

```
#TANGCTRL ON [A34.35]
#TANGCTRL ON [A90, F300]
#TANGCTRL ON [B-45, W15.123, F300]
#TANGCTRL ON [F300]
```

Combining both programming formats.

The two programming formats may be combined in the same part-program. For example, the instruction may be used to set the positioning feedrate and function G45 to active tangential control.

```
#TANGCTRL ON [F1000]
G45 W45
```

Programming the positioning angle.

The positioning angle is defined in degrees (± 359.9999). The angle is defined with respect to the path to follow, positive angle for counterclockwise positioning and negative angle for clockwise positioning.

16.

TANGENTIAL CONTROL:
Turning tangential control on and off.

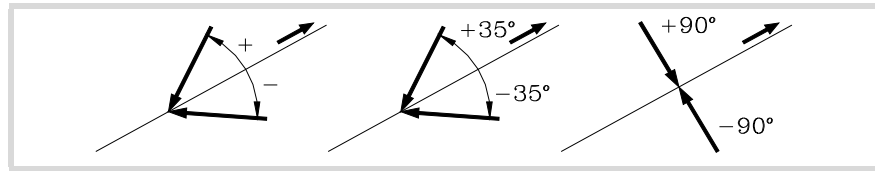
FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

16.

TANGENTIAL CONTROL:
Turning tangential control on and off.



The positioning angle is only maintained when tangential control is "frozen"; in the rest of the cases, it must be programmed every time tangential control is activated. See "16.2 Freezing tangential control." on page 264.

Positioning feedrate for the tangential axis.

The feedrate for the tangential axes is set with the instruction #TANGCTRL. This feedrate is only applied to the movements of the tangential axes; not to the axes of the plane whose feedrate is F.

```
#TANGCTRL ON [F1000]
```

The tangential feedrate stays active even when tangential control is canceled. This means that the feedrate will be applied the next time tangential control is activated.

If no feedrate has been defined for the tangential axis, it behaves as follows. In any case, the maximum feedrate for each tangential axis will be limited by its machine parameter MAXFEED.

- If the tangential axis must move alone, it moves at the feedrate set by machine parameter MAXFEED.
- If the tangential axis moves with the axes of the plane, its feedrate is the same as the one for those axes.

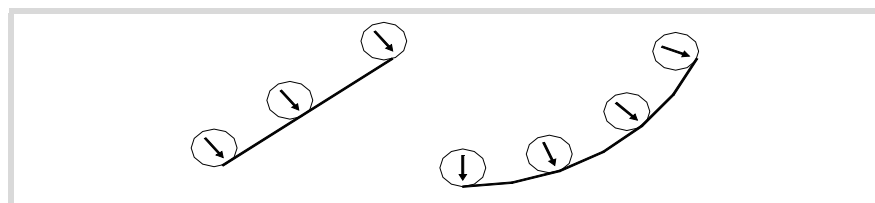
Operation of the tangential control.

Every time tangential control is activated, the CNC acts as follows:

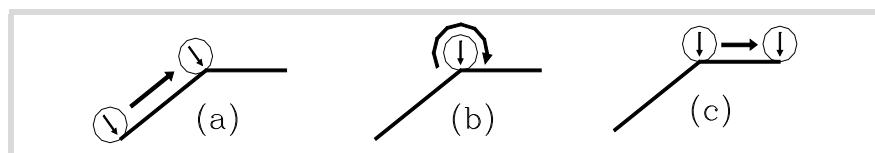
- 1 The CNC orients the tangential axis with respect to the first section and it puts it in the programmed position.



- 2 The interpolation of the plane axes starts once the tangential axis has been positioned. The orientation of the tangential axis is maintained in linear sections whereas in circular interpolations the programmed orientation is maintained throughout the whole travel.



- 3 Blending two sections requires a new orientation of the tangential axis, the CNC ends the section in progress, then orients the tangential axis with respect to the next section and goes on with the execution.



CNC 8060
CNC 8065

(REF. 1402)

Cancel tangential control.

Tangential control is canceled with function G45 or with the instruction #TANGCTRL.

Programming format (1).

This function cancels tangential control in all the axes of the channel.

G45

```
G45
```

Programming format (2).

This instruction cancels tangential control in one or several axes. If no axis is programmed, it cancels tangential control in all the axes of the channel.

The programming format is: Optional parameters are indicated between angle brackets.

```
#TANGCTRL OFF <[X~C]>
```

X~C Optional. Axis where tangential control is canceled.

```
#TANGCTRL OFF
#TANGCTRL OFF [A]
#TANGCTRL OFF [B, W, V]
```

Cancellation of tangential control during tool radius compensation.

Tangential control may be canceled even if tool radius compensation is active. However, it is recommended to freeze tangential control instead of canceling it. This is because the instruction #TANGCTRL OFF, besides canceling tangential control, it generates some additional blocks at the beginning and end of the tool compensation.

16.

TANGENTIAL CONTROL.
Turning tangential control on and off.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

16.2 Freezing tangential control.

Freezing tangential control is a special cancellation where the CNC remembers the programmed angle. When restoring tangential control, the CNC orients the axis with the same angle it had when tangential control was frozen. Freezing tangential control does not cancel tool radius compensation.

16.

TANGENTIAL CONTROL.
Freezing tangential control.

Turning tangential control on.

While tangential control is frozen, the movements are programmed for the axes of the active work plane. No movement of the tangential axis can be programmed.

Tangential control is frozen with function G145 or with the instruction #TANGCTRL.

Programming format (1).

This instruction freezes tangential control in one or several axes. If no axis is programmed, it freezes tangential control in all the axes of the channel.

The programming format is: Optional parameters are indicated between angle brackets.

```
G145 <K0> <X~C>
```

K0 Optional. Freeze tangential control.

X~C Optional. Axis where tangential control is frozen.

Parameter κ may take the values $\cdot 0 \cdot$ and $\cdot 1 \cdot$. When set to $\cdot 1 \cdot$, it means that a previously frozen tangential axis is to be resumed. If the parameter κ is not programmed, the CNC assumes $\kappa 0$.

```
G145 K0
G145 K0 A
G145 K0 B W C
G145 B A
```

Programming format (2).

This statement freezes tangential control in one or several axes. If no axis is programmed, it freezes tangential control in all the axes of the channel.

The programming format is: Optional parameters are indicated between angle brackets.

```
#TANGCTRL SUSP <[X~C]>
```

X~C Optional. Axis where tangential control is frozen.

```
#TANGCTRL SUSP
#TANGCTRL SUSP [A]
#TANGCTRL SUSP [B, W]
```

Canceling tangential control.

Tangential control is resumed with function G145 or with the instruction #TANGCTRL.

Programming format (1).

This function resumes tangential control in one or several axes. If no axis is programmed, it resumes tangential control in all the axes of the channel.

The programming format is: Optional parameters are indicated between angle brackets.

```
G145 K1 <X~C>
```

K1 Restoring tangential control.

X~C Optional. Axis where tangential control is resumed.

Parameter κ may take the values $\cdot 0 \cdot$ and $\cdot 1 \cdot$. When set to $\cdot 0 \cdot$, it means that tangential control is to be frozen.

```
G145 K1
G145 K1 A
G145 K1 B W C
```



CNC 8060
CNC 8065

(REF. 1402)

Programming format (2).

This instruction resumes tangential control in one or several axes. If no axis is programmed, it resumes tangential control in all the axes of the channel.

The programming format is: Optional parameters are indicated between angle brackets.

```
#TANGCTRL RESUME <[X~C]>
```

X~C Optional. Axis where tangential control is resumed.

```
#TANGCTRL RESUME  
#TANGCTRL RESUME [A]  
#TANGCTRL RESUME [B, W, C]
```

16.

TANGENTIAL CONTROL.
Freezing tangential control.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

16.3 Obtaining information on tangential control.

16.

Checking the configuration of the angular transformation.

The configuration data of the tangential control may be checked directly in the machine parameter table or using the following variables.

Is it a module type of rotary axis?

(V.)[n].MPA.AXISMODE.Xn

The variable indicates the type of rotary axis; if it is a module type, the variable must return the value of -0-.

Consulting the tangential control data.

(V.)A.TANGAN.Xn

This variable returns the angle programmed in the Xn axis.

(V.)G.TANGFEED

This variable returns the positioning feedrate programmed for tangential control.

Consulting the tangential control status.

(V.)PLC.TANGACTIVCn

This variable indicates whether tangential control is active in channel nor not. A -1- if the tangential control is active and a -0- if otherwise.

(V.)PLC.TANGACTx

This variable indicates whether tangential control is active in the x axis or not. A -1- if the tangential control is active and a -0- if otherwise.

(V.)[n].G.TGCTRLST

It returns the status of tangential control in the channel. A value of -0- if tangential control is off, a -1- if it is on and a -2- if frozen (suspended).

(V.)[n].A.TGCTRLST.Xn

It returns the status of tangential control in the axis. A value of -0- if tangential control is off, a -1- if it is on and a -2- if frozen (suspended).

Initialization of the variables.

Canceling tangential control initializes all the variables except (V.)A.TANGFEED, because the programmed feedrate is maintained for a possible future tangential control.

When freezing (suspending) tangential control, the variables act as follows.

(V.)A.TANGAN.Xn	It maintains the value of the programmed angle.
(V.)G.TANGFEED	It is not initialized.
(V.)PLC.TANGACTIVCn	It is not initialized.
(V.)PLC.TANGACTx	It is initialized.



CNC 8060
CNC 8065

(REF. 1402)

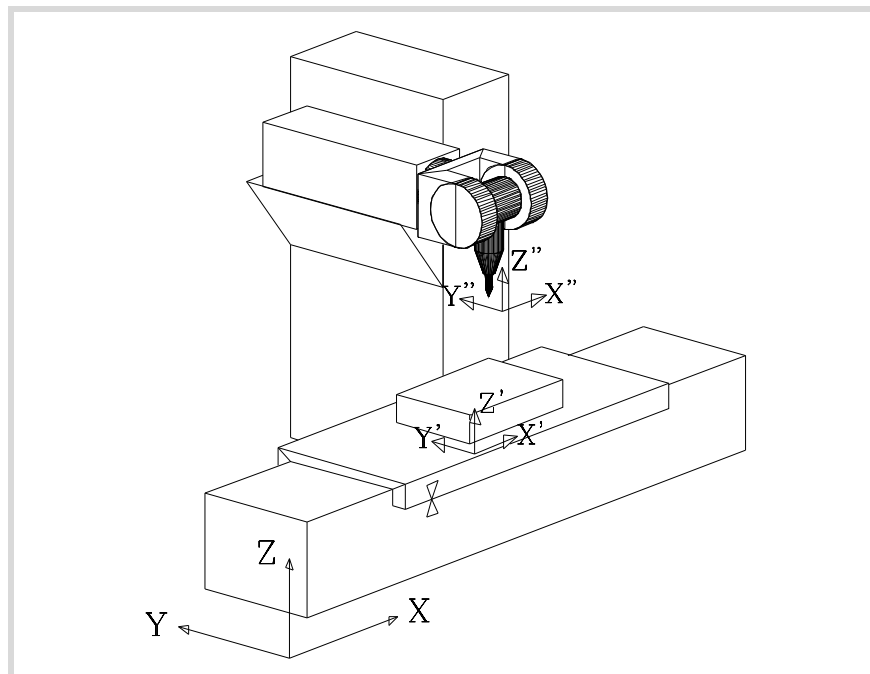
The description of the general coordinate transformation is divided into these basic functionalities:

- Selection of the kinematics. #KIN ID instruction.
- Definition and selection of the machining coordinate system (inclined plane). #CS instruction.
- Definition and selection of the fixture coordinate system. #ACS instruction.
- RTCP (Rotating Tool Center Point) transformation. #RTCP instruction.
- Orient the tool perpendicular to the work plane (parallel to the third axis). #TOOL ORI instruction.
- Tool length compensation adaptation implicit in the program. #TLC instruction.

For clarity's sake, the following examples show three coordinate systems:

- XYZ Machine coordinate system.
- X' Y' Z' Part coordinate system.
- X'' Y'' Z'' Tool coordinate system.

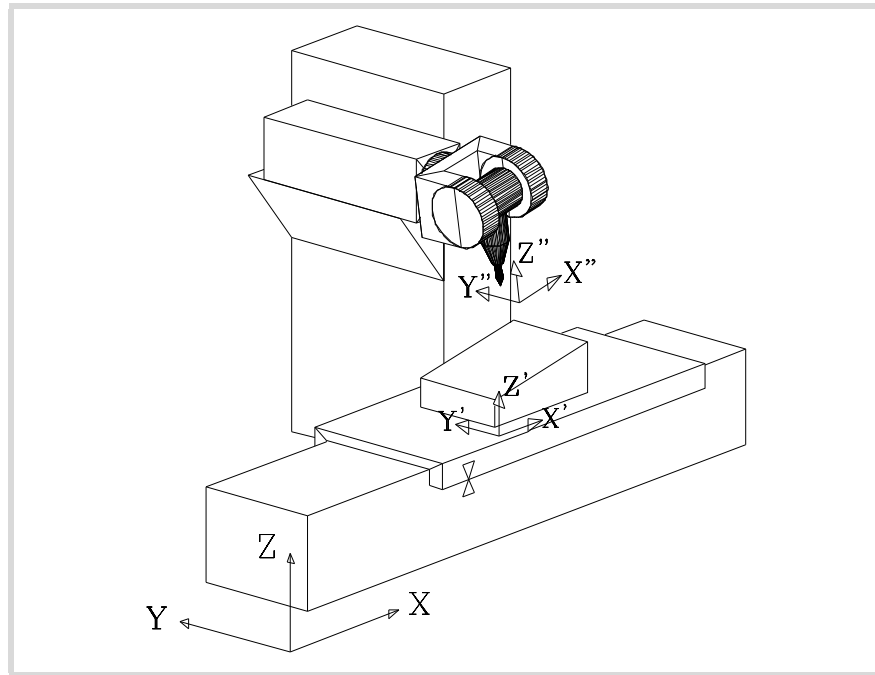
When no transformation has been made and the spindle is in the starting position, the three coordinate systems coincide.



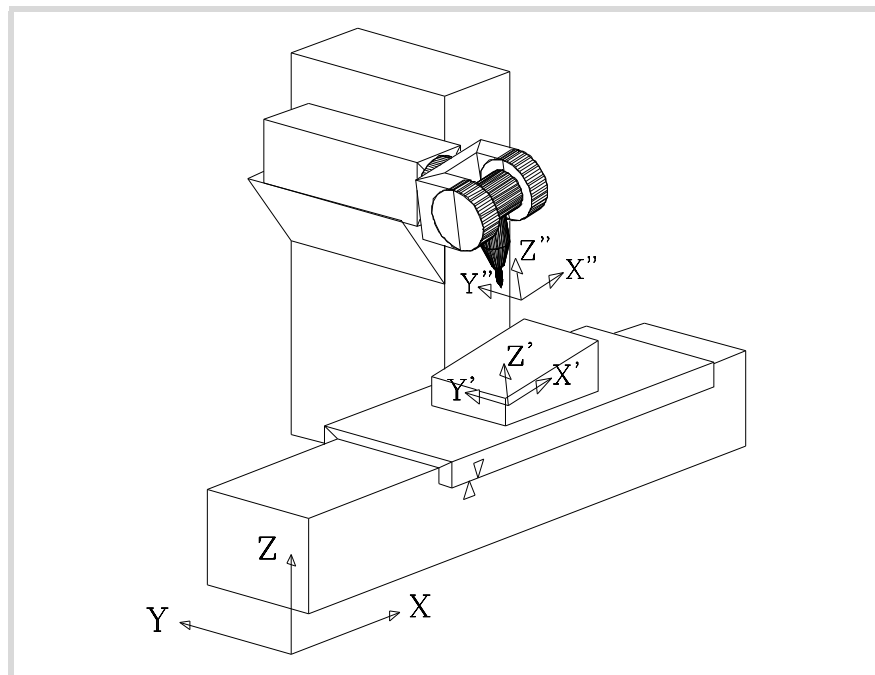
When turning the spindle, the tool coordinate system (X'' Y'' Z'') changes.

17.

COORDINATE TRANSFORMATION



If besides this, a new machining coordinate system is selected (#CS instruction) or fixture coordinate system (#ACS instruction) the part coordinate system will also change (X' Y' Z').

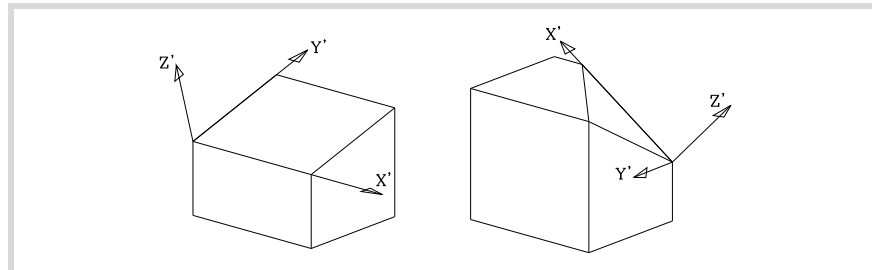


17.1 Movement in an inclined plane

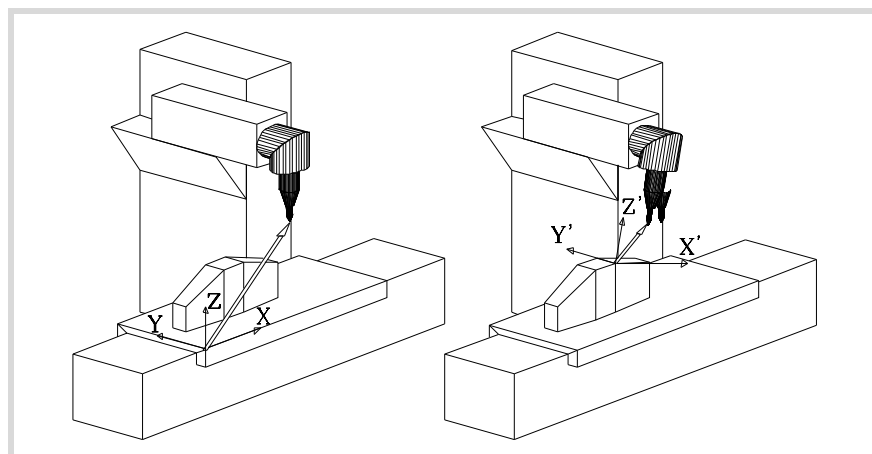
An inclined plane is any plane in space resulting from the coordinate transformation of the XYZ axes.

Any plane in space may be selected to carry out machining operations in it.

To define the inclined plane corresponding to the machining operations, use the #CS and #ACS instructions that are described later on in this chapter.

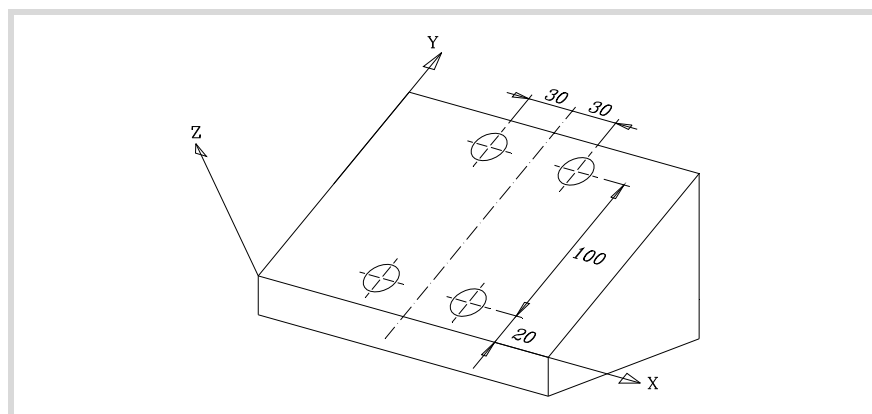


The new coordinates (right figure) are referred to the new part zero assuming that the tool is positioned perpendicular to the new plane.



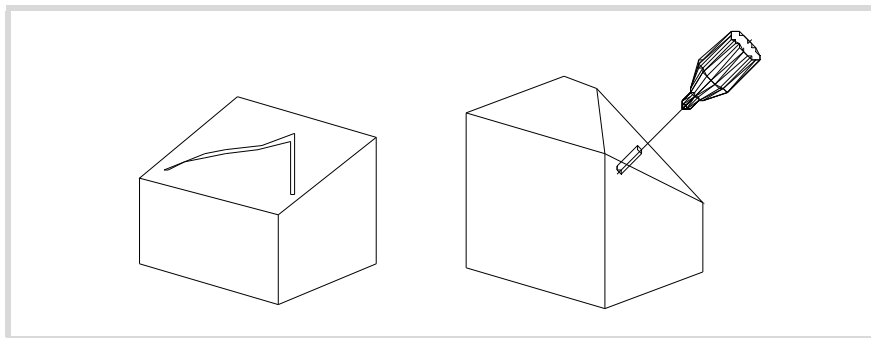
To place the tool at that position, use the #TOOL ORI instruction or the kinetics related variables that indicate the position that each rotary axis of the spindle head must occupy. See "[17.8 Kinematics related variables](#)" on page 295.

From this moment on, the programming and the X, Y movements are carried out along the selected plane and those of the Z axis will be perpendicular to it.



To orient the tool and work with it perpendicular to the plane, use the instruction #TOOL ORI that is described later on in this chapter.

17.

17.**COORDINATE TRANSFORMATION**
Movement in an inclined plane

17.2 Kinematics selection (#KIN ID)

The OEM may set up to 6 different kinematics for the machine. Each one of them indicates the type of spindle being used, its characteristics and dimensions.

To work with coordinate transformation, the kinematics being used must be indicated. Usually, the OEM defines the kinematics number being used by default by means of general machine parameter KINID.

If several kinematics have been defined, it is possible to activate the desired one via part-program using the instruction #KIN ID. If there is only one and it has been set as the default kinematics, there is no need to program this instruction.

Format to activate a particular kinematics:

```
#KIN ID [n]
n      Kinetics number
```

Functions #RTCP, #TLC and #TOOL ORI must always be activated after selecting a kinematics. The kinematics cannot be changed while function #RTCP or #TLC is active.

N50 #KIN ID[2]	(Activating kinematics Nr 2)
N60 #RTCP ON	(Activating RTCP with kinematics 2)
...	
N70 #RTCP OFF	(Turn RTCP off)
N80 M30	

17.

COORDINATE TRANSFORMATION
Kinematics selection (#KIN ID)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

17.3 Coordinate systems (#CS) (#ACS)

There are two different types of coordinate systems, namely the machining coordinate system and the fixture coordinate system. Each one is handled with its associated instruction.

#CS	With the #CS instruction, up to 5 machining coordinate systems may be defined, stored, activated and deactivated.
#ACS	With the #ACS instruction, up to 5 fixture coordinate systems may be defined, stored, activated and deactivated. It is used to compensate for workpiece inclination due to the fixtures used to secure them.

Both instructions use the same programming format and may be used independently or combined as indicated in the following sections.

Several #ACS and #CS coordinate systems may be combined. When activating a new one, it is added to the current coordinate system. See "[17.4 How to combine several coordinate systems](#)" on page 286.

It is recommended to start the program with #CS NEW or #ACS NEW to avoid undesired planes. This happens, for example, after interrupting the program and resuming execution.

The coordinate systems and part zero

The origin of the coordinate system is referred to the current part zero. While being a #CS or #ACS activated, new part zeros may be preset in the plane.

Deactivating an inclined plane, if not defined otherwise, restores the part zero that was defined before activating the inclined plane. Optionally, it is possible to define whether to keep or not the current part zero.

Sometimes, it could happen that when activating a #CS or a #ACS saved earlier, the coordinate origin of the plane may not be desired one. This happens if the part zero is modified between the definition and the application of the #CS or #ACS.

Considerations for the two functions

Both coordinate systems, (#CS and #ACS) are kept active after a Reset or an M30. On start-up, the CNC either keeps or cancels the coordinate system according to the setting of machine parameter CSCANCEL.

Operations with the coordinate systems

Both instructions (#CS and #ACS) use the same programming format. The meaning of the parameters used by both instructions is the following. The brackets [] must be programmed. The parameters defined between angular brackets "< >" are optional.

n	Number of the coordinate system (1..5). Up to 5 different ones may be defined and stored to be activated at any time.
MODE m	Definition mode used (1..6).
V1...V3	Components of the translation vector.
φ1...φ3	Rotation angles.
<0/1>	Plane alignment (value 0/1). Only in modes 3, 4 and 5.
<KEEP>	It keeps the part zero defined in the transformation.
<FIRST/SECOND>	Axis orientation. Only in mode 6.

Definition mode

The MODE definition mode sets the order in which the axes rotate to reach the desired plane. In some cases, the resolution of the plane presents two solutions; the selection is carried out defining which axis of the coordinate system stays aligned with the plane.

17.

COORDINATE TRANSFORMATION
Coordinate systems (#CS) (#ACS)



CNC 8060
CNC 8065

(REF. 1402)

Keep the part zero when deactivating a transformation

Turning a transformation off, if not defined otherwise, restores the part zero that was defined before activating the inclined plane.

The <KEEP>command may be used to keep the current part zero. This command is only admitted in instructions that deactivate a coordinate system.

Programming formats

- Format to define and store:

```
#CS DEF [n] [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>]
#ACS DEF [n] [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>]
```

- Format to define, store and activate:

```
#CS DEF [n] [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>]
#ACS ON [n] [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>]
```

- Format to define and activate (without storing):

```
#CS ON [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>]
#ACS ON [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>]
```

Only one of them may be defined; to define another one, the previous one must be canceled. It may be used, until canceled, as any other coordinate system stored in memory.

- Format to deactivate and delete all the current #CS or #ACS and define and activate a new one:

```
#CS NEW <KEEP> [n] [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>]
#ACS NEW <KEEP> [n] [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>]
```

- Format to deactivate and delete all the current #CS or #ACS and define and activate a new one (without storing):

```
#CS NEW <KEEP> [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>]
#ACS NEW <KEEP> [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>]
```

- Format to assume and store the current coordinates as a #CS or a #ACS:

```
#CS DEF ACT [n]
#ACS DEF ACT [n]
```

- Format to activate one that has been stored:

```
#CS ON [n]
#ACS ON [n]
```

- Format to activate the one stored last:

```
#CS ON
#ACS ON
```

- Format to deactivate the one activated last:

```
#CS OFF <KEEP>
#ACS OFF <KEEP>
```

- Format to deactivate all the activated #CS or #ACS:

```
#CS OFF ALL
#ACS OFF ALL
```

45° spindles (Huron type)

Huron type spindles have two solutions for orienting the tool perpendicular to the new work plane. For this type of spindles, it is possible to select which one of the two solutions is to be applied. See "[17.3.7 Operation with 45° spindles \(Huron type\)](#)" on page 285.

17.

COORDINATE TRANSFORMATION
Coordinate systems (#CS) (#ACS)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

Programming example

```
#CS NEW [3] [MODE 1,2,15,5,2,3,4.5]
  (It deletes the current CS)
  (It defines it and stores it as CS3)
#CS DEF [2] [MODE 1,P1,15,5,2,3,4.5]
  (It defines it and stores it as CS2)
#CS DEF [5] [MODE 2,0,1,2,0,30,30]
  (It defines it and stores it as CS5)
#CS ON
  (It activates the CS programmed last, the CS5)
#CS OFF
  (It cancels the CS5)
#CS ON [3]
  (It activates the CS3)
#CS DEF [2] [MODE 1,1,1.2,1.3,0,0,33]
  (It redefines the stored CS2, the CS3 stays active)
M30
```



In "Editing - simulation" mode, the user can access an editor that makes programming inclined planes easier using the instructions #CS and #ACS. See the operating manual for further information on the editor of inclined planes.

17.

COORDINATE TRANSFORMATION
Coordinate systems (#CS) (#ACS)



CNC 8060
CNC 8065

(REF. 1402)

17.3.1 Coordinate system definition MODE 1

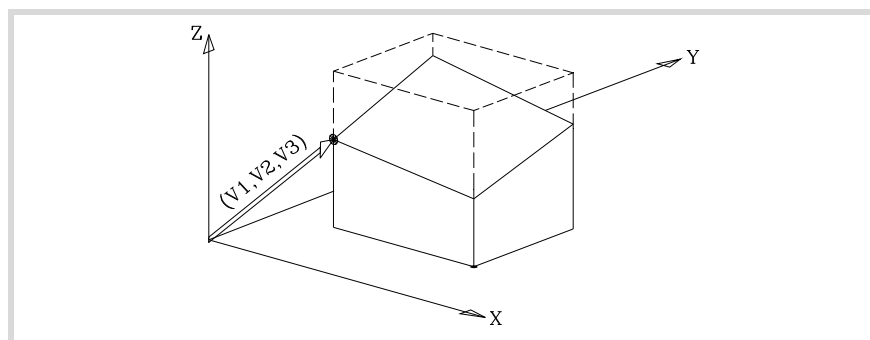
Both instructions use the same programming format and may be used together or separately.

```
#CS DEF [n] [MODE 1, V1, V2, V3, φ1, φ2, φ3]
#ACS DEF [n] [MODE 1, V1, V2, V3, φ1, φ2, φ3]
```

It defines the inclined plane resulting from rotating the amounts indicated in ϕ_1, ϕ_2, ϕ_3 first around the first axis, then around the second axis and finally around the third axis respectively.

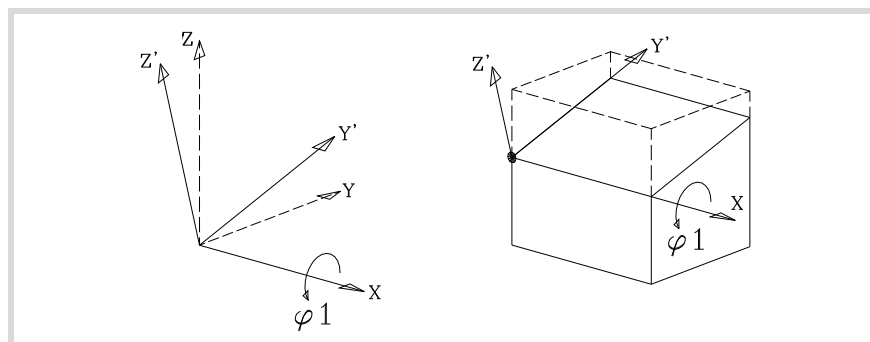
V1, V2, V3

Define the coordinate origin of the inclined plane with respect to the current part zero.



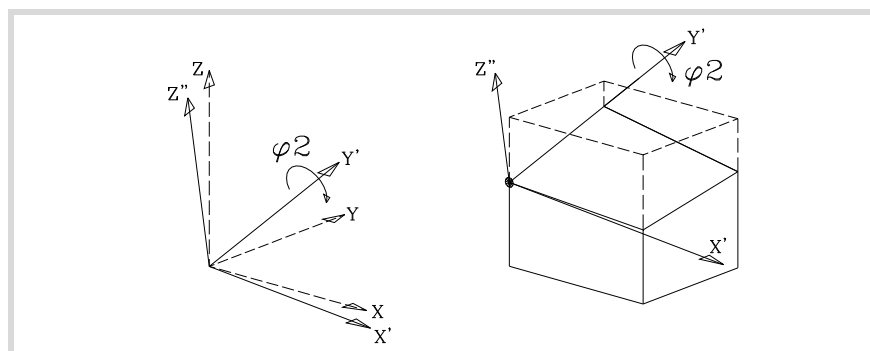
ϕ_1, ϕ_2, ϕ_3

Define the inclined plane resulting from having rotated first around the first axis (X), the amount indicated by ϕ_1 .



In the figure, the new coordinate system resulting from this transformation is called X' Y' Z' because the Y, Z axes have been rotated.

Then, rotate around the 2nd axis (Y'), the ϕ_2 amount.



In the figure, the new coordinate system resulting from this transformation is called X' Y' Z'' because the X, Z axes have been rotated.

And last, rotate around the Z'' axis the amount indicated by ϕ_3 .

17.

COORDINATE TRANSFORMATION
Coordinate systems (#CS) (#ACS)



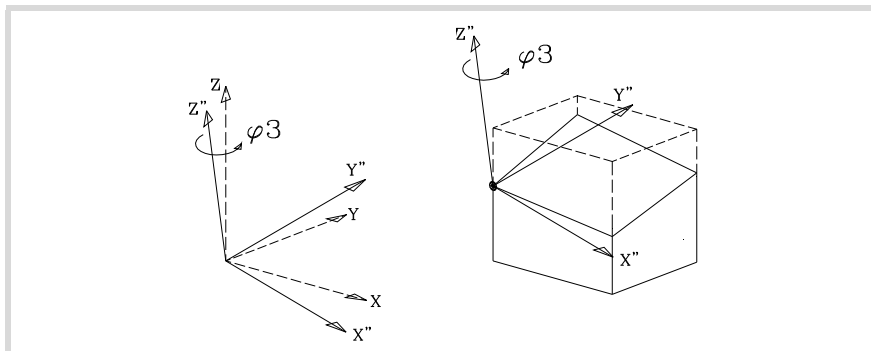
CNC 8060
CNC 8065

(REF. 1402)

17.

COORDINATE TRANSFORMATION

Coordinate systems (#CS) (#ACS)



17.3.2 Coordinate system definition MODE 2

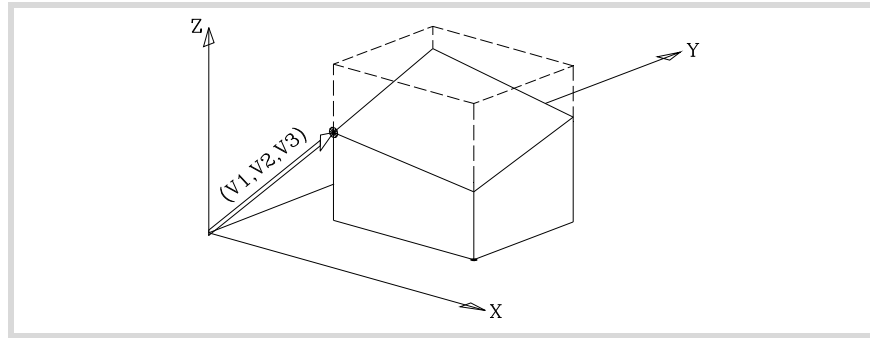
Both instructions use the same programming format and may be used together or separately.

```
#CS DEF [n] [MODE 2, V1, V2, V3, φ1, φ2, φ3]
#ACS DEF [n] [MODE 2, V1, V2, V3, φ1, φ2, φ3]
```

They define, in spherical coordinates, the inclined plane resulting from having rotated around the 3rd axis, then around the 2nd one and then again around the 3rd axis the amounts indicated by φ1, φ2, φ3 respectively.

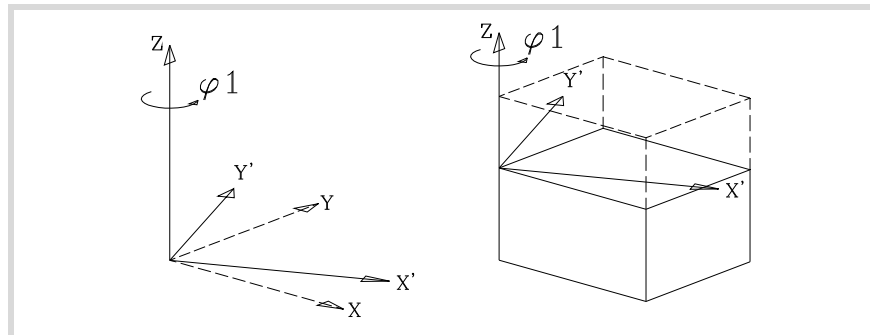
V1, V2, V3

Define the coordinate origin of the inclined plane with respect to the current part zero.



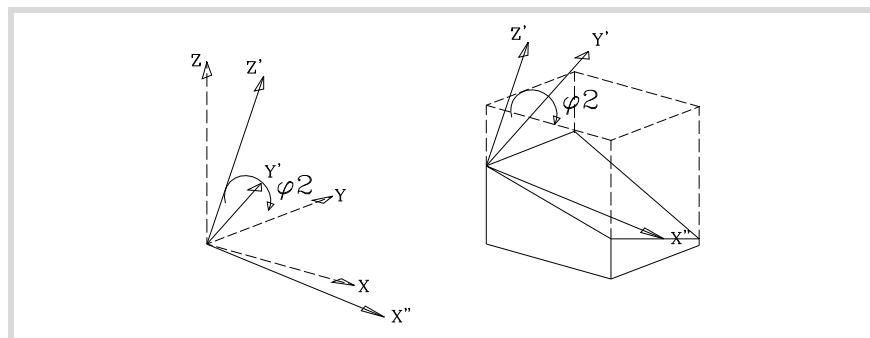
φ1, φ2, φ3

Define the inclined plane resulting from having rotated first around the 3rd axis (Z), the amount indicated by φ1.



In the figure, the new coordinate system resulting from this transformation is called X' Y' Z' because the X, Y axes have been rotated.

Then, it must be rotated around the Y' axis the φ2 amount.



In the figure, the new coordinate system resulting from this transformation is called X'' Y' Z'' because the X, Z axes have been rotated.

And last, rotate around the Z'' axis the amount indicated by φ3.

17.

COORDINATE TRANSFORMATION
 Coordinate systems (#CS) (#ACS)

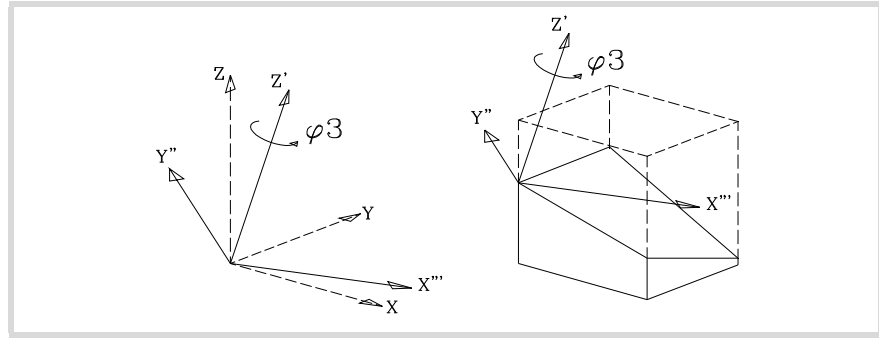


CNC 8060
CNC 8065

(REF. 1402)

17.**COORDINATE TRANSFORMATION**

Coordinate systems (#CS) (#ACS)

CNC 8060
CNC 8065

(REF. 1402)

17.3.3 Coordinate system definition MODE 3

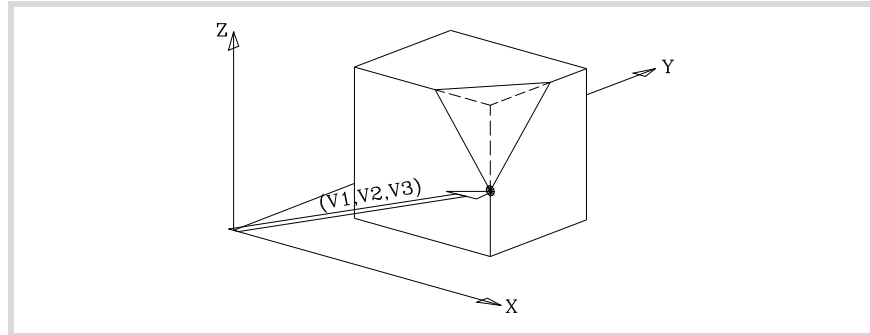
Both instructions use the same programming format and may be used together or separately.

```
#CS DEF [n] [MODE 3, V1, V2, V3, φ1, φ2, φ3, <0/1>]
#ACS DEF [n] [MODE 3, V1, V2, V3, φ1, φ2, φ3, <0/1>]
```

The inclined plane is defined with the angles it forms with respect to the 1st and 2nd axes (X Y) of the machine's coordinate system.

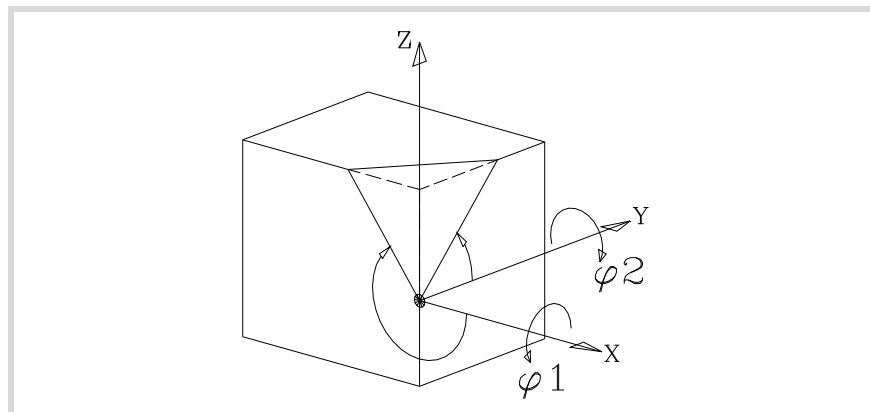
V1, V2, V3

Define the coordinate origin of the inclined plane with respect to the current part zero.



φ1, φ2

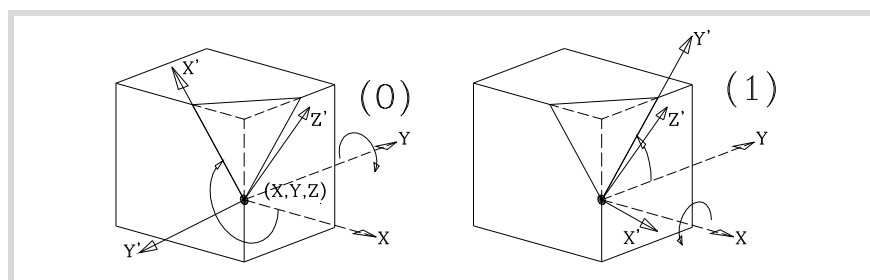
Define the angles that the inclined plane forms with the 1st and 2nd axes (X Y) of the machine's coordinate system.



0/1

Defines which of the axes of the new plane (X' Y') is aligned with the edge.

If <0> the X' axis and if <1> the Y' axis. If not programmed, it assumes <0>.



φ3

Permits defining and applying a coordinate rotation in the new cartesian plane X' Y'.

17.3.4 Coordinate system definition MODE 4

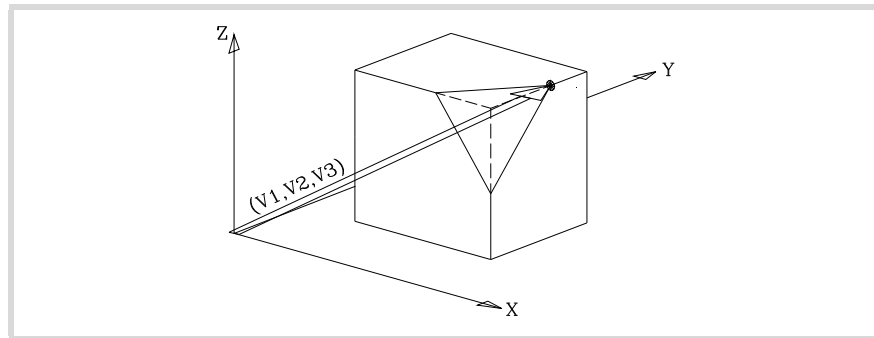
Both instructions use the same programming format and may be used together or separately.

#CS DEF [n] [MODE 4, V1, V2, V3, ϕ 1, ϕ 2, ϕ 3, <0/1>]
 #ACS DEF [n] [MODE 4, V1, V2, V3, ϕ 1, ϕ 2, ϕ 3, <0/1>]

The inclined plane is defined with the angles it forms with respect to the 1st and 3rd axes (X Z) of the machine's coordinate system.

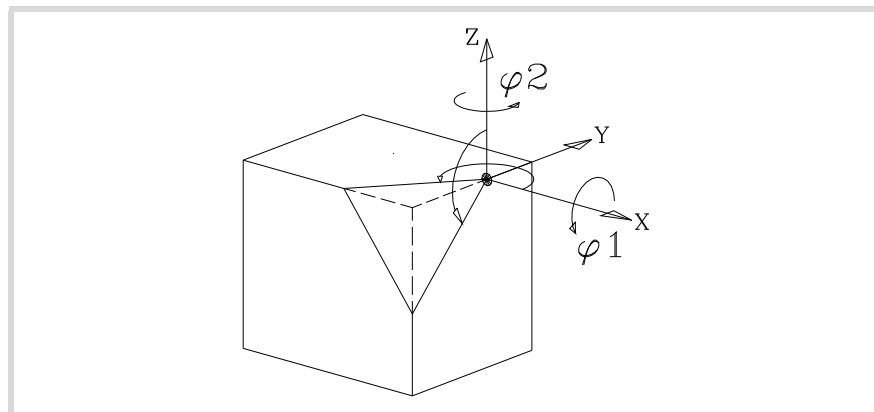
V1, V2, V3

Define the coordinate origin of the inclined plane with respect to the current part zero.



ϕ 1, ϕ 2

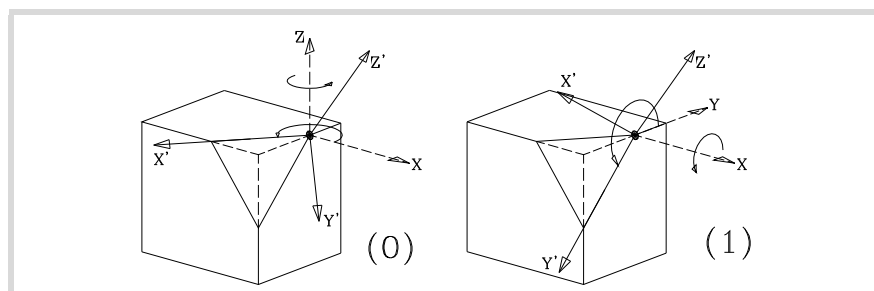
Define the angles that the inclined plane forms with the 1st and 3rd axes (X Z) of the machine's coordinate system.



0/1

Defines which of the axes of the new plane (X' Y') is aligned with the edge.

If <0> the X' axis and if <1> the Y' axis. If not programmed, it assumes <0>.



ϕ 3

Permits defining and applying a coordinate rotation in the new cartesian plane X' Y'.

17.

COORDINATE TRANSFORMATION
Coordinate systems (#CS) (#ACS)



CNC 8060
CNC 8065

(REF. 1402)

17.3.5 Coordinate system definition MODE5

Both instructions use the same programming format and may be used together or separately.

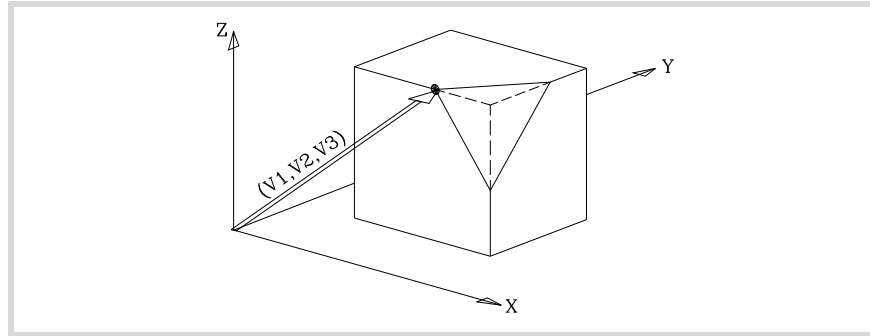
#CS DEF [n] [MODE 5, V1, V2, V3, ϕ 1, ϕ 2, ϕ 3, <0/1>]

#ACS DEF [n] [MODE 5, V1, V2, V3, ϕ 1, ϕ 2, ϕ 3, <0/1>]

The inclined plane is defined with the angles it forms with respect to the 2nd and 3rd axes (Y Z) of the machine's coordinate system.

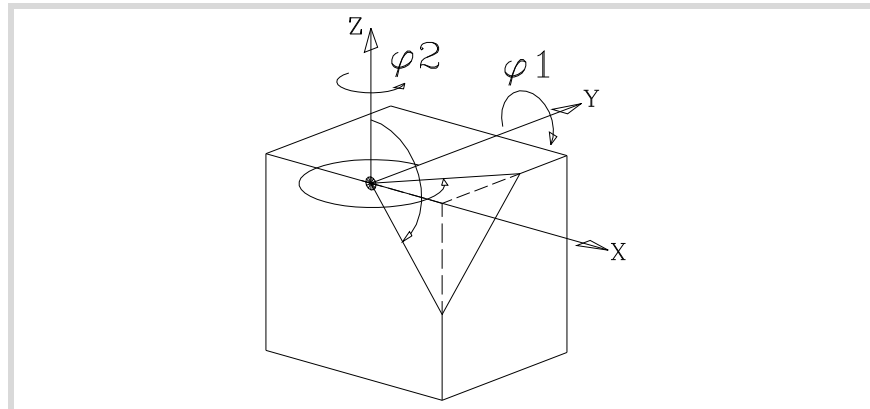
V1, V2, V3

Define the coordinate origin of the inclined plane with respect to the current part zero.



ϕ 1, ϕ 2

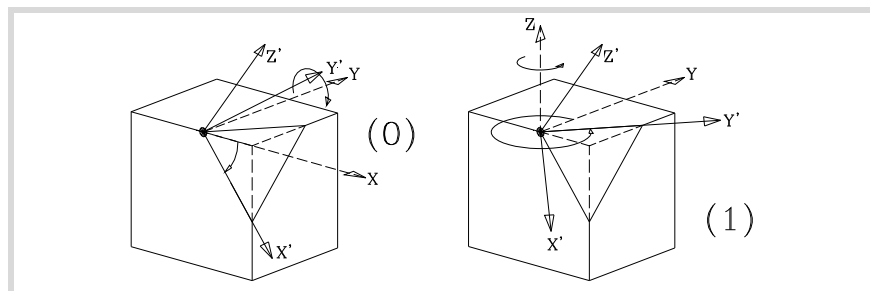
Define the angles that the inclined plane forms with the 2nd and 3rd axes (Y Z) of the machine's coordinate system.



0/1

Defines which of the axes of the new plane (X' Y') is aligned with the edge.

If <0> the X' axis and if <1> the Y' axis. If not programmed, it assumes <0>.



ϕ 3

Permits defining and applying a coordinate rotation in the new cartesian plane X' Y'.

17.3.6 Coordinate system definition MODE6



In order to use this definition, while setting up the machine, the tool position when it is parallel to the Z axis of the machine must be set as the spindle's rest position.

Both instructions use the same programming format and may be used together or separately.

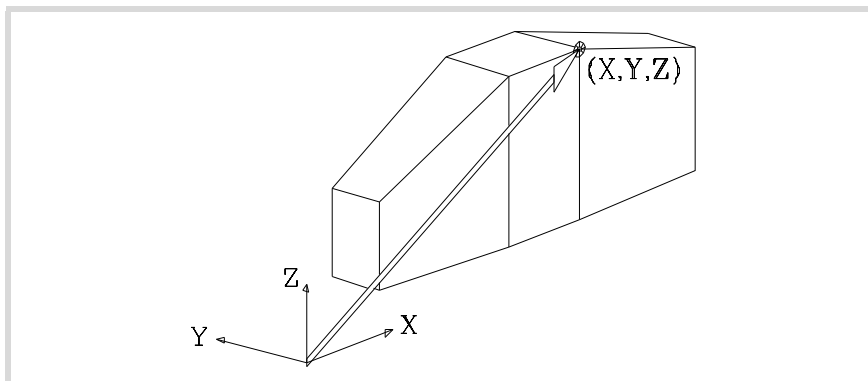
#CS DEF [n] [MODE 6, V1, V2, V3, φ1, <FIRST/SECOND>]

#ACS DEF [n] [MODE 6, V1, V2, V3, φ1, <FIRST/SECOND>]

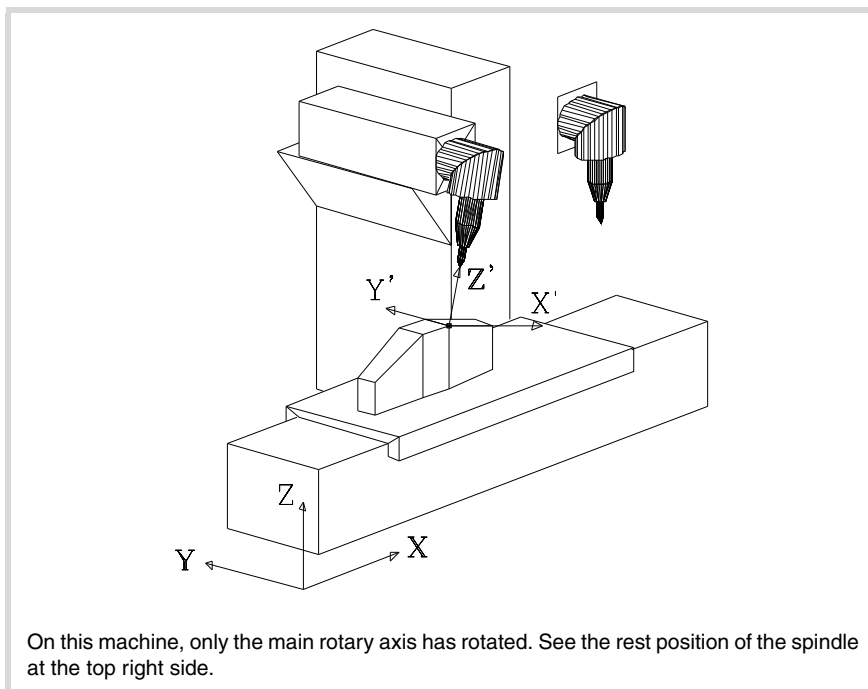
It defines a new work plane (inclined plane) perpendicular to the direction of the tool.

V1, V2, V3

Define the coordinate origin of the inclined plane with respect to the current part zero.



The new work plane assumes the orientation of the tool's coordinate system.



On this machine, only the main rotary axis has rotated. See the rest position of the spindle at the top right side.

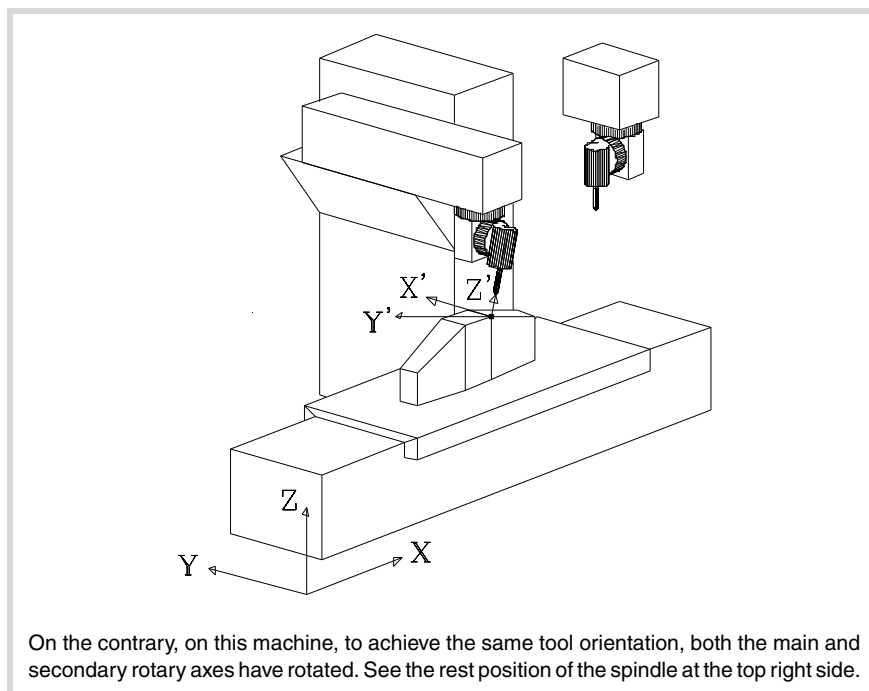
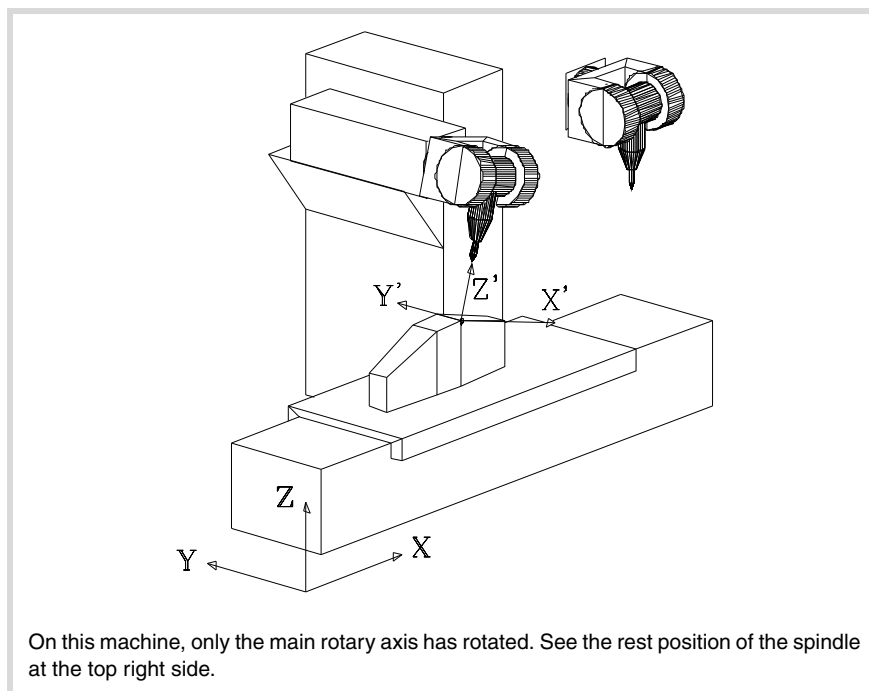
17.

COORDINATE TRANSFORMATION
Coordinate systems (#CS) (#ACS)



CNC 8060
CNC 8065

(REF. 1402)



The main axis has rotated 90° and, therefore, the X' Y' axes of the plane are rotated 90°.

φ1

Permits defining and applying a coordinate rotation in the new cartesian plane X' Y'.

If on the last machine, we wanted to orient the X', Y' axes like in the other two cases, we would have to program the following:

```
#CS DEF [n] [MODE 6, V1, V2, V3, -90]
```

**<FIRST/
SECOND>**

When defining an inclined plane perpendicular to the tool, the third axis of the plane is completely defined with the tool orientation. On the other hand, the location of the first and second axes of the new plane depends on the type of spindle; being 45° spindles, it is hard to predict.

17.

COORDINATE TRANSFORMATION
Coordinate systems (#CS) (#ACS)



CNC 8060
CNC 8065

(REF. 1402)

Depending on the programmed option, it behaves as follow.

- When programming the <FIRST> command, the projection of the new first axis of the inclined plane is oriented with the first axis of the machine.
- When programming the <SECOND> command, the projection of the new second axis of the inclined plane is oriented with the second axis of the machine.
- If none of them is programmed, it is impossible to determine the orientation of the axes in advance, it will depend on the type of spindle.

17.

COORDINATE TRANSFORMATION Coordinate systems (#CS) (#ACS)



CNC 8060
CNC 8065

(REF. 1402)

17.3.7 Operation with 45° spindles (Huron type)

Huron type spindles have two solutions for orienting the tool perpendicular to the new work plane.

- The first solution is the one that involves the smallest movement of the main rotary axis (the articulation closest to the ram or furthest away from the tool) referred to the zero position.
- The second solution is the one that involves the largest movement of the main rotary axis referred to the zero position.

The selected solution will be applied to calculate the offsets of the spindle and for the instruction `#TOOL ORI`, placing the tool perpendicular to the work plane. See "[17.5 Tool perpendicular to the plane \(#TOOL ORI\)](#)" on page 288.

Selecting one of the solutions to orient the spindle

When defining a new coordinate system, it is possible to define which of the two solutions to be applied. For this type of spindles, programming the `<sol2>` command applies the second solution; otherwise, it applies the first solution.

```
#CS DEF [n] [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>, <SOL2>]
#CS ON [n] [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>, <SOL2>]
#CS ON [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>, <SOL2>]
#CS NEW [n] [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>, <SOL2>]
#CS NEW [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>, <SOL2>]
```

```
#ACS DEF [n] [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>, <SOL2>]
#ACS ON [n] [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>, <SOL2>]
#ACS ON [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>, <SOL2>]
#ACS NEW [n] [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>, <SOL2>]
#ACS NEW [MODE m, V1, V2, V3, φ1, φ2, φ3, <0/1>, <SOL2>]
```

Consulting the position for each axis to occupy

The position for each rotary axis to occupy may be consulted in the following variables.

- For the first solution.

```
(V.)G.TOOLORIF1      Main rotary axis position.
(V.)G.TOOLORIS1      Secondary rotary axis position.
```

- For the second solution.

```
(V.)G.TOOLORIF2      Main rotary axis position.
(V.)G.TOOLORIS2      Secondary rotary axis position.
```

The CNC updates these variables every time a new plane is selected using the instructions `#CS` or `#ACS`.

17.

COORDINATE TRANSFORMATION
Coordinate systems (#CS) (#ACS)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

17.4 How to combine several coordinate systems

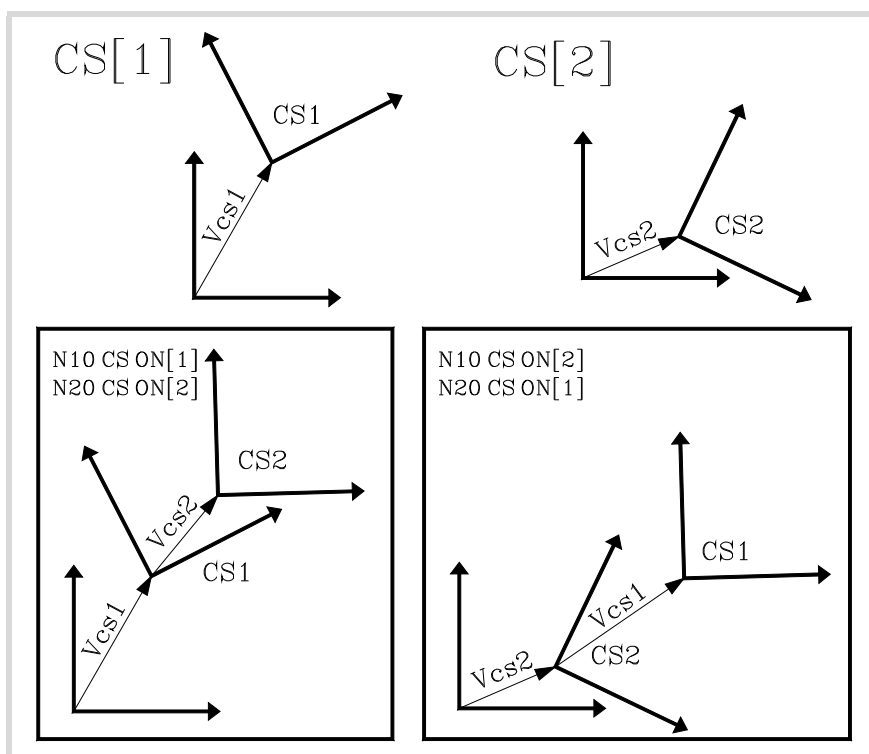
Several #ACS and #CS coordinate systems may be combined to construct new coordinate systems.

For example, the #ACS inclination generated by a fixture on the part may be combined with the #CS coordinate system that defines the inclined plane of the part to be machined.

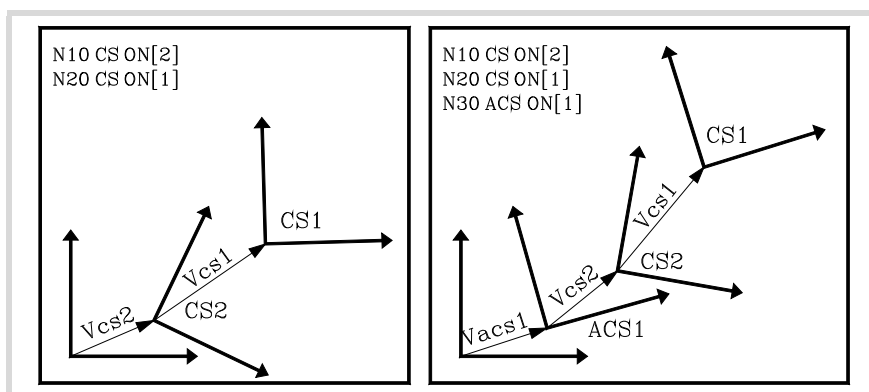
Up to ten #ACS or #CS coordinate systems may be combined. The CNC acts as follows:

- 1 First, it checks the #ACS and applies them sequentially in the programmed order, resulting in an #ACS transformation.
- 2 Then, it checks the #CS and applies them sequentially in the programmed order, resulting in a #CS transformation.
- 3 And last, it applies the resulting #CS over the resulting #ACS to obtain the new coordinate system.

The result of the combination depends on the order they are activated as may be observed in the figure below.



Every time a #ACS or #CS is activated, the resulting coordinate system is recalculated as can be observed in the figure below.



17.

COORDINATE TRANSFORMATION
How to combine several coordinate systems



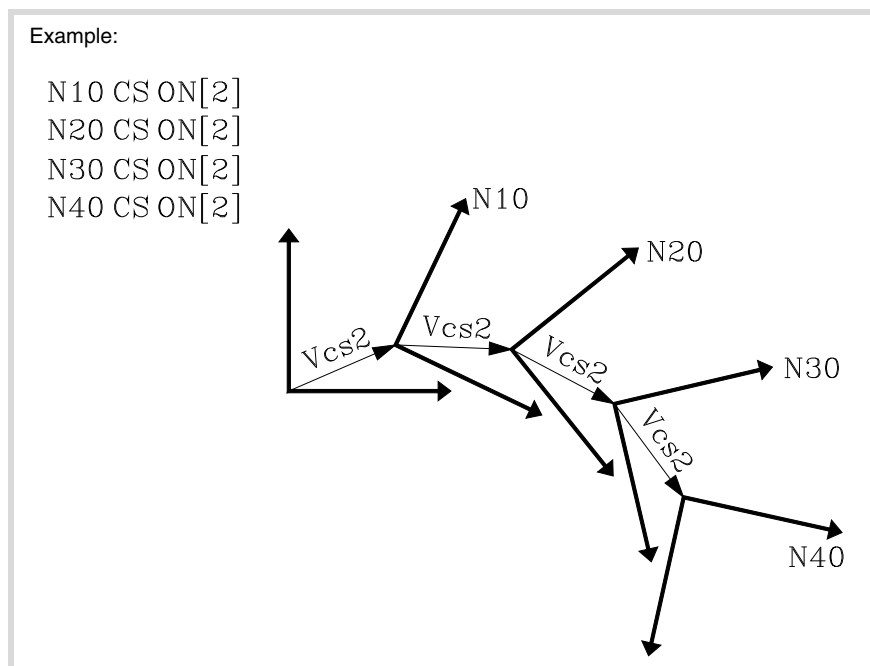
CNC 8060
CNC 8065

(REF. 1402)

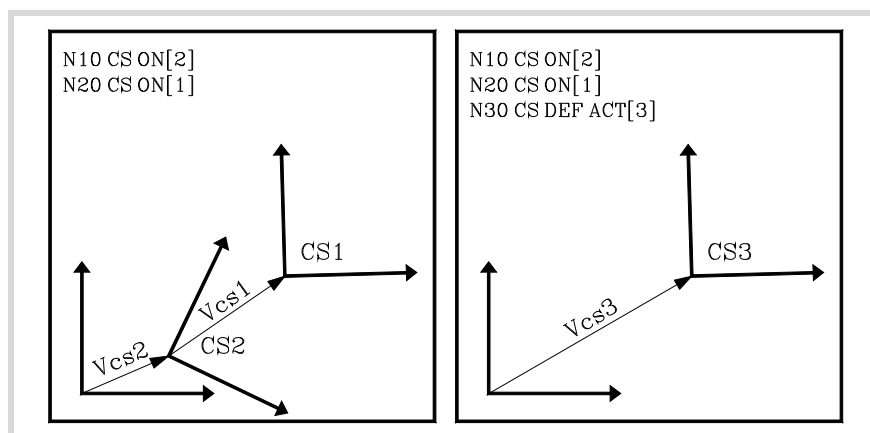
The #ACS OFF and #CS OFF instructions deactivate the last #ACS or #CS activated, respectively.

N100 #CS ON [1]	(CS[1])
N110 #ACS ON [2]	(ACS[2] + CS[1])
N120 #ACS ON [1]	(ACS[2] + ACS[1] + CS[1])
N130 #CS ON [2]	(ACS[2] + ACS[1] + CS[1] + CS[2])
N140 #ACS OFF	(ACS[2] + CS[1] + CS[2])
N140 #CS OFF	(ACS[2] + CS[1])
N150 #CS ON [3]	(ACS[2] + CS[1] + CS[3])
N160 #ACS OFF ALL	(CS[1] + CS[3])
N170 #CS OFF ALL	
M30	

A #ACS or #CS coordinate system may be activated several time.



The figure below shows an example of the instruction #CS DEF ACT [n] to assume and store the current coordinate system as a #CS.



17.

COORDINATE TRANSFORMATION
How to combine several coordinate systems



CNC 8060
CNC 8065

(REF. 1402)

17.5 Tool perpendicular to the plane (#TOOL ORI)

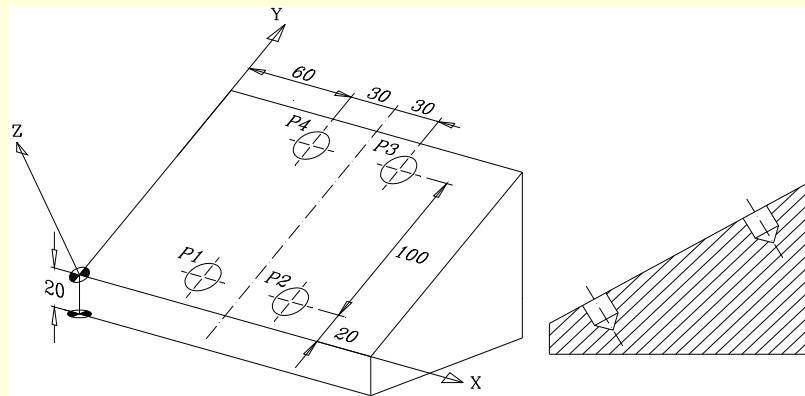
The #TOOL ORI instruction is used to position the tool perpendicular to the work plane.

After executing the #TOOL ORI instruction, the tool is positioned perpendicular to the plane, parallel to the third axis of the active coordinate system at the first motion programmed next.

17.

COORDINATE TRANSFORMATION

Tool perpendicular to the plane (#TOOL ORI)



```
#CS ON [1] [MODE 1, 0, 0, 20, 30, 0, 0]
```

(Defines the inclined plane)

```
#TOOL ORI
```

(Perpendicular tool, request)

```
G90 G0 X60 Y20 Z3
```

(Position at point P1)

(The spindle orients perpendicular to the plane during this positioning move)

```
G1 G91 Z-13 F1000
```

(Drilling)

```
G0 Z13
```

(Withdrawal)

```
G0 G90 X120 Y20
```

(Position at point P2)

```
G1 G91 Z-13 F1000
```

(Drilling)

```
G0 Z13
```

(Withdrawal)

```
G0 G90 X120 Y120
```

(Position at point P3)

```
G1 G91 Z-13 F1000
```

(Drilling)

```
G0 Z13
```

(Withdrawal)

```
G0 G90 X60 Y120
```

(Position at point P4)

```
G1 G91 Z-13 F1000
```

(Drilling)

```
G0 Z13
```

(Withdrawal)

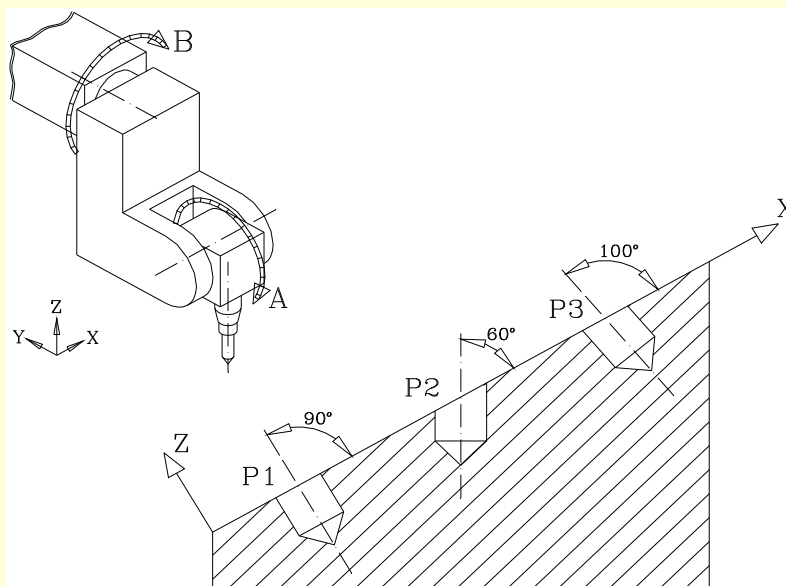
```
M30
```

The following example shows how to drill three holes with different inclination in the same plane:



CNC 8060
CNC 8065

(REF. 1402)



```

#CS ON [1] [MODE .....]      (Defines the inclined plane)
#TOOL ORI                      (Perpendicular tool, request)
G0 <P1>                        (Movement to point P1)
    (The spindle orients perpendicular to the plane during this positioning move)
G1 G91 Z-10 F1000              (Drilling)
G0 Z10                          (Withdrawal)
G0 <P2>                        (Movement to point P2)
G90 B0                          (Orients the tool with machine coordinates)
#MCS ON                         (Programming in machine coordinates)
G1 G91 Z-10 F1000              (Drilling)
G0 Z10                          (Withdrawal)
#MCS OFF                        (End of programming in machine coordinates. It recovers plane
                                coordinates)

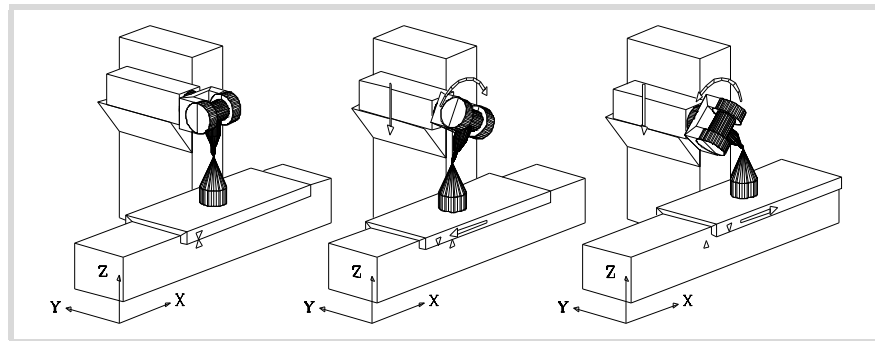
G0 <P3>                        (Movement to point P3)
G90 B-100                       (Positions the tool at 100°)
#CS OFF
#CS ON [2] [MODE6 .....]      (Defines the inclined plane perpendicular to the tool)
G1 G91 Z-10 F1000              (Drilling)
G0 Z30                          (Withdrawal)
#CS OFF
M30
    
```

17.

COORDINATE TRANSFORMATION
Tool perpendicular to the plane (#TOOL ORI)

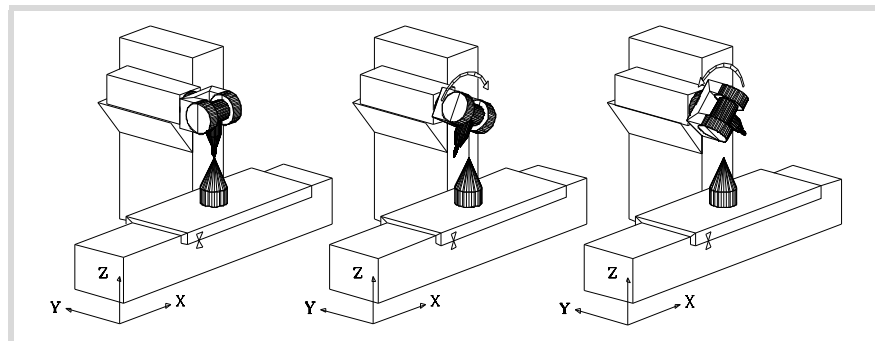
17.6 Using RTCP (Rotating Tool Center Point)

The orientation of the tool may be changed without modifying the position occupied by its tip on the part. The RTCP represents a length compensation in space.



Obviously, the CNC must move several axes in order to maintain the tool tip position at all times.

The figure below shows what happens when turning the spindle when NOT working with RTCP.

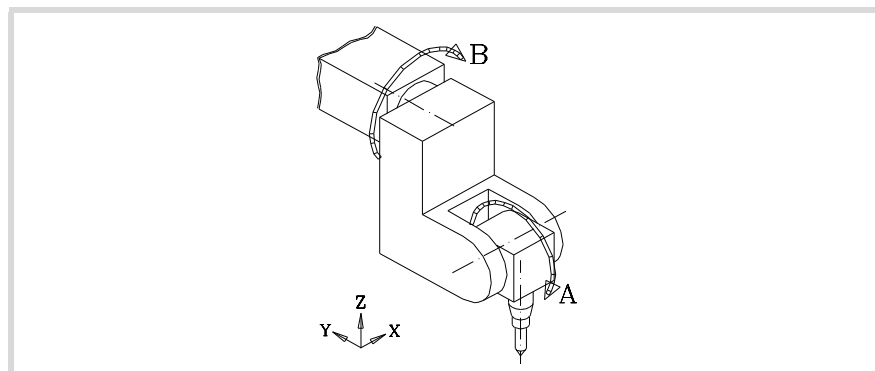


Use the following instructions for working with RTCP transformation:

- #RTCP ON Activate RTCP transformation
- #RTCP OFF Cancel RTCP transformation

Once RTCP transformation is active, spindle positioning may be combined with linear and circular interpolations. The RTCP function cannot be selected while the TLC function is active.

The following examples use a double swivel rectangular spindle head:



17.

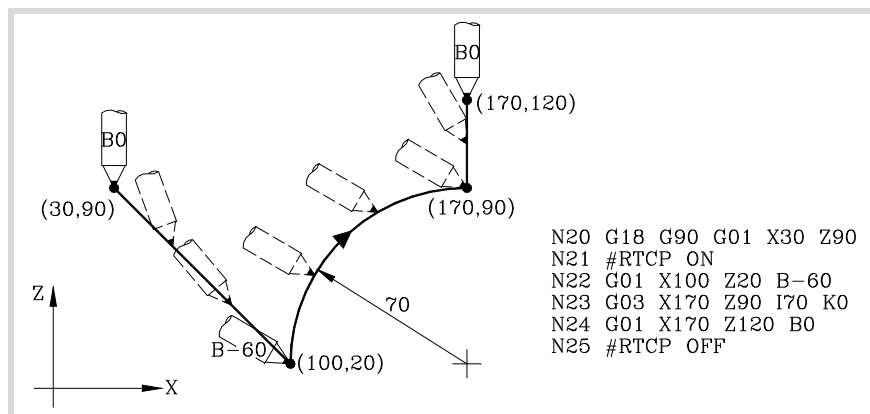
COORDINATE TRANSFORMATION
Using RTCP (Rotating Tool Center Point)



CNC 8060
CNC 8065

(REF. 1402)

Example ·1· Circular interpolation maintaining tool orientation



Block N20 selects the ZX plane (G18) and positions the tool at the starting point (30,90).

Block N21 turns RTCP on.

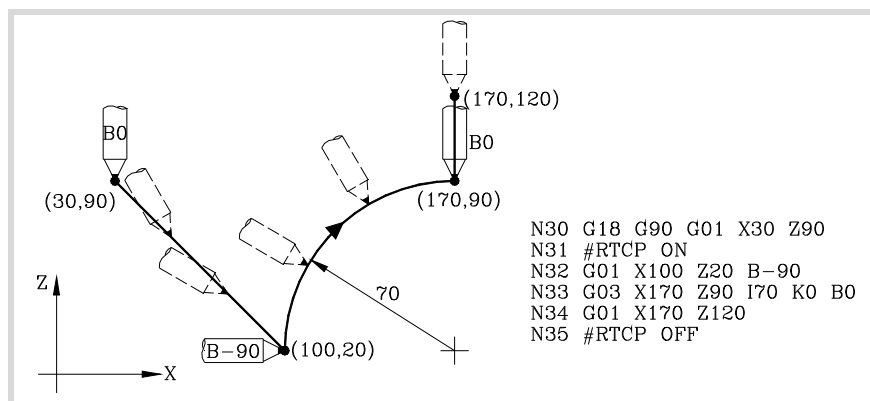
Block N22 contains a movement to point (100,20) and a tool orientation from 0° to -60°. The CNC interpolates the X, Z and B axes in such a way that the tool is being oriented along the movement.

Block N23 makes a circular interpolation to point (170,90) maintaining the same tool orientation along the whole path.

Block N24 contains a movement to point (170,120) and a tool orientation from -60° to 0°. The CNC interpolates the X, Z and B axes in such a way that the tool is being oriented along the movement.

Block N25 turns RTCP off.

Example ·2· Circular interpolation with tool perpendicular to its path



Block N30 selects the ZX plane (G18) and positions the tool at the starting point (30,90).

Block N31 turns RTCP on.

Block N32 contains a movement to point (100,20) and a tool orientation from 0° to -90°. The CNC interpolates the X, Z and B axes in such a way that the tool is being oriented along the movement.

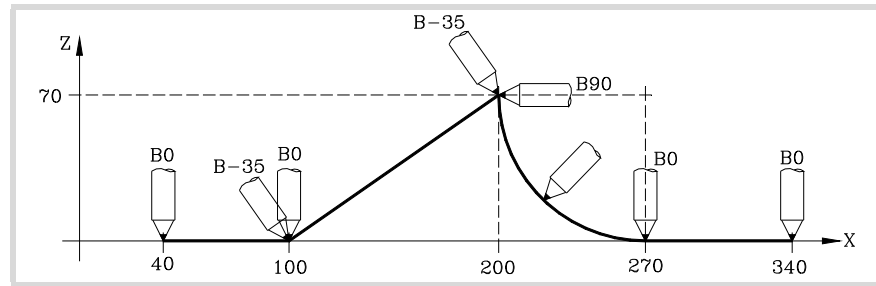
Block N33 contains a circular interpolation to point (170,90) maintaining the tool perpendicular to the path at all times.

At the starting point, it is oriented to -90° and at the endpoint, it must end be 0°. The CNC interpolates the X, Z and B axes maintaining the tool perpendicular to its path at all times.

Block N34 moves the tool to point (170,120) maintaining the orientation of 0°.

Block N35 cancels RTCP.

Example ·3· Machining a profile



G18 G90	Selects the ZX plane (G18)
#RTCP ON	It activates RTCP transformation
G01 X40 Z0 B0 F1000	Positions the tool at (40,0) oriented to (0°)
X100	Movement to (100,0) with tool oriented to (0°)
B-35	Orients the tool to (-35°)
X200 Z70	Movement to (200,70) with tool oriented to (-35°)
B90	Orients the tool to (90°)
G02 X270 Z0 R70 B0	Circular interpolation to (270,0) maintaining the tool perpendicular to the path.
G01 X340	Movement to (340,0) with tool oriented to (0°)
#RTCP OFF	It cancels RTCP transformation

17.

COORDINATE TRANSFORMATION
Using RTCP (Rotating Tool Center Point)



CNC 8060
CNC 8065

(REF. 1402)

17.6.1 Considerations about the RTCP function

In order to work with RTCP transformation, the X, Y, Z axes must be defined, they must form a trihedron and be linear. X, Y and Z may be GANTRY axes.

The RTCP transformation is kept active even after executing M02 or M30, after an Emergency or a Reset and after turning the CNC off.

While RTCP is on, the following operations are possible:

- Zero offsets G54-G59, G159.
- Presetting (G92).
- Movements in continuous / incremental jog and handwheel.

If the RTCP transformation is active, home search (G74) is only allowed on the axes not involved in RTCP .

When working with inclined planes and RTCP transformation, it is recommended to follow this programming order (sequence):

#RTCP ON	(Turn RTCP on)
#CS ON	(Define the inclined plane)
#TOOL ORI	(Tool perpendicular to the plane)
G	(Start machining)
	(End machining)
#CS OFF	(Cancel the inclined plane)
#RTCP OFF	(Turn RTCP off)
M30	(End of part-program)

RTCP should be turned on first because it allows orienting the tool without modifying the tool tip position.

17.

COORDINATE TRANSFORMATION
Using RTCP (Rotating Tool Center Point)

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

17.7 Tool length compensation (#TLC)

It must be used with CAD-CAM generated programs and the tool to be used does not have the same dimensions.

The #TLC compensates for the length difference; but it does not compensate for the radius difference.

CAD-CAM programs take the tool length into consideration and generate the coordinates for the tool base.

When using the #TLC function (Tool Length Compensation) the CNC compensates the length difference between both tools, the actual (real) one and the theoretical (calculated) one.

To work with tool length compensation (#TLC), use the following instructions:

#TLC ON [n] Turn TLC on.
 n: Tool length difference (real - theoretical).

#TLC OFF Turn TLC off.

The TLC function cannot be selected while the RTCP function is active.

N10 #TLC ON [1.5]	(Turn TLC on with a tool that is 1.5mm. longer)
N100 #TLC OFF	(Turn TLC off)
N200 #TLC ON [-2]	(Turn TLC on with a tool that is 2mm. shorter)
N300 #TLC OFF	(Turn TLC off)
N200 M30	

17.

COORDINATE TRANSFORMATION
Tool length compensation (#TLC)



CNC 8060
CNC 8065

(REF. 1402)

17.8 Kinematics related variables

These variables indicate the position occupied by the rotary axes of the spindle head and the one (target) they must occupy in order to position the tool perpendicular to the defined plane.

They are very useful when the spindle is not fully motorized (mono-rotary or manual spindles).

Variables that indicate the position of the rotary axes. They can be read and written (R/W) and are given in degrees.

(V.)G.POSROTF	Main rotary axis position.
(V.)G.POSROTS	Secondary rotary axis position.

Variables that indicate the position the rotary axes must occupy in order for the tool to be perpendicular to the define work plane. They are read-only (R) and are given in degrees. Here are the two possible solutions for swivel spindles:

The one involving the shortest movement of the main rotary axis with respect to the zero position.

(V.)G.TOOLORIF1	Position of the main rotary axis in order to position perpendicular to the inclined plane.
(V.)G.TOOLORIS1	Position of the secondary rotary axis in order to position perpendicular to the inclined plane.

The one involving the longest movement of the main rotary axis with respect to the zero position.

(V.)G.TOOLORIF2	Position of the main rotary axis in order to position perpendicular to the inclined plane.
(V.)G.TOOLORIS2	Position of the secondary rotary axis in order to position perpendicular to the inclined plane.

The CNC updates the (V.)G.TOOLORI* variables every time a new plane is selected using the instructions #CS or #ACS.

17.

COORDINATE TRANSFORMATION
Kinematics related variables

FAGOR 

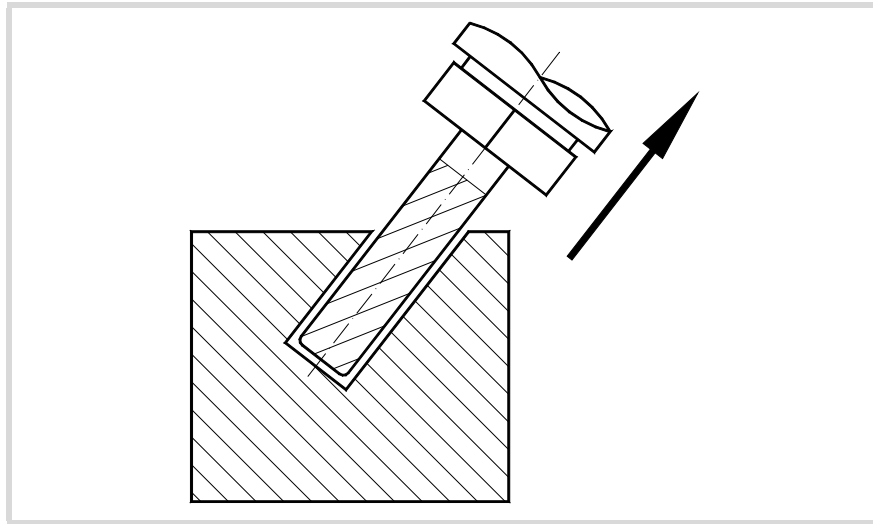
CNC 8060
CNC 8065

(REF. 1402)

17.9 How to withdraw the tool when losing the plane

If the CNC is turned off and back on while working with kinematics, the work plane that was selected gets lost.

If the tool is inside the part, proceed as follows to withdraw it:



Use the #KIN ID [n] instruction to select the kinematics that was being used.

Use the coordinate system definition MODE6 so the CNC selects a plane perpendicular to the direction of the tool as the work plane.

```
#CS ON [n] [MODE 6, 0, 0, 0, 0]
```

Move the tool along the longitudinal axis until it is away from the part.

This movement may be made in jog mode or by program, for example, G0 G91 Z20.

17.

COORDINATE TRANSFORMATION

How to withdraw the tool when losing the plane



CNC 8060
CNC 8065

(REF. 1402)

Nowadays, lots of parts are designed using CAD-CAM systems. This type of information is later post-processed to generate a CNC program, usually made up of a large number of very short blocks of several mm or just a few tenths of a micron.

In this type of parts, the CNC must be capable to analyze a large number of points in advance so it can generate a continuous path that goes through (or near) the points of the program while keeping (the best way possible) the programmed feedrate and the restrictions of maximum acceleration, jerk, etc of each axis and of the path.

The command to execute programs made up of lots of small blocks, typical of high speed machining, is carried out with a single instruction #HSC. This function offers several ways to work; optimizing the contouring error or the machining speed.

18.1 HSC mode. Optimization of surface error.

It is the recommended way (mode) to work. The maximum chordal error permitted is a parameter of this mode. This mode optimizes the velocity profile through intelligent algorithms and error compensation by means of filters.

This mode offers excellent results in time and surface quality solving jerk problems that may appear depending on the profile to be machined. This is the best mode for roughing and semi-finishing operations besides finishing operations where surface quality is most important.

18.

HSC. HIGH SPEED MACHINING
HSC mode. Optimization of surface error.

Activating the HSC mode.

The CNC does not allow activating an HSC mode if there is one active already. Before activating an HSC mode, cancel the previous one. See "[18.4 Canceling the HSC mode.](#)" on page 303.

This mode is turned on with the instruction #HSC and the SURFACE command. The chordal can also be programmed as an option (parameter E).

```
#HSC ON [SURFACE <,E {error} >]
```

E {error} Optional. Maximum chordal error allowed (millimeters or inches).

The default machining mode is set in machine parameter HSCDEFAULTMODE. If the default mode is SURFACE, this command may be left out in the #HSC instruction.

The parameter of this instruction is the maximum contour error permitted between the programmed path and the resulting path. Programming it is optional; if not defined, it assumes as maximum contouring error the value set in machine parameter MAXROUND.

```
#HSC ON [SURFACE]
#HSC ON [SURFACE, E0.1]
```



CNC 8060
CNC 8065

(REF. 1402)

18.2 HSC mode. Optimizing the contouring error.

The maximum contour error permitted is a parameter of this mode. From this instruction on, the CNC modifies the geometry through intelligent algorithms for eliminating unnecessary points and automatically generating splines and polynomial transitions between blocks. This way, the contour is traveled at a variable feedrate according to the curvature and the parameters (programmed acceleration and feedrate) but without going beyond the set error limits.

Activating the HSC mode optimizing the chordal error.

The CNC does not allow activating an HSC mode if there is one active already. Before activating an HSC mode, cancel the previous one. See ["18.4 Canceling the HSC mode."](#) on page 303.

This mode is turned on with the instruction #HSC and the command CONTERROR. Optionally, the CORNER command may be programmed.

```
#HSC ON [<CONTERROR {error}><,CORNER {angle}>]
```

CONTERROR {error} Optional. Maximum contouring error allowed.

CORNER {angle} Optional. Maximum angle between two paths (between 0° and 180°), under which the machining operation is carried out in square corner mode.

The default machining mode is set in machine parameter HSCDEFAULTMODE. If the default mode is CONTERROR, this command may be left out in the #HSC instruction.

The parameter of this instruction is the maximum contour error permitted between the programmed path and the resulting path. Programming it is optional; if not defined, it assumes as maximum contouring error the value set in machine parameter MAXROUND.

```
#HSC ON [CONTERROR 0.01]
#HSC ON [CONTERROR 0.01, CORNER 150]
#HSC ON [CORNER 150]
```

Programming the CORNER command is optional; if not programmed, it assumes the percentage set in machine parameter CORNER.

Recommendations for machining. Selecting the chordal error at the CNC and at the post-processed CAM.

As mentioned earlier, the error caused by the CNC between the programmed part and the resulting part is never greater than the programmed value. On the other hand, the CAM system also generates an error when processing the original part and converting the paths into a CNC program. The resulting error may be the sum of the two; therefore, the desired maximum error must be spread between both processes.

Selecting a large chordal error when generating the program and a small chordal error when executing it, the execution is slower and of lower quality. In this case ridges will appear because the CNC perfectly follows the CAM generated polyhedron. It is recommended to post-process at the CAM with a smaller error than the desired one for high speed cutting HSC (between 10% and 20%). For example, for a maximum error of 50 microns, we should post-process with an error of 5 or 10 microns and program the 50 microns in the HSC command (#HSC ON [CONTERROR 0.050]). This way of programming lets CNC modify the profile while respecting the dynamics of each axis without causing undesired effects like ridges. If CAM post-processing is done with the desired error and a very small error is programmed in HSC CONTERROR, as a result, the CNC will follow faithfully the facets generated by the CAM.

18.

HSC. HIGH SPEED MACHINING
HSC mode. Optimizing the contouring error.

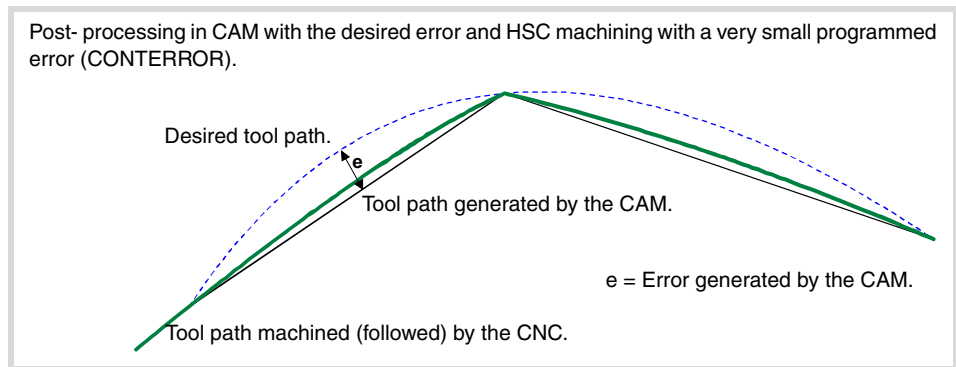
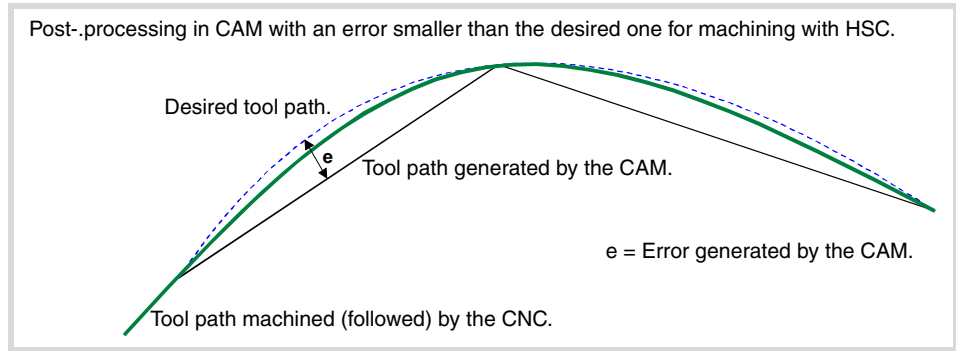
FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

18.

HSC. HIGH SPEED MACHINING
HSC mode. Optimizing the contouring error.



Recommendations for machining. The part-program.

Since the CNC works with an accuracy of nanometers, better results may be obtained if the coordinate have 4 or 5 decimals than if they only have 2 or 3. This has no negative effect because the block processing time does not change noticeably. The slight increase in the size of the programs does not represent a problem for storing them thanks to the high capacity hard disk or for transmitting them thanks to Ethernet.

18.3 HSC mode. Optimizing the machining speed.

In spite of the recommendations for generating CAM programs, it is possible to have programs already generated that do not show a continuity between the error generated by the CAM, the block size and the error required by the HSC. For this type of programs, the HSC mode has a fast mode where the CNC generates paths trying to restore this continuity and thus be able to work on a smoother surface and obtain a more continuous speed. It is also the most appropriate mode for programs where the feedrate has been programmed in each block.

This mode is recommended for machines whose dynamics does not respond in a wide frequency range; i.e. the ones that could show some resonance or have a limited bandwidth. It is also recommended for 5-axis machines due to dynamic limitations of rotary axes.

Activating the HSC mode optimizing the machining speed.

The CNC does not allow activating an HSC mode if there is one active already. Before activating an HSC mode, cancel the previous one. See ["18.4 Canceling the HSC mode."](#) on page 303.

This mode is turned on with the instruction #HSC and the command FAST. Optionally, it will be possible to program the desired feedrate percentage (parameter FAST), the chordal error (parameter E) and the maximum angle to machine in "sharp corner" (parameter CORNER).

```
#HSC ON [FAST <{%feed}>><,E {error}>><,CORNER {angle}>>]
```

{%feed}	Optional. Percentage of the desired speed (between 0.01% and 100%).
E {error}	Optional. Maximum chordal error allowed (millimeters or inches).
CORNER {angle}	Optional. Maximum angle between two paths (between 0° and 180°), under which the machining operation is carried out in square corner mode.

The default machining mode is set in machine parameter HSCDEFAULTMODE. If FAST is the default mode, this command may be left out in the #HSC instruction.

A parameter of this instruction is the percentage of machining speed to be reached, of the maximum that the CNC can reach. Programming it is optional; if not defined, it assumes the percentage set in machine parameter FASTFACTOR. The value of the FAST parameter may be programmed with a value lower than 100% when running machining tests and the speed reached is considered to be too high.

```
#HSC ON [FAST]
  (FAST factor= machine parameter FASTFACTOR)
  (Chordal error = machine parameter ROUNDPAR)
  (Angle = machine parameter CORNER)
```

```
#HSC ON [FAST, E 0.05]
  (FAST factor= machine parameter FASTFACTOR)
  (Chordal error = 0.05)
  (Angle = machine parameter CORNER)
```

```
#HSC ON [FAST 93.5, E 0.05]
  (FAST factor = 93.5%)
  (Chordal error = 0.05)
  (Angle = machine parameter CORNER)
```

```
#HSC ON [FAST 93.5, E 0.01, CORNER 130]
  (FAST factor = 93.5%)
  (Chordal error = 0.01)
  (Angle = 130°)
```

```
#HSC ON [FAST, CORNER 130]
  (FAST factor= machine parameter FASTFACTOR)
  (Chordal error = machine parameter ROUNDPAR)
  (Angle = 130°)
```

18.

HSC. HIGH SPEED MACHINING
HSC mode. Optimizing the machining speed.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

Programming the CORNER command is optional; if not programmed, it assumes the angle set in machine parameter CORNER.

Programming the chordal error improves the accuracy in curved segments or circles; however, and considering the peculiarities of executing in FAST mode, the contour error at the corners is not ensured. Programming the E command is optional; if not programmed, it assumes the error set in machine parameter ROUNDPAR.

18.

HSC. HIGH SPEED MACHINING

HSC mode. Optimizing the machining speed.



CNC 8060
CNC 8065

(REF. 1402)

18.4 Canceling the HSC mode.

The HSC mode is canceled with the instruction #HSC.

```
#HSC OFF
```

```
#HSC OFF
```

HSC is also canceled when programming any of the functions G05, G07 or G50. Functions G60 and G61 do not cancel the HSC mode. Activating a second HSC mode does not cancel the previous HSC mode.

Influence of the reset, turning the CNC off and of the M30.

The HSC mode is canceled on power-up, after executing an M02 or M30 and after an emergency or reset.

18.

HSC. HIGH SPEED MACHINING
Canceling the HSC mode.

FAGOR 

**CNC 8060
CNC 8065**

(REF. 1402)

18.

HSC. HIGH SPEED MACHINING

Canceling the HSC mode.



CNC 8060
CNC 8065

(REF. 1402)

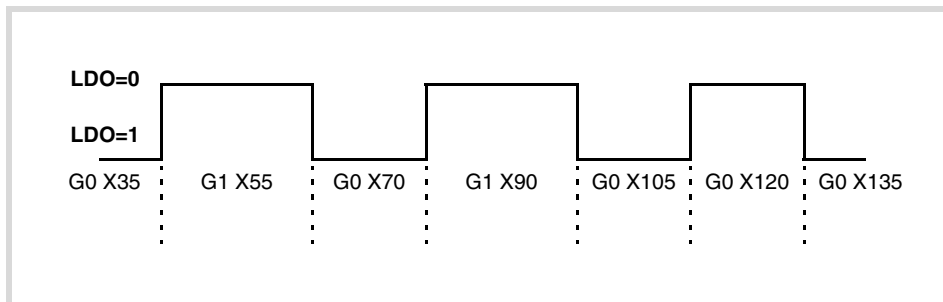
19.1 Synchronized switching.

Synchronized switching refers to the process to control a local digital output of the CNC depending on the type of movement programmed on the axes. This feature is valid for all kinds of corner G7, G60, G5, G61, G50 or HSC that is programmed.

- The selected digital output is activated when switching from G0 to G1/G2/G3.
- The selected digital output is canceled when switching from G1/G2/G3 to G0.

This way, since it is possible to make the signal status depend on the type of tool path, this feature may be used for various applications such as laser cutting systems or others that require a digital output signal synchronized with the type of tool path.

The following diagram shows the state of the local digital output (LDO) depending on programmed G0-to-G1 transitions or vice versa. The digital output associated with synchronized switching is defined in machine parameter SWTOUTPUT.



19.1.1 Activate synchronized switching.

The instruction #SWTOUT may be used to activate synchronized switching. After executing this instruction, a G0-to-G1/G2/G3 transition activates the associated digital output and a G1/G2/G3-to-G0 transition cancels it. Once the digital output is activated, it stays active until a transition to G0 occurs or M30, reset is executed or synchronized switching is canceled (#SWTOUT OFF) PLC signals STOP and _FEEDHOL also cancel the digital output; when these signals disappear, if the digital output was active before, it activates it again.

19.

LASER.
Synchronized switching.

Programming.

When defining this instruction, it is optionally possible to define an offset (in time or distance) to anticipate or delay the activation or cancellation of the digital output.

Programming format.

The programming format is the following; the list of arguments appears between curly brackets and the optional ones between angle brackets.

```
#SWTOUT ON [<TON={ time }>,<TOF={ time }>,<PON={ long }>,<POF={ long }>]
```

TON={ time }	Optional. Time shift (milliseconds) to anticipate the activation of the digital output.
TOF={ time }	Optional. Time shift (milliseconds) to anticipate the deactivation of the digital output.
PON={ long }	Optional. Distance shift (offset, mm(inches) to anticipate the activation of the digital output.
POF={ long }	Optional. Distance shift (offset, mm(inches) to anticipate the deactivation of the digital output.

```
#SWTOUT ON
#SWTOUT ON [TON=50 TOF=40]
#SWTOUT ON [TON50 TOF40]
#SWTOUT ON [PON=0.3]
```

Offsets (time or distance) to anticipate or delay the activation or deactivation of the digital output.

Parameters TON, TOF, PON and POF are optional; it is possible to program all of them, some or none of them and in any order. A positive value of these parameters anticipates the activation or deactivation of the digital output and a negative value delays it. The "=" sign may be left out.

On CNC power-up, the offset wears are set to zero. After power-up, the values programmed for the offsets (using the instruction #SWTOUT or the variables) are maintained even after an error, a reset or an M30.

When it is necessary to anticipate or delay the activation or deactivation of the digital output.

Depending on the feedrate, the cycle time, the type of servo system used, etc. it might be necessary to program the offset values TON, TOF, PON, POF so the switch-on/off of the output can take place at the desired point.

For example, when observing that the system used in the application is switching on too late, either set a positive TON value to anticipate the activation by that amount of time or a positive PON value to anticipate the activation by that distance. If on the other hand, the system used in the application is switching on too early, either set a negative TON value to delay the activation by that amount of time or a negative PON value to delay the activation by that distance. Proceed the same way for switch-off, but using offsets TOF and POF.



CNC 8060
CNC 8065

(REF. 1402)

19.1.2 Cancel synchronized switching.

The instruction #SWTOUT cancels synchronized switching. Synchronized switching is also canceled after a reset or an M30.

Programming.

Program the instruction alone in the block.

Programming format.

The programming format is:

```
#SWTOUT OFF
```

```
#SWTOUT OFF
```

19.

LASER.
Synchronized switching.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

19.1.3 Variables related to synchronized switching.

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W).

Variable.	PRG	Meaning.
V.G.TON	R/W	Time shift (milliseconds) to anticipate the activation of the digital output.
V.G.TOF	R/W	Time shift (milliseconds) to anticipate the deactivation of the digital output.
V.G.PON	R/W	Distance shift (offset, mm(inches) to anticipate the activation of the digital output.
V.G.POF	R/W	Distance shift (offset, mm(inches) to anticipate the deactivation of the digital output.

The value of these variables is equivalent to the one programmed in parameters TON, TOF, PON and POF of the instruction #SWTOUT. When executing the instruction without parameters and the variables have a value assigned to it, the CNC assumes these latter values as active. Likewise, when programming the parameters of the instruction, the variables assume these values as their own.

These variables interrupt block preparation. To modify the offset values without interrupting block preparation, change these values from the PLC or use the parameters of the instruction #SWTOUT. If these variables are changed from the PLC, the new values are assumed when executing the instruction #SWTOUT. If the instruction is active when changing the values from the PLC, it assumes the new values.

19.

LASER.
Synchronized switching.



CNC 8060
CNC 8065

(REF. 1402)

19.2 PWM (Pulse-Width Modulation).

Pulse width modulation, known as PWM, may be used to control a duty cycle of the Laser signal and therefore modify the power of the Laser. One of the applications of the PWM is using the Laser like a drill by applying and removing power to it very fast.



Besides the Laser, the PWM may be used in other applications such as plasma cutting.

Activating the PWM via PLC

The PWM may be activated from the PLC (PWMON mark) and from the part-program. Activating the Laser via PLC has priority over activating it from the CNC.

Laser status from the CNC. Variable (V.)G.PWMON	Laser status from the PLC. PWMON mark	Laser status.
0	0	Laser OFF.
1	0	Laser ON from the CNC.
0	1	Laser ON from the PLC.
1	1	Laser ON from the PLC.

19.

LASER.
PWM (Pulse-Width Modulation).

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

19.2.1 Activate the PWM.

The #PWMOUT ON instruction may be used to activate the PWM. This function interrupts block preparation. The PWM may be activated from any channel, but only the channel that activates it will have control over it. When trying to activate the PWM from a channel when it is already active, the CNC will issue the corresponding error message.

The modifications to the PWM both via program or via PLC are updated (refreshed) without waiting for the PWM cycle in progress to be completed and as continuously as possible with respect to the previous conditions; in other words, it does not wait for the default signal to set to zero or to one in each change.

Programming.

When defining this instruction, there is an option to set the frequency and the percentage of the duty cycle.

Programming format.

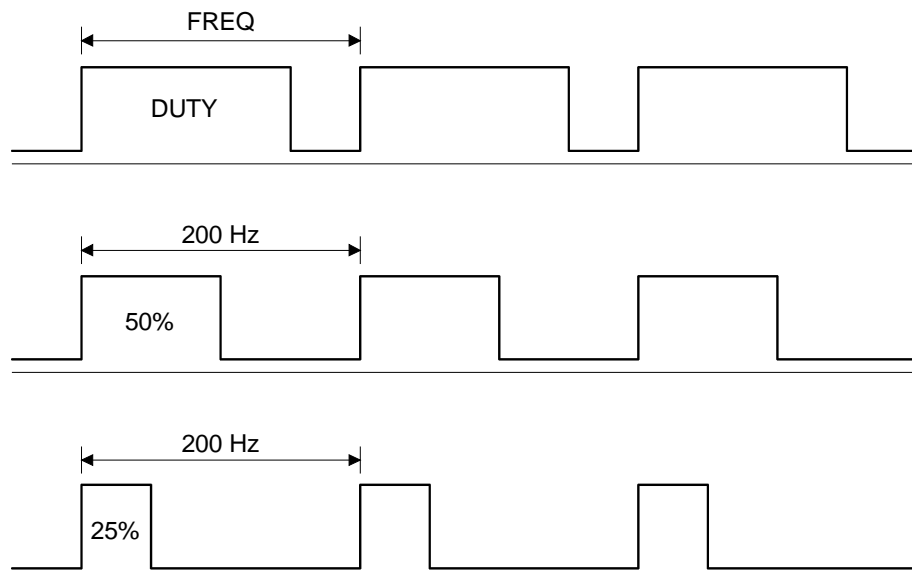
The programming format is the following; the list of arguments appears between curly brackets and the optional ones between angle brackets.

```
#PWMOUT ON [<FREQ/F{Hz}>,<DUTY/D{ %}>,<PWMBTIME{ms}>,<PWMBEND>]
```

{Hz} Optional. Frequency of the PWM (between 2 and 5000 Hz).

{%} Optional. Percentage of duty cycle (between 0.1 and 100%).

{ms} Optional. Duration of burst mode.



```
#PWMOUT ON [FREQ 200, DUTY 50]
```

```
#PWMOUT ON [F200, D50]
```

```
#PWMOUT ON [FREQ 200, DUTY 25, PWMBTIME 50, PWMBEND 1]
```

PWM frequency.

The frequency is programmed using the FREQ command (or simply F) and may be a value between 2 Hz and 5000 Hz. This command is optional; if not programmed, the CNC assumes the last programmed value or 0 if there is no previous value.

Percentage of duty cycle.

The percentage of duty cycle is programmed using the DUTY program (or simply D) and may have a value between 0.1% and 100%. This command is optional; if not programmed, the CNC assumes the last programmed value or 50 if there is no previous value.

19.

LASER.
PWM (Pulse-Width Modulation).



CNC 8060
CNC 8065

(REF. 1402)

Burst mode.

The burst mode consists in activating the PWM, but for a set time period (PWMBTIME) so when done, the PWM output will stay at a set level (PWMBEND).

The PWMBTIME command sets the time the PWM stays active (in ms rounded up to loop units). The PWMBEND command indicates the level (0/1) at which the PWM stays after the time period set by PWMBTIME; if set to "1", the PWM stays active, but if set to "0" not programmed, the PWM is turned off.

Remarks.

- None of the simulation modes activates the PWM signal.
- During tool inspection, the CNC does not deactivate the PWM. The OEM may have programmed the PLC to deactivate the PWM during tool inspection and resume the PWM at the end of the inspection.

19.2.2 Cancel the PWM.

The #PWMOUT OFF instruction may be used to deactivate the PWM. The way the PWM will behave after executing an M30 or after a reset depends on the setting of parameter PWMCANCEL. On power-up and when canceling the PWM, all the variables and marks of the CNC and PLC recover their initial values.

Programming.

Program the instruction alone in the block.

Programming format.

The programming format is:

```
#PWMOUT OFF
```

```
#PWMOUT OFF
```

19.

LASER.

PWM (Pulse-Width Modulation).



CNC 8060
CNC 8065

(REF. 1402)

19.2.3 PWM variables.

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W). Reading these variables interrupts block preparation. These variables only work when the Laser is active via CNC.

Variable.	PRG	Meaning.
(V.)G.PWMON	R	PWM status, when it has been activated via CNC. (0 = Laser OFF; 1 = Laser ON)
(V.)G.PWMFREQ	R	Frequency of the PWM (between 2 and 5000 Hz; by default, 0), when the PWM has been activated via CNC.
(V.)G.PWMDUTY	R	Work cycle of the PWM (between 0.1 and 100%; by default, 50%), when the PWM has been activated via CNC. The reading from the PLC will be given in tenths (x10); i.e. for a value ·0.1·, the PLC reading will return a value of ·1·. For a value of ·100·, the reading from the PLC will return a value of ·1000·.

Syntax of the variables.

V.G.PWMON
V.G.PWMFREQ
V.G.PWMDUTY

19.

LASER.

PWM (Pulse-Width Modulation).

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

19.

LASER.

PWM (Pulse-Width Modulation).

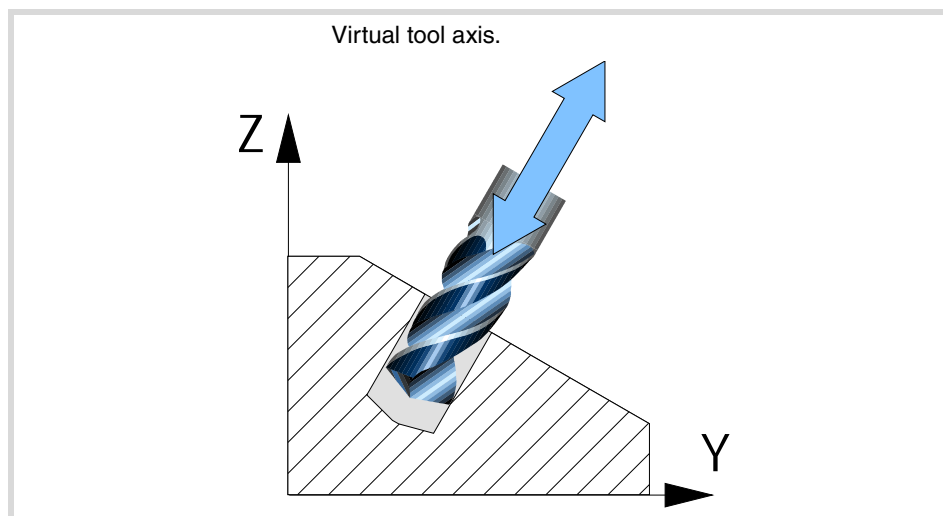


CNC 8060
CNC 8065

(REF. 1402)

A virtual axis of the tool is a fictitious axis that always moves in the direction in which the tool is oriented. This virtual tool axis facilitates the movement in the tool direction when it is not aligned with the axes of the machine, but it is oriented in any other direction depending on the position of the bi-rotary or tri-rotary spindle.

This way and depending on the applied kinematics, it will move the corresponding X Y Z axes so the tool moves along its axis. This function facilitates drilling operations, withdrawing the tool in its direction as well as increasing or decreasing the depth of the pass while machining a part.



Considerations about the virtual tool axis.

- There can be one virtual tool axis per channel.
- The virtual axis of the tool must be linear and it must belong to the channel. The virtual tool axis cannot be part of the main trihedron when it is active.
- The virtual tool axis, since it is an axis of the channel, may be moved like any other axis in the various work modes, automatic, jog, tool inspection, axis repositioning, etc.
- The virtual tool axis has travel limits that may be set by machine parameter and by program.

20.1 Activate the virtual tool axis.

The instruction #VIRTAX is used to activate the transformation of the virtual tool axis.

Programming.

When defining this instruction, it is also possible to define the position value (coordinate) where the axis is located.

Programming format.

The programming format is the following; the list of arguments appears between curly brackets and the optional ones between angle brackets.

```
#VIRTAX ON
#VIRTAX ON <[pos]>
{pos}           Optional. Axis position.
```

```
#VIRTAX
  Activate the transformation of the virtual tool axis, in its current position.
#VIRTAX ON
  Activate the transformation of the virtual tool axis, in its current position.
#VIRTAX ON [15]
  Activate the transformation of the virtual tool axis considering that it is positioned
  in position 15.
#VIRTAX [0]
  Activate the transformation of the virtual tool axis considering that it is positioned
  in position 0.
```

Programming the ON command is optional.

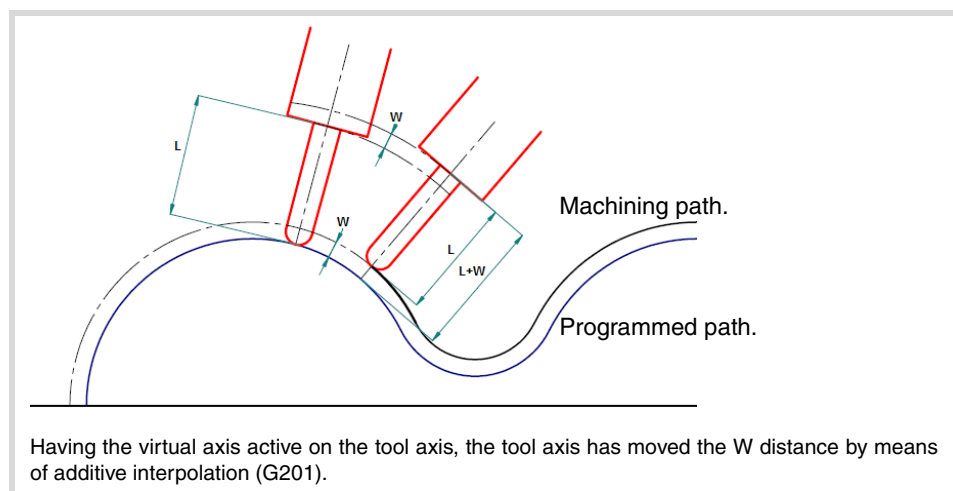
Axis position.

This parameter may be used to activate the transformation of the virtual tool axis considering that it is positioned in a specific position. If 0 is programmed, the CNC considers that the virtual tool axis is positioned in the 0 position.

If the axis position is not programmed, the CNC activates the virtual axis taking its current position into account.

Example 1. Increase or decrease the machining pass depth while machining

Functions #VIRTAX and G201 are active in the program being executed. In this case, the virtual tool axis may be moved while the program is running.



20.

VIRTUAL TOOL AXIS.
Activate the virtual tool axis.



CNC 8060
CNC 8065

(REF. 1402)

Example 2. Increase or decrease the machining pass depth while machining

Functions #VIRTAX and G201 are not active in the program being executed. The steps to change the machining pass are the following.

- (1) Interrupt program execution with the [STOP] key.
- (2) Go into tool inspection mode.
- (3) From MDI mode, execute #VIRTAX[0].
- (4) Move the axis the desired distance via MDI, jog, etc.
- (5) Resume execution without repositioning the axes.

20.2 Cancel the virtual tool axis.

The instruction #VIRTAX OFF cancels the transformation of the virtual tool axis. The way the virtual tool axis will behave after executing an M30 or after a reset depends on the setting of parameter VIRTAXCANCEL.

Programming.

Program the instruction alone in the block.

Programming format.

The programming format is:

```
#VIRTAX OFF
```

```
#VIRTAX OFF
```

20.

VIRTUAL TOOL AXIS.
Cancel the virtual tool axis.

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

20.3 Variables associated with the virtual tool axis.

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W).

Variable.	R/W	Meaning.
(V.)[ch].G.VIRTAXIS	R	Logic number of the virtual tool axis.
(V.)[ch].G.VIRTAXST	R	Status of the virtual tool axis: (0) inactive / (1) active.
(V.)[ch].A.VIRTAXOF.xn	R	Distance traveled by the axis, due to the movement of the virtual tool axis.

Syntax of the variables.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

V.[2].G.VIRTAXS	Channel ·2·.
V.A.VIRTAXOF.Z	Z axis.
V.A.VIRTAXOF.4	Axis with logic number ·4·.
V.[2].A.VIRTAXOF.1	Axis with index ·1· in the channel ·2·.

20.

VIRTUAL TOOL AXIS.
 Variables associated with the virtual tool axis.



CNC 8060
CNC 8065

(REF. 1402)

There are two types of high level language commands, programming instructions and flow controlling instructions.

Programming instructions

They are defined with the "#" sign followed by the name of the instruction and its associated parameters.

They are used for various operations such as.

- Displaying errors, messages, etc.
- Programming movements referred to machine reference zero (home).
- Executing blocks and programs.
- Synchronizing channels.
- Coupling, parking and swapping axes.
- Swapping spindles,
- Activating collision detection.
- Activating manual intervention.

Flow controlling instructions

They are defined with the "\$" sign followed by the name of the instruction and its related data.

They are used to make loops and program jumps.

21.1 Programming statements

21.1.1 Display instructions. Display an error on the screen

It interrupts program execution and displays the indicated error message. It is programmed using the instruction `#ERROR`, selecting either the number of the error to be displayed or the error text.

#ERROR

Display an error by selecting its number

It displays the indicated error number and its associated text according to the CNC's error listing. If the indicated error number does not exist in the CNC's error listing, it does not display any text.

The programming format is:

```
#ERROR [<number>]
```

Parameter	Meaning
<number>	Error number.

The error number, that must be an integer, may be defined with a numerical constant, a parameter or an arithmetic expression. When using local parameters, they must be programmed as P0-P25.

```
#ERROR [100000]
#ERROR [P100]
#ERROR [P10+34]
```

OEM errors in various languages.

Errors between 10000 and 20000 are reserved for the OEM so he can create his own warning or error texts in different languages. Each `mtb\data\lang\language` folder contains the file `cncError.txt` that contains the OEM messages and errors in different languages. If an error text is not in the folder of the language active at the CNC, it looks for it in the english folder; if it is not there, the CNC will issue the relevant error message.

#ERROR

Display an error by selecting its text

It displays the indicated error text. If no text is defined, it shows an empty error window.

The programming format is:

```
#ERROR [<text>]
```

Parameter	Meaning
<number>	Error text.

The error text must be defined between quote marks. Certain special characters are defined as follows.

```
\ "      Inserts quote marks in the text.
%%      Inserts the % character.
```

```
#ERROR ["Message"]
#ERROR ["Parameter \"P100\" is wrong"]
#ERROR ["Difference between P12 and P14 > 40%%"]
```

21.

STATEMENTS AND INSTRUCTIONS
Programming statements



CNC 8060
CNC 8065

(REF. 1402)

Including external values in the error text

The identifier %D or %d may be used to insert external values (parameters or variables) into the text. The data whose value is to be displayed must be defined after the text.

```
#ERROR ["Wrong %d value", 120]  
#ERROR ["Tool %D expired", V.G.TOOL]  
#ERROR ["Wrong %D - %D values", 18, P21]
```

Up to 5 identifiers %D or %d may be defined, but there must be as many data values as identifiers.

21.1.2 Display instructions. Display a warning on the screen

The display of warnings on the screen may be programmed using the instruction #WARNINGSTOP or #WARNING depending on whether the program execution is to be interrupted or not. In either case, the CNC shows the warning during block preparation, not when it executes it.

#WARNING This instruction does not interrupt the execution of the program.
 #WARNINGSTOP This instruction interrupts program execution at the point where the instruction is located. It's up to the user to either resume the execution at this point, [START] key, or abort the program, [RESET] key.

Both instructions are programmed by selecting either the text to be displayed or the warning number according to the list of errors and warnings of the CNC.

#WARNING
Display a warning by selecting its number

#WARNINGSTOP
Display a warning by selecting its number and interrupt the execution.

It displays the indicated warning number and its associated text according to the CNC's error listing. If the indicated warning number does not exist in the CNC's error listing, it does not display any text.

The programming format is:

```
#WARNING [<number>]
#WARNINGSTOP [<number>]
```

Parameter	Meaning
<number>	Warning number.

The warning number, that must be an integer, may be defined with a numerical constant, a parameter or an arithmetic expression. When using local parameters, they must be programmed as P0-P25.

```
#WARNING [100000]
#WARNING [P100]
#WARNING [P10+34]
```

#WARNING
Display a warning by selecting its text

#WARNINGSTOP
Display a warning by selecting its text and interrupt the execution.

It displays the indicated warning text. If no text is defined, it shows an empty warning window.

The programming format is:

```
#WARNING [<text>]
#WARNINGSTOP ["<text>"]
```

Parameter	Meaning
<number>	Warning text.

The warning text must be defined between quote marks. Certain special characters are defined as follows.

```
\ " Inserts quote marks in the text.
%% Inserts the % character.
```

21.

```
#WARNING ["Message"]  
#WARNING ["Parameter \"P100\" is wrong"]  
#WARNING ["Difference between P12 and P14 > 40%%"]
```

Including external values in the error text

The identifier %D or %d may be used to insert external values (parameters or variables) into the text. The data whose value is to be displayed must be defined after the text.

```
#WARNING ["Wrong %d value",120]  
#WARNING ["Tool %D expired",V.G.TOOL]  
#WARNING ["Wrong %D - %D values",18,P21]
```

Up to 5 identifiers %D or %d may be defined, but there must be as many data values as identifiers.

21.1.3 Display instructions. Display a message on the screen

The indicated message appears at the top of the screen and it does not interrupt the execution of the program. The message will stay active until a new message is activated, another program is executed or a reset is carried out.

The text to be displayed is programmed using the #MSG instruction

#MSG Display a message

The programming format is:

```
#MSG ["<text>"]
```

Parameter	Meaning
<text>	Message text.

The text of the message must be defined between quote marks. Certain special characters are defined as follows.

\ " Inserts quote marks in the text.

%% Inserts the % character.

If no text is defined, the message is erased from the screen.

```
#MSG ["User message"]
#MSG ["The tool \"T1\" is a finishing tool"]
#MSG ["80%% of feedrate is being used"]
#MSG [""]
```

Including external values in the error text

The identifier %D or %d may be used to insert external values (parameters or variables) into the message. The data whose value is to be displayed must be defined after the text.

```
#MSG ["Part number %D", P2]
#MSG ["The current tool is %D", V.G.TOOL]
#MSG ["Finishing F=%D mm/min. and S=%D RPM", P21, 1200]
```

Up to 5 identifiers %D or %d may be defined, but there must be as many data values as identifiers.

21.

21.1.4 Display instructions. Define the size of the the graphics area

#DGWZ

Defines the graphics area

The graphic area can be defined with the instruction #DGWZ (Define Graphics Work Zone). The new graphic area defined is kept until another one is defined, modified at the graphics window or the CNC is turned off. On power-up, the CNC assumes the graphic area defined by default.

Programming on a mill model.

On a mill model, the programming format is the following.

```
#DGWZ [<Xmin>,<Xmax>,<Ymin>,<Ymax>,<Zmin>,<Zmax>]
```

<Xmin>	Lower X axis limit.
<Xmax>	Upper X axis limit.
<Ymin>	Lower Y axis limit.
<Ymax>	Upper Y axis limit.
<Zmin>	Lower Z axis limit.
<Zmax>	Upper Z axis limit.

Both limits may be positive or negative, but the lower limits of an axis must always be smaller than the upper limits for that axis.

Programming on a lathe model.

On a lathe model, the programming format is the following.

```
#DGWZ [<Zmin>,<Zmax>,<Xmin>,<Xmax>]
```

<Zmin>	Lower Z axis limit.
<Zmax>	Upper Z axis limit.
<Xmin>	Inside radius or diameter.
<Xmax>	Outside radius or diameter.

Both limits may be positive or negative, but the lower limits of an axis must always be smaller than the upper limits for that axis.

21.

21.1.5 Enabling and disabling instructions

#ESBLK
Beginning of the single-block treatment

#DSBLK
End of the single-block treatment

The #ESBLK and #DSBLK instructions activate and deactivate the single block treatment.

When executing the #ESBLK instruction, the CNC executes the following blocks as if they were a single block. This single block treatment remains active until canceled by executing the #DSBLK instruction.

```
G01 X20 Y0 F850
G01 X20 Y20
#ESBLK
  (Beginning of single block)
G01 X30 Y30
G02 X20 Y40 I-5 J5
G01 X10 Y30
G01 X20 Y20
#DSBLK
  (End of single block)
G01 X20 Y0
M30
```

This way, when executing a program in "SINGLE BLOCK" mode, the group of blocks located between #ESBLK and #DSBLK will be executed in a row. In other words, the execution will not be interrupted after each block; it will go on until reaching the #DSBLK instruction.

#ESTOP
Enable the CYCLE STOP signal

#DSTOP
Disable the CYCLE STOP signal

The #ESTOP and #DSTOP instructions enable and disable the CYCLE STOP signal whether it comes from the operator panel or from the PLC.

When executing the #DSTOP statement, the CNC disables the CYCLE STOP key of the operator panel and the CYCLE STOP signal coming from the PLC. It is kept disabled until canceled by the #ESTOP instruction.

#EFHOLD
Enable the feed-hold signal

#DFHOLD
Disable the feed-hold signal

The #EFHOLD and #DFHOLD instructions enable and disable the FEED-HOLD coming from the PLC.

When executing the #DFHOLD instruction, the CNC disables the FEED-HOLD input coming from the PLC. It is kept disabled until canceled by the #EFHOLD instruction.

21.

21.1.6 Electronic axis slaving

Two axes may be slaved to each other so the movement of one of them (slave) depends on the movement of the other one (master).

It is possible to have several axis couplings (slaving) at the same time.

Axis coupling is activated with the #LINK instruction and canceled with the #UNLINK instruction. When reaching the end of program with a coupled pair of axes, this slaving is canceled after executing an M02 or M30.

Considerations about axis coupling

Although the #LINK instruction admits several sets (pairs) of axes, the following limitations must be taken into account:

- The main axes (the first three axes of the channel) cannot be slaves.
- Both axes of the master-slave pair must be of the same type (linear or rotary).
- The master axis of a pair cannot be the slave of another pair.
- An axis cannot be slaved to more than one master axis.

Likewise, a new slaving (coupling) cannot be activated without deactivating the pairs previously slaved.

#LINK

Activate the electronic coupling (slaving) of axes

This instruction defines and activates the electronic coupling of axes. Several couplings may be activated at the same time. When executing this instruction, all the axes defined as slaves depend on their relevant masters. On these slave axes, no movement may be programmed while they stay coupled.

This instruction may also be used to define the maximum following error difference allowed between the master axis and its slave.

The programming format is:

```
#LINK [<master>,<slave>,<error>][...]
```

Parameter	Meaning
<master>	Master axis.
<slave>	Slave axis.
<error>	Optional. Maximum difference allowed between the following errors of both axes.

Programming the amount of error is optional; if not programmed, this test is not carried out. The maximum error will be defined in millimeters or inches for linear axes and in degrees for rotary axes.

```
#LINK [X,U][Y,V,0.5]
#LINK [X,U,0.5][Z,W]
#LINK [X,U][Y,V][Z,W]
```

#UNLINK

Cancel the electronic coupling (slaving) of axes

This instruction deactivates the active axis slaving.

```
#UNLINK
(Cancels axis coupling)
```

When reaching the end of program with a coupled pair of axes, this slaving is canceled after executing an M02 or M30.

21.1.7 Axis parking

Some machines, depending on the type of machining, may have two different configurations (axes and spindles). In order to prevent the elements not present in one of the configurations from causing an error message (drives, feedback systems, etc.) the CNC allows parking them.

For example, a machine that swaps a normal spindle with a rectangular one may have the following axes configurations:

- With a normal spindle, X Y Z axes configuration.
- With an orthogonal spindle, X Y Z A B axes configuration.

In this case, when working with the normal spindle, the A and B axes may be parked to ignore their signals.

Several axes and spindles may stay parked at the same time, but they must always be parked (and unparked) one by one.

Use the #PARK instruction to park the axes and spindles and #UNPARK to cancel (unpark) them. The axes and spindles stay parked after executing an M02 or M30, after a RESET and even after turning the CNC off and back on.

Considerations about axis parking

The CNC will not allow parking an axis in the following cases.

- If the axis belongs to the active kinematics.
- If the axis belongs to an active transformation #AC or #ACS.
- If the axis belongs to an active angular transformation #ANGAX .
- If the axis belongs to a gantry, tandem pair or is a slaved axis.
- If the axis belongs to an active tangential control #TANGCTRL.

Considerations about spindle parking

The CNC will not allow parking a spindle in the following cases.

- If the spindle is not stopped.
- If the spindle is working as a C axis.
- If G96 or G63 is active and it is the master spindle of the channel.
- If G33 or G95 is active and it is the master spindle of the channel or the spindle is used to synchronize the feedrate.
- If the spindle belongs to a tandem pair or is a synchronized spindle, be it the master or the slave.

If after parking the spindles, there is only one spindle left in the channel, it will become the new master. If a spindle is unparked and it is the only spindle of the channel, it is also assumed as the new master spindle.

#PARK

Parks an axis

This instruction is used to park the selected axis or spindle. When any of them is parked, the CNC interprets that it no longer belongs to the machine configuration and no longer controls it (ignoring the signals from the drive and from the feedback systems, etc.).

Once an axis or spindle has been parked, the part-program cannot mention it (movements, speed, M functions, etc.).

The programming format is:

```
#PARK <axis/spindle>
```

Each element (axis or spindle) must be parked separately. However, a second element can be parked without having to unpark the first one.

21.

STATEMENTS AND INSTRUCTIONS
Programming statements



CNC 8060
CNC 8065

(REF. 1402)

When trying to park an axis or spindle that is already parked, the programming is ignored.

```
#PARK A
  (It parks the "A" axis)
#PARK S2
  (It parks spindle "S2")
```

#UNPARK **Unparks an axis**

This instruction is used to unpark the selected axis or spindle. When unparking one of them, the CNC interprets that it belongs to the machine configuration and starts controlling it.

The programming format is:

```
#UNPARK <axis/spindle>
```

The axes must be unparked one by one.

When trying to unpark an axis or spindle that is already unparked, the programming is ignored.

```
#UNPARK A
  (It unparks the "A" axis)
#UNPARK S
  (It unparks the "S" spindle)
```

21.1.8 Modifying the configuration of the axes of a channel

Initially, each channel has some axes assigned to it as set by the machine parameters. While executing a program, a channel may release its axes or request new axes. This possibility is determined by machine parameter `AXISEXCH`, which establishes whether an axis can change channels or whether this change is permanent or not.

A permanent change is maintained after the end of the program, after a reset and on power-up. The original configuration may be restored either by validating the general parameters and restarting or by a part-program that undoes the changes.



It also restores the machine parameter settings if a checksum error occurs when powering up the CNC.

Knowing if an axis can change channels

Machine parameter `AXISEXCH` may be consulted using the following variable.

`V.MPA.AXISEXCH.Xn`

Replace "Xn" with the name or logic number of the axis.

Value	Meaning
0	It cannot change channels.
1	The change is temporary.
2	The change is permanent.

Knowing in which channel the axis is

It is possible to know in which channel the axis is by using the following variable.

`V.[n].A.ACTCH.Xn`

Replace "Xn" with the name or logic number of the axis.

Replace the "n" letter with the channel number.

Value	Meaning
0	It is not in any channel.
1-4	Channel number.

Commands for modifying the axis configuration via program

The following instructions are used to modify the configuration of the axes. It is possible to add or remove axes, change their names and even redefine the main axes of the channel by swapping their names.

Changing the configuration of the axes cancels the active polar origin, the pattern rotation, the mirror image and the scaling factor.

In the configuration of the axes (if G17 is active), the axis that occupies the first position must be the abscissa axis, the second will be the ordinate axis, the third will be the axis perpendicular to the work plane, the fourth will be the first auxiliary axis and so on.

#SET AX

Sets the axis configuration

Defines a new axis configuration in the channel. The channel axes not programmed in the instruction and the nonexistent programmed ones will be added. The axes are placed in the channel in the positions as they are programmed in the instruction `#SET AX`. Optionally, one or several offsets may be applied to the defined axes.

It is the same as programming a `#FREE AX` of all the axes and then a `#CALL AX` of all the new axes.

The instruction `#SET AX` may also be used only to order the existing axes in the channel differently.

The programming format is:

```
#SET AX [<Xn>, ...] <offset> <...>
```

21.

STATEMENTS AND INSTRUCTIONS
Programming statements



CNC 8060
CNC 8065

(REF. 1402)

Parameter	Meaning
<Xn>	Axes that make up the new configuration. If instead of defining an axis, a zero is written, an empty space (without an axis) appears in this position.
<offset>	Optional. It sets which offset is applied to the axes. Several offsets may be applied.

```
#SET AX [X,Y,Z]
#SET AX [X,Y,V1,0,A]
```

Offset setting

The offsets that may be applied to the axes are identified with the following commands. To apply several offsets, program the relevant commands separated by a blank space.

Command	Meaning
ALL	Include all the offsets.
LOCOF	Include the offset of the reference search.
FIXOF	Include the fixture offset.
ORGOF	Include zero offset.
MEASOF	Include measurement offset.
MANOF	Include the offset of the manual operations.

```
#SET AX [X,Y,Z] ALL
#SET AX [X,Y,V1,0,A] ORGOF FIXOF
```

If when defining a new configuration only the order of the axes in the channel is swapped, the offsets are ignored.

Screen display

At first, the axes appear ordered as they have been defined in the general machine parameter table (by channels) and then as the swapping is defined.

Y 00000.0000 ? 00000.0000 ? 00000.0000 Z 00000.0000 A 00000.0000	X 00125.1500 Y 00089.5680 Z 00000.0000 ? 00000.0000 ? 00000.0000
#SET AX [Y, 0, 0, Z, A]	#SET AX [X, Y, Z] FIXOF ORGOF
Screen display of the different configurations. Let us suppose a machine with 5 axes X-Y-Z-A-W.	

#CALL AX Adds an axis to the configuration

it adds one or more axes to the preset configuration and it also allows defining its position. If the axis already exists in the configuration, it is placed in the new position. If the axis already exists and no position is programmed, the axis stays in its original position. Optionally, one or several offsets may be applied to the defined axes.

The programming format is:

```
#CALL AX [<Xn>,<pos>...] <offset> <...>
```

21.

Parameter	Meaning
<Xn>	Axes to be added to the configuration. If the axis already exists, it is placed in the new position.
<pos>	Optional. Position of the axis in the new configuration. If not programmed, the axis is placed after the last one. If the position is occupied, the relevant error message will be issued.
<offset>	Optional. It sets which offset is applied to the axes. Several offsets may be applied.

```
#CALL AX [X,A]
  (It adds the X and A axes to the configuration, after the last existing axis)
#CALL AX [V,4,C]
  (It adds the V axis to position 4 and the C axis after the last one)
```

Offset setting

The offsets that may be applied to the axes are identified with the following commands. To apply several offsets, program the relevant commands separated by a blank space.

Command	Meaning
ALL	Include all the offsets.
LOCOF	Include the offset of the reference search.
FIXOF	Include the fixture offset.
ORGOF	Include zero offset.
MEASOF	Include measurement offset.
MANOF	Include the offset of the manual operations.

```
#CALL AX [X] ALL
#CALL AX [V1,4,Y] ORGOF FIXOF
```

Screen display

At first, the axes appear ordered as they have been defined in the general machine parameter table (by channels) and then as the swapping is defined.

Y 00000.0000	Axis configuration #SET AX [Y, 0, 0, Z] Y: Abscissa axis. Z: First auxiliary axis. #CALL AX [X,2, W, 3] Y: Abscissa axis. X: Ordinate axis. W: Axis perpendicular to the plane. Z: First auxiliary axis.
X 00000.0000	
W00000.0000	
Z 00000.0000	
? 00000.0000	

#FREE AX**Frees an axis from the configuration**

Removes the programmed axes from the current configuration. After removing an axis, the position is free, but the order of the axes that remain in the channel does not change.

The programming format is:

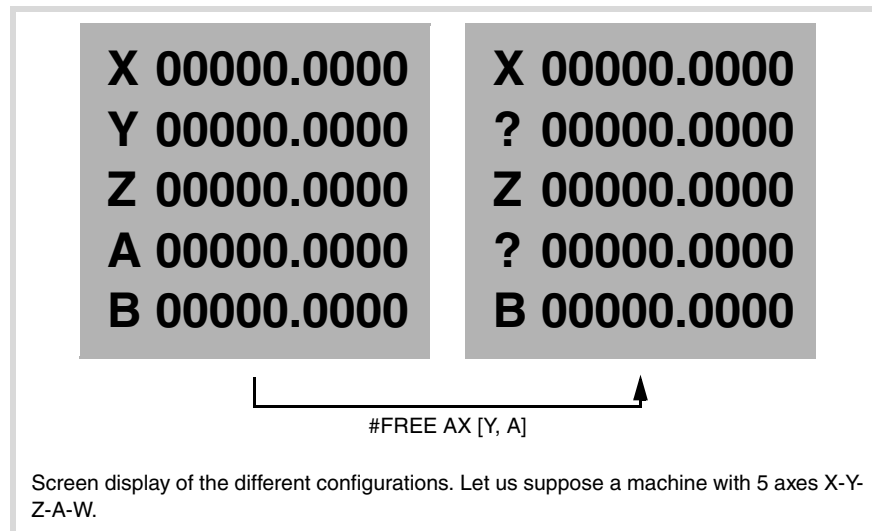
```
#FREE AX [<Xn>, ...]
```

Parameter	Meaning
<Xn>	Axis to be removed from the configuration

```
#FREE AX [X,A]
    (It removes the X and A axes from the configuration)
#FREE AX ALL
    (Removes all the axes from the channel)
```

Screen display

At first, the axes appear ordered as they have been defined in the general machine parameter table (by channels) and then as the swapping is defined.



#RENAME AX Renames the axes

It changes the name of the axes. For each programmed axis pair, the first axis takes the name of the second one. If the second axis is present in the configuration, it takes the name of the first one. Any axis may be renamed with any name, whether it is in any channel or not.

The programming format is:

```
#RENAME AX [<Xn1>,<Xn2>][...]
```

Parameter	Meaning
<Xn1>	Axis whose name is to be changed
<Xn2>	new axis name.

```
#RENAME AX [X,X1]
    (The X axis is now called X1. If X1 already exists in the channel, it is called X)
#RENAME AX [X1,Y][Z,V2]
```

Parameter **RENAMECANCEL** indicates whether the CNC keeps or cancel the names of the axes and spindles (**#RENAME**) after executing M02 or M30, after a reset or at the beginning of a new part-program in the same channel.

After turning the CNC off and on, the axes and spindles always keep their new name, except after a checksum error or after validating the machine parameters that involve restoring the original configuration of channels, axes or spindles. In either case, the axes and the spindles will recover their original names.

When a channel releases (frees) an axis (instruction **#SET** or **#FREE**), the axis always recovers its original name.

Even if the **#RENAME** is kept (parameter **RENAMECANCEL**), the CNC cancels it if the channel recovers an axis with the same name after a reset or after a new program starts. This occurs if the **#RENAME** uses the name of an axis whose channel change permission is temporary or **no_exchange** (parameter **AXISEXCH**), that is not in the channel at the time.

Accessing the variables of a renamed axis.

After changing the name of an axis, the new name of the axis must be used to access its variables from the part-program or MDI. The access to the variables from the PLCA or from an interface does not change; the original name of the axis remains unchanged.

#RENAME AX OFF

Cancel rename.

This instruction cancels the renaming of the indicated axes, regardless of what parameter RENAMECANCEL indicates; if no axis is defined, it cancels the renaming of all the axes of the channel.

The programming format is:

```
#RENAME AX OFF [<Xn>, <Xn>, ...]
```

Parameter	Meaning
<Xn>	Renamed axis.

```
#RENAME AX OFF [X]
  (Cancel the renaming of the X axis).
#RENAME AX OFF
  (Cancel the renaming of all the axes).
```

21.

STATEMENTS AND INSTRUCTIONS
Programming statements



CNC 8060
CNC 8065

(REF. 1402)

21.1.9 Modifying the configuration of the spindles of a channel

The CNC can have up to four spindles distributed between the various channels of the system. A channel may have one, several or no spindles associated with it.

Initially, each channel has some spindles assigned to it as set by the machine parameters. While executing a program, a channel may release its spindles or request new spindles. This possibility is determined by machine parameter `AXISEXCH`, which establishes whether a spindle can change channels or whether this change is permanent or not.

A permanent change is maintained after the end of the program, after a reset and on power-up. The original configuration may be restored either by validating the general parameters and restarting or by a part-program that undoes the changes.



It also restores the machine parameter settings if a checksum error occurs when powering up the CNC.

Knowing if a spindle can change channels

Machine parameter `AXISEXCH` may be consulted using the following variable.

`V.MPA.AXISEXCH.Sn`

Replace "Sn" with the spindle name.

Value	Meaning
0	It cannot change channels.
1	The change is temporary.
2	The change is permanent.

Knowing in which channel the spindle is

It is possible to know in which channel the spindle is by using the following variable.

`V.[n].A.ACTCH.Sn`

Replace "Sn" with the spindle name.

Replace the "n" letter with the channel number.

Value	Meaning
0	It is not in any channel.
1-4	Channel number.

Commands for modifying the spindle configuration via program

The following instructions are used to modify the configuration of the spindles of the channel. It is possible to add or remove spindles, change the name of the spindles and define which one is the master spindle of the channel.

#FREE SP

Frees a spindle from the configuration

Removes the defined spindles from the current configuration.

The programming format is:

```
#FREE SP [<Sn>, ...]
#FREE SP ALL
```

Parameter	Meaning
<Sn>	Spindle name.
ALL	Frees all the spindles of the channel.

```
#FREE SP [S]
  (It removes the spindle S from the configuration)
#FREE SP [S1,S4]
  (It removes spindles S1 and S4 from the configuration)
#FREE SP ALL
  (It removes all the spindles from the configuration)
```

21.

#CALL SP**Add a spindle to the configuration**

It adds one or several spindles to the current configuration. The position of the spindles in the channel is not relevant. To add a spindle to the channel, the spindle must be free; it must not be in another channel.

The programming format is:

```
#CALL SP [<Sn>, ...]
```

Parameter	Meaning
<Sn>	Spindle name.

```
#CALL SP [S1]
```

(It adds spindle S1 to the configuration)

```
#CALL SP [S,S2]
```

(It adds spindles S and S2 to the configuration)

#SET SP**Sets the spindle configuration**

Defines a new spindle configuration. The spindles existing in the channel and not programmed in #SET SP are removed and those programmed that are not already in the channel will be added.

It is the same as programming a #FREE SP of all the spindles and then a #CALL SP of all the new spindles. The programming format is:

```
#SET SP [<Sn>, ...]
```

Parameter	Meaning
<Sn>	Spindle name.

```
#SET SP [S]
```

(Configuring one spindle)

```
#SET SP [S1,S2]
```

(Configuring two spindles)

#RENAME SP**Rename the spindles**

It changes the name of the spindles. For each programmed spindle pair, the first spindle takes the name of the second one. If the second spindle is present in the configuration, it takes the name of the first one. Any axis may be renamed with any name, whether it is in any channel or not.

The programming format is:

```
#RENAME SP [<Sn>,<Sn>][...]
```

Parameter	Meaning
<Sn>	Spindle name.

```
#RENAME SP [S,S1]
```

```
#RENAME SP [S1,S2][S3,S]
```

Parameter RENAMECANCEL indicates whether the CNC keeps or cancel the names of the axes and spindles (#RENAME) after executing M02 or M30, after a reset or at the beginning of a new part-program in the same channel.

After turning the CNC off and on, the axes and spindles always keep their new name, except after a checksum error or after validating the machine parameters that involve restoring the original configuration of channels, axes or spindles. In either case, the axes and the spindles will recover their original names.

When a channel releases (frees) a spindle (instruction #SET or #FREE), the axis always recovers its original name.

Even if the #RENAME is kept (parameter RENAMECANCEL), the CNC cancels it if the channel recovers a spindle with the same name after a reset or after a new program starts. This occurs if the #RENAME uses the name of a spindle whose channel change permission is temporary or no_exchange (parameter AXISEXCH), that is not in the channel at the time.

Accessing the variables of a renamed axis.

After changing the name of an axis, the new name of the axis must be used to access its variables from the part-program or MDI. The access to the variables from the PLCA or from an interface does not change; the original name of the axis remains unchanged.

#RENAME SP OFF Cancel rename.

This instruction cancels the renaming of the indicated spindles, regardless of what parameter RENAMECANCEL indicates; if no spindle is defined, it cancels the renaming of all the spindles of the channel.

The programming format is:

```
#RENAME SP OFF [<Sn>, <Sn>, ...]
```

Parameter	Meaning
<Sn>	Renamed spindle.

```
#RENAME SP OFF [S3]
  (Cancel the renaming of spindle S3).
#RENAME SP OFF
  (Cancel the renaming of all the spindles).
```

21.1.10 Spindle synchronization

This mode may be used to set the movement of a spindle (slave) synchronized with that of another spindle (master) through a given ratio. The spindle synchronization is always programmed in the channel the slave spindle belongs to, both to activate it or deactivate it and to reset it.

There are two types of synchronization; in speed and in position. The activation and cancellation of the different types of synchronization are programmed with the following instructions.

- #SYNC - Spindle synchronization based on the real (actual) coordinate.
- #TSYNC - Spindle synchronization based on the theoretical coordinate.
- #UNSYNC - Cancellation of spindle synchronization.

#SYNC

- Spindle synchronization based on the real (actual) coordinate.

#TSYNC

- Spindle synchronization based on the theoretical coordinate.

The programming format for each of them is the following. Optional parameters are indicated between the <> characters.

```
#SYNC [{master}, {slave} <,N{nratio}, D{dratio}> <,O{posync}> <,{looptype}>
<,{keepsync}>][··]
#TSYNC [{master}, {slave} <,N{nratio}, D{dratio}> <,O{posync}> <,{looptype}>
<,{keepsync}>][··]
```

Each pair of brackets defines a synchronism between two spindles.

Parameter	Meaning
{master}	Master spindle of the synchronization.
{slave}	Master spindle of the synchronization.
{nratio} {dratio}	Optional. This pair of numbers define the gear ratio (nratio/dratio) between the synchronized spindles. Both values can be positive or negative.
{posync}	Optional. This parameter determines that they are synchronized in position and it also sets the offset (shift) between the two spindles. The values may be positive or negative and greater than 360°.
{looptype}	Optional. This parameter indicates the type of loop for the master spindle. "CLOOP" for the spindle in closed loop. "OLOOP" for the spindle in open loop. If not programmed, the statement assumes the value "CLOOP".
{keepsync}	Optional. This parameter indicates whether the CNC cancels spindle synchronization or not after executing M02, M30 or after an error or reset. With "CANCEL" value, the CNC cancels the synchronism; with "NOCANCEL" value, it doesn't cancel it. If not programmed, the statement assumes the value set by the OEM (parameter SYNCCANCEL).

#SYNC [S,S1]

The spindles are synchronized in speed. The slave spindle S1 turns at the same speed as the master spindle S.

#SYNC [S,S1,N1,D2]

The slave spindle S1 turns at half the speed (1/2) of the master spindle S.

#SYNC [S,S1,N1,D2,O15]

After synchronizing in speed and in position, the slave spindle S1 follows the master S with the indicated offset, which could be for example 15°.

#SYNC [S,S1,O30,OLOOP]

Synchronism in speed and position with 30° offset. The master spindle works in open loop.

#SYNC [S,S1,O30,CLOOP, CANCEL]

Synchronism in speed and position with 30° offset. The master spindle works in closed loop. The CNC cancels the synchronism after M30, an error or a reset.

21.

STATEMENTS AND INSTRUCTIONS
Programming statements



CNC 8060
CNC 8065

(REF. 1402)

Considerations for the synchronization

The #SYNC function may be executed either in open loop (M3 or M4) or in closed loop (M19). In the synchronization, the master spindle can work in either open or closed loop; the slave spindle is always in closed loop.

Several pairs of synchronized spindles may be programmed in the same instruction #SYNC or #TSYNC. Several consecutive #SYNC instructions can also be programmed with an additive effect as long as they do not conflict with the previous ones.

The slave spindle must be in the channel where the synchronization is activated whereas the master spindle may be in any channel. Several slave spindles may have the same master spindle, but a slave spindle cannot be the master of a third one; thus preventing synchronism loops.

It is possible to program either first the synchronism in speed and then in position or both at the same time. Once a pair has been synchronized, it is possible to change its speed ratio and/or its offset (shift); if necessary, the spindles will get out of synchronism and back in synchronism to make the change.

Both spindles must operate in closed loop in order to assure proper follow-up. Once they're both in closed loop, the slave spindle switches from its current speed to the synchronism speed. The master spindle may be turning when programming the synchronization and it will switch to closed loop maintaining its rotation.

Programming the master and slave spindles

For the slave spindle, it is not possible to program the speed, the spindle functions M3, M4, M5 and M19, gear changes M41 through M44 or change the speed override.

The following functions may be programmed for the master spindle:

- Change the spindle speed via PLC or CNC.
- Execute the speed functions G94, G95, G96 and G97.
- Execute the auxiliary functions M3, M4, M5 and M19.
- Change the spindle speed override via PLC, CNC or keyboard.
- Change the spindle speed limit via PLC or CNC.
- If the C axis is activated, define the XC or ZC plane.

When defining the synchronization or when it is active, the master spindle can work as C axis or in G63. Functions G33, G95 or G96 may also be active at the master spindle. Functions G33 and G95 may also be active at the slave spindle, but function G96 will stay temporarily "frozen" and without effect during the synchronization.

On the other hand, the synchronized spindles cannot change channels and it is not possible to change gears M41 through M44. If the gear change is automatic and the new speed requires a gear change, the relevant error message will be issued.

Work gear

The spindles may have different gears (ranges). If at the time of the synchronization, the spindles are not in the same state, the slave "freezes" its state, changes the gear indicated by machine parameter SYNCSET and is forced to follow the master.

If the master belongs to the same channel, it also changes to the gear indicated by its parameter SYNCSET. If the master is in another channel, the gear must be activated before activating the synchronization. Therefore, it is up to the user to prepare the master spindle so the slave can synchronize it with it.

Home search

The slave spindle must be homed (referenced) before activating the position synchronism if it has been homed before. If the master spindle is in the same channel and has not been homed, it is also forced to be homed. An error message will be issued if the master spindle is in another channel and it has not been homed.

#UNSYNC**Uncouple one or several spindles**

The programming format is: Optional parameters are indicated between the <> characters.

```
#UNSYNC
#UNSYNC [slave1 <,slave2> ...]
```

All the spindles are uncoupled if no parameter is defined.

Parameter	Meaning
slave	Slave spindle to be synchronized.

```
#UNSYNC
  All the spindles of the channel are uncoupled.
#UNSYNC [S1,S2]
  Slave spindles S1 and S2 are uncoupled from the master spindle with which they were
  synchronized.
```

Considerations for uncoupling

The synchronism is also canceled with M30 and RESET.

When canceling the synchronism, the master spindle stays in its current state and the slave stops. The slave does not recover the M function prior to the synchronization, but it keeps the synchronism gear until a new S function is programmed.

Variables associated with the synchronization movement

These variables are read/write (R/W) synchronous and are evaluated during execution. They have generic names.

- Replace the "n" character with the channel number, maintaining the brackets. The first channel is identified with the number 1, "0" is not a valid number.
- Replace the "Xn" character by the name, logic number or index in the channel of the axis.

Adjust the speed synchronism ratio**(V.)[n].A.GEARADJ.Xn**

Read-only from the PRG, PLC and INT. The PLC reading comes in hundredths (x100)

Fine adjustment of the gear ratio during the synchronization itself. It is programmed as a percentage of the original adjustment value.

Velocity synchronization**(V.)[n].A.SYNCVELW.Xn**

Read-only from the PRG, PLC e INT.

When the spindles are synchronized in speed, the slave spindle turns at the same speed as the master spindle (taking the ratio into account). If the value defined in this variable is exceeded, the SYNSPEED signal goes low; the movement is not stopped and no error message is issued.

Its default value is that of machine parameter DSYNCVELW.

(V.)[n].A.SYNCVELOFF.Xn

Read-only from the PRG, PLC e INT.

Velocity offset of the slave spindle synchronization.

21.

Position synchronization

(V.)[n].A.SYNCPOSW.Xn

Read-only from the PRG, PLC e INT.

When the spindles are synchronized in position, the slave spindle follows the master keeping the programmed offset (taking the ratio into account). If the value defined in this variable is exceeded, the SYNCPOSI signal goes low; the movement is not stopped and no error message is issued.

Its default value is that of machine parameter DSYNCPOSW.

(V.)[n].A.SYNCPOSOFF.Xn

Read-only from the PRG, PLC e INT.

Position offset.

21.1.11 Selecting the loop for an axis or a spindle. Open loop or closed loop



This function is not available for Sercos position-drives (axis or spindle). In this case, the CNC cannot open or close the loop, the drive controls the loop, instead.

When working in open loop, the command does not depend on the feedback. When working in closed loop, the command is generated according to the feedback.

The spindle usually works in open loop when it is in M3 or M4 and in closed loop when it is in M19. In spindle synchronization, the slave always works in closed loop and the master can work either in open loop or closed loop depending on the programming parameters of the #SYNC instruction. However, it is possible to work in closed loop with functions M3 and M4 to make the following adjustments on a spindle:

- Adjust the loop for M19.
- Adjust a loop for when the spindle is the master of a synchronization.

The axes usually work in closed loop. A rotary axis may be controlled in open loop as if it were a spindle.

The following instructions may be used to open and close the loops and they are valid for axes as well as for spindles.

```
#SERVO ON      - Activates the closed loop mode.
#SERVO OFF     - Activates the open loop mode.
```

#SERVO ON Activates the closed loop mode

Programming this instruction switches the axis or spindle to closed loop mode.

The spindle must be homed before switching to closed loop mode; otherwise the loop will be closed and a warning message will be issued.

The programming format is:

```
#SERVO ON [axis/spindle]
```

Parameter	Meaning
axis/spindle	Name of the axis or spindle.

The loop of each axis or spindle must be closed separately.

```
#SERVO ON [S]
  Closes the loop of spindle S.
#SERVO ON [S2]
  Closes the loop of spindle S2.
#SERVO ON [X]
  Closes the loop of the X axis.
```

#SERVO OFF Activates the open loop mode

Programming this instruction switches the axis or spindle to open loop mode. For a spindle, it cancels the closed-loop mode programmed #SERVO ON and restores the status that the spindle had before closing the loop.

- If the spindle was in M19, it stays in closed loop after programming this instruction.
- In spindle synchronization, the #SERVO OFF instruction cannot be programmed for the slave spindle; if programmed, the CNC will issue an error message.

If the synchronization has been defined with the master spindle working in closed loop, it stays in closed loop after programming #SERVO OFF. If the synchronization has been defined with the master spindle working in open loop and, later on, it is closed with #SERVO ON, the loop of the master spindle will be opened after programming #SERVO OFF.

- If the spindle was in M3, M4 or M5 without synchronism, the CNC opens the loop.

21.

The programming format is:

```
#SERVO ON [axis/spindle]
```

Parameter	Meaning
axis/spindle	Name of the axis or spindle.

The loop of each axis or spindle must be opened separately.

```
#SERVO OFF [S]
```

It cancels the closed loop of spindle S.

```
#SERVO OFF [Z2]
```

The Z2 axis switches to open loop mode.

Considerations for programming the loops

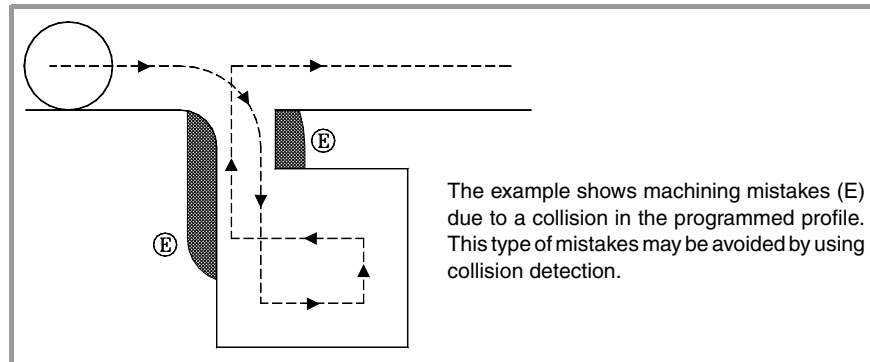
Function M19 involves always working in closed loop. Functions M3, M4 and M5 work in open loop by default , but they can also work in closed loop when programming a spindle synchronization or the instruction #SERVO ON .

When a spindle becomes a C axis or is interpolated with the rest of the axes (for rigid tapping, for example) does not lose its open-loop or closed-loop condition that it had. When done with these instructions, the previous situation is restored.

On power-up, the spindle is set in open loop. After executing an M30 or reset, the CNC opens the loop and cancels the instruction #SERVO ON, except when the reset is for the master spindle of a synchronization (which could be in a different channel from that of the slave); in that case, the synchronization is not canceled and it does not switch to open loop. This case issues a warning message.

21.1.12 Collision detection

With this option, the CNC analyzes in advance the blocks to be executed in order to detect loops (intersections of the profile with itself) or collisions in the programmed profile. The operator may define up to 200 blocks on an 8065 CNC and 40 blocks on an 8060 to be analyzed.



When detecting a loop or a collision, the CNC will not execute the blocks that cause it and the screen will display a warning to let the operator know that the programmed profile has been modified. It will display a warning for each loop or collision eliminated.

The information contained in the blocks removed if it is not a movement in the active plane will be executed (including the movements of the other axes).

Considerations for the collision detecting process.

- Collision detection may be applied even when tool radius compensation is not active.
- Being collision detection active, it is possible to apply zero offsets, coordinate presetting and tool changes. However, home searches and measurements are NOT possible.
- Changing the work plane will interrupt the collision detecting process. The CNC checks for collisions in the blocks stored so far and resumes the process with the new plane starting with the new motion blocks.
- The collision detecting process will be interrupted when programming a instruction (explicit or implicit) that involves synchronizing block preparation and execution (e.g. #FLUSH). The process will resume after executing that instruction.
- Collision detection cannot be activated if a Hirth axis is active and it is part of the main plane. Likewise, while collision detection is active, an axis cannot be activated as a Hirth axis and the work plane cannot be changed if one of the axis is a Hirth axis.

#CD ON

Activate collision detection

It activates the collision detecting process. Being collision detection already active, it lets modify the number of blocks to be analyze.

The programming format is:

```
#CD ON [<blocks>]
```

Parameter	Meaning
<blocks>	Optional. Number of blocks to analyze.

Defining the number of blocks to be analyzed is optional. If not defined, the CNC assumes the maximum (200 blocks). The horizon of blocks may be changed at any time, even while collision detection is active.

21.

STATEMENTS AND INSTRUCTIONS
Programming statements



CNC 8060
CNC 8065

(REF. 1402)

#CD OFF
Cancels collision detection

It cancels the collision detecting process.

The process will also be canceled automatically after executing an M02 or M30 and after an error or a reset.

Example of a profile with a loop.

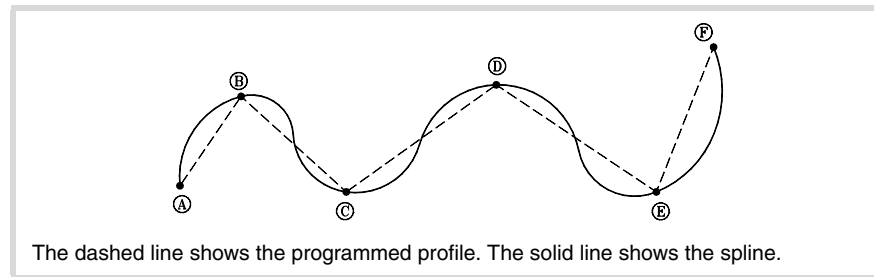
```
#CD ON [50]
G01 X0 Y0 Z0 F750
X100 Y0
Y -50
X90
Y20
X40
Y -50
X0
Y0
#CD OFF
```

Example of profile collision.

```
#CD ON
G01 G41 X0 Y0 Z0 F750
X50
Y -50
X100
Y -10
X60
Y0
X150
Y -100
X0
G40 X0 Y0
#CD OFF
M30
```

21.1.13 Spline interpolation (Akima)

This type of machining adapts the programmed contour to a spline type curve that goes through all the programmed points.



The contour to be splined is defined with straight paths (G00/G01). When defining an arc (G02/G03), the spline is interrupted while machining it and it resumes on the next straight path. The transitions between the arc and the spline is done tangentially.

#SPLINE ON Activate spline adaptation.

When executing this instruction, the CNC interprets that the points programmed next are part of the spline and begins making the curve.

The programming format is:

```
#SPLINE ON
```

The machining of splines cannot be activated if tool radius compensation (G41/G42) with linear transition between blocks (G137) or viceversa.

#SPLINE OFF Cancel spline adaptation.

When executing this instruction, the CNC ends the spline and goes on machining as the path were programmed.

The programming format is:

```
#SPLINE OFF
```

The spline can only be canceled if at least 3 points have been programmed. When defining the initial and final tangents of the spline, 2 points will be enough.

#ASPLINE MODE Select type of tangent.

This instruction sets the type of initial and final tangents of the spline that determines the transition from the previous and to the next path. It is optional; if not defined, the tangent is calculated automatically.

The programming format is:

```
#ASPLINE MODE [<initial>, <final>]
```

Parameter	Meaning
<initial>	Initial tangent.
<final>	Final tangent

The initial and final tangent of the spline may take one of the following values. If not programmed, it assumes a value of 1.

21.

STATEMENTS AND INSTRUCTIONS
Programming statements



CNC 8060
CNC 8065

(REF. 1402)

Value	Meaning
1	The tangent is calculated automatically.
2	Tangent to the previous /next block.
3	Tangent as specified.

If defined with a value of -3, the initial tangent is defined using the #ASPLINE STARTTANG instruction and the final tangent using the #ASPLINE ENDTANG instruction. If not defined, it applies the values used last.

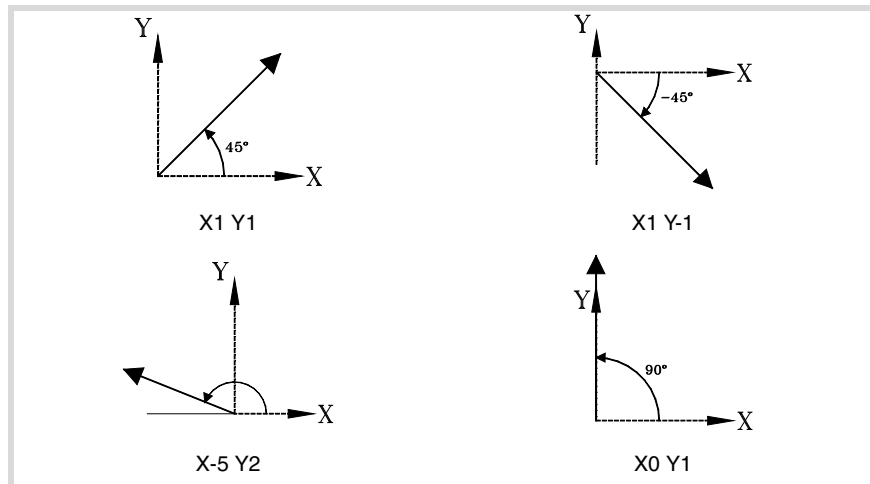
#ASPLINE STARTTANG
Initial tangent

#ASPLINE ENDTANG
Final tangent

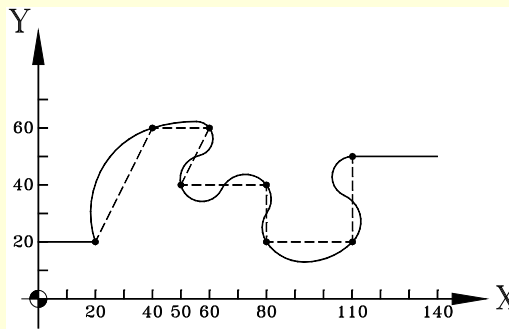
These instructions define the initial and final tangents of the spline. The tangent is determined by giving its vectorial direction along the different axes.

The programming format is:

#ASPLINE STARTTANG <axes>
#ASPLINE ENDTANG <axes>



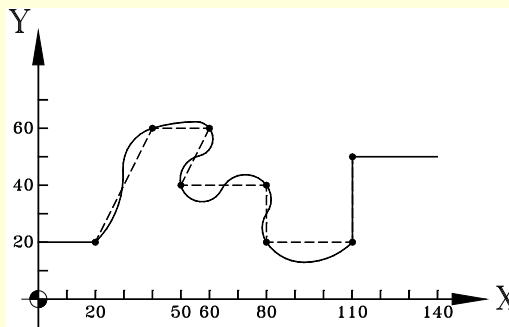
21.

STATEMENTS AND INSTRUCTIONS
Programming statements

```

N10 G00 X0 Y20
N20 G01 X20 Y20 F750           (Starting point of the spline)
N30 #ASPLINE MODE [1,2]       (Type of initial and final tangent)
N40 #SPLINE ON                 (Activation of the spline)
N50 X40 Y60
N60 X60
N70 X50 Y40
N80 X80
N90 Y20
N100 X110
N110 Y50                       (Last point of the spline)
N120 #SPLINE OFF              (Cancellation of the spline)
N130 X140
N140 M30

```



```

N10 G00 X0 Y20
N20 G01 X20 Y20 F750           (Starting point of the spline)
N30 #ASPLINE MODE [3,3]       (Type of initial and final tangent)
N31 #ASPLINE STARTTANG X1 Y1
N32 #ASPLINE ENDTANG X0 Y1
N40 #SPLINE ON                 (Activation of the spline)
...
N120 #SPLINE OFF              (Cancellation of the spline)
N130 X140
N140 M30

```

21.1.14 Polynomial interpolation

The CNC permits interpolating straight lines and arcs and the #POLY instruction may be used to interpolate complex curves, like a parabola.

#POLY Polynomial interpolation

This type of interpolation lets machining a curve given by a polynomial of up to a 4th degree where the interpolation parameter is the length of the arc.

The programming format is:

```
#POLY [<axis1>[a,b,c,d,e] <axis2>[a,b,c,d,e] .. SP<sp> EP<ep>]
```

Parameter	Meaning
<axis>	Axis to interpolate.
a,b,c,d,e	Coefficients of the polynomial.
<sp>	Initial parameter of the interpolation.
<ep>	Final parameter of the interpolation.

The coefficients define the axis path as a function for each axis.

```
#POLY [X[ax,bx,cx,dx,ex] Y[ay,by,cy,dy,ey] Z[az,bz,cz,dz,ez] .. SP<sp> EP<ep>]
```

$$X(p) = ax + bx \cdot p + cx \cdot p^2 + dx \cdot p^3 + ex \cdot p^4$$

$$Y(p) = ay + by \cdot p + cy \cdot p^2 + dy \cdot p^3 + ey \cdot p^4$$

$$Z(p) = az + bz \cdot p + cz \cdot p^2 + dz \cdot p^3 + ez \cdot p^4$$

Where "p" is the same parameter in all the axes. Parameters sp and ep define the initial and final values of "p", as the ends between which the path for each axis will be generated.

Programming a parabola. The polynomial may be represented as follows:

- Coefficients of the X axis: [0,60,0,0,0]
- Coefficients of the Y axis: [1,0,3,0,0]
- Starting parameter: 0
- End parameter: 60

The part program stays as follows.

```
G0 X0 Y1 Z0
G1 F1000
#POLY [X[0,60,0,0,0] Y[1,0,3,0,0] SP0 EP60]
M30
```

21.

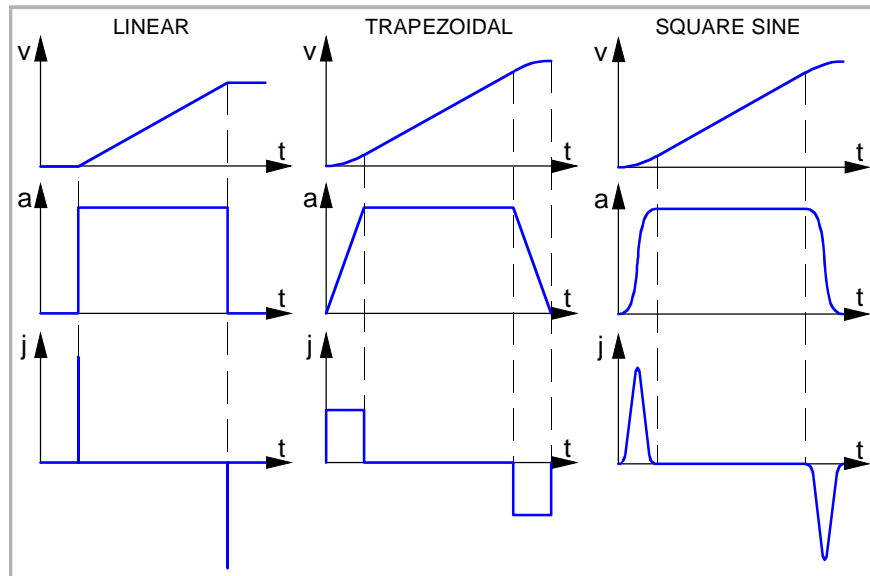
21.1.15 Acceleration control

The acceleration and the jerk (variation of acceleration) applied on the movements are set by machine parameters. However, those values may be changed from the program using the following functions.

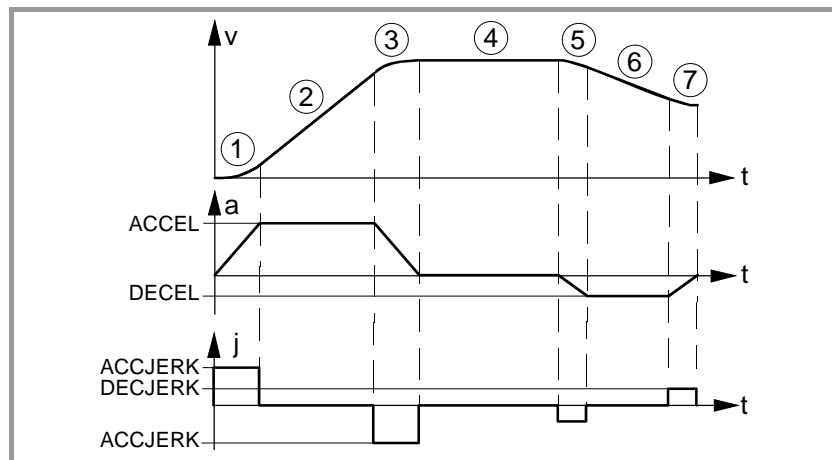
G130 or G131 Percentage of acceleration and deceleration to be applied.

G132 or G133 Percentage of acceleration jerk and deceleration jerk to be applied.

The figure below shows the graphs for velocity (v), acceleration graph (a) and jerk (j) for each case.



We now show an example of the dynamics of the trapezoidal acceleration.



- 1 The axis starts moving with a uniform increasing acceleration, with a slope limited by the percentage of acceleration jerk indicated with function G132 or G133 until reaching the acceleration percentage indicated by function G130 or G131.
- 2 The acceleration is now constant.
- 3 Before reaching the programmed feedrate, there is a steady deceleration with a slope limited by the percentage of acceleration jerk.
- 4 It goes on at the programmed feedrate and with no acceleration.
- 5 To slow down or stop the axis, a deceleration is applied with a slope limited by the percentage of deceleration jerk.
- 6 The deceleration becomes constant and its value is the percentage of deceleration.
- 7 Before reaching the programmed feedrate, or stopping, there is a deceleration with a slope limited by the percentage of deceleration jerk.

21.

#SLOPE**It sets the behavior of the acceleration**

This instruction sets the influence of the values defined with functions G130, G131, G132 and G133 in the behavior of the acceleration.

The programming format is:

```
#SLOPE [<type>,<jerk>,<accel>,<move>]
```

Parameter	Meaning
<type>	Acceleration type.
<jerk>	Optional. It sets the influence of the jerk.
<accel>	Optional. It sets the influence of the acceleration.
<mov>	Optional. It affects the movements in G00.

```
#SLOPE [1,1,0,0]
#SLOPE [1]
#SLOPE [2,,,1]
```

It is not necessary to program all the parameters. The values that each parameter may take are the following.

- The <type> parameter determines the type of acceleration.

Value	Meaning
0	Linear acceleration.
1	Trapezoidal acceleration.
2	Square sine (bell shaped) acceleration.

By default, it assumes a value of .0.

- The optional <jerk> parameter sets the influence of the Jerk defined with functions G132 and G133. It will only be taken into account in trapezoidal and square-sine type acceleration.

Value	Meaning
0	It modifies the jerk of the acceleration and deceleration stage.
1	It modifies the jerk of the acceleration stage.
2	It modifies the jerk of the deceleration stage.

By default, it assumes a value of .0.

- The optional <accel> parameter sets the influence of the acceleration set with functions G130 and G131.

Value	Meaning
0	Always applied.
1	Only applied in the acceleration stage.
2	Only applied in the deceleration stage.

By default, it assumes a value of .0.

- The optional <move> parameter determines whether functions G130, G131, G132 and G133 affect the G00 movements or not.

Value	Meaning
0	They affect the G00 movements.
1	They do NOT affect the G00 movements.

By default, it assumes a value of .0.

21.1.16 Definition of macros

Macros may be used to define a program block or part of it with their own names in the format "MacroName" = "CNCblock". Once the macro has been defined, programming "MacroName" will be the same as programming "CNCblock". When executing a macro from the program (or MDI), the CNC will execute its associated program block.

The macros defined via program (or MDI) are stored in a CNC table; this way, they are available for the rest of the programs without having to define them again. This table is initialized on CNC power-up and it can also be initialized from the part-program using the #INIT MACROTAB instruction, thus deleting the macros saved.

#DEF: Macro definition

Up to 50 different macros may be defined at the CNC. The defined macros may be accessed from any program. When trying to define too many macros, the CNC issues the relevant error message. The macro table may be initialized (erasing all the macros) using the instruction #INIT MACROTAB.

The definition of the macro must be programmed alone in the block.

The programming format is:

```
#DEF "MacroName" = "BloqueCNC"
```

Parameter	Meaning
MacroName	Name used to identify the macro in the program. It may have up to 30 characters and consist of letters and numbers.
CNCBlock	Program block. It may be up to 140 characters long.

Several macros may be defined in a block as follows.

```
#DEF "Macro1"="Block1" "Macro2"="Block2" ...
```

(Definition of macros)

```
#DEF "READY"="GO X0 Y0 Z10"
```

```
#DEF "START"="SP1 M3 M41" "STOP"="M05"
```

(Execution of macros)

```
"READY" (same as programming GO X0 Y0 Z10)
```

```
P1=800 "START" F450 (same as programming S800 M3 M41)
```

```
G01 Z0
```

```
X40 Y40
```

```
"STOP" (same as programming M05)
```

Definition of arithmetic operations in the macros.

When including arithmetic operations in the definition of a macro, the whole arithmetic operation must be included.

Correct definition of a macro.

```
#DEF "MACRO1"="P1*3"
```

```
#DEF "MACRO2"="SIN [\"MACRO1\"]"
```

The following macros are defined wrong.

```
#DEF "MACRO1"="56 +"
```

```
#DEF "MACRO2"="12"
```

```
#DEF "MACRO3"="\"MACRO1\" \"MACRO2\""
```

```
#DEF "MACRO4"="SIN["
```

```
#DEF "MACRO5"="45]"
```

```
#DEF "MACRO6"="\"MACRO4\" \"MACRO5\""
```

21.

Concatenating of macros. Including macros in the definition of other macros.

The definition of a macro can include other macros. In this case, one of the macros included in the definition must be delimited with the \ characters (\ "macro\ ").

```
Example1
#DEF "MACRO1"="X20 Y35"
#DEF "MACRO2"="S1000 M03"
#DEF "MACRO3"="G01 \ "MA1\" F100 \ "MA2\"""

Example 2
#DEF "POS"="G1 X0 Y0 Z0"
#DEF "START"="S750 F450 M03"
#DEF "MACRO"="\ "POS\" \ "START\"""
```

#INIT MACROTAB
Resetting the table of macros

When defining a macro from a program (or MDI), it is stored in a CNC table so it is available for all the rest of the programs. This instruction resets the table of macros erasing the ones stored in it.

21.1.17 Block repetition

This instruction may be used to execute a portion of the program defined between two blocks which will be identified with labels. The label of the last block must be programmed alone.

Optionally, it is possible to define the number of repetitions of the execution; if not defined, it will be repeated once.

The number of blocks to be repeated must be defined in the same program or subroutine from which this instruction is executed. They may also be after the program (after function M30)

Up to 20 nesting levels are allowed.

#RPT Block repetition

The programming format is:

```
#RPT [<blk1>,<blk2>,<n>]
```

Parameter	Meaning
<blk1>	First block.
<blk2>	Last block.
<n>	Optional. Number of repetitions.

Since the labels to identify the blocks may be of two types (number and name), the #RPT instruction may be programmed as follows:

- The label is the block number.

In the blocks containing the first and last labels, program the ":" character after the block number. This is required in every label that is the target of a jump.

```
N10 #RPT [N50,N70]

N50: G01 G91 X15 F800           (first block)
X-10 Y-10
X20
X-10 Y10
N70:                             (last block)
```

- The label is the block name.

```
N10 #RPT [[BEGIN],[END]]

[BEGIN] G01 G91 F800           (first block)
X-10 Y-10
X20
X-10 Y10
G90
[END]                             (last block)
```

Once the repetition is done, the execution resumes at the block after the one containing the #RPT instruction.

Considerations

The labels of the first and last blocks must be different. To repeat the execution of a single block, program as follows:

```
N10 #RPT [N10,N20,4]

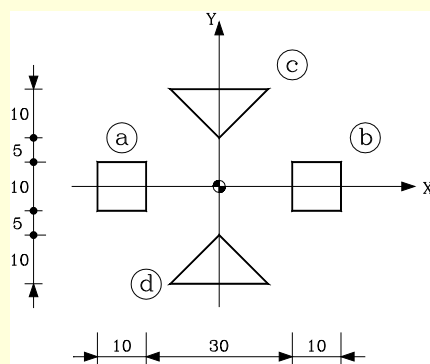
N10: G01 G91 F800           (first block)
N20:                             (last block)
```

21.

The execution of a block can also be repeated with the "NR" command. See "[Programming in ISO code.](#)" on page 34.

It is not possible to repeat a group of blocks that close a control loop if the opening of the control loop is not within the instructions being repeated.

```
N10 #RPT [N10,N20]
N10: $FOR P1=1,10,1
G0 XP1
$ENDFOR
G01 G91 F800
N20:
```



```
%PROGRAM
G00 X-25 Y-5
N10: G91 G01 F800          (Definition of profile "a")
X10
Y10
X -10
Y -10
G90
N20:
G00 X15
#RPT [N10, N20]          (Block repetition. Profile "b")
#RPT [[INIT], [END], 2] (Block repetition. Profiles "c" and "d")
M30

[INIT]
G1 G90 X0 Y10
G1 G91 X10 Y10
X -20
X10 Y-10
G73 Q180
[END]
```

21.

STATEMENTS AND INSTRUCTIONS
Programming statements



CNC 8060
CNC 8065

(REF. 1402)

21.1.18 Communication and synchronization between channels

Each channel may execute its own program simultaneously and independently from other channels. But, besides this, it can also communicate with other channels, transfer information or synchronize in specific points.

The communication takes place on the basis of a number of marks managed by the part-programs of each channel. These marks establish whether the channel is waiting to be synchronized or it may be synchronized, etc.

There are two different ways to synchronize, each offers a different solution.

- Using the #MEET instruction.

The easiest way to synchronize. It stops the execution in all the channels involved in the synchronization.

The set of marks being used are initialized after executing an M02 or an M30, after a reset or on power-up.

- Using the instructions #WAIT - #SIGNAL - #CLEAR.

This method is somewhat more complicated than the previous one, but more versatile. It does not stop the execution in all the channels in order to synchronize.

The set of marks being used are maintained after executing an M02 or an M30, after a reset or on power-up.

The synchronism marks of the two methods are independent from each other. The marks managed by the #MEET instruction neither affect nor are affected by the rest of the instructions.

Other ways to synchronize channels

The common arithmetic parameters can also be used to communicate and synchronize channels. By writing a certain value from a channel and later reading it from another channel, it is possible to set the condition to follow up on the execution of a program.

Accessing the variables of a channel from another channel can also be used as a way to communicate.

Swapping axes between channels also makes it possible to synchronize processes, because a channel cannot grab an axis until it has been released by another one.

CHANNEL 1	CHANNEL 2	CHANNEL 3
G1 F1000	X1=0 Y1=0 Z1=0	G1 F1000
S3000 M3	G1 F1000	X2=20 Z2=10
#FREE AX[Z]	#FREE AX[Z1]	#FREE AX[Z2]
(Frees the Z axis)	(Frees the Z1 axis)	(Frees the Z2 axis)
X30 Y0	G2 X1=-50 Y1=0 I-25	X2=100 Y2=50
#CALL AX [Z1,Z2]	#CALL AX [Z]	#CALL AX[Z2]
(It adds the Z1 and Z2 axes)	(Adds the Z axis)	(Recovers the Z2 axis)
X90 Y70 Z1=-30 Z2=-50	G1 X1=50 Z20	G0 X2=0 Y2=0 Z2=0
#FREE AX [Z1,Z2]	#FREE AX[Z]	M30
(Frees the Z1 and Z2 axes)	(Frees the Z axis)	
X0	X1=20	
#CALL AX [Z]	#CALL AX [Z1]	
(Recovers the Z axis)	(Recovers the Z1 axis)	
G0 X0 Y0 Z0	G0 X1=0 Y1=0 Z1=0	
M30	M30	

Consultation variables

The information about the status of the synchronization marks may be consulted using the following variables.

- MEET or WAIT type mark expected by the "n" channel from the "m" channel

V.[n].G.MEETCH[m]

V.[n].G.WAITCH[m]

Replace the letters "n" and "m" with the channel number.

21.

- Status of the MEET or WAIT type "m" mark in the "n" channel

V.[n].G.MEETST[m]

V.[n].G.WAITST[m]

#MEET

It activates the mark indicated in the channel and waits for it to be activated in the rest of the programmed channels.

This instruction, after activating the mark in its own channel, waits for it to also be active in the programmed channels before resuming the execution. Each channel has 100 marks that are numbered from 1 to 100.

Programming the same instruction in several channels, all of them stop and wait for the rest to reach the indicated point before they all resume the execution at the same time from that point on.

The programming format is:

```
#MEET [<mark>, <channel>, ...]
```

Parameter	Meaning
<mark>	Synchronization mark that is activated in the channel itself and must be activated in the rest of the channels before going on.
<channel>	Channel or channels where the same mark must be activated.

There is no need to include the number of its own channel in each instruction because the mark is activated when executing the #MEET instruction. However, it is recommended to program it in order to make the program more understandable.

Operation

Programming the same instruction in each channel, all of them are synchronized at that point and the execution resumes from there on. It works as follows.

- 1 It activates the mark selected in its own channel.
- 2 It waits for the mark to be activated in all the indicated channels.
- 3 After synchronizing the channels, it deletes the mark from its own channel and goes on executing the program.

Each channel stops on its #MEET. When the last one of them reaches the command and checks that all the marks are active, the process unlocks for all of them at the same time.

In the following example, it waits for mark ·5· to be active in channels ·1·, ·2· and ·3· to synchronize the channels and resume the execution.

CHANNEL 1	CHANNEL 2	CHANNEL 3
%PRG_1	%PRG_2	%PRG_3
...
...	#MEET [5,1,2,3]	...
#MEET [5,1,2,3]
...
...	...	#MEET [5,1,2,3]
M30	M30	M30

#WAIT

It waits for the mark to be activated in the indicated channel

The #WAIT instruction waits for the indicated mark to be active in the specified channels. If the mark is already active when executing the command, the execution is not interrupted and the program keeps running.

Each channel has 100 marks that are numbered from 1 to 100.

The programming format is:

```
#WAIT [<mark>, <channel>, ...]
```

Parameter	Meaning
<mark>	Synchronization mark waited for to be activated.
<channel>	Channel or channels that must activate the mark.

As opposed to the #MEET instruction, it does not activate the indicated mark of its own channel. The marks of the channel are activated using the instruction #SIGNAL.

21.

#SIGNAL

It activates the mark in its own channel

The #SIGNAL instruction activates the indicated marks in its own channel. Each channel has 100 marks that are numbered from 1 to 100. These marks correspond to the #WAIT instructions.

This instruction does not perform any wait; it goes on executing. Once synchronized, the marks are deactivated, if so wished, using the #CLEAR instruction.

The programming format is:

```
#SIGNAL [<mark>, ...]
```

Parameter	Meaning
<mark>	Synchronization marks that is activated in the channel.

#CLEAR

It clears the synchronism marks of the channel

This instruction activates the indicated marks in its own channel. If no marks are programmed, it deletes all of them.

The programming format is:

```
#CLEAR  
#CLEAR [<mark>, ...]
```

Parameter	Meaning
<mark>	Synchronization marks that is deleted in the channel.

In the following example, channels ·1· and ·2· wait for mark ·5· to be active in channel ·3· to synchronize. When mark ·5· is activated in channel ·3·, it resumes the execution in all three channels.

CHANNEL 1	CHANNEL 2	CHANNEL 3
%PRG_1	%PRG_2	%PRG_3
...
...	#WAIT [5,3]	...
#WAIT [5,3]
...	...	#SIGNAL [5]
...
...	...	#CLEAR [5]
M30	M30	M30

21.1.19 Movements of independent axes



This function has a specific manual. This manual that you are reading now only offers some information about this function. Refer to the specific documentation to obtain further information regarding the requirements and operation of the independent axes.

The CNC has the possibility of executing independent positioning and synchronization. For this type of movements, each CNC axis has an independent interpolator that keeps track of the current position on its own without depending on the tracking of the general interpolator of the CNC.

It is possible to execute an independent movement and general movement simultaneously. The result will be the sum of the two interpolators.

The CNC stores up to a maximum of two independent-motion instructions per axis. The rest of instructions sent when there are two pending execution imply a wait from the part-program.

Treatment of a rotary axis as an infinite axis.

Axis synchronization permits treating a rotary axis as an infinite axis making it possible to increase the feedback count of the axis indefinitely (without limits) regardless of the value of the module. This type of axis is activated when programming by adding the prefix ACCU to the name of the master axis. From the moment this is programmed, the CNC uses the variable V.A.ACCUDIST.xn that may be initialized at any time to follow up on the axis.

This feature is useful, for example, for a rotary axis or encoder that moves an endless conveyor belt on where the part is. The infinite axis treatment allows synchronizing the coordinate of the conveyor belt with an external event and, this way, count the moving distance of the part beyond the value of the module of the rotary axis that moves the belt.

Restrictions for the independent axes

Any axis of the channel may be moved independently using the associated instructions. However, this function presents the following restrictions.

- A spindle can only move independently when set in axis mode with the instruction #CAX . However, it can always be the master of a synchronization.
- A rotary axis may be of any module, but the lower limit must always be zero.
- A Hirth axis cannot move independently.

Synchronizing the interpolators

In order for the incremental movements to take the real coordinate of the machine into account, each interpolator must be synchronized with that real coordinate. The synchronization is done from the part-program using the instruction #SYNC POS.

Resetting the CNC synchronizes the theoretical coordinates of both interpolators with the real coordinate. These synchronizations will only be necessary when inserting instructions of both types of interpolators.

Every time the program is initiated or an MDI block is executed, the coordinate of the general interpolator of the CNC is synchronized and every new independent instruction (without any one pending) also synchronizes the coordinate of the independent interpolator.

Influence of the movements in block preparation

None of these blocks interrupt block preparation, but they do interrupt the interpolation. Therefore, it will not blend two blocks, there will be an intermediate one.

Positioning move (#MOVE)

The various types of positioning are programmed with the following instructions.

- | | |
|-----------|--|
| #MOVE | - Absolute positioning move. |
| #MOVE ADD | - Incremental positioning move. |
| #MOVE INF | - Infinite (endless) positioning move. |

The programming format for each of them is the following. Optional parameters are indicated between the <> characters.

```
#MOVE <ABS> [Xpos < ,Fn> < ,blend>]
#MOVE ADD [Xpos < ,Fn> < ,blend>]
#MOVE INF [X+/- < ,Fn> < ,blend>]
```

[Xpos] Axis and position to reach

Axis and position to reach. With #MOVE ABS it will be defined in absolute coordinates whereas with #MOVE ADD it will be defined in incremental coordinates.

The moving direction is determined by the coordinate or the increment programmed. For rotary axes, the moving direction is determined by the type of axis. If normal, via the shortest path; if unidirectional, in the preset direction.

[X+/-] Axis and moving direction

Axis (without coordinate) to position. The sign indicates the moving direction.

It is used with #MOVE INF to execute an endless (infinite) movement until the axis limit is reached or until the movement is interrupted.

[Fn] Positioning feedrate

Positioning feedrate.

Feedrate given in mm/min, inches/min or degrees/min.

Optional parameter. If not defined, it assumes the feedrate set by machine parameter POSFEED.

[blend] Dynamic blend with the next block

Optional parameter. The feedrate used to reach the position (dynamic blend with the next block) is defined by an optional parameter.

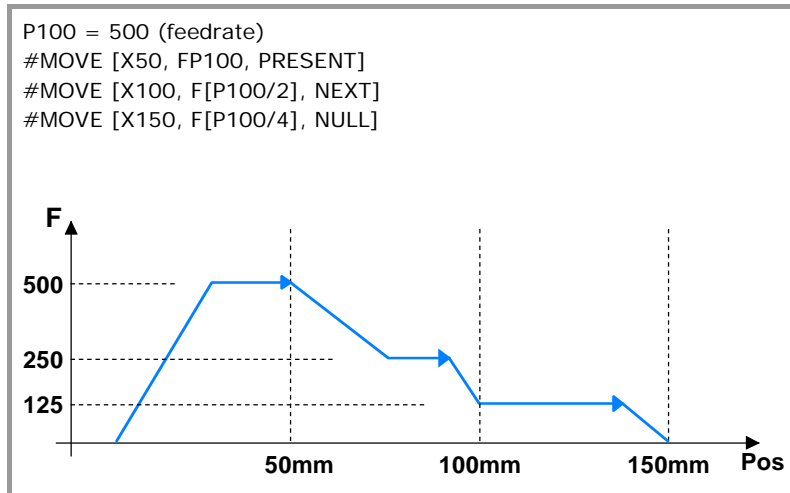
The feedrate used to reach the position is given by one of these elements:

[blend]	Type of dynamic blend
PRESENT	It reaches the indicated position at the positioning feedrate specified for the block itself.
NEXT	It reaches the indicated position at the positioning feedrate specified in the next block.
NULL	The indicated position is reached at zero feedrate.
WAITINPOS	The indicated position is reached at zero feedrate and it waits to be in position before executing the next block.

Programming this parameter is optional. If not programmed, the dynamic blend is carried out according to machine parameter ICORNER as follows.

ICORNER	Type of dynamic blend
G5	According to the setting for the PRESENT value.
G50	According to the setting for the NULL value.
G7	According to the setting for the WAITINPOS value.

21.



Synchronization move (#FOLLOW ON)

The activation and cancellation of the different types of synchronization are programmed with the following instructions.

- #FOLLOW ON - Activates the synchronization movement (real coordinates).
- #TFOLLOW ON - Activates the synchronization movement (theoretical coordinates).
- #FOLLOW OFF - Cancels the synchronization movement.

The programming format for each of them is the following. Optional parameters are indicated between the <> characters.

- #FOLLOW ON [master, slave, Nratio, Dratio <,synctype>]
- #TFOLLOW ON [master, slave, Nratio, Dratio <,synctype>]
- #FOLLOW OFF [slave]

Executing the #FOLLOW OFF instruction involves eliminating the synchronization speed of the slave. The axis will take some time to brake and the instruction will stay in execution during that time.

[master] Master axis

Name of the master axis.

To treat a rotary axis as an infinite axis making it possible to increase the feedback count of the axis indefinitely (without limits) regardless of the value of the module, program the master axis with the prefix ACCU. This way, the CNC does a follow-up of the axis through the variable V.A.ACCUDIST.xn.

[slave] Slave axis

Name of the slave axis.

[Nratio] Gear ratio (slave axis)

Numerator of the gear ratio. Turns of the slave axis.

[Dratio] Gear ratio (master axis)

Denominator of the gear ratio. Rotations of the master axis.

[synctype] Type of synchronization

Optional parameter. Indicator that determines whether it is a velocity or position synchronization type.

[synctype]	Type of synchronization
POS	It is a position synchronization.
VEL	It is a velocity synchronization.

Programming it is an option. If not programmed, it executes a velocity synchronization.

```
#FOLLOW ON [X, Y, N1, D1]
#FOLLOW ON [A1, U, N2, D1, POS]
#FOLLOW OFF [Y]
#FOLLOW ON [ACCUX, Y, N1, D1]
```

21.

STATEMENTS AND INSTRUCTIONS

Programming statements



CNC 8060
CNC 8065

(REF. 1402)

21.1.20 Electronic cams.



This function has a specific manual. This manual that you are reading now only offers some information about this function. Refer to the specific documentation to obtain further information regarding the requirements and operation of the electronic cams.

The electronic cam mode may be used to move a slave axis based on a positions-table or on a cam profile. If while executing a cam profile, a second cam profile is executed, the second one gets ready and waiting for the current execution to be over. Once the current cam profile has ended, it starts executing the second cam blending both profiles like when blending (joining) two positioning blocks. Executing the cam synchronization ending instruction (#CAM OFF) will end the current cam, but not immediately, it will do so the next time it goes through the end of the cam profile.

After executing the synchronization of the cam, no positioning moves of the independent axis (MOVE) are possible. It makes no sense to add another movement to the cam synchronization movement that would brake the synchronization already achieved.

Position - position cam

On this type of cam, it is possible to obtain non-linear electronic synchronization ratios between two axes. Thus, the position of the slave axis is synchronized with the position of the master axis using a cam profile.

Position - time cam

On this type of cam, it is possible to obtain other movement profiles different from trapezoidal or S-shaped ones.

Electronic-cam editor.

Before activating an electronic cam, it must have been defined at the cam editor which is accessed from the machine parameters. This editor offers a friendly assistance to analyze the behavior of the cam projected through graphically assisted data entry for speed, acceleration and jerk.

It is up to the user to select the parameters and the functions to design an electronic cam and must make sure that his design is coherent with the required specifications.

Activating and canceling the file cam from the part-program.

The data of a cam may be defined in a file that may be loaded from the CNC or from the PLC. When executing a cam from a file, the CNC reads its data dynamically and, consequently, the number of points to define the cam is unlimited. After selecting a file cam, it stays available until the cam table of the machine parameters is validated or the CNC is turned off.

To select or cancel a file cam, use the following instructions. The following instructions only defines the location of the cam; to activate it, use the instruction #CAM ON.

```
#CAM SELECT      - Select a file cam.
#CAM DESELECT    - Cancel the cam of a file.
```

The programming format for each of them is the following.

```
#CAM SELECT [cam, file]
#CAM DESELECT [cam]
```

Parameter.	Meaning.
cam	Cam number.
path/file	Name and path of the file containing the cam data.

```
#CAM SELECT [6, "C:\USERCAM\cam.txt"]
  (The CNC uses for cam ·6· the data defined in the cam.txt file)
#CAM DESELECT [6]
  (The CNC stops using for cam ·6· the data defined in the cam.txt file)
```

Activating and canceling the electronic cam (#CAM).

The activation and cancellation of the electronic cam is programmed with the following instructions.

- #CAM ON - Activates the cam (real coordinates).
- #TCAM ON - Activates the cam (theoretical coordinates).
- #CAM OFF - Cancel the electronic cam.

The programming format for each of them is the following. Optional parameters are indicated between the <> characters.

```
#CAM ON [cam, master/"TIME", slave, master_off, slave_off, range_master,
range_slave <,type>]
#TCAM ON [cam, master/"TIME", slave, master_off, slave_off, range_master,
range_slave <,type>]
#CAM OFF [slave]
```

Executing the #CAM OFF instruction involves eliminating the synchronization of the cam. Once this instruction has been programmed, the cam ends when reaching the end of its profile.

[cam] Cam number.

To activate a cam, it must have been previously defined at the cam editor, within the machine parameters.

[master/"TIME"] Master axis.

Name of the master axis, when it is a position cam. When programming the "TIME" command instead of an axis name, the cam is interpreted as being a time cam.

In a position cam, treating a rotary axis as an infinite axis making it possible to increase the feedback count of the axis indefinitely (without limits) regardless of the value of the module, program the master axis with the prefix ACCU. This way, the CNC does a follow-up of the axis through the variable V.A.ACCUDIST.xn.

```
#CAM ON [1, X, Y, 30, 0, 100, 100]
#CAM ON [1, ACCUX, Y, 30, 0, 100, 100]
#CAM ON [1, TIME, A2, 0, 0, 6, 3, ONCE]
#CAM OFF [Y]
```

[slave] Slave axis.

Name of the slave axis.

[master_off] Offset of the master axis or time offset.

In a position cam, this offset sets the position where the cam is activated. The offset is subtracted from the master axis position to calculate the entry position of the cam table.

In a time cam this offset may be used to set a time for the cam trigger.

[slave_off] Offset of the master axis.

The slave_off and range_slave values allow moving the slave axis positions outside the range of values set by the cam function.

[Range_master] Master axis activation scale or range.

The position cam is activated when the master axis is between the positions "master_off" and "master_off + range_master". The cam only regulates the slave axis position within this range.

In a time cam, this parameter sets the time range or total duration of the cam.

[Range_slave] Slave axis activation scale or range.

The cam applies it to the slave axis when the slave axis is between "slave_off" and "slave_off + range_slave".

21.

[type] Cam type.

Depending on the execution mode, the time cams and the position cams may be of two different types; i.e. periodic or non-periodic. This selection is made using the following commands:

[type]	Meaning.
ONCE	Non-periodic cam. This mode maintains the synchronization for the range defined for the master axis. If the master axis moves backwards or if it is a module, the slave axis will keep on executing the cam profile until the cancellation is programmed.
CONT	Periodic cam. In this mode, when reaching the end of the range of the master axis, it calculates the offset to execute the cam again shifted in the amount of that range. In other words, identical cams are executed along the path of the master axis.

If the master axis is a rotary module and the cam definition range is that module, the two execution modes are equivalent. Either mode maintains synchronization until the #CAM OFF instruction is executed. When reaching that instruction, the execution of the cam will end the next time the end of the cam profile is reached.

21.1.21 Additional programming instructions

21.

#FLUSH

Interruption of block preparation

The CNC reads several blocks ahead (preparation) of the one being executed in order to calculate in advance the path to follow.

The #FLUSH instruction interrupts this block preparation in advance, executes the last prepared blocs, synchronizes the preparation and execution of blocks and then goes on with the program. When resuming, it begins preparing blocks again in advance.

The programming format is:

```
#FLUSH
```

The blocks have data that is analyzed when it is read; to analyze it when it is executed, then use the #FLUSH instruction.

This instruction is useful to evaluate a "block skip" condition at the time of the execution.

```
...
N110 #FLUSH
/N120 G01 X100
...
```

It must be borne in mind that interrupting block preparation may result in compensated paths different from the one programmed, undesired joints when working with very short moves, jerky axis movements, etc.

#WAIT FOR

Wait for an event

This instruction interrupts block preparation until the programmed condition is met. This instruction does not synchronize block preparation and execution; for synchronizing, use function #FLUSH.

The programming format is:

```
#WAIT FOR [<condition>]
```

```
P100=1
#FLUSH
#WAIT FOR [P100==0]
```

It is possible to compare numbers, parameters or arithmetic expressions whose result is a number.

21.2 Flow controlling instructions

21.2.1 Jump to a block (\$GOTO)

\$GOTO N<EXPRESIÓN>
\$GOTO [<ETIQUETA>]

One of the following parameters is defined in this instruction:

<expression>	It may be a number, parameter or arithmetic expression whose result is a number.
<label>	It may be a sequence of up to 14 characters consisting of uppercase and lowercase letters and numbers (neither blank spaces nor quote marks allowed).

This instruction provokes a jump to the block defined with "N<expression>" or "[<label>]", that may be defined at a point before or after the \$GOTO instruction. Program execution continues after the jump at the indicated block.

The \$GOTO instruction may be programmed in two ways:

- With a block number.

In these blocks that are the target of a jump, the label must be programmed followed by "·".

Destination	N<number>·
Call	\$GOTO N<number> or N<number>·

- With a label.

Destination	[<label>]
Call	\$GOTO [<label>]

The call instruction and the destination block must be in the same program or subroutine. There cannot be a jump to a subroutine or between subroutines.

N10 \$GOTO N60	N40·	N10 \$GOTO [LABEL]
...
N60· ...	N90 \$GOTO N40·	N40 [LABEL]

There cannot be jumps to blocks contained in another instruction (\$IF, \$FOR, \$WHILE, etc.)

Although the flow controlling instructions must be programmed alone in the block, the \$GOTO instruction may added to an \$IF instruction in the same block. This way, it is possible to exit the blocks contained in an instruction (\$IF, \$FOR, \$WHILE, etc.) without having to end the loop.

```
N10 P0=10
N20 $WHILE P0<=10
N30 G01 X[P0*10] F400
N40 P0=P0-1
N50 $IF P0==1 $GOTO N100
N60 $ENDWHILE
N100: G00 Y30
M30
```

21.

STATEMENTS AND INSTRUCTIONS
Flow controlling instructions

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

21.2.2 Conditional execution (\$IF)

\$IF <CONDITION>... \$ENDIF

The following parameter is defined in this instruction:

<code><condition></code>	It may be a comparison between two numbers, parameters or arithmetic expressions whose result is a number.
--------------------------------	--

This instruction analyzes the programmed condition.

- If the condition is true, it executes the blocks contained between the \$IF and \$ENDIF instruction.
- If the condition is false, the execution continues at the block after \$ENDIF.

```
...
N20 $IF P1==1
N30...
N40...
N50 $ENDIF
N60 ...
```

If P1 is equal to 1, it will execute blocks N30 through N40.
If P1 is other than 1, the execution continues at N60.

The \$IF instruction always ends with a \$ENDIF, except when adding a \$GOTO instruction, in which case, it must NOT be programmed.

```
...
N20 $IF P1==1 $GOTO N40
N30...
N40: ...
N50...
```

If P1 is equal to 1, the execution continues at block N40.
If P1 is other than 1, the execution continues at N30.

As an option, the \$ELSE and \$ELSEIF instructions may be inserted between \$IF and \$ENDIF.

\$IF <CONDITION> ... \$ELSE ... \$ENDIF

This instruction analyzes the programmed condition.

- If the condition is true, it executes the blocks contained between \$IF and \$ELSE and the execution continues at the block after \$ENDIF.
- If the condition is false, it executes the blocks contained between \$ELSE and \$ENDIF.

```
N20 $IF P1==1
N30...
N40...
N50 $ELSE
N60...
N70...
N80 $ENDIF
N90 ...
```

If P1 is equal to 1, it will execute blocks N30 through N40. The execution continues at N90.
If P1 other than 1, the execution continues at N50.

21.

STATEMENTS AND INSTRUCTIONS
Flow controlling instructions



CNC 8060
CNC 8065

(REF. 1402)

\$IF <CONDITION1>... \$ELSEIF<CONDITION2>... \$ENDIF

This instruction analyzes the following programmed conditions.

- If <condition1> is true, it executes the blocks contained between \$IF and \$ELSEIF.
- If <condition1> is false, it analyzes <condition2>. If true, it executes the blocks contained between \$ELSEIF and \$ENDIF (or the next \$ELSEIF if any).
- If all the conditions are false, the execution continues at the block after \$ENDIF.

As many \$ELSEIF instructions as necessary may be programmed.

```
N20 $IF P1==1
N30...
N40...
N50 $ELSEIF P2==[-5]
N60...
N70 $ELSE
N80...
N90 $ENDIF
N100 ...
```

If P1 is equal to 1, it will execute blocks N30 through N40. The execution continues at N100.

- If P1 is other than 1 and P2 is equal to -5, it executes block N60. The execution continues at N100.
- If P1 is other than 1 and P2 is other than -5, it executes block N80 and the execution continues at N100.

An \$ELSE instruction may also be included. In this case, if all the conditions are false, it will execute the blocks contained between \$ELSE and \$ENDIF.

21.

21.2.3 Conditional execution (\$SWITCH)

```
$SWITCH <EXPRESSION1>... $CASE<EXPRESSION2>...
$ENDSWITCH
```

The following parameters are defined in this instruction:

<expression> It may be a number, parameter or arithmetic expression whose result is a number.

This instruction calculates the result of <expression1> and executes the blocks contained between the \$CASE instruction, whose <expression2> has the same value as the calculated result and the corresponding \$BREAK instruction.

The \$SWITCH instruction always ends with a \$ENDSWITCH.

The \$CASE instruction always ends with a \$BREAK. As many \$CASE instructions as necessary may be programmed.

As an option, a \$DEFAULT instruction may be inserted in such a way that if the result of <expression1> does not coincide with the value of any <expression2>, it executes the blocks contained between \$DEFAULT and \$ENDSWITCH.

```
N20 $SWITCH [P1+P2/P4]
N30 $CASE 10
N40...
N50...
N60 $BREAK
N70 $CASE [P5+P6]
N80...
N90...
N100 $BREAK
N110 $DEFAULT
N120...
N130...
N140 $ENDSWITCH
N150...
```

If the result of the expression [P1+P2/P4].

- Is "10", it executes blocks N40 through N50. The execution continues at N150.
- Is equal to [P5+P6], it executes blocks N80 through N90. The execution continues at N150.
- Is other than "10" and [P5+P6], it executes blocks N120 and N130. The execution continues at N150.

21.

21.2.4 Block repetition (\$FOR)

\$FOR <N> = <EXPR1>, <EXPR2>, <EXPR3>... \$ENDFOR

The following parameters are defined in this instruction.

<n>	It may be an arithmetic parameter of a write variable.
<expr>	It may be a number, parameter or arithmetic expressing whose result is a number.

When executing this instruction, <n> takes the value of <expr1> and it changes its value up to the value of <expr2>, in steps indicated by <expr3>. At each step, it executes the blocks contained between \$FOR and \$ENDFOR.

```
...
N20 $FOR P1=0,10,2
N30...
N40...
N50...
N60 $ENDFOR
N70...
```

It executes blocks N30 through N50 from P1=0 until P1=10, in steps of 2 (thus 6 times).

```
...
N12 $FOR V.P.VAR_NAME=20,15,-1
N22...
N32...
N42 $ENDFOR
N52...
```

It executes blocks N22 through N32 in steps of -1 (thus 5 times) from V.P.VAR_NAME=20 to V.P.VAR_NAME=15.

The \$BREAK instruction lets ending block repetition even if the stop condition is not met. The execution of the program will continue at the block after \$ENDFOR.

```
...
N20 $FOR P1= 1,10,1
N30...
N40 $IF P2==2
N50 $BREAK
N60 $ENDIF
N70...
N80 $ENDFOR
...
```

Block repetition stops if P1 is greater than 10, or if P2 = 2.

The \$CONTINUE instruction starts the next repetition even when the current one has not finished. The blocks programmed after \$CONTINUE up to \$ENDFOR will be ignored in this repetition.

21.

21.2.5 Conditional block repetition (\$WHILE)

\$WHILE <CONDITION>... \$ENDWHILE

The following parameter is defined in this instruction:

<condition> It may be a comparison between two numbers, parameters or arithmetic expressions whose result is a number.

While the condition is true, it executes the blocks contained between \$WHILE and \$ENDWHILE. The condition is analyzed at the beginning of each new repetition.

```
...
N20 $WHILE P1<= 10
N30 P1=P1+1
N40...
N50...
N60 $ENDWHILE
...
```

While P1 is smaller than or equal to 10, it executes blocks N30 through N50.

The \$BREAK instruction lets ending block repetition even if the stop condition is not met. The execution of the program will continue at the block after \$ENDWHILE.

```
...
N20 $WHILE P1<= 10
N30...
N40 $IF P2==2
N50 $BREAK
N60 $ENDIF
N70...
N80 $ENDWHILE
...
```

Block repetition stops if P1 is greater than 10, or if P2 = 2.

The \$CONTINUE instruction starts the next repetition even when the current one has not finished. The blocks programmed after \$CONTINUE up to \$ENDWHILE will be ignored in this repetition.

```
...
N20 $WHILE P1<= 10
N30...
N40 $IF P0==2
N50 $CONTINUE
N60 $ENDIF
N70...
N80...
N80 $ENDWHILE
...
```

If P0=2, it ignores blocks N70 through N80 and it starts a new repetition at N20.

21.

STATEMENTS AND INSTRUCTIONS
Flow controlling instructions



CNC 8060
CNC 8065

(REF. 1402)

21.2.6 Conditional block repetition (\$DO)

\$DO ... \$ENDDO <CONDITION>

The following parameter is defined in this instruction:

<condition> It may be a comparison between two numbers, parameters or arithmetic expressions whose result is a number.

While the condition is true, it repeats the execution of the blocks contained between \$DO and \$ENDDO. The condition is analyzed at the end of each repetition, therefore the group of blocks is executed at least once.

```
...
N20 $DO
N30 P1=P1+1
N40...
N50...
N60 $ENDDO P1<=10
N70...
```

Blocks N30 through N50 are executed while P1 is smaller than or equal 10.

The \$BREAK instruction lets ending block repetition even if the stop condition is not met. The execution of the program continues at the block after \$ENDDO.

```
...
N20 $DO
N30...
N40 $IF P2==2
N50 $BREAK
N60 $ENDIF
N70...
N80 $ENDDO P1<= 10
...
```

Block repetition stops if P1 is greater than 10, or if P2 = 2.

The \$CONTINUE instruction starts the next repetition even when the current one has not finished. The blocks programmed after \$CONTINUE up to \$ENDDO will be ignored in this repetition.

```
...
N20 $DO
N30...
N40 $IF P0==2
N50 $CONTINUE
N60 $ENDIF
N70...
N80...
N80 $ENDDO P1<= 10
...
```

If P0=2, it ignores blocks N70 through N80 and it starts a new repetition at N20.

21.

STATEMENTS AND INSTRUCTIONS
Flow controlling instructions

FAGOR 

CNC 8060
CNC 8065

(REF. 1402)

21.

STATEMENTS AND INSTRUCTIONS

Flow controlling instructions



CNC 8060
CNC 8065

(REF. 1402)

22.1 Understanding how variables work.

Accessing the variables.

The internal CNC variables may be accessed from the part program, MDI/MDA, PLC and from any application or external interface (for example FGUIM). Each variable must indicate whether it can only be read or read and written.

Access the variables from a part-program. Accessing during execution or during block preparation.

The CNC reads several blocks ahead of the one being executed in order to calculate in advance the path to follow. This prior reading is known as "block preparation".

The CNC checks certain variables during block preparation and others during execution. The variables that use the execution value interrupt block preparation temporarily and the CNC resumes it when it is done reading/writing the variable. Accessing the variables from the PLC or from an external interface never interrupts block preparation.

Be careful with the variables that interrupt block preparation because when they are inserted between machining blocks with compensation may cause undesired profiles. Interrupting block preparation may result in compensated paths different from the one programmed, undesired joints when working with very short moves, etc.

In any case, it is possible to use the #FLUSH instruction to force the evaluation of a variable when it is being executed. This instruction interrupts block preparation in advance, executes the last prepared blocs, synchronizes the preparation and execution of blocks and then goes on with the execution of the program and block preparation.

Accessing the variables. from the PLC. Synchronous or asynchronous access.

PLC access to the variable, both for reading and writing, may be either synchronous or asynchronous. A synchronous access is resolved immediately whereas an asynchronous access takes several PLC cycles to resolve.

The tool variables will be read asynchronously when the tool is neither the active one nor in the magazine. The tool variables will be written asynchronously whether the tool is the active one or not.

Example of how to access asynchronous variables.

Reading of the radius value of offset ·1· of tool ·9· when it is not in the tool magazine.

```
<condition> AND NOT M11 = CNCRD (TM.TORT.[9][1], R11, M11)
```

The PLC activates mark M11 when the operation begins and keeps it active until the operation ends.

```
DFD M11 AND CPS R11 EQ 3 = ...
```

It waits for the consultation to end before evaluating the data.

22.

CNC VARIABLES.
Understanding how variables work.

Example of how to access synchronous variables.

Reading the actual (real) feedrate.

<condition> = CNCRD (G.FREAL, R12, M12)

The PLC activates mark M12 when the operation begins and keeps it active until the operation ends.

CPS R12 GT 2000 = ...

There is no need to wait for consulting the data because the synchronous variables are resolved immediately.

It resets the clock enabled by the PLC with the value contained in register R13.

<condition> = CNCWR (R13, PLC.TIMER, M13)

Accessing the variables. from the PLC. Accessing numeric variables

When the PLC accesses numeric variables that can have decimals, the values will be given in PLC units.



CNC 8060
CNC 8065

(REF: 1402)

22.1.1 Accessing numeric variables from the PLC.

When the PLC accesses numeric variables that can have decimals, the values will be given as follows.

- The coordinates will be given in ten-thousandths if they are in mm or hundred-thousandths if they are in inches.

Units.	Reading from the PLC.
1 millimeter.	10000.
1 inch.	100000.
1 degree.	10000.

- The feedrate of the axes is given in tenth-thousandths if mm or in hundred-thousandths if inches.

Units.	Reading from the PLC.
1 millimeter/minute.	10000.
1 inch/minute.	100000.
1 degree/minute.	10000.

- The spindle speed will be given in ten-thousandths.

Units.	Reading from the PLC.
G97. 1 rpm.	10000.
G96. 1 meter/minute.	10000.
G96. 1 foot/minute.	10000.
G192. 1 rpm.	10000.
M19. 1 degree/minute.	10000.

- The percentages will be given with the real value, in tenths or in hundredths depending on the variable. If not indicated otherwise, the PLC will read the actual value. If not so, it will indicate if the variable will be read in tenths (x10) or in hundredths (x100).

Units.	Reading from the PLC.
1 %.	1.
1 % (x10).	10.
1 % (x100).	100.

- Time will be given in thousandths.

Units.	Reading from the PLC.
1 second.	1000.

- Voltage will given as follows. The variables associated with the machine parameter table return the actual value (in millivolts). For the rest of the variables (in volts), the reading will appear in ten-thousandths.

Units.	Reading from the PLC.
1 volt.	10000.

22.

CNC VARIABLES.

Understanding how variables work.



CNC 8060
CNC 8065

(REF: 1402)

22.2 Variables in a single-channel system.

The generic mnemonic associated with the variables is written as follows.

```
(V){prefix}.{variable}
(V){prefijo}.{variable}.{eje/cabecal}
```

The –V.– indicator.

Programming the -V.- indicator depends on where the variable is used. To access the variables from the part-program or in MDI/MDA mode, the mnemonic begins with the indicator -V.- Accessing the variables from the PLC or from an external interface requires NOT using the -V.- indicator.

All mnemonics in this manual show this indicator as (V.), indicating that it must only be programmed when necessary.

Mnemoni.	Part-program. MDI/MDA mode.	PLC. External interface.
(V.)MPG.NAXIS	V.MPG.NAXIS	MPG.NAXIS

Prefixes of the variables.

The prefix must always be programmed. Prefixes make it possible to easily identify the group the variable belongs to.

Prefix.	Meaning.
A	Axis and/or spindle variables.
C	Canned cycle or subroutine calling parameters.
E	Interface related variables.
G	General variables.
MPA	Variables related to axis and/or spindle machine parameters.
MPG	Variables related to general machine parameters.
MPK	Variables related to kinematic machine parameters.
MPM	Variables related to machine parameters for M functions.
MPMAN	Variables related to machine parameters for JOG mode.
MTB	Variables related to OEM machine parameters.
P	User local variables.
PLC	PLC related variables.
S	User global variables.
SP	Spindle related variables.
TM	Variables related to tools or tool magazines.

Axis and spindle variables.

Axis and spindle variables are identified with the prefix –A.–. When these variables refer to a spindle, they may also be accessed with the prefix –SP.–.

```
(V.)A.{variable}.{axis/spindle}
(V.)SP.{variable}.{spindle}
(V.)SP.{variable}
```

The variables of the axis machine parameters (–MPA.– prefix) can also be accessed using the –SP– prefix when referring to a spindle.

```
(V.)MPA.{variable}.{axis/spindle}
(V.)SP.{variable}.{spindle}
(V.)SP.{variable}
```

22.

CNC VARIABLES.
Variables in a single-channel system.



CNC 8060
CNC 8065

(REF: 1402)

Identifying the axes and the spindles in the variables.

In these variables one must indicate which axis or spindle they refer to. The axis may be referred to by its name or logic number; the spindle may be referred to by its name, logic number or index in the spindle system.

In these variables one must indicate which axis or spindle they refer to. In variables with the prefix -A.- and -MPA.-, the axes and the spindles are identified with their name or logic number. In variables with the prefix -SP.-, the spindles are identified with their name or spindle index. If no spindle is selected in the variables with -SP.- prefix, the variable refers to the master spindle.

Mnemoni.	Meaning when the variable is executed by the part-program, the MDI/MDA mode or the PLC.
V.MPA.variable.Z V.A.variable.Z	Z axis.
V.MPA.variable.S V.A.variable.S V.SP.variable.S	Spindle S.
V.MPA.variable.4 V.A.variable.4	Axis or spindle with logic number -4-.
V.SP.variable.2	Spindle with index -2- in the system.
V.SP.variable	Master spindle.

Mnemoni.	Meaning when the variable is executed by an external interface.
V.MPA.variable.Z V.A.variable.Z	Z axis.
V.MPA.variable.S V.A.variable.S V.SP.variable.S	Spindle S.
V.MPA.variable.4 V.A.variable.4	Axis with logic number -4-.
V.SP.variable.2	Spindle with index -2- in the system.
V.SP.variable	Master spindle.

The logic number of the axes is determined by the order in which they have been defined in the machine parameter table (AXISNAME). The first axis of the table will be logic axis -1- and so on.

The logic number of the spindles is determined by the order in which they have been defined in the machine parameter table (NAXIS + SPDLNAME). The logic numbering of the spindles continues from the last logic axis; hence, in a 5-axis system, the first spindle of the table will be logic spindle -6- and so on.

The index of a spindle in the system is determined by the order in which they have been defined in the machine parameter table (SPDLNAME). The index of the first spindle of the table will be -1- and so on.

AXISNAME	SPDLNAME	Logic order.	Index of the spindle in the system.
AXISNAME 1		Logic number 1.	
AXISNAME 2		Logic number 2.	
AXISNAME 3		Logic number 3.	
AXISNAME 4		Logic number 4.	
AXISNAME 5		Logic number 5.	
	SPDLNAME 1	Logic number 6.	Index 1.
	SPDLNAME 2	Logic number 7.	Index 2.

22.

CNC VARIABLES.

Variables in a single-channel system.


FAGOR
CNC 8060
CNC 8065

(REF: 1402)

Variables of the master spindle.

In a multi-spindle system, the master spindle is the main spindle, the one receiving the commands when no specific spindle is mentioned. In a single-spindle channel, that will be its master spindle.

The variables of the master spindle are identified with the prefix –SP.– but without indicating the spindle. These variables may be used to access the data of the master spindle without knowing its name or logic number. These variables are meant for displaying data and programming cycles.

22.

CNC VARIABLES.
Variables in a single-channel system.



CNC 8060
CNC 8065

(REF: 1402)

22.3 Variables in a multi-channel system.

The generic mnemonic associated with the variables is written as follows.

```
(V.)[channel].{prefix}.{variable}
(V.)[canal].{prefix}.{variable}.{axis/spindle}
```

The -V.- indicator.

Programming the -V.- indicator depends on where the variable is used. To access the variables from the part-program or in MDI/MDA mode, the mnemonic begins with the indicator -V.- Accessing the variables from the PLC or from an external interface requires NOT using the -V.- indicator.

All mnemonics in this manual show this indicator as (V.), indicating that it must only be programmed when necessary.

Mnemoni.	Part-program. MDI/MDA mode.	PLC. External interface.
(V.)[2].MPG.NAXIS	V.[2].MPG.NAXIS	[2].MPG.NAXIS

Programming the channel.

Programming the channel makes it possible to access the variables of the channel itself or another channel from a channel. The first channel is identified with the number 1, "0" is not a valid number.

Programming the channel number is optional; if not programmed, it works as follows depending on who executes the variable. The following table does not apply to the axis and spindle variables.

Where they are executed.	Meaning when no channel has been programmed.
Part-program. MDI/MDA mode.	Channel that is executing the variable.
PLC	First channel or main channel.
External interface.	Active channel.

Prefixes of the variables.

The prefix must always be programmed. Prefixes make it possible to easily identify the group the variable belongs to.

Prefix.	Meaning.
A	Axis and/or spindle variables.
C	Canned cycle or subroutine calling parameters.
E	Interface related variables.
G	General variables.
MPA	Variables related to axis and/or spindle machine parameters.
MPG	Variables related to general machine parameters.
MPK	Variables related to kinematic machine parameters.
MPM	Variables related to machine parameters for M functions.
MPMAN	Variables related to machine parameters for JOG mode.
MTB	Variables related to OEM machine parameters.
P	User local variables.
PLC	PLC related variables.

22.

CNC VARIABLES.

Variables in a multi-channel system.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

Prefix.	Meaning.
S	User global variables.
SP	Spindle related variables.
TM	Variables related to tools or tool magazines.

Axis and spindle variables.

Axis and spindle variables are identified with the prefix –A.–. When these variables refer to a spindle, they may also be accessed with the prefix –SP.–.

(V.)[channel].A.{variable} .{axis/spindle}
 (V.)[channel].SP.{variable} .{spindle}
 (V.)[channel].SP.{variable}

The variables of the axis machine parameters (–MPA.– prefix) can also be accessed using the –SP– prefix when referring to a spindle.

(V.)[channel].MPA.{variable} .{axis/spindle}
 (V.)[channel].SP.{variable} .{spindle}
 (V.)[channel].SP.{variable}

Identifying the axes and the spindles in the variables.

In these variables one must indicate which axis or spindle they refer to. In variables with the prefix –A.– and –MPA.–, the axes and the spindles are identified with their name, logic number or index in the channel. In variables with the prefix –SP.–, the spindles are identified with their name, index in the channel or spindle index. If no spindle is selected in the variables with -SP- prefix, the variable refers to the master spindle.

Mnemoni.	Meaning when the variable is executed by the part-program, the MDI/MDA mode or the PLC.
V.MPA.variable.Z V.A.variable.Z	Z axis.
V.MPA.variable.S V.A.variable.S V.SP.variable.S	Spindle S.
V.MPA.variable.4 V.A.variable.4	Axis or spindle with logic number -4-.
V.[2].MPA.variable.4 V.[2].A.variable.4	Axis with index -4- in the channel -2-.
V.SP.variable.2	Spindle with index -2- in the system.
V.[2].SP.variable.1	Spindle with index -1- in the channel -2-.
V.SP.variable	Master spindle of the channel. If the variable is executed by the PLC, spindle master of the first channel.
V.[2].SP.variable	Master spindle of channel -2-.

Mnemoni.	Meaning when the variable is executed by an external interface.
V.MPA.variable.Z V.A.variable.Z	Z axis.
V.MPA.variable.S V.A.variable.S V.SP.variable.S	Spindle S.
V.MPA.variable.4 V.A.variable.4	Axis with logic number -4-.
V.[2].MPA.variable.4 V.[2].A.variable.4	Axis with index -4- in the channel -2-.
V.SP.variable.2	Spindle with index -2- in the active channel.
V.[2].SP.variable.1	Spindle with index -1- in the channel -2-.
V.SP.variable	Master spindle of the active channel.
V.[2].SP.variable	Master spindle of channel -2-.

22.

CNC VARIABLES.
Variables in a multi-channel system.



CNC 8060
CNC 8065

(REF: 1402)

When referring to the axis or spindle by its name, programming the channel they are in is not a determining factor; thus, programming them in this case is irrelevant. When programming the channel, if the axis or spindle is not in it, its programming is ignored.

The logic number of the axes is determined by the order in which they have been defined in the machine parameter table (AXISNAME). The first axis of the table will be logic axis -1- and so on.

The logic number of the spindles is determined by the order in which they have been defined in the machine parameter table (NAXIS + SPDLNAME). The logic numbering of the spindles continues from the last logic axis; hence, in a 5-axis system, the first spindle of the table will be logic spindle -6- and so on.

The index of a spindle in the system is determined by the order in which they have been defined in the machine parameter table (SPDLNAME). The index of the first spindle of the table will be -1- and so on.

AXISNAME	SPDLNAME	Logic order.	Index of the spindle in the system.
AXISNAME 1		Logic number 1.	
AXISNAME 2		Logic number 2.	
AXISNAME 3		Logic number 3.	
AXISNAME 4		Logic number 4.	
AXISNAME 5		Logic number 5.	
	SPDLNAME 1	Logic number 6.	Index 1.
	SPDLNAME 2	Logic number 7.	Index 2.

The index of an axis in the channel is determined by the order in which they have been defined in the machine parameter table (CHAXISNAME). The index of the first axis of the table will be -1- and so on.

The index of a spindle in the channel is determined by the order in which they have been defined in the machine parameter table (CHSPDLNAME). The index of the first spindle of the table will be -1- and so on.

CHAXISNAME CHSPDLNAME	Index of the axis in the channel.	Index of the spindle in the channel.
CHAXISNAME 1	Index 1.	
CHAXISNAME 2	Index 2.	
CHAXISNAME 3	Index 3.	
CHSPDLNAME 1		Index 1.
CHSPDLNAME 2		Index 2.

Variables of the master spindle.

In a multi-spindle system, the master spindle is the main spindle of the channel, the one receiving the commands when no specific spindle is mentioned. Each channel has a master spindle. In a single-spindle channel, that will always be the master spindle.

The variables of the master spindle are identified with the prefix -SP.- but without indicating the spindle. These variables may be used to access the data of the master spindle without knowing its name or logic number. These variables are meant for displaying data and programming cycles.

Programming the channel number is optional; if not programmed, it works as follows depending on who executes the variable.

Where they are executed.	Meaning when no channel has been programmed.
Part-program. MDI/MDA mode.	Channel that is executing the variable.
PLC	First channel or main channel.
External interface.	Active channel.

Syntax.

·nb· Number of the tandem pair

V.MPG.TSLAVEAXIS[2]	Second tandem pair.
---------------------	---------------------

(V.)MPG.TORQDIST[nb]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Tandem pair [nb]. Torque distribution (percentage required from the master motor).

By torque distribution, we mean the torque percentage supplied by each motor to obtain the total necessary torque in the tandem. This variable shows the percentage of the total torque to be supplied by the master axis. The difference between this value and 100% will be the percentage to be supplied by the slave motor.

Syntax.

·nb· Number of the tandem pair

V.MPG.TORQDIST[2]	Second tandem pair.
-------------------	---------------------

Remarks.

The reading from the PLC will be given in hundredths (x100); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·1000·.

(V.)MPG.PRELOAD[nb]

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

The variable returns the execution value; reading it interrupts block preparation.

Tandem pair [nb]. Preload between both motors.

Preload is the torque difference to be applied between the master motor and the slave motor. The preload sets a traction between both motors in order to eliminate the backlash when the tandem is in rest position. This variable shows the percentage of the rated torque of the master motor that is applied as preload.

If the variable returns a ·0· value, it means that preload is disabled.

Syntax.

·nb· Number of the tandem pair

V.MPG.PRELOAD[2]	Second tandem pair.
------------------	---------------------

Remarks.

The reading from the PLC will be given in hundredths (x100); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·1000·.

(V.)MPG.PRELFITI[nb]

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

The variable returns the execution value; reading it interrupts block preparation.

Tandem pair [nb]. Filter time to apply the preload.

This filter sets the time during which preload is applied gradually. If the variable returns a ·0· value, it means that the filter is disabled.

Syntax.

·nb· Number of the tandem pair

V.MPG.PRELFITI[2]	Second tandem pair.
-------------------	---------------------

22.**CNC VARIABLES.**

Variables related to general machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to general machine parameters.

(V.)MPG.TPROGAIN[nb]

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

The variable returns the execution value; reading it interrupts block preparation.

Tandem pair [nb]. Proportional gain (Kp) for the tandem.

The proportional controller generates an output proportional to the torque error between the two motors. If the variable returns a -0- value, it means that no proportional gain is applied.

Syntax.

·nb· Number of the tandem pair

V.MPG.TPROGAIN[2]	Second tandem pair.
-------------------	---------------------

Remarks.

The reading from the PLC will be given in hundredths (x100); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·1000·.

(V.)MPG.TINTIME[nb]

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

The variable returns the execution value; reading it interrupts block preparation.

Tandem pair [nb]. Integral gain (Ki) for the tandem.

The integral controller generates an output proportional to the integral of the torque error between the two motors. If the variable returns a -0- value, it means that no integral gain is applied.

Syntax.

·nb· Number of the tandem pair

V.MPG.TINTIME[2]	Second tandem pair.
------------------	---------------------

(V.)MPG.TCOMPLIM[nb]

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

The variable returns the execution value; reading it interrupts block preparation.

Tandem pair [nb]. Compensation limit.

Syntax.

·nb· Number of the tandem pair

V.MPG.TCOMPLIM[2]	Second tandem pair.
-------------------	---------------------

Remarks.

The reading from the PLC will be given in hundredths (x100); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·1000·.

CONFIGURATION OF A GANTRY AXIS..**(V.)MPG.MASTERAXIS[nb]**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Gantry axis [nb]. Logic number of the master axis.

If no axis has been defined, the variable will return a -0- value.



CNC 8060
CNC 8065

(REF: 1402)

Syntax.

·nb· Number of the gantry pair

V.MPG.MASTERAXIS[2]	Second gantry pair.
---------------------	---------------------

(V.)MPG.SLAVEAXIS[nb]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Gantry axis [nb]. Logic number of the slave axis.

If no axis has been defined, the variable will return a ·0· value.

Syntax.

·nb· Number of the gantry pair

V.MPG.SLAVEAXIS[2]	Second gantry pair.
--------------------	---------------------

(V.)MPG.WARNCOUPE[nb]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Gantry axis [nb]. Difference between the following errors of both axes for issuing a warning.

This variable shows the maximum difference allowed between the following errors of both axes for issuing a warning.

Syntax.

·nb· Number of the gantry pair

V.MPG.WARNCOUPE[2]	Second gantry pair.
--------------------	---------------------

(V.)MPG.MAXCOUPE[nb]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Gantry axis [nb]. Maximum difference allowed between the following errors of both axes.

This variable shows the maximum difference allowed between the following errors of both axes.

Syntax.

·nb· Number of the gantry pair

V.MPG.MAXCOUPE[2]	Second gantry pair.
-------------------	---------------------

(V.)MPG.DIFFCOMP[nb]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Gantry axis [nb]. Compensate for the coordinate (position) difference between the two axes after G74.

Syntax.

·nb· Number of the gantry pair

V.MPG.DIFFCOMP[2]	Second gantry pair.
-------------------	---------------------

22.**CNC VARIABLES.**

Variables related to general machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)MPG.MAXDIFF[nb]

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Gantry axis [nb]. Maximum position value (coordinate) difference between both axes to compensate for.

Syntax.

·nb· Number of the gantry pair

V.MPG.MAXDIFF[2] Second gantry pair.

CONFIGURATION OF A MULTI-AXIS GROUP.**(V.)MPG.MULNGROUP**

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Number of multi-axis groups in the system.

V.MPG.MULNGROUP

(V.)MPG.MULNAXIS[nb]

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Number of axes and/or spindles making up the multi-axis group.

Syntax.

·nb· Multi-axis group number.

V.MPG.MULNAXIS[2] Second multi-axis group.

(V.)MPG.MULAXISNAMExn[nb]

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Name of the axes and/or spindles that make up the multi-axis group.

Syntax.

·nb· Multi-axis group number.

·xn· Name of the axis and/or spindle within the multi-axis group.

V.MPG.MULAXISNAME4[2] Fourth axis of the second multi-axis group.

Values of the variable.

The values returned by this variable are encoded as follows.

X=10	X1=11	X2=12	X3=13	X4=14	... X9=19
Y=20	Y1=21	Y2=22	Y3=23	Y4=24	... Y9=29
Z=30	Z1=31	Z2=32	Z3=33	Z4=34	... Z9=39
U=40	U1=41	U2=42	U3=43	U4=44	... U9=49

V=50	V1=51	V2=52	V3=53	V4=54	... V9=59
W=60	W1=61	W2=62	W3=63	W4=64	... W9=69
A=70	A1=71	A2=72	A3=73	A4=74	... A9=79
B=80	B1=81	B2=82	B3=83	B4=84	... B9=89
C=90	C1=91	C2=92	C3=93	C4=94	... C9=99
S=100	S1=101	S2=102	S3=103	S4=104	... S9=109

CONFIGURING THE SPINDLES OF THE SYSTEM

22.

CNC VARIABLES.

Variables related to general machine parameters.

(V.)MPG.NSPDL

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of spindles governed by the CNC.

V.MPG.NSPDL

(V.)MPG.SPDLNAME n

Variable to be read via interface.

Name of logic spindle "n".

Syntax.

Replace the "n" letter with the spindle logic number.

MPG.SPDLNAME2 Spindle with logic number -2.

Remarks.

The logic number of the spindles is determined by the order in which they have been defined in the machine parameter table. The logic numbering of the spindles continues from the last logic axis; hence, in a 5-axis system, the first spindle of the table will be logic spindle -6- and so on.

TIME SETTING (SYSTEM).

(V.)MPG.LOOPTIME

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

CNC cycle (loop) time in milliseconds.

V.MPG.LOOPTIME

(V.)MPG.PRGFREQ

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Frequency of the PLC's PRG module (in cycles).

The frequency of the module means how often (every how many CNC cycles) a full scan of the PLC program is executed.

V.MPG.PRGFREQ

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

SERCOS BUS CONFIGURATION

(V.)MPG.SERBRATE

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Sercos transmission speed

V.MPG.SERBRATE

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	4 Mbps.
1	2 Mbps.
2	16 Mbps.
3	8 Mbps.

(V.)MPG.SERPOWSE

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Sercos optical power.

V.MPG.SERPOWSE

MECHATROLINK BUS CONFIGURATION.

(V.)MPG.MLINK

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Mechatrolink mode.

V.MPG.MLINK

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Mlink-I
2	Mlink-II

(V.)MPG.DATASIZE

Telegram size in Mlink-II mode.

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Telegram size in Mlink-II mode.

V.MPG.DATASIZE

22.

CNC VARIABLES.
Variables related to general machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	17 bytes.
1	32 bytes.

CAN BUS CONFIGURATION.

(V.)MPG.CANMODE

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

CAN bus protocol.

V.MPG.CANMODE

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	CANfagor protocol.
1	CANopen protocol.

(V.)MPG.CANLENGTH

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

CANfagor bus cable length (in meters)

V.MPG.CANLENGTH

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.	Value.	Meaning.
0	Up to 20 meters.	7	Up to 90 meters.
1	Up to 30 meters.	8	Up to 100 meters.
2	Up to 40 meters.	9	Up to 110 meters.
3	Up to 50 meters.	10	Up to 120 meters.
4	Up to 60 meters.	11	Up to 130 meters.
5	Up to 70 meters.	12	More than 130 meters.
6	Up to 80 meters.		

SERIAL LINE CONFIGURATION.

(V.)MPG.RSTYPE

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Type of serial line.

V.MPG.RSTYPE

22.

CNC VARIABLES.
Variables related to general machine parameters.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	RS232.
2	RS485.
3	RS422.

DEFAULT CONDITIONS (SYSTEM).**(V.)MPG.INCHES**

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Default measuring units.

V.MPG.INCHES

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Millimeters.
1	Inches.

ARITHMETIC PARAMETERS.**(V.)MPG.MAXLOCP**

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Upper limit for local arithmetic parameters.

V.MPG.MAXLOCP

(V.)MPG.MINLOCP

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Lower limit for local arithmetic parameters.

V.MPG.MINLOCP

(V.)MPG.MAXGLBP

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Upper limit for global arithmetic parameters.

V.MPG.MAXGLBP

(V.)MPG.MINGLBP

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Lower limit for global arithmetic parameters.

V.MPG.MINGLBP



CNC 8060
CNC 8065

(REF: 1402)

(V.)MPG.ROPARMIN

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Lower limit for global read-only arithmetic parameters.

The variable will return a ·0· if no range has been defined or the range is wrong.

V.MPG.ROPARMIN

(V.)MPG.ROPARMAX

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Upper limit for global read-only arithmetic parameters.

The variable will return a ·0· if no range has been defined or the range is wrong.

V.MPG.ROPARMAX

(V.)MPG.MAXCOMP

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Upper limit for common arithmetic parameters.

V.MPG.MAXCOMP

(V.)MPG.MINCOMP

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Lower limit for common arithmetic parameters.

V.MPG.MINCOMP

(V.)MPG.BKUPCUP

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of common non-volatile arithmetic parameters.

V.MPG.BKUPCUP

CROSS COMPENSATION TABLE.**(V.)MPG.MOVAXIS[tbl]**

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Cross compensation table [tbl]. Logic number of the master axis.

The variable will return a ·0· value if the table has not been defined.

Syntax.

·tbl· Table number.

V.MPG.MOVAXIS[3] Third cross compensation table.

(V.)MPG.COMPAXIS[tbl]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Cross compensation table [tbl]. Logic number of the compensated axis.

22.**CNC VARIABLES.**

Variables related to general machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

The variable will return a -0- value if the table has not been defined.

Syntax.

.tbl- Table number.

V.MPG.COMPAXIS[3] Third cross compensation table.

(V.)MPG.NPCROSS[tbl]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Cross compensation table [tbl]. Number of points in the table.

Syntax.

.tbl- Table number.

V.MPG.NPCROSS[3] Third cross compensation table.

(V.)MPG.TYPCROSS[tbl]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Cross compensation table [tbl]. Type of compensation (type of coordinates).

Syntax.

.tbl- Table number.

V.MPG.TYPCROSS[3] Third cross compensation table.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Compensation is applied using real coordinates.
1	Compensation is applied using theoretical coordinates.

(V.)MPG.BIDIR[tbl]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Cross compensation table [tbl]. Bi-directional compensation.

When using bi-directional compensation, the table permits using a different compensation for each moving direction. If the compensation is not bidirectional, it applies the same compensation in both directions.

Syntax.

.tbl- Table number.

V.MPG.BIDIR[3] Third cross compensation table.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

22.

CNC VARIABLES.
Variables related to general machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

(V.)MPG.REFNEED[tbl]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Cross compensation table [tbl]. Both axes must be homed in order to apply the compensation.

Syntax.

.tbl· Table number.

V.MPG.REFNEED[3] Third cross compensation table.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)MPG.POSITION[tbl][pt]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Cross compensation table [tbl]; point [pt] Position of the master axis.

Syntax.

.tbl· Table number.

.pt· Point in the table.

V.MPG.POSITION[3][14] Point 14 of the third cross compensation table.

(V.)MPG.POSERROR[tbl][pt]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Cross compensation table [tbl]; point [pt] Amount of error to compensate when moving in the positive direction.

If there is no bi-directional, it indicates the amount of error to compensate in both directions.

Syntax.

.tbl· Table number.

.pt· Point in the table.

V.MPG.POSERROR[3][14] Point 14 of the third cross compensation table.

(V.)MPG.NEGERROR[tbl][pt]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Cross compensation table [tbl]; point [pt] Amount of error to compensate when moving in the negative direction.

Syntax.

.tbl· Table number.

.pt· Point in the table.

V.MPG.NEGERROR[3][14] Point 14 of the third cross compensation table.

22.**CNC VARIABLES.**

Variables related to general machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

VOLUMETRIC COMPENSATION TABLES.

(V.)MPG.VCOMPAXIS1[tbl]

(V.)MPG.VCOMPAXIS2[tbl]

(V.)MPG.VCOMPAXIS3[tbl]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Volumetric compensation table [tbl]. Axis to be compensated.

Syntax.

.tbl- Table number.

V.MPG.COMPAXIS2[1]

Second axis to be compensated in the first volumetric compensation table.

(V.)MPG.VCOMPFILE[tbl]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Volumetric compensation table [tbl]. File containing volumetric compensation data.

Syntax.

.tbl- Table number.

V.MPG.VCOMPFILE[1]

File for defining the first volumetric compensation.

EXECUTION TIMES.

(V.)MPG.MINAENDW

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Minimum duration of the AUXEND signal (in milliseconds).

V.MPG.MINAENDW

(V.)MPG.REFTIME

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Estimated home searching time (in milliseconds).

V.MPG.REFTIME

(V.)MPG.HTIME

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Estimated time for an "H" function (in milliseconds).

V.MPG.HTIME

(V.)MPG.DTIME

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Estimated time for an "D" function (in milliseconds).

V.MPG.DTIME

22.

CNC VARIABLES.
Variables related to general machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

(V.)MPG.TTIME

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Estimated time for an "T" function (in milliseconds).

V.MPG.TTIME

NUMBERING OF THE DIGITAL INPUTS AND OUTPUTS.

(V.)MPG.NDIMOD

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Total number of digital input modules.

V.MPG.NDIMOD

This variable indicates the number of these modules connected to the same CAN bus. In remote modules with CANopen protocol each double module of digital inputs and outputs counts as two.

(V.)MPG.NDOMOD

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Total number of digital output modules.

V.MPG.NDOMOD

This variable indicates the number of these modules connected to the same CAN bus. In remote modules with CANopen protocol each double module of digital inputs and outputs counts as two.

(V.)MPG.DIMODADDR[nb]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Base address of the digital input modules.

Syntax.

·nb· Module number.

V.MPG.DIMODADDR[4] Fourth digital input module.

(V.)MPG.DOMODADDR[nb]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Base address of the digital output modules.

Syntax.

·nb· Module number.

V.MPG.DOMODADDR[4] Fourth digital output module.

22.

CNC VARIABLES.

Variables related to general machine parameters.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

NUMBERING OF ANALOG INPUTS FOR TEMPERATURE SENSORS PT100.

(V.)MPG.NPT100

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Total number of active PT100 inputs.

V.MPG.NPT100

(V.)MPG.PT100[nb]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Analog input associated with the PT100 input.

Syntax.

·nb· PT100 input number.

V.MPG.NPT100[3] Third PT100 input.

PROBE SETTING.

(V.)MPG.PROBE

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

There is a probe.

V.MPG.PROBE

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)MPG.PROBETYPE1

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Probe type 1, depending on where it is connected.

V.MPG.PROBETYPE1

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Remote probe.
1	Local probe.

22.

CNC VARIABLES.
Variables related to general machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

(V.)MPG.PROBETYPE2

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Probe type 2, depending on where it is connected.

V.MPG.PROBETYPE2

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Remote probe.
1	Local probe.

(V.)MPG.PRBDI 1

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of the input associated with probe 1.

For remote probes, this parameter indicates the digital input number; for local probes, it indicates the local probe input number.

The variable will return a -0- value if no digital input has been defined.

V.MPG.PRBDI1

(V.)MPG.PRBDI 2

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of the input associated with probe 2.

For remote probes, this parameter indicates the digital input number; for local probes, it indicates the local probe input number.

The variable will return a -0- value if no digital input has been defined.

V.MPG.PRBDI2

(V.)MPG.PRBPULSE1

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Logic level to activate probe 1.

V.MPG.PRBPULSE1

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Low logic level (0 V).
1	High logic level (5 V / 24 V).

(V.)MPG.PRBPULSE2

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Logic level to activate probe 2.

V.MPG.PRBPULSE2



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to general machine parameters.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Low logic level (0 V).
1	High logic level (5 V / 24 V).

SHARED PLC MEMORY.**(V.)MPG.PLCDATASIZE**

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Size (in bytes) of the PLC's shared data area.

V.MPG.PLCDATASIZE

MANAGEMENT OF LOCAL I/O.**(V.)MPG.NLOCOUT**

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Number of local digital outputs.

V.MPG.NLOCOUT

(V.)MPG.EXPSCHK

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Activate the 24 V monitoring at the local digital outputs.

V.MPG.EXPSCHK

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

SYNCHRONIZED SWITCHING.**(V.)MPG.SWTOUTPUT**

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Local digital output associated with synchronized switching.

V.MPG.SWTOUTPUT

(V.)MPG.SWTDELAY

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*



CNC 8060
CNC 8065

(REF: 1402)

Delay of the device associated with synchronized switching.

V.MPG.SWTDELAY

PWM (PULSE-WIDTH MODULATION).

(V.)MPG.PWMOUTPUT

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Local digital output associated with the PWM.

(V.)MPG.PWMOUTPUT

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	PWM missing.
1	Local output 1 (pin LI/O1).
2	Local output 2 (pin LI/O2).

(V.)MPG.PWMCANCEL

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Cancel the PWM after an M30 or a reset.

(V.)MPG.PWMCANCEL

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

BACKUP OF NON-VOLATILE DATA.

(V.)MPG.BKUPREG

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Number of non-volatile PLC registers.

V.MPG.BKUPREG

(V.)MPG.BKUPCOUN

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Number of non-volatile PLC counters.

V.MPG.BKUPCOUN

22.

CNC VARIABLES.

Variables related to general machine parameters.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

TOOL OFFSET AND WEAR.

(V.)MPG.TOOLFSG

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Sign criteria for tool offsets and tool wear.

The offsets are used to define the tool dimensions in each axis. The dimensions of the turning tools are defined using these offsets; either these offsets or tool length and radius may be used for the dimensions of the rest of the tools.

V.MPG.TOOLFSG

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Positive.
1	Negative.

SPINDLE SYNCHRONIZATION.

(V.)MPG.SYNCCANCEL

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Cancel the spindle synchronization.

V.MPG.SYNCCANCEL

This parameter indicates whether the CNC cancels spindle synchronization or not after executing M02, M30 or after an error or reset.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

DEFINE THE NUMBER OF JOG PANELS AND THEIR RELATIONSHIP WITH THE CHANNELS.

(V.)MPG.NKEYBD

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of jog panels.

V.MPG.NKEYBD

(V.)MPG.KEYBDCH[jog]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel to which the jog panel is assigned.

22.

CNC VARIABLES.
Variables related to general machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

Syntax.

.jog· Number of jog panel.

V.MPG.KEYBDCH[2] Jog panel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Active channel.
1	Channel ·1·.
2	Channel ·2·.
3	Channel ·3·.
4	Channel ·4·.

Remarks.

The CNC numbers the operator panels following the order (sequence) that they occupy in the CAN bus (Address switch). The first jog panel will be the one with the lowest number and so on.

PLC TYPE.

(V.)MPG.PLCTYPE

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

PLC type.

V.MPG.PLCTYPE

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	IEC.
1	IEC+Fagor.
2	Fagor.

RENAME THE AXES AND THE SPINDLES.

(V.)MPG.RENAMECANCEL

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Cancel the name change for the axes and the spindles.

V.MPG.RENAMECANCEL

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

ZERO OFFSETS.

(V.)MPG.FINEORG

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Fine definition of zero offsets.

V.MPG.FINEORG

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

REMOTE MODULE RCS-S (SERCOS COUNTER).

(V.)MPG.NSERCOUNT

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of RCS-S modules in the bus.

V.MPG.NSERCOUNT

(V.)MPG.SERCOUNTID[n]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Thumbwheel identifiers associated with Sercos counters.

Syntax.

·num· RCS-S module number.

V.MPG.SERCOUNTID[2] Module RCS-S ·2·.

22.

CNC VARIABLES.
Variables related to general machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

22.5 Variables related to the machine parameters of the channels.

CHANNEL CONFIGURATION.

(V.)[ch].MPG.GROUPID

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Group the channel belongs to.

If the variable returns a -0- value, it means that the channel is not associated with any group.

Syntax.

·ch· Channel number.

V.[2].MPG.GROUPID Channel -2-.

(V.)[ch].MPG.CHTYPE

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Channel type.

A channel may be governed from the CNC, from the PLC or from both. Channels governed by the PLC are not displayed in automatic, jog or edisimu modes. The tables can be accessed.

Syntax.

·ch· Channel number.

V.[2].MPG.CHTYPE Channel -2-.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	CNC channel.
1	PLC channel.
2	CNC and PLC channel.

(V.)[ch].MPG.HIDDENCH

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Hidden channel.

Hidden channels are not displayed and cannot be selected.

Syntax.

·ch· Channel number.

V.[2].MPG.HIDDENCH Channel -2-.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

22.

CNC VARIABLES.

Variables related to the machine parameters of the channels.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

CONFIGURING THE AXES OF THE CHANNEL.

(V.) [ch].MPG.CHNAXIS

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Number of the channel axes without including spindles.

It is possible to change the configuration of the axes of a channel via part-program, (defining a new configuration, adding or removing axes) using the instructions #SET AX, #FREE AX and #CALL AX.

Syntax.

·ch· Channel number.

V.[2].MPG.CHNAXIS Channel -2-.

(V.) [ch].MPG.CHAXISNAME_n

Variable to be read via interface.

Channel [ch]. Name the "n" axis of the channel.

Syntax.

·ch· Channel number.

·n· Index of the axis in the channel.

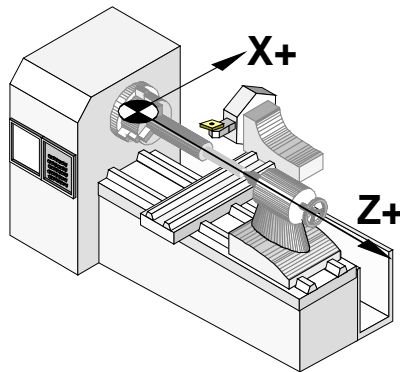
[2].MPG.CHAXISNAME4 Axis with index -1- in the channel -2-.

(V.) [ch].MPG.GEOCONFIG

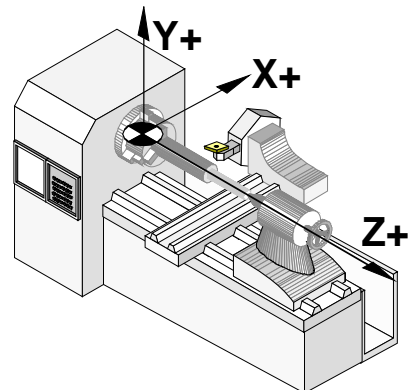
Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Geometrical configuration of the axes of the channel.

On the lathe model, the geometric configuration of the axes may be a "trihedron" type or a "plane" type.



Configuration of "plane" type axes.



Configuration of trihedron type axes.

Syntax.

·ch· Channel number.

V.[2].MPG.GEOCONFIG Channel -2-.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Configuration of "plane" type axes.
1	Configuration of "Trihedron" type axes.

22.

CNC VARIABLES.
Variables related to the machine parameters of the channels.



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

Configuration.	Configuration properties.
"Trihedron" type.	In this configuration, there are three axes forming a Cartesian XYZ type trihedron like on a milling machine. There may be more axes besides those forming the trihedron. With this configuration, the planes behave in the same way as on a milling machine except that the usual work plane will be G18 (if it has been configured like that).
"plane" type.	In this configuration, there are two axes forming the usual work plane. There may be more axes, but they cannot be part of the trihedron; there must be auxiliary, rotary, etc. With this configuration, the work plane is always G18 and will be formed by the first two axes defined in the channel. In this configuration, the second axis of the channel is considered as longitudinal axis. If the X (first) and Z (second) axes have been defined, the work plane will be the ZX (Z as abscissa and X as ordinate) and Z the longitudinal axis. Tool length compensation is applied on this longitudinal axis when using milling tools. With lathe tools, tool length compensation is applied on all the axes where a tool offset has been defined.

CONFIGURING THE SPINDLES OF THE CHANNEL.**(V.)[ch].MPG.CHNSPDL**

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Number of spindles of the channel.

It is possible to change the configuration of the spindles of a channel via part-program, (defining a new configuration, adding or remove spindles) using the instructions #SET SP, #FREE SP and #CALL SP.

Syntax.

·ch· Channel number.

V.[2].MPG.CHSPDL Channel ·2·.

(V.)[ch].MPG.CHSPDLNAME_n

Variable to be read via interface.

Channel [ch]. Name the "n" spindle of the channel.

Syntax.

·ch· Channel number.

·n· Index of the spindle in the channel.

[2].MPG.CHSPDLNAME1 Spindle with index ·1· in the channel ·2·.

CONFIGURATION OF THE C AXIS.**(V.)[ch].MPG.CAXNAME**

Variable to be read via interface.

Channel [ch]. Name of the axis working as "C" axis (by default).

When setting more than one C axis, use the program instruction #CAX to indicate the one that is active. Only one C axis may be active in each channel.

22.**CNC VARIABLES.**

Variables related to the machine parameters of the channels.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the machine parameters of the channels.

Syntax.

·ch· Channel number.

[2].MPG.CAXNAME Channel -2.

(V.) [ch].MPG.ALIGNC

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. "C" axis alignment for diametrical machining.

This parameter indicates whether the C axis must be aligned for diameter machining (ALIGNC = Yes) or the tool can machine the whole surface diametrically in a single fixturing operation (ALIGNC = NO).

Syntax.

·ch· Channel number.

V.[2].MPG.ALIGNC Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

TIME SETTING (CHANNEL).**(V.) [ch].MPG.PREPFREQ**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Maximum number of blocks to prepare per cycle.

Syntax.

·ch· Channel number.

V.[2].MPG.PREPFREQ Channel -2.

(V.) [ch].MPG.ANTIME

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Anticipation time.

Syntax.

·ch· Channel number.

V.[2].MPG.ANTIME Channel -2.

CONFIGURATION OF THE HSC MODE (CHANNEL).**(V.) [ch].MPG.FEEDAVRG**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Calculating feedrate average.

Syntax.

.ch· Channel number.

V.[2].MPG.FEEDAVRG Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPG.SMOOTHFREQ

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Smoothing frequency in the interpolation.

Syntax.

.ch· Channel number.

V.[2].MPG.SMOOTHFREQ Channel -2.

(V.)[ch].MPG.CORNER

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Maximum angle of the corner to machine it in square corner mode.

Syntax.

.ch· Channel number.

V.[2].MPG.CORNER Channel -2.

(V.)[ch].MPG.HSCFILTFREQ

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Filter frequency (CONTERROR mode).

Syntax.

.ch· Channel number.

V.[2].MPG.HSCFILTFREQ Channel -2.

(V.)[ch].MPG.FASTFACTOR

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Default feedrate percentage (FAST mode).

Syntax.

.ch· Channel number.

V.[2].MPG.FASTFACTOR Channel -2.

(V.)[ch].MPG.FTIMELIM

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Time difference permitted in feedrate interpolation (FAST mode).

22.**CNC VARIABLES.**

Variables related to the machine parameters of the channels.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.

Variables related to the machine parameters of the channels.

Syntax.

·ch· Channel number.

V.[2].MPG.FTIMELIM	Channel -2.
--------------------	-------------

(V.) [ch].MPG.MINCORFEED*Variable that can only be read from the program, PLC and interface.**The variable returns the value of block preparation.*

Channel [ch]. Minimum feedrate at the corners.

Syntax.

·ch· Channel number.

V.[2].MPG.MINCORFEED	Channel -2.
----------------------	-------------

(V.) [ch].MPG.FSMOOTHFREQ*Variable that can only be read from the program, PLC and interface.**The variable returns the value of block preparation.*

Channel [ch]. Smoothing frequency in the interpolation (FAST mode).

Syntax.

·ch· Channel number.

V.[2].MPG.FSMOOTHFREQ	Channel -2.
-----------------------	-------------

(V.) [ch].MPG.FASTFILTFREQ*Variable that can only be read from the program, PLC and interface.**The variable returns the value of block preparation.*

Channel [ch]. Filter frequency (FAST mode).

Syntax.

·ch· Channel number.

V.[2].MPG.FASTFILTFREQ	Channel -2.
------------------------	-------------

(V.) [ch].MPG.FREQRES*Variable that can only be read from the program, PLC and interface.**The variable returns the value of block preparation.*

Channel [ch]. First resonance frequency of the machine.

Syntax.

·ch· Channel number.

V.[2].MPG.FREQRES	Channel -2.
-------------------	-------------

(V.) [ch].MPG.SOFTFREQ*Variable that can only be read from the program, PLC and interface.**The variable returns the value of block preparation.*

Channel [ch]. Path filter frequency for profiles of linear acceleration.

Syntax.

·ch· Channel number.

V.[2].MPG.SOFTFREQ	Channel -2.
--------------------	-------------



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPG.HSCROUND

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Default value of the maximum path error in HSC.

Syntax.

.ch· Channel number.

V.[2].MPG.HSCROUND Channel -2.

(V.)[ch].MPG.SURFFILFREQ

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Axis filter frequency (SURFACE mode).

Syntax.

.ch· Channel number.

V.[2].MPG.SURFFILFREQ Channel -2.

(V.)[ch].MPG.HSCDEFAULTMODE

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Default mode when programming #HSC ON.

Syntax.

.ch· Channel number.

V.[2].MPG.HSCDEFAULTMODE Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	SURFACE
1	CONTERROR
2	FAST

VIRTUAL TOOL AXIS.

(V.)[ch].MPG.VIRTAXISNAME

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Name of the virtual tool axis.

Syntax.

.ch· Channel number.

V.[2].MPG.VIRTAXISNAME Channel -2.

Values of the variable.

The values returned by this variable are encoded as follows.

X=10	X1=11	X2=12	X3=13	X4=14	... X9=19
Y=20	Y1=21	Y2=22	Y3=23	Y4=24	... Y9=29
Z=30	Z1=31	Z2=32	Z3=33	Z4=34	... Z9=39

22.

CNC VARIABLES.

Variables related to the machine parameters of the channels.



CNC 8060
CNC 8065

(REF: 1402)

U=40	U1=41	U2=42	U3=43	U4=44	... U9=49
V=50	V1=51	V2=52	V3=53	V4=54	... V9=59
W=60	W1=61	W2=62	W3=63	W4=64	... W9=69
A=70	A1=71	A2=72	A3=73	A4=74	... A9=79
B=80	B1=81	B2=82	B3=83	B4=84	... B9=89
C=90	C1=91	C2=92	C3=93	C4=94	... C9=99

22.

CNC VARIABLES.
Variables related to the machine parameters of the channels.

(V.) [ch].MPG.VIRTAXCANCEL

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Cancel the virtual tool axis after an M30 and Reset.

Syntax.

·ch· Channel number.

V.[2].MPG.VIRTAXCANCEL Channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

DEFAULT CONDITIONS (CHANNEL).**(V.) [ch].MPG.KINID**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Default kinematics number.

The channel assumes the default value on power-up, after executing an M02 or M30 or after a Reset. To select another kinematics from the part-program, use the #KIN ID instruction.

Syntax.

·ch· Channel number.

V.[2].MPG.KINID Channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	The CNC restores the last kinematics active.
1..6	Default kinematics number.
255	There is no default kinematics.

(V.) [ch].MPG.CSCANCEL

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Cancel the inclined plane on start-up.

This parameter indicates whether on start-up the CNC cancels the inclined plane (#CS/#ACS) that was active when the CNC was turned off.



CNC 8060
CNC 8065

(REF: 1402)

Syntax.

.ch· Channel number.

V.[2].MPG.CSCANCEL Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPG.LINKCANCEL

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Cancel axis coupling by default.

The channel assumes the default value after executing an M02 or M30, after an emergency or a Reset. To couple axes from the part-program, use the #LINK instruction.

Syntax.

.ch· Channel number.

V.[2].MPG.LINKCANCEL Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPG.MIRRORCANCEL

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Cancel mirror image (G11/G12/G13/G14) after M30 and reset.

Syntax.

.ch· Channel number.

V.[2].MPG.MIRRORCANCEL Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPG.SLOPETYPE

Variable that can only be read from the program, PLC and interface.
Variable that may be modified during setup.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Default acceleration type.

It indicates the type of acceleration applied by default in automatic movements. When working in manual (JOG) mode, the CNC always applies linear acceleration.

The channel assumes the default value on power-up, after executing an M02 or M30 or after a Reset. To select another acceleration from the part-program, use the #SLOPE instruction.

Syntax.

·ch· Channel number.

V.[2].MPG.SLOPETYPE Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Linear acceleration.
1	Trapezoidal acceleration.
2	Square sine (bell shaped) acceleration.

(V.)[ch].MPG.IPLANE

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Channel [ch]. Main work plane (G17/G18) by default.

The channel assumes the default value on power-up, after executing an M02 or M30 or after a Reset. To change the work plane via part-program, use function G17, G18, G19 or G20.

Syntax.

·ch· Channel number.

V.[2].MPG.IPLANE Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	G17.
1	G18.

(V.)[ch].MPG.ISYSTEM

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Channel [ch]. Default type of coordinates (G90/G91).

The coordinates of a point may be defined either in absolute coordinates (G90) referred to part zero or in incremental coordinates (G91) referred to the current position.

The channel assumes the default value on power-up, after executing an M02 or M30 or after a Reset. To change the type of coordinates via part-program, use function G90 or G91.

Syntax.

·ch· Channel number.

V.[2].MPG.ISYSTEM Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	G90.
1	G91.

22.

CNC VARIABLES.
Variables related to the machine parameters of the channels.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPG.IMOVE

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Default type of movement (G0/G1).

The channel assumes the default value on power-up, after executing an M02 or M30 or after a Reset. To change the type of movement via part-program, use function G0 or G1.

Syntax.

.ch· Channel number.

V.[2].MPG.IMOVE Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	G00.
1	G01.

(V.)[ch].MPG.IFEED

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Default type of feedrate (G94/G95).

The channel assumes the default value on power-up, after executing an M02 or M30 or after a Reset. To change the type of feedrate via part-program, use function G93 or G94.

Syntax.

.ch· Channel number.

V.[2].MPG.IFEED Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	G94.
1	G95.

(V.)[ch].MPG.FPRMAN

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Function G95 admitted in jog mode.

Syntax.

.ch· Channel number.

V.[2].MPG.FPRMAN Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPG.LCOMPTYP

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Keep the longitudinal axis when changing planes (G17/G18/G19).

Syntax.

·ch· Channel number.

V.[2].MPG.LCOMPTYP Channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPG.ICORNER

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Default type of corner (G5/G7/G50).

The channel assumes the default value on power-up, after executing an M02 or M30 or after a Reset. To change the type of corner via part-program, use function G5, G7 or G50.

Syntax.

·ch· Channel number.

V.[2].MPG.ICORNER Channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	G50.
1	G05.
2	G07.

(V.)[ch].MPG.IRCOMP

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Tool radius compensation mode (G136/G137) by default

The channel assumes the default value on power-up, after executing an M02 or M30 or after a Reset. To change the type of tool radius compensation via part-program, use function G136 or G137.

Syntax.

·ch· Channel number.

V.[2].MPG.IRCOMP Channel ·2·.

22.

CNC VARIABLES.
Variables related to the machine parameters of the channels.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	G136.
1	G137.

(V.) [ch].MPG.COMPCANCEL

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Channel [ch]. How to cancel tool radius compensation.

This parameter indicates whether tool radius compensation is canceled in the first motion block, even if the plane axes are not involved, or if it requires a movement of the plane axes.

Syntax.

·ch· Channel number.

V.[2].MPG.COMPCANCEL Channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Without moving the plane axes.
1	Moving the plane axes.

(V.) [ch].MPG.ROUNDTYPE

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Channel [ch]. Default rounding type in G5.

The rounding may be executed by limiting the chordal error or the feedrate. The chordal error (#ROUNDPAR [1]) defines the maximum deviation allowed between the programmed point and the resulting profile. The feedrate (#ROUNDPAR [2]) defines the percentage of the active feedrate to be used for machining.

The channel assumes the default value on power-up, after executing an M02 or M30 or after a Reset. To change the type of rounding via program, use the #ROUNDPAR instruction.

Syntax.

·ch· Channel number.

V.[2].MPG.ROUNDTYPE Channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Chordal error.
1	Percentage of feedrate.

(V.) [ch].MPG.MAXROUND

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Channel [ch]. Maximum rounding error in G5.

22.

CNC VARIABLES.
Variables related to the machine parameters of the channels.

Syntax.

·ch· Channel number.

V.[2].MPG.MAXROUND	Channel ·2·.
--------------------	--------------

(V.) [ch].MPG.ROUNDFEED

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Percentage of feedrate in G5.

Syntax.

·ch· Channel number.

V.[2].MPG.ROUNDFEED	Channel ·2·.
---------------------	--------------

ARC CENTER CORRECTION.**(V.) [ch].MPG.CIRINERR**

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Maximum absolute radius error permitted.

Syntax.

·ch· Channel number.

V.[2].MPG.CIRINERR	Channel ·2·.
--------------------	--------------

(V.) [ch].MPG.CIRINFACT

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Maximum relative radius error permitted.

The relative error is given as a percentage over the radius.

Syntax.

·ch· Channel number.

V.[2].MPG.CIRINFACT	Channel ·2·.
---------------------	--------------

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·100·.

BEHAVIOR OF THE FEEDRATE AND THE FEEDRATE OVERRIDE.**(V.) [ch].MPG.MAXOVR**

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Maximum override (%) permitted.

Syntax.

·ch· Channel number.

V.[2].MPG.MAXOVR	Channel ·2·.
------------------	--------------



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·100·.

(V.)[ch].MPG.RAPIDOVR

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Override acts in G00 (from 0 to 100%).

This parameter indicates whether the feedrate % may be modified (between 0% and 100%) or not when working in G0; if not possible, it will stay fixed at 100%.

Regardless of the value assigned to this parameter, the override always attends to the 0% position and never acts over 100%. It is always possible to change the % of feedrate when moving in jog mode.

Syntax.

·ch· Channel number.

```
V.[2].MPG.RAPIDOVR Channel ·2·
```

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPG.FEEDND

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Apply the programmed feedrate to all the axes of the channel

This parameter indicates whether the programmed feedrate is applied to all the axes of the channel or only to the main axes. If it is only applied to the main axes, the rest of the axes move at their corresponding feedrate to end the movement of them all at the same time.

Syntax.

·ch· Channel number.

```
V.[2].MPG.FEEDND Channel ·2·
```

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No. The programmed feedrate is only applied to the main axes.
1	Yes. The programmed feedrate is applied to all the axes of the channel.

VERRIDE OF THE DYNAMICS FOR HSC.

(V.)[ch].MPG.MINDYNOVR

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Minimum override for the dynamics in HSC.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the machine parameters of the channels.

Syntax.

·ch· Channel number.

V.[2].MPG.MINDYNOVR Channel -2.

(V.) [ch].MPG.MAXDYNOVR

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Maximum override for the dynamics in HSC.

Syntax.

·ch· Channel number.

V.[2].MPG.MAXDYNOVR Channel -2.

(V.) [ch].MPG.STEPDYNOVR

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Override step for the dynamics in HSC.

Syntax.

·ch· Channel number.

V.[2].MPG.STEPDYNOVR Channel -2.

MOVEMENT OF THE INDEPENDENT AXES.**(V.) [ch].MPG.IMOVEMACH**

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Movement of the independent axis referred to machine coordinates.

Syntax.

·ch· Channel number.

V.[2].MPG.IMOVEMACH Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.) [ch].MPG.XFITOIND

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. The transfer inhibit of the channel affects the independent axes.

This parameter indicates whether the transfer inhibit of the channel (_XFERINH mark) affects the independent axis or not.

Syntax.

·ch· Channel number.

V.[2].MPG.XFITOIND Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

DEFINITION OF THE SUBROUTINES.

(V.)[ch].MPG.TOOLSUB

Variable to be read via interface.

Channel [ch]. Subroutine associated with "T".

Syntax.

.ch- Channel number.

[2].MPG.TOOLSUB Channel -2.

(V.)[ch].MPG.REFPSUB

Variable to be read via interface.

Channel [ch]. Subroutine associated with function G74.

Syntax.

.ch- Channel number.

[2].MPG.REFPSUB Channel -2.

(V.)[ch].MPG.OEMSUB1

..

(V.)[ch].MPG.OEMSUB10

Variable to be read via interface.

Channel [ch]. Subroutines associated with G180 through G189.

Syntax.

.ch- Channel number.

[2].MPG.OEMSUB1 Channel -2.

(V.)[ch].MPG.OEMSUB11

..

(V.)[ch].MPG.OEMSUB30

Variable to be read via interface.

Channel [ch]. Subroutines associated with G380 through G399.

Syntax.

.ch- Channel number.

[2].MPG.OEMSUB11 Channel -2.

(V.)[ch].MPG.INT1SUB

..

(V.)[ch].MPG.INT4SUB

Variable to be read via interface.

Channel [ch]. Interruption subroutines.

22.

CNC VARIABLES.
Variables related to the machine parameters of the channels.

Syntax.

.ch- Channel number.

[2].MPG.INT1SUB	Channel -2-.
-----------------	--------------

(V.) [ch].MPG.SUBPATH

Variable to be read via interface.

Channel [ch]. Path of the program subroutines.

Syntax.

.ch- Channel number.

[2].MPG.SUBPATH	Channel -2-.
-----------------	--------------

TABLETOP PROBE POSITION.**(V.) [ch].MPG.PR1MIN**

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Channel [ch]. Minimum probe coordinate (abscissa axis).

Syntax.

.ch- Channel number.

V.[2].MPG.PR1MIN	Channel -2-.
------------------	--------------

(V.) [ch].MPG.PR1MAX

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Channel [ch]. Maximum probe coordinate (abscissa axis).

Syntax.

.ch- Channel number.

V.[2].MPG.PR1MAX	Channel -2-.
------------------	--------------

(V.) [ch].MPG.PR2MIN

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Channel [ch]. Minimum probe coordinate (ordinate axis).

Syntax.

.ch- Channel number.

V.[2].MPG.PR2MIN	Channel -2-.
------------------	--------------

(V.) [ch].MPG.PR2MAX

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Channel [ch]. Maximum probe coordinate (ordinate axis).

Syntax.

.ch- Channel number.

V.[2].MPG.PR2MAX	Channel -2-.
------------------	--------------



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPG.PR3MIN

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Minimum probe coordinate (axis perpendicular to the plane).

Syntax.

·ch· Channel number.

V.[2].MPG.PR3MIN Channel -2.

(V.)[ch].MPG.PR3MAX

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Maximum probe coordinate (axis perpendicular to the plane).

Syntax.

·ch· Channel number.

V.[2].MPG.PR3MAX Channel -2.

BLOCK SEARCH.

(V.)[ch].MPG.FUNPLC

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Send the M, H, S functions to the PLC during block search.

Syntax.

·ch· Channel number.

V.[2].MPG.FUNPLC Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

INTERRUPTION SUBROUTINES.

(V.)[ch].MPG.SUBINTSTOP

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Execute interruption subroutines while the program is stopped or while no program is in execution.

Syntax.

·ch· Channel number.

V.[2].MPG.SUBINTSTOP Channel -2.

22.

CNC VARIABLES.

Variables related to the machine parameters of the channels.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.

Variables related to the machine parameters of the channels.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

MACHINING FEEDRATE.**(V.) [ch].MPG.MAXFEED***Variable that can only be read from the program, PLC and interface.**The variable returns the value of block preparation.*

Channel [ch]. Maximum machining feedrate.

If the variable returns a ·0· value, the machining feedrate is not limited, the CNC assumes for all the movements the one set in machine parameter G00FEED as the maximum feedrate.

Syntax.

·ch· Channel number.

V.[2].MPG.MAXFEED	Channel ·2·.
-------------------	--------------

(V.) [ch].MPG.DEFAULTFEED*Variable that can only be read from the program, PLC and interface.**The variable returns the value of block preparation.*

Channel [ch]. Assume MAXFEED for movements in G1/G2/G3 without active feedrate

If parameter MAXFEED = 0, the variable will always return a 0 value.

Syntax.

·ch· Channel number.

V.[2].MPG.DEFAULTFEED	Channel ·2·.
-----------------------	--------------

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

RAPID TRAVERSE FOR THE AUTOMATIC MODE.**(V.) [ch].MPG.RAPIDEN***Variable that can only be read from the program, PLC and interface.**The variable returns the value of block preparation.*

Channel [ch]. It may be used to enable the rapid traverse for the automatic mode while executing a program.

Syntax.

·ch· Channel number.

V.[2].MPG.RAPIDEN	Channel ·2·.
-------------------	--------------



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Disabled. Rapid traverse is not available for the automatic mode.
1	EXRAPID or rapid key. To activate the rapid feed, just activate the PLC mark EXRAPID or press the "rapid" key of the jog panel.
2	EXRAPID and rapid key. To activate the rapid feed, just activate the PLC mark EXRAPID and press the "rapid" key of the jog panel.

(V.) [ch].MPG.FRAPI DEN

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Rapid traverse in the channel when rapid traverse for the automatic mode is active (parameter RAPIDEN). If the parameter is set with a "0" value, the feedrate is not limited.

This parameter does not the movements programmed in G00 or the threads. Movements in G0 are carried out at the feedrate set in parameter G00FEED. The threads are executed at the programmed feedrate.

Syntax.

.ch· Channel number.

V.[2].MPG.FRAPI DEN Channel ·2·.

Remarks.

Rapid traverse cannot exceed the value set in axis parameters G00FEED and FRAPI DEN or the maximum feedrate set by PLC (variable (V.)PLC.G00FEED). Rapid traverse cannot exceed the value set in axis parameter MAXFEED of the channel and the active feedrate set by PLC (variable (V.)PLC.F).

MAXIMUM ACCELERATION AND JERK ON THE TOOL PATH.**(V.) [ch].MPG.MAXACCEL**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Maximum acceleration on the machining path.

Syntax.

.ch· Channel number.

V.[2].MPG.MAXACCEL Channel ·2·.

(V.) [ch].MPG.MAXJERK

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Maximum jerk on the machining path.

Syntax.

.ch· Channel number.

V.[2].MPG.MAXJERK Channel ·2·.

22.**CNC VARIABLES.**

Variables related to the machine parameters of the channels.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

"RETRACE" FUNCTION.**(V.) [ch].MPG.RETRACAC**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Permit enabling the "retrace" function

Syntax.

·ch· Channel number.

V.[2].MPG.RETRACAC Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.) [ch].MPG.NRETBK

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Maximum number of blocks allowed for the "retrace" function.

Syntax.

·ch· Channel number.

V.[2].MPG.NRETBK Channel -2.

(V.) [ch].MPG.RETMFUNC

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Treatment of the M functions with the "retrace" function.

This parameter sets the behavior of the "retrace" function when executing M functions. When the CNC finds an M function, it can either ignore it and keep executing blocks in retrace or cancel the "retrace" function.

Syntax.

·ch· Channel number.

V.[2].MPG.RETMFUNC Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Ignore the M function and go on.
1	Turn the retrace function off.

Remarks.

This parameter does not affect the following "M" functions.

- Functions M00 and M01 are always executed; they are sent to the PLC and [CYCLE START] must be pressed to resume execution in retrace.
- Functions M03 and M04 are always ignored; the CNC does not start the spindle nor does it change its turning direction.
- Function M05 cancels the "retrace" function; the CNC does not stop the spindle.

22.

CNC VARIABLES.
Variables related to the machine parameters of the channels.



CNC 8060
CNC 8065

(REF: 1402)

MASTER SPINDLE.

(V.)[ch].MPG.MASTERSPDL

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Master spindle maintained.

This parameter indicates whether the master spindle of a channel maintains its master condition or not after executing M02, M30 or after an emergency or reset or restarting the CNC.

Syntax.

·ch· Channel number.

V.[2].MPG.MASTERSPDL Channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Temporary.
1	Maintained.

22.

CNC VARIABLES.

Variables related to the machine parameters of the channels.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.6 Variables related to axis and spindle machine parameters.

BELONGING TO THE CHANNEL.

(V.)[ch].MPA.AXISEXCH.xn
 (V.)[ch].MPA.AXISEXCH.sn
 (V.)[ch].SP.AXISEXCH.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Channel change permission.

This variable shows whether it is possible for the axis or spindle to change channels via part-program and, if so, whether the change is temporary or permanent; in other words, whether the change is maintained after an M02, M30 or a reset.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.AXISEXCH.Z	Z axis.
V.MPA.AXISEXCH.S	Spindle S.
V.SP.AXISEXCH.S	Spindle S.
V.SP.AXISEXCH	Master spindle.
V.MPA.AXISEXCH.4	Axis or spindle with logic number ·4·.
V.[2].MPA.AXISEXCH.1	Axis with index ·1· in the channel ·2·.
V.SP.AXISEXCH.2	Spindle with index ·2· in the system.
V.[2].SP.AXISEXCH.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	It is not possible to change the axis or the spindle of the channel.
1	The change is temporary.
2	The change is permanent.

TYPE OF AXIS AND DRIVE.

(V.)[ch].MPA.AXISTYPE.xn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Type of axis.

Syntax.

- ch· Channel number.

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

·xn· Name, logic number or index of the axis.

V.MPA.AXISTYPE.Z	Z axis.
V.MPA.AXISTYPE.4	Axis with logic number ·4·.
V.[2].MPA.AXISTYPE.1	Axis with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Linear axis.
2	Rotary axis.

(V.)[ch].MPA.DRIVETYPE.xn

(V.)[ch].MPA.DRIVETYPE.sn

(V.)[ch].SP.DRIVETYPE.sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for an analog, position-Sercos, velocity-Sercos and Mechatrolink drive.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Type of drive.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.MPA.DRIVETYPE.Z	Z axis.
V.MPA.DRIVETYPE.S	Spindle S.
V.SP.DRIVETYPE.S	Spindle S.
V.SP.DRIVETYPE	Master spindle.
V.MPA.DRIVETYPE.4	Axis or spindle with logic number ·4·.
V.[2].MPA.DRIVETYPE.1	Axis with index ·1· in the channel ·2·.
V.SP.DRIVETYPE.2	Spindle with index ·2· in the system.
V.[2].SP.DRIVETYPE.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Analog drive.
2	Sercos drive.
16	Simulated drive.
32	Mechatrolink drive.

(V.)[ch].MPA.POSUNITS.xn

(V.)[ch].MPA.POSUNITS.sn

(V.)[ch].SP.POSUNITS.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for a position-Sercos, velocity-Sercos and Mechatrolink drive.

The variable returns the value of block preparation.

Channel [ch]. Units system used by the feedback system.

22.

CNC VARIABLES.

Variables related to axis and spindle machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.POSUNITS.Z	Z axis.
V.MPA.POSUNITS.S	Spindle S.
V.SP.POSUNITS.S	Spindle S.
V.SP.POSUNITS	Master spindle.
V.MPA.POSUNITS.4	Axis or spindle with logic number ·4·.
V.[2].MPA.POSUNITS.1	Axis with index ·1· in the channel ·2·.
V.SP.POSUNITS.2	Spindle with index ·2· in the system.
V.[2].SP.POSUNITS.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Measurement (mm or degrees).
1	Pulses.

CONFIGURING A SERCOS DRIVE.

(V.)[ch].MPA.DRIVEID.xn

(V.)[ch].MPA.DRIVEID.sn

(V.)[ch].SP.DRIVEID.sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for a position-Sercos, velocity-Servos and Mechatrolink drive.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Address (node) of the drive.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.DRIVEID.Z	Z axis.
V.MPA.DRIVEID.S	Spindle S.
V.SP.DRIVEID.S	Spindle S.
V.SP.DRIVEID	Master spindle.
V.MPA.DRIVEID.4	Axis or spindle with logic number ·4·.
V.[2].MPA.DRIVEID.1	Axis with index ·1· in the channel ·2·.
V.SP.DRIVEID.2	Spindle with index ·2· in the system.
V.[2].SP.DRIVEID.1	Spindle with index ·1· in the channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.OPMODEP.xn

(V.)[ch].MPA.OPMODEP.sn

(V.)[ch].SP.OPMODEP.sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for a position-Sercos, velocity-Servos and Mechatrolink drive.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Main operating mode of the Sercos drive or Mechatrolink device.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.OPMODEP.Z	Z axis.
V.MPA.OPMODEP.S	Spindle S.
V.SP.OPMODEP.S	Spindle S.
V.SP.OPMODEP	Master spindle.
V.MPA.OPMODEP.4	Axis or spindle with logic number ·4·.
V.[2].MPA.OPMODEP.1	Axis with index ·1· in the channel ·2·.
V.SP.OPMODEP.2	Spindle with index ·2· in the system.
V.[2].SP.OPMODEP.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Sercos drive or Mechatrolink servo. Position command.
1	Sercos drive or Mechatrolink servo. Velocity command.
2	Inverter Mechatrolink.

(V.)[ch].MPA.FBACKSRC.xn

(V.)[ch].MPA.FBACKSRC.sn

(V.)[ch].SP.FBACKSRC.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for position-Sercos drives and velocity-Sercos drives.

The variable returns the value of block preparation.

Channel [ch]. Type of feedback.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.FBACKSRC.Z	Z axis.
V.MPA.FBACKSRC.S	Spindle S.
V.SP.FBACKSRC.S	Spindle S.
V.SP.FBACKSRC	Master spindle.
V.MPA.FBACKSRC.4	Axis or spindle with logic number ·4·.
V.[2].MPA.FBACKSRC.1	Axis with index ·1· in the channel ·2·.
V.SP.FBACKSRC.2	Spindle with index ·2· in the system.
V.[2].SP.FBACKSRC.1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to axis and spindle machine parameters.



**CNC 8060
CNC 8065**

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Internal feedback (motor feedback).
1	External feedback (direct feedback).
2	Combined feedback (internal + external).

(V.)[ch].MPA.FBACKDIFF.xn

(V.)[ch].MPA.FBACKDIFF.sn

(V.)[ch].SP.FBACKDIFF.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for position Sercos drives.

The variable returns the value of block preparation.

Channel [ch]. Maximum difference between feedbacks.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.FBACKDIFF.Z	Z axis.
V.MPA.FBACKDIFF.S	Spindle S.
V.SP.FBACKDIFF.S	Spindle S.
V.SP.FBACKDIFF	Master spindle.
V.MPA.FBACKDIFF.4	Axis or spindle with logic number ·4·.
V.[2].MPA.FBACKDIFF.1	Axis with index ·1· in the channel ·2·.
V.SP.FBACKDIFF.2	Spindle with index ·2· in the system.
V.[2].SP.FBACKDIFF.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.FBMIXTIME.xn

(V.)[ch].MPA.FBMIXTIME.sn

(V.)[ch].SP.FBMIXTIME.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for position Sercos drives.

The variable returns the value of block preparation.

Channel [ch]. Time constant for combined feedback.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.FBMIXTIME.Z	Z axis.
V.MPA.FBMIXTIME.S	Spindle S.
V.SP.FBMIXTIME.S	Spindle S.
V.SP.FBMIXTIME	Master spindle.
V.MPA.FBMIXTIME.4	Axis or spindle with logic number ·4·.
V.[2].MPA.FBMIXTIME.1	Axis with index ·1· in the channel ·2·.
V.SP.FBMIXTIME.2	Spindle with index ·2· in the system.
V.[2].SP.FBMIXTIME.1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

HIRTH AXIS CONFIGURATION.

(V.)[ch].MPA.HIRTH.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Hirth toothed axis.

A Hirth axis is the one that can only be positioned at positions multiple of a given value.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.HIRTH.Z	Z axis.
V.MPA.HIRTH.4	Axis with logic number ·4·.
V.[2].MPA.HIRTH.1	Axis with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.HPITCH.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Hirth axis pitch.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.HPITCH.Z	Z axis.
V.MPA.HPITCH.4	Axis with logic number ·4·.
V.[2].MPA.HPITCH.1	Axis with index ·1· in the channel ·2·.

AXIS CONFIGURATION FOR LATHE TYPE MACHINES.

(V.)[ch].MPA.FACEAXIS.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Cross axis on a lathe.

Syntax.

·ch· Channel number.

22.

CNC VARIABLES.

Variables related to axis and spindle machine parameters.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

·xn· Name, logic number or index of the axis.

V.MPA.FACEAXIS.Z	Z axis.
V.MPA.FACEAXIS.4	Axis with logic number ·4·.
V.[2].MPA.FACEAXIS.1	Axis with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.LONGAXIS.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Longitudinal axis on a lathe.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.LONGAXIS.Z	Z axis.
V.MPA.LONGAXIS.4	Axis with logic number ·4·.
V.[2].MPA.LONGAXIS.1	Axis with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

CONFIGURATION OF THE ROTARY AXES.

(V.)[ch].MPA.AXISMODE.xn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Working method with the rotary axis.

This variable indicates how the rotary axis will behave in relation to the number of turns and position display.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.AXISMODE.Z	Z axis.
V.MPA.AXISMODE.4	Axis with logic number ·4·.
V.[2].MPA.AXISMODE.1	Axis with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to axis and spindle machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	"Module" type rotary axis.
1	"Linearlike" type rotary axis.

(V.) [ch].MPA.UNIDIR.xn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Unidirectional rotation.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.UNIDIR.Z	Z axis.
V.MPA.UNIDIR.4	Axis with logic number ·4·.
V.[2].MPA.UNIDIR.1	Axis with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.) [ch].MPA.SHORTESTWAY.xn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Positioning via shortest way.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.SHORTESTWAY.Z	Z axis.
V.MPA.SHORTESTWAY.4	Axis with logic number ·4·.
V.[2].MPA.SHORTESTWAY.1	Axis with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

22.**CNC VARIABLES.**

Variables related to axis and spindle machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

MODULE CONFIGURATION (ROTARY AXES AND SPINDLE).

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.

(V.) [ch].MPA.MODCOMP.xn**(V.) [ch].MPA.MODCOMP.sn****(V.) [ch].SP.MODCOMP.sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified during setup.**Variable valid for rotary axes and spindles.**Variable valid for analog drives and velocity-Sercos drives.**The variable returns the execution value; reading it interrupts block preparation.*

Channel [ch]. Module compensation.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MODCOMP.Z	Z axis.
V.MPA.MODCOMP.S	Spindle S.
V.SP.MODCOMP.S	Spindle S.
V.SP.MODCOMP	Master spindle.
V.MPA.MODCOMP.4	Axis or spindle with logic number -4-.
V.[2].MPA.MODCOMP.1	Axis with index -1- in the channel -2-.
V.SP.MODCOMP.2	Spindle with index -2- in the system.
V.[2].SP.MODCOMP.1	Spindle with index -1- in the channel -2-.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

CONFIGURATION OF THE C AXIS.

(V.) [ch].MPA.CAXIS.xn**(V.) [ch].MPA.CAXIS.sn****(V.) [ch].SP.CAXIS.sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary axes and spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Channel [ch]. Possibility of working as C axis.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.CAXIS.Z	Z axis.
V.MPA.CAXIS.S	Spindle S.
V.SP.CAXIS.S	Spindle S.
V.SP.CAXIS	Master spindle.
V.MPA.CAXIS.4	Axis or spindle with logic number -4-.



CNC 8060
CNC 8065

(REF: 1402)

V.[2].MPA.CAXIS.1	Axis with index ·1· in the channel ·2·.
V.SP.CAXIS.2	Spindle with index ·2· in the system.
V.[2].SP.CAXIS.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.CAXSET.xn

(V.)[ch].MPA.CAXSET.sn

(V.)[ch].SP.CAXSET.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary axes and spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Set of parameters for working as C axis.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.CAXSET.Z	Z axis.
V.MPA.CAXSET.S	Spindle S.
V.SP.CAXSET.S	Spindle S.
V.SP.CAXSET	Master spindle.
V.MPA.CAXSET.4	Axis or spindle with logic number ·4·.
V.[2].MPA.CAXSET.1	Axis with index ·1· in the channel ·2·.
V.SP.CAXSET.2	Spindle with index ·2· in the system.
V.[2].SP.CAXSET.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.PERCAX.xn

(V.)[ch].MPA.PERCAX.sn

(V.)[ch].SP.PERCAX.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary axes and spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. C axis kept active after end of program, emergency or reset.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.PERCAX.Z	Z axis.
V.MPA.PERCAX.S	Spindle S.
V.SP.PERCAX.S	Spindle S.
V.SP.PERCAX	Master spindle.
V.MPA.PERCAX.4	Axis or spindle with logic number ·4·.
V.[2].MPA.PERCAX.1	Axis with index ·1· in the channel ·2·.
V.SP.PERCAX.2	Spindle with index ·2· in the system.
V.[2].SP.PERCAX.1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to axis and spindle machine parameters.



**CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

CONFIGURATION OF THE SPINDLE.

(V.)[ch].MPA.AUTOGEAR.sn

(V.)[ch].SP.AUTOGEAR.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Automatic gear change.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.MPA.AUTOGEAR.S	Spindle S.
V.SP.AUTOGEAR.S	Spindle S.
V.SP.AUTOGEAR	Master spindle.
V.MPA.AUTOGEAR.4	Spindle with logic number ·4·.
V.SP.AUTOGEAR.2	Spindle with index ·2· in the system.
V.[2].SP.AUTOGEAR.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.LOSPD LIM.sn

(V.)[ch].SP.LOSPD LIM.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Lower "rpm OK" percentage.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.MPA.LOSPD LIM.S	Spindle S.
V.SP.LOSPD LIM.S	Spindle S.
V.SP.LOSPD LIM	Master spindle.
V.MPA.LOSPD LIM.4	Spindle with logic number ·4·.
V.SP.LOSPD LIM.2	Spindle with index ·2· in the system.
V.[2].SP.LOSPD LIM.1	Spindle with index ·1· in the channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·100·.

(V.)[ch].MPA.UPSPDLIM.sn

(V.)[ch].SP.UPSPDLIM.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Upper "rpm OK" percentage.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.MPA.UPSPDLIM.S	Spindle S.
V.SP.UPSPDLIM.S	Spindle S.
V.SP.UPSPDLIM	Master spindle.
V.MPA.UPSPDLIM.4	Spindle with logic number ·4·.
V.SP.UPSPDLIM.2	Spindle with index ·2· in the system.
V.[2].SP.UPSPDLIM.1	Spindle with index ·1· in the channel ·2·.

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·100·.

(V.)[ch].MPA.SPDLTIME.sn

(V.)[ch].SP.SPDLTIME.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Estimated time for an S function.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.MPA.SPDLTIME.S	Spindle S.
V.SP.SPDLTIME.S	Spindle S.
V.SP.SPDLTIME	Master spindle.
V.MPA.SPDLTIME.4	Spindle with logic number ·4·.
V.SP.SPDLTIME.2	Spindle with index ·2· in the system.
V.[2].SP.SPDLTIME.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.SPDLSTOP.sn

(V.)[ch].SP.SPDLSTOP.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Functions M2 and M30, an error or a reset stop the spindle.

Syntax.

·ch· Channel number.

22.**CNC VARIABLES.**

Variables related to axis and spindle machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

·sn· Name, logic number or index of the spindle.

V.MPA.SPDLSTOP.S	Spindle S.
V.SP.SPDLSTOP.S	Spindle S.
V.SP.SPDLSTOP	Master spindle.
V.MPA.SPDLSTOP.4	Spindle with logic number ·4·.
V.SP.SPDLSTOP.2	Spindle with index ·2· in the system.
V.[2].SP.SPDLSTOP.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.SREVM05.sn

(V.)[ch].SP.SREVM05.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. With G84, the spindle must be stopped to change the turning direction.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.MPA.SREVM05.S	Spindle S.
V.SP.SREVM05.S	Spindle S.
V.SP.SREVM05	Master spindle.
V.MPA.SREVM05.4	Spindle with logic number ·4·.
V.SP.SREVM05.2	Spindle with index ·2· in the system.
V.[2].SP.SREVM05.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.M19SPDLEREV.sn

(V.)[ch].SP.M19SPDLEREV.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for analog drives.

The variable returns the value of block preparation.

Channel [ch]. The SPDLEREV mark (reverse turning direction) affects the spindle in M19.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.MPA.M19SPDLEREV.S	Spindle S.
V.SP.M19SPDLEREV.S	Spindle S.
V.SP.M19SPDLEREV	Master spindle.
V.MPA.M19SPDLEREV.4	Spindle with logic number ·4·.
V.SP.M19SPDLEREV.2	Spindle with index ·2· in the system.
V.[2].SP.M19SPDLEREV.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.STEPOVR.sn

(V.)[ch].SP.STEPOVR.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Spindle override step.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.MPA.STEPOVR.S	Spindle S.
V.SP.STEPOVR.S	Spindle S.
V.SP.STEPOVR	Master spindle.
V.MPA.STEPOVR.4	Spindle with logic number ·4·.
V.SP.STEPOVR.2	Spindle with index ·2· in the system.
V.[2].SP.STEPOVR.1	Spindle with index ·1· in the channel ·2·.

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·100·.

(V.)[ch].MPA.MINOVR.sn

(V.)[ch].SP.MINOVR.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Minimum override (%) permitted for the spindle.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.MPA.MINOVR.S	Spindle S.
V.SP.MINOVR.S	Spindle S.
V.SP.MINOVR	Master spindle.



22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.

V.MPA.MINOVR.4	Spindle with logic number -4-.
V.SP.MINOVR.2	Spindle with index -2- in the system.
V.[2].SP.MINOVR.1	Spindle with index -1- in the channel -2-.

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is -10-, the PLC reading will return a value of -100-.

(V.) [ch].MPA.MAXOVR.sn

(V.) [ch].SP.MAXOVR.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Maximum override (%) permitted for the spindle.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.MPA.MAXOVR.S	Spindle S.
V.SP.MAXOVR.S	Spindle S.
V.SP.MAXOVR	Master spindle.
V.MPA.MAXOVR.4	Spindle with logic number -4-.
V.SP.MAXOVR.2	Spindle with index -2- in the system.
V.[2].SP.MAXOVR.1	Spindle with index -1- in the channel -2-.

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is -10-, the PLC reading will return a value of -100-.

SYNCHRONIZATION OF AXES AND SPINDLES.

(V.) [ch].MPA.SYNCSET.xn

(V.) [ch].SP.SYNCSET.sn

(V.) [ch].SP.SYNCSET.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary axes and spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Set of parameters for synchronization.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.SYNCSET.Z	Z axis.
V.MPA.SYNCSET.S	Spindle S.
V.SP.SYNCSET.S	Spindle S.
V.SP.SYNCSET	Master spindle.
V.MPA.SYNCSET.4	Axis or spindle with logic number -4-.
V.[2].MPA.SYNCSET.1	Axis with index -1- in the channel -2-.
V.SP.SYNCSET.2	Spindle with index -2- in the system.
V.[2].SP.SYNCSET.1	Spindle with index -1- in the channel -2-.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Do not force the parameter set.
1 to 4	Parameter set.

(V.) [ch].MPA.DSYNCVELW.xn

(V.) [ch].SP.DSYNCVELW.sn

(V.) [ch].SP.DSYNCVELW.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Velocity synchronization window.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.DSYNCVELW.Z	Z axis.
V.MPA.DSYNCVELW.S	Spindle S.
V.SP.DSYNCVELW.S	Spindle S.
V.SP.DSYNCVELW	Master spindle.
V.MPA.DSYNCVELW.4	Axis or spindle with logic number ·4·.
V.[2].MPA.DSYNCVELW.1	Axis with index ·1· in the channel ·2·.
V.SP.DSYNCVELW.2	Spindle with index ·2· in the system.
V.[2].SP.DSYNCVELW.1	Spindle with index ·1· in the channel ·2·.

(V.) [ch].MPA.DSYNCPOSW.xn

(V.) [ch].SP.DSYNCPOSW.sn

(V.) [ch].SP.DSYNCPOSW.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Position synchronization window.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.DSYNCPOSW.Z	Z axis.
V.MPA.DSYNCPOSW.S	Spindle S.
V.SP.DSYNCPOSW.S	Spindle S.
V.SP.DSYNCPOSW	Master spindle.
V.MPA.DSYNCPOSW.4	Axis or spindle with logic number ·4·.
V.[2].MPA.DSYNCPOSW.1	Axis with index ·1· in the channel ·2·.
V.SP.DSYNCPOSW.2	Spindle with index ·2· in the system.
V.[2].SP.DSYNCPOSW.1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to axis and spindle machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

SOFTWARE AXIS LIMITS.

(V.) [ch].MPA.POSLIMIT.xn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Positive software limit.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.POSLIMIT.Z	Z axis.
V.MPA.POSLIMIT.4	Axis with logic number ·4·.
V.[2].MPA.POSLIMIT.1	Axis with index ·1· in the channel ·2·.

(V.) [ch].MPA.NEGLIMIT.xn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Negative software limit.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.NEGLIMIT.Z	Z axis.
V.MPA.NEGLIMIT.4	Axis with logic number ·4·.
V.[2].MPA.NEGLIMIT.1	Axis with index ·1· in the channel ·2·.

(V.) [ch].MPA.SWLIMITTOL.xn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Software limit tolerance.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.SWLIMITTOL.Z	Z axis.
V.MPA.SWLIMITTOL.4	Axis with logic number ·4·.
V.[2].MPA.SWLIMITTOL.1	Axis with index ·1· in the channel ·2·.

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

OVERRIDE CHANGE WHILE THREADING.

(V.)[ch].MPA.THREADOVR.sn

(V.)[ch].SP.THREADOVR.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Maximum variation allowed for the override while threading.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.MPA.THREADOVR.S	Spindle S.
V.SP.THREADOVR.S	Spindle S.
V.SP.THREADOVR	Master spindle.
V.MPA.THREADOVR.4	Spindle with logic number ·4·.
V.SP.THREADOVR.2	Spindle with index ·2· in the system.
V.[2].SP.THREADOVR.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.OVRFILTER.sn

(V.)[ch].SP.OVRFILTER.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Time to make the override change effective.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.MPA.OVRFILTER.S	Spindle S.
V.SP.OVRFILTER.S	Spindle S.
V.SP.OVRFILTER	Master spindle.
V.MPA.OVRFILTER.4	Spindle with logic number ·4·.
V.SP.OVRFILTER.2	Spindle with index ·2· in the system.
V.[2].SP.OVRFILTER.1	Spindle with index ·1· in the channel ·2·.

RUNAWAY PROTECTION AND TENDENCY TEST.

(V.)[ch].MPA.TENDENCY.xn

(V.)[ch].SP.TENDENCY.sn

(V.)[ch].SP.TENDENCY.sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drives.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Activation of tendency test.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

22.

CNC VARIABLES.

Variables related to axis and spindle machine parameters.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.

·sn· Name, logic number or index of the spindle.

V.MPA.TENDENCY.Z	Z axis.
V.MPA.TENDENCY.S	Spindle S.
V.SP.TENDENCY.S	Spindle S.
V.SP.TENDENCY	Master spindle.
V.MPA.TENDENCY.4	Axis or spindle with logic number ·4·.
V.[2].MPA.TENDENCY.1	Axis with index ·1· in the channel ·2·.
V.SP.TENDENCY.2	Spindle with index ·2· in the system.
V.[2].SP.TENDENCY.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.TENDTIME.xn

(V.)[ch].SP.TENDTIME.sn

(V.)[ch].SP.TENDTIME.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drives.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Time to detect axis runaway.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.MPA.TENDTIME.Z	Z axis.
V.MPA.TENDTIME.S	Spindle S.
V.SP.TENDTIME.S	Spindle S.
V.SP.TENDTIME	Master spindle.
V.MPA.TENDTIME.4	Axis or spindle with logic number ·4·.
V.[2].MPA.TENDTIME.1	Axis with index ·1· in the channel ·2·.
V.SP.TENDTIME.2	Spindle with index ·2· in the system.
V.[2].SP.TENDTIME.1	Spindle with index ·1· in the channel ·2·.

PLC OFFSET.

(V.)[ch].MPA.PLCOINC.xn

(V.)[ch].MPA.PLCOINC.sn

(V.)[ch].SP.PLCOINC.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. PLC offset increment per cycle.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.



CNC 8060
CNC 8065

(REF: 1402)

·sn· Name, logic number or index of the spindle.

V.MPA.PLCOINC.Z	Z axis.
V.MPA.PLCOINC.S	Spindle S.
V.SP.PLCOINC.S	Spindle S.
V.SP.PLCOINC	Master spindle.
V.MPA.PLCOINC.4	Axis or spindle with logic number ·4·.
V.[2].MPA.PLCOINC.1	Axis with index ·1· in the channel ·2·.
V.SP.PLCOINC.2	Spindle with index ·2· in the system.
V.[2].SP.PLCOINC.1	Spindle with index ·1· in the channel ·2·.

DWELL FOR DEAD AXES.

(V.)[ch].MPA.DWELL.xn

(V.)[ch].MPA.DWELL.sn

(V.)[ch].SP.DWELL.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Dwell for dead axes.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.MPA.DWELL.Z	Z axis.
V.MPA.DWELL.S	Spindle S.
V.SP.DWELL.S	Spindle S.
V.SP.DWELL	Master spindle.
V.MPA.DWELL.4	Axis or spindle with logic number ·4·.
V.[2].MPA.DWELL.1	Axis with index ·1· in the channel ·2·.
V.SP.DWELL.2	Spindle with index ·2· in the system.
V.[2].SP.DWELL.1	Spindle with index ·1· in the channel ·2·.

PROGRAMMING IN RADIUS OR DIAMETERS.

(V.)[ch].MPA.DIAMPROG.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Programming in diameters.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.DIAMPROG.Z	Z axis.
V.MPA.DIAMPROG.4	Axis with logic number ·4·.
V.[2].MPA.DIAMPROG.1	Axis with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to axis and spindle machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

HOME SEARCH.

(V.)[ch].MPA.REFDIREC.xn

(V.)[ch].MPA.REFDIREC.sn

(V.)[ch].SP.REFDIREC.sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Direction of the search.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.REFDIREC.Z	Z axis.
V.MPA.REFDIREC.S	Spindle S.
V.SP.REFDIREC.S	Spindle S.
V.SP.REFDIREC	Master spindle.
V.MPA.REFDIREC.4	Axis or spindle with logic number ·4·.
V.[2].MPA.REFDIREC.1	Axis with index ·1· in the channel ·2·.
V.SP.REFDIREC.2	Spindle with index ·2· in the system.
V.[2].SP.REFDIREC.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Movement in the negative direction.
1	Movement in the positive direction.

(V.)[ch].MPA.DECINPUT.xn

(V.)[ch].MPA.DECINPUT.sn

(V.)[ch].SP.DECINPUT.sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. The axis/spindle has a home switch.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.



CNC 8060
CNC 8065

(REF: 1402)

.sn· Name, logic number or index of the spindle.

V.MPA.DECINPUT.Z	Z axis.
V.MPA.DECINPUT.S	Spindle S.
V.SP.DECINPUT.S	Spindle S.
V.SP.DECINPUT	Master spindle.
V.MPA.DECINPUT.4	Axis or spindle with logic number .4.
V.[2].MPA.DECINPUT.1	Axis with index .1 in the channel .2.
V.SP.DECINPUT.2	Spindle with index .2 in the system.
V.[2].SP.DECINPUT.1	Spindle with index .1 in the channel .2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.REFINI.sn
(V.)[ch].SP.REFINI.sn

Variable that can only be read from the program, PLC and interface.
Variable valid for spindles.
Variable valid for analog drive, position Sercos and velocity Sercos.
The variable returns the value of block preparation.

Channel [ch]. Home search in the first movement

Syntax.

.ch· Channel number.
 .sn· Name, logic number or index of the spindle.

V.MPA.REFINI.S	Spindle S.
V.SP.REFINI.S	Spindle S.
V.SP.REFINI	Master spindle.
V.MPA.REFINI.4	Spindle with logic number .4.
V.SP.REFINI.2	Spindle with index .2 in the system.
V.[2].SP.REFINI.1	Spindle with index .1 in the channel .2.

CONFIGURATION OF THE PROBING MOVEMENT.

(V.)[ch].MPA.PROBEAXIS.xn

Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes.
Variable valid for analog drive, position Sercos and velocity Sercos.
The variable returns the value of block preparation.

Channel [ch]. The axis can participate in probing movements.

Syntax.

.ch· Channel number.
 .xn· Name, logic number or index of the axis.

V.MPA.PROBEAXIS.Z	Z axis.
V.MPA.PROBEAXIS.4	Axis with logic number .4.
V.[2].MPA.PROBEAXIS.1	Axis with index .1 in the channel .2.



CNC 8060
 CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.) [ch].MPA.PROBERANGE.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Maximum braking distance.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.PROBERANGE.Z	Z axis.
V.MPA.PROBERANGE.4	Axis with logic number ·4·.
V.[2].MPA.PROBERANGE.1	Axis with index ·1· in the channel ·2·.

(V.) [ch].MPA.PROBEFEED.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Maximum probing feedrate.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.PROBEFEED.Z	Z axis.
V.MPA.PROBEFEED.4	Axis with logic number ·4·.
V.[2].MPA.PROBEFEED.1	Axis with index ·1· in the channel ·2·.

(V.) [ch].MPA.PROBEDELAY.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Delay for the "probe 1" signal.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.PROBEDELAY.Z	Z axis.
V.MPA.PROBEDELAY.4	Axis with logic number ·4·.
V.[2].MPA.PROBEDELAY.1	Axis with index ·1· in the channel ·2·.

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.PROBEDELAY2.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Delay for the "probe 2" signal.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.PROBEDELAY2.Z	Z axis.
V.MPA.PROBEDELAY2.4	Axis with logic number ·4·.
V.[2].MPA.PROBEDELAY2.1	Axis with index ·1· in the channel ·2·.

REPOSITIONING OF THE AXES IN TOOL INSPECTION.**(V.)[ch].MPA.REPOSFEED.xn**

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Maximum repositioning feedrate.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.REPOSFEED.Z	Z axis.
V.MPA.REPOSFEED.4	Axis with logic number ·4·.
V.[2].MPA.REPOSFEED.1	Axis with index ·1· in the channel ·2·.

CONFIGURATION OF THE INDEPENDENT AXIS.**(V.)[ch].MPA.POSFEED.xn****(V.)[ch].MPA.POSFEED.sn****(V.)[ch].SP.POSFEED.sn**

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Positioning feedrate

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.MPA.POSFEED.Z	Z axis.
V.MPA.POSFEED.S	Spindle S.
V.SP.POSFEED.S	Spindle S.
V.SP.POSFEED	Master spindle.
V.MPA.POSFEED.4	Axis or spindle with logic number ·4·.

22.**CNC VARIABLES.**

Variables related to axis and spindle machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.

V.[2].MPA.POSFEED.1	Axis with index -1- in the channel -2-.
V.SP.POSFEED.2	Spindle with index -2- in the system.
V.[2].SP.POSFEED.1	Spindle with index -1- in the channel -2-.

CONFIGURE THE MAXIMUM SAFETY LIMIT FOR THE FEEDRATE AND FOR THE SPEED.

(V.)[ch].MPA.FLIMIT.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Maximum safety limit for axis feedrate.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.FLIMIT.Z	Z axis.
V.MPA.FLIMIT.4	Axis with logic number -4-.
V.[2].MPA.FLIMIT.1	Axis with index -1- in the channel -2-.

(V.)[ch].MPA.SLIMIT.sn

(V.)[ch].SP.SLIMIT.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Maximum safety limit for spindle speed.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.MPA.SLIMIT.S	Spindle S.
V.SP.SLIMIT.S	Spindle S.
V.SP.SLIMIT	Master spindle.
V.MPA.SLIMIT.4	Spindle with logic number -4-.
V.SP.SLIMIT.2	Spindle with index -2- in the system.
V.[2].SP.SLIMIT.1	Spindle with index -1- in the channel -2-.

JOG MODE. CONTINUOUS JOG.

(V.)[ch].MPA.JOGFEED.xn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Feedrate in continuous jog.

Syntax.

·ch· Channel number.



CNC 8060
CNC 8065

(REF: 1402)

·xn· Name, logic number or index of the axis.

V.MPA.JOGFEED.Z	Z axis.
V.MPA.JOGFEED.4	Axis with logic number ·4·.
V.[2].MPA.JOGFEED.1	Axis with index ·1· in the channel ·2·.

(V.)[ch].MPA.JOGRAPFEED.xn

Variable that can only be read from the program, PLC and interface.
 Variable that may be modified during setup.
 Variable valid for rotary and linear axes.
 Variable valid for analog drive, position Sercos and velocity Sercos.
 The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Rapid feedrate in continuous jog.

Syntax.

·ch· Channel number.
 ·xn· Name, logic number or index of the axis.

V.MPA.JOGRAPFEED.Z	Z axis.
V.MPA.JOGRAPFEED.4	Axis with logic number ·4·.
V.[2].MPA.JOGRAPFEED.1	Axis with index ·1· in the channel ·2·.

(V.)[ch].MPA.MAXMANFEED.xn

Variable that can only be read from the program, PLC and interface.
 Variable that may be modified during setup.
 Variable valid for rotary and linear axes.
 Variable valid for analog drive, position Sercos and velocity Sercos.
 The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Maximum feedrate in continuous jog.

Syntax.

·ch· Channel number.
 ·xn· Name, logic number or index of the axis.

V.MPA.MAXMANFEED.Z	Z axis.
V.MPA.MAXMANFEED.4	Axis with logic number ·4·.
V.[2].MPA.MAXMANFEED.1	Axis with index ·1· in the channel ·2·.

(V.)[ch].MPA.MAXMANACC.xn

Variable that can only be read from the program, PLC and interface.
 Variable valid for rotary and linear axes.
 Variable valid for analog drive, position Sercos and velocity Sercos.
 The variable returns the value of block preparation.

Channel [ch]. Maximum acceleration in continuous JOG.

Syntax.

·ch· Channel number.
 ·xn· Name, logic number or index of the axis.

V.MPA.MAXMANACC.Z	Z axis.
V.MPA.MAXMANACC.4	Axis with logic number ·4·.
V.[2].MPA.MAXMANACC.1	Axis with index ·1· in the channel ·2·.



JOG MODE. INCREMENTAL JOG.

(V.) [ch].MPA.INCJOGDIST[pos].xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Distance to move in incremental jog, [pos] dial position.

pos=1 for position -1-, pos=2 for position -10- and so on.

Syntax.

- ch· Channel number.
- pos· Position of the switch on the operator panel.
- xn· Name, logic number or index of the axis..

V.MPA.INCJOGDIST[4].Z	Z axis.
V.MPA.INCJOGDIST[4].4	Axis with logic number -4-.
V.[2].MPA.INCJOGDIST[4].1	Axis with index -1- in the channel -2-.

(V.) [ch].MPA.INCJOGFEED[pos].xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Feedrate in incremental jog, [pos] dial position.

pos=1 for position -1-, pos=2 for position -10- and so on.

Syntax.

- ch· Channel number.
- pos· Position of the switch on the operator panel for incremental jog.
- xn· Name, logic number or index of the axis.

V.MPA.INCJOGFEED[4].Z	Z axis.
V.MPA.INCJOGFEED[4].4	Axis with logic number -4-.
V.[2].MPA.INCJOGFEED[4].1	Axis with index -1- in the channel -2-.

JOG MODE. HANDWHEELS.

(V.) [ch].MPA.MPGRESOL[pos].xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Resolution of the handwheel, in the [pos] position.

pos=1 for position -1-, pos=2 for position -10- and pos=3 for -100-.

Syntax.

- ch· Channel number.
- pos· Position of the switch on the operator panel for the handwheel mode.
- xn· Name, logic number or index of the axis.

V.MPA.MPGRESOL[2].Z	Z axis.
V.MPA.MPGRESOL[2].4	Axis with logic number -4-.
V.[2].MPA.MPGRESOL[2].1	Axis with index -1- in the channel -2-.

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.MPGFILTER.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Filter time for the handwheel.

Syntax.

- ch· Channel number.
- pos· Position of the switch on the operator panel for the handwheel mode.
- xn· Name, logic number or index of the axis.

V.MPA.MPGFILTER[2].Z	Z axis.
V.MPA.MPGFILTER[2].4	Axis with logic number -4-.
V.[2].MPA.MPGFILTER[2].1	Axis with index -1- in the channel -2-.

JOG MODE. MANUAL INTERVENTION.**(V.)[ch].MPA.MANPOSSW.xn**

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Maximum positive travel with G201.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

V.MPA.MANPOSSW.Z	Z axis.
V.MPA.MANPOSSW.4	Axis with logic number -4-.
V.[2].MPA.MANPOSSW.1	Axis with index -1- in the channel -2-.

(V.)[ch].MPA.MANNEGSW.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Maximum negative travel with G201.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

V.MPA.MANNEGSW.Z	Z axis.
V.MPA.MANNEGSW.4	Axis with logic number -4-.
V.[2].MPA.MANNEGSW.1	Axis with index -1- in the channel -2-.

(V.)[ch].MPA.MANFEEDP.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Maximum % of jog feedrate in G201.

22.**CNC VARIABLES.**

Variables related to axis and spindle machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.

Variables related to axis and spindle machine parameters.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

V.MPA.MANFEEDP.Z	Z axis.
V.MPA.MANFEEDP.4	Axis with logic number -4-.
V.[2].MPA.MANFEEDP.1	Axis with index -1- in the channel -2-.

(V.)[ch].MPA.IPOFEEDP.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Maximum % of execution feedrate in G201.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

V.MPA.IPOFEEDP.Z	Z axis.
V.MPA.IPOFEEDP.4	Axis with logic number -4-.
V.[2].MPA.IPOFEEDP.1	Axis with index -1- in the channel -2-.

(V.)[ch].MPA.MANACCP.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Maximum % of jog acceleration in G201.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

V.MPA.MANACCP.Z	Z axis.
V.MPA.MANACCP.4	Axis with logic number -4-.
V.[2].MPA.MANACCP.1	Axis with index -1- in the channel -2-.

(V.)[ch].MPA.IPOACCP.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Maximum % of execution acceleration in G201.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

V.MPA.IPOACCP.Z	Z axis.
V.MPA.IPOACCP.4	Axis with logic number -4-.
V.[2].MPA.IPOACCP.1	Axis with index -1- in the channel -2-.



CNC 8060
CNC 8065

(REF: 1402)

LEADSCREW ERROR COMPENSATION.

(V.)[ch].MPA.LSCRWCOMP.xn**(V.)[ch].MPA.LSCRWCOMP.sn****(V.)[ch].SP.LSCRWCOMP.sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Channel [ch]. Leadscrew error compensation.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.LSCRWCOMP.Z	Z axis.
V.MPA.LSCRWCOMP.S	Spindle S.
V.SP.LSCRWCOMP.S	Spindle S.
V.SP.LSCRWCOMP	Master spindle.
V.MPA.LSCRWCOMP.4	Axis or spindle with logic number -4.
V.[2].MPA.LSCRWCOMP.1	Axis with index -1 in the channel -2.
V.SP.LSCRWCOMP.2	Spindle with index -2 in the system.
V.[2].SP.LSCRWCOMP.1	Spindle with index -1 in the channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.NPOINTS.xn**(V.)[ch].MPA.NPOINTS.sn****(V.)[ch].SP.NPOINTS.sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Channel [ch]. Number of points in the table.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.NPOINTS.Z	Z axis.
V.MPA.NPOINTS.S	Spindle S.
V.SP.NPOINTS.S	Spindle S.
V.SP.NPOINTS	Master spindle.
V.MPA.NPOINTS.4	Axis or spindle with logic number -4.
V.[2].MPA.NPOINTS.1	Axis with index -1 in the channel -2.
V.SP.NPOINTS.2	Spindle with index -2 in the system.
V.[2].SP.NPOINTS.1	Spindle with index -1 in the channel -2.

22.**CNC VARIABLES.**

Variables related to axis and spindle machine parameters.

FAGOR **CNC 8060**
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.

(V.) [ch].MPA.TYPLSCRW.xn**(V.) [ch].MPA.TYPLSCRW.sn****(V.) [ch].SP.TYPLSCRW.sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Channel [ch]. Type of compensation (type of coordinates).

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.TYPLSCRW.Z	Z axis.
V.MPA.TYPLSCRW.S	Spindle S.
V.SP.TYPLSCRW.S	Spindle S.
V.SP.TYPLSCRW	Master spindle.
V.MPA.TYPLSCRW.4	Axis or spindle with logic number -4-.
V.[2].MPA.TYPLSCRW.1	Axis with index -1- in the channel -2-.
V.SP.TYPLSCRW.2	Spindle with index -2- in the system.
V.[2].SP.TYPLSCRW.1	Spindle with index -1- in the channel -2-.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Compensation is applied using real coordinates.
1	Compensation is applied using theoretical coordinates.

(V.) [ch].MPA.BIDIR.xn**(V.) [ch].MPA.BIDIR.sn****(V.) [ch].SP.BIDIR.sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Channel [ch]. Bi-directional compensation.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.BIDIR.Z	Z axis.
V.MPA.BIDIR.S	Spindle S.
V.SP.BIDIR.S	Spindle S.
V.SP.BIDIR	Master spindle.
V.MPA.BIDIR.4	Axis or spindle with logic number -4-.
V.[2].MPA.BIDIR.1	Axis with index -1- in the channel -2-.
V.SP.BIDIR.2	Spindle with index -2- in the system.
V.[2].SP.BIDIR.1	Spindle with index -1- in the channel -2-.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.REFNEED.xn

(V.)[ch].MPA.REFNEED.sn

(V.)[ch].SP.REFNEED.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. The axis must be homed in order to apply the compensation.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.REFNEED.Z	Z axis.
V.MPA.REFNEED.S	Spindle S.
V.SP.REFNEED.S	Spindle S.
V.SP.REFNEED	Master spindle.
V.MPA.REFNEED.4	Axis or spindle with logic number ·4·.
V.[2].MPA.REFNEED.1	Axis with index ·1· in the channel ·2·.
V.SP.REFNEED.2	Spindle with index ·2· in the system.
V.[2].SP.REFNEED.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.POSITION[pt].xn

(V.)[ch].MPA.POSITION[pt].sn

(V.)[ch].SP.POSITION[pt].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Master axis position for point [pt]

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.
- pt· Point in the table.

V.MPA.POSITION[13].Z	Z axis.
V.MPA.POSITION[13].S	Spindle S.
V.SP.POSITION[13].S	Spindle S.
V.SP.POSITION[13]	Master spindle.

22.

CNC VARIABLES.

Variables related to axis and spindle machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.

Variables related to axis and spindle machine parameters.

V.MPA.POSITION[13].4	Axis or spindle with logic number -4.
V.[2].MPA.POSITION[13].1	Axis with index -1- in the channel -2.
V.SP.POSITION[13].2	Spindle with index -2- in the system.
V.[2].SP.POSITION[13].1	Spindle with index -1- in the channel -2.

(V.) [ch].MPA.POSERROR[pt].xn**(V.) [ch].MPA.POSERROR[pt].sn****(V.) [ch].SP.POSERROR[pt].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Channel [ch]. Amount of error at point [pt] in the positive direction

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.
- pt· Point in the table.

V.MPA.POSERROR[13].Z	Z axis.
V.MPA.POSERROR[13].S	Spindle S.
V.SP.POSERROR[13].S	Spindle S.
V.SP.POSERROR[13]	Master spindle.
V.MPA.POSERROR[13].4	Axis or spindle with logic number -4.
V.[2].MPA.POSERROR[13].1	Axis with index -1- in the channel -2.
V.SP.POSERROR[13].2	Spindle with index -2- in the system.
V.[2].SP.POSERROR[13].1	Spindle with index -1- in the channel -2.

(V.) [ch].MPA.NEGERROR[pt].xn**(V.) [ch].MPA.NEGERROR[pt].sn****(V.) [ch].SP.NEGERROR[pt].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Channel [ch]. Amount of error at point [pt] in the negative direction.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.
- pt· Point in the table.

V.MPA.NEGERROR[13].Z	Z axis.
V.MPA.NEGERROR[13].S	Spindle S.
V.SP.NEGERROR[13].S	Spindle S.
V.SP.NEGERROR[13]	Master spindle.
V.MPA.NEGERROR[13].4	Axis or spindle with logic number -4.
V.[2].MPA.NEGERROR[13].1	Axis with index -1- in the channel -2.
V.SP.NEGERROR[13].2	Spindle with index -2- in the system.
V.[2].SP.NEGERROR[13].1	Spindle with index -1- in the channel -2.



CNC 8060
CNC 8065

(REF: 1402)

FILTERS TO ELIMINATE FREQUENCIES.

(V.)[ch].MPA.ORDER[nb].xn

(V.)[ch].MPA.ORDER[nb].sn

(V.)[ch].SP.ORDER[nb].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Filter order.

Syntax.

- ch· Channel number.
- nb· Filter number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.ORDER[3].Z	Z axis.
V.MPA.ORDER[3].S	Spindle S.
V.SP.ORDER[3].S	Spindle S.
V.SP.ORDER[3]	Master spindle.
V.MPA.ORDER[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.ORDER[3].1	Axis with index ·1· in the channel ·2·.
V.SP.ORDER[3].2	Spindle with index ·2· in the system.
V.[2].SP.ORDER[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.TYPE[nb].xn

(V.)[ch].MPA.TYPE[nb].sn

(V.)[ch].SP.TYPE[nb].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Type of filter.

Syntax.

- ch· Channel number.
- nb· Filter number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.TYPE[3].Z	Z axis.
V.MPA.TYPE[3].S	Spindle S.
V.SP.TYPE[3].S	Spindle S.
V.SP.TYPE[3]	Master spindle.
V.MPA.TYPE[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.TYPE[3].1	Axis with index ·1· in the channel ·2·.
V.SP.TYPE[3].2	Spindle with index ·2· in the system.
V.[2].SP.TYPE[3].1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to axis and spindle machine parameters.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Low passing filter.
1	Anti-resonance Filter.
2	FAGOR low passing filter.

(V.) [ch].MPA.FREQUENCY[nb].xn

(V.) [ch].MPA.FREQUENCY[nb].sn

(V.) [ch].SP.FREQUENCY[nb].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Cutoff or center frequency

Syntax.

- ch· Channel number.
- nb· Filter number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.FREQUENCY[3].Z	Z axis.
V.MPA.FREQUENCY[3].S	Spindle S.
V.SP.FREQUENCY[3].S	Spindle S.
V.SP.FREQUENCY[3]	Master spindle.
V.MPA.FREQUENCY[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.FREQUENCY[3].1	Axis with index -1- in the channel -2-.
V.SP.FREQUENCY[3].2	Spindle with index -2- in the system.
V.[2].SP.FREQUENCY[3].1	Spindle with index -1- in the channel -2-.

(V.) [ch].MPA.NORBWIDTH[nb].xn

(V.) [ch].MPA.NORBWIDTH[nb].sn

(V.) [ch].SP.NORBWIDTH[nb].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Normal bandwidth.

Syntax.

- ch· Channel number.
- nb· Filter number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.NORBWIDTH[3].Z	Z axis.
V.MPA.NORBWIDTH[3].S	Spindle S.
V.SP.NORBWIDTH[3].S	Spindle S.
V.SP.NORBWIDTH[3]	Master spindle.
V.MPA.NORBWIDTH[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.NORBWIDTH[3].1	Axis with index -1- in the channel -2-.
V.SP.NORBWIDTH[3].2	Spindle with index -2- in the system.
V.[2].SP.NORBWIDTH[3].1	Spindle with index -1- in the channel -2-.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.SHARE[nb].xn**(V.)[ch].MPA.SHARE[nb].sn****(V.)[ch].SP.SHARE[nb].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Channel [ch]. Percentage of signal going through the filter.

Syntax.

- ch· Channel number.
- nb· Filter number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.SHARE[3].Z	Z axis.
V.MPA.SHARE[3].S	Spindle S.
V.SP.SHARE[3].S	Spindle S.
V.SP.SHARE[3]	Master spindle.
V.MPA.SHARE[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.SHARE[3].1	Axis with index ·1· in the channel ·2·.
V.SP.SHARE[3].2	Spindle with index ·2· in the system.
V.[2].SP.SHARE[3].1	Spindle with index ·1· in the channel ·2·.

PARAMETER SETS.**(V.)[ch].MPA.NPARSETS.xn****(V.)[ch].MPA.NPARSETS.sn****(V.)[ch].SP.NPARSETS.sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Channel [ch]. Number of parameter sets available.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.NPARSETS.Z	Z axis.
V.MPA.NPARSETS.S	Spindle S.
V.SP.NPARSETS.S	Spindle S.
V.SP.NPARSETS	Master spindle.
V.MPA.NPARSETS.4	Axis or spindle with logic number ·4·.
V.[2].MPA.NPARSETS.1	Axis with index ·1· in the channel ·2·.
V.SP.NPARSETS.2	Spindle with index ·2· in the system.
V.[2].SP.NPARSETS.1	Spindle with index ·1· in the channel ·2·.

22.**CNC VARIABLES.**

Variables related to axis and spindle machine parameters.

FAGOR **CNC 8060**
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to axis and spindle machine parameters.

(V.)[ch].MPA.DEFAULTSET.xn

(V.)[ch].MPA.DEFAULTSET.sn

(V.)[ch].SP.DEFAULTSET.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Default parameter set on power-up.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.DEFAULTSET.Z	Z axis.
V.MPA.DEFAULTSET.S	Spindle S.
V.SP.DEFAULTSET.S	Spindle S.
V.SP.DEFAULTSET	Master spindle.
V.MPA.DEFAULTSET.4	Axis or spindle with logic number ·4·.
V.[2].MPA.DEFAULTSET.1	Axis with index ·1· in the channel ·2·.
V.SP.DEFAULTSET.2	Spindle with index ·2· in the system.
V.[2].SP.DEFAULTSET.1	Spindle with index ·1· in the channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

22.7 Variables related to the sets of machine parameters.

FEEDBACK RESOLUTION.

(V.)[ch].MPA.PITCH[set].xn
 (V.)[ch].MPA.PITCH[set].sn
 (V.)[ch].SP.PITCH[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Leadscrew pitch.

Depending on the type of feedback, this parameter means the following.

- On a linear axis with a rotary encoder and leadscrew, it defines the leadscrew pitch.
- On a linear axis with a linear encoder (scale), it defines the pitch of the scale.
- On a rotary axis, it sets the number of degrees per turn of the encoder.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.PITCH[3].Z	Z axis.
V.MPA.PITCH[3].S	Spindle S.
V.SP.PITCH[3].S	Spindle S.
V.SP.PITCH[3]	Master spindle.
V.MPA.PITCH[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.PITCH[3].1	Axis with index ·1· in the channel ·2·.
V.SP.PITCH[3].2	Spindle with index ·2· in the system.
V.[2].SP.PITCH[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.INPUTREV[set].xn
 (V.)[ch].MPA.INPUTREV[set].sn
 (V.)[ch].SP.INPUTREV[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Gear ratio; turns of the motor shaft.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.INPUTREV[3].Z	Z axis.
V.MPA.INPUTREV[3].S	Spindle S.
V.SP.INPUTREV[3].S	Spindle S.
V.SP.INPUTREV[3]	Master spindle.
V.MPA.INPUTREV[3].4	Axis or spindle with logic number ·4·.

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

V.[2].MPA.INPUTREV[3].1	Axis with index -1- in the channel -2-.
V.SP.INPUTREV[3].2	Spindle with index -2- in the system.
V.[2].SP.INPUTREV[3].1	Spindle with index -1- in the channel -2-.

(V.)[ch].MPA.OUTPUTREV[set].xn

(V.)[ch].MPA.OUTPUTREV[set].sn

(V.)[ch].SP.OUTPUTREV[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Gear ratio; turns of the machine shaft.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.OUTPUTREV[3].Z	Z axis.
V.MPA.OUTPUTREV[3].S	Spindle S.
V.SP.OUTPUTREV[3].S	Spindle S.
V.SP.OUTPUTREV[3]	Master spindle.
V.MPA.OUTPUTREV[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.OUTPUTREV[3].1	Axis with index -1- in the channel -2-.
V.SP.OUTPUTREV[3].2	Spindle with index -2- in the system.
V.[2].SP.OUTPUTREV[3].1	Spindle with index -1- in the channel -2-.

(V.)[ch].MPA.NPULSES[set].xn

(V.)[ch].MPA.NPULSES[set].sn

(V.)[ch].SP.NPULSES[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drives and velocity-Sercos drives.

The variable returns the execution value; reading it interrupts block preparation.

Number of encoder pulses.

With linear encoders (scales), this parameter will be set to -0-. When using a gear reduction on the axis, the whole assembly must be taken into account when defining the number of pulses per turn.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.NPULSES[3].Z	Z axis.
V.MPA.NPULSES[3].S	Spindle S.
V.SP.NPULSES[3].S	Spindle S.
V.SP.NPULSES[3]	Master spindle.
V.MPA.NPULSES[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.NPULSES[3].1	Axis with index -1- in the channel -2-.
V.SP.NPULSES[3].2	Spindle with index -2- in the system.
V.[2].SP.NPULSES[3].1	Spindle with index -1- in the channel -2-.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.PITCH2[set].xn**(V.)[ch].MPA.PITCH2[set].sn****(V.)[ch].SP.PITCH2[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified during setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for position-Sercos drives and velocity-Sercos drives.**The variable returns the execution value; reading it interrupts block preparation.*

Leadscrew pitch (external feedback).

Depending on the type of feedback, this parameter means the following.

- On a linear axis with a rotary encoder and leadscrew, it defines the leadscrew pitch.
- On a linear axis with a linear encoder (scale), it defines the pitch of the scale.
- On a rotary axis, it sets the number of degrees per turn of the encoder.

Syntax.**.ch** Channel number.**.set** Parameter set.**.xn** Name, logic number or index of the axis.**.sn** Name, logic number or index of the spindle.

V.MPA.PITCH2[3].Z	Z axis.
V.MPA.PITCH2[3].S	Spindle S.
V.SP.PITCH2[3].S	Spindle S.
V.SP.PITCH2[3]	Master spindle.
V.MPA.PITCH2[3].4	Axis or spindle with logic number .4 .
V.[2].MPA.PITCH2[3].1	Axis with index .1 in the channel .2 .
V.SP.PITCH2[3].2	Spindle with index .2 in the system.
V.[2].SP.PITCH2[3].1	Spindle with index .1 in the channel .2 .

(V.)[ch].MPA.INPUTREV2[set].xn**(V.)[ch].MPA.INPUTREV2[set].sn****(V.)[ch].SP.INPUTREV2[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified during setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for position-Sercos drives and velocity-Sercos drives.**The variable returns the execution value; reading it interrupts block preparation.*

Gear ratio; turns of the motor shaft (external feedback).

Syntax.**.ch** Channel number.**.set** Parameter set.**.xn** Name, logic number or index of the axis.**.sn** Name, logic number or index of the spindle.

V.MPA.INPUTREV2[3].Z	Z axis.
V.MPA.INPUTREV2[3].S	Spindle S.
V.SP.INPUTREV2[3].S	Spindle S.
V.SP.INPUTREV2[3]	Master spindle.
V.MPA.INPUTREV2[3].4	Axis or spindle with logic number .4 .
V.[2].MPA.INPUTREV2[3].1	Axis with index .1 in the channel .2 .
V.SP.INPUTREV2[3].2	Spindle with index .2 in the system.
V.[2].SP.INPUTREV2[3].1	Spindle with index .1 in the channel .2 .

22.**CNC VARIABLES.**

Variables related to the sets of machine parameters.

FAGOR **CNC 8060**
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.

(V.)[ch].MPA.OUTPUTREV2[set].xn**(V.)[ch].MPA.OUTPUTREV2[set].sn****(V.)[ch].SP.OUTPUTREV2[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified during setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for position-Sercos drives and velocity-Sercos drives.**The variable returns the execution value; reading it interrupts block preparation.*

Gear ratio; turns of the machine shaft (external feedback).

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.OUTPUTREV2[3].Z	Z axis.
V.MPA.OUTPUTREV2[3].S	Spindle S.
V.SP.OUTPUTREV2[3].S	Spindle S.
V.SP.OUTPUTREV2[3]	Master spindle.
V.MPA.OUTPUTREV2[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.OUTPUTREV2[3].1	Axis with index -1- in the channel -2-.
V.SP.OUTPUTREV2[3].2	Spindle with index -2- in the system.
V.[2].SP.OUTPUTREV2[3].1	Spindle with index -1- in the channel -2-.

(V.)[ch].MPA.NPULSES2[set].xn**(V.)[ch].MPA.NPULSES2[set].sn****(V.)[ch].SP.NPULSES2[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified during setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for position-Sercos drives and velocity-Sercos drives.**The variable returns the execution value; reading it interrupts block preparation.*

Number of pulses per turn of the encoder (external feedback).

With linear encoders (scales), this parameter will be set to -0-. When using a gear reduction on the axis, the whole assembly must be taken into account when defining the number of pulses per turn.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.NPULSES2[3].Z	Z axis.
V.MPA.NPULSES2[3].S	Spindle S.
V.SP.NPULSES2[3].S	Spindle S.
V.SP.NPULSES2[3]	Master spindle.
V.MPA.NPULSES2[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.NPULSES2[3].1	Axis with index -1- in the channel -2-.
V.SP.NPULSES2[3].2	Spindle with index -2- in the system.
V.[2].SP.NPULSES2[3].1	Spindle with index -1- in the channel -2-.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.SINMAGNI[set].xn**(V.)[ch].MPA.SINMAGNI[set].sn****(V.)[ch].SP.SINMAGNI[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified during setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Multiplying factor for sinusoidal feedback signal.

For square feedback signals, it will be set to -0.; the CNC applies a x4 factor.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.SINMAGNI[3].Z	Z axis.
V.MPA.SINMAGNI[3].S	Spindle S.
V.SP.SINMAGNI[3].S	Spindle S.
V.SP.SINMAGNI[3]	Master spindle.
V.MPA.SINMAGNI[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.SINMAGNI[3].1	Axis with index -1- in the channel -2-.
V.SP.SINMAGNI[3].2	Spindle with index -2- in the system.
V.[2].SP.SINMAGNI[3].1	Spindle with index -1- in the channel -2-.

(V.)[ch].MPA.ABSFEEDBACK[set].xn**(V.)[ch].MPA.ABSFEEDBACK[set].sn****(V.)[ch].SP.ABSFEEDBACK[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified during setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Absolute feedback system.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.ABDFEEDBACK[3].Z	Z axis.
V.MPA.ABDFEEDBACK[3].S	Spindle S.
V.SP.ABDFEEDBACK[3].S	Spindle S.
V.SP.ABDFEEDBACK[3]	Master spindle.
V.MPA.ABDFEEDBACK[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.ABDFEEDBACK[3].1	Axis with index -1- in the channel -2-.
V.SP.ABDFEEDBACK[3].2	Spindle with index -2- in the system.
V.[2].SP.ABDFEEDBACK[3].1	Spindle with index -1- in the channel -2-.

22.**CNC VARIABLES.**

Variables related to the sets of machine parameters.

FAGOR **CNC 8060**
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.FBACKAL[set].xn

(V.)[ch].MPA.FBACKAL[set].sn

(V.)[ch].SP.FBACKAL[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drives.

The variable returns the execution value; reading it interrupts block preparation.

Activate feedback alarm.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.FBACKAL[3].Z	Z axis.
V.MPA.FBACKAL[3].S	Spindle S.
V.SP.FBACKAL[3].S	Spindle S.
V.SP.FBACKAL[3]	Master spindle.
V.MPA.FBACKAL[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.FBACKAL[3].1	Axis with index ·1· in the channel ·2·.
V.SP.FBACKAL[3].2	Spindle with index ·2· in the system.
V.[2].SP.FBACKAL[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

LOOP SETTING.

(V.)[ch].MPA.LOOPCH[set].xn

(V.)[ch].MPA.LOOPCH[set].sn

(V.)[ch].SP.LOOPCH[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Velocity command sign change.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.



CNC 8060
CNC 8065

(REF: 1402)

·sn· Name, logic number or index of the spindle.

V.MPA.LOOPCH[3].Z	Z axis.
V.MPA.LOOPCH[3].S	Spindle S.
V.SP.LOOPCH[3].S	Spindle S.
V.SP.LOOPCH[3]	Master spindle.
V.MPA.LOOPCH[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.LOOPCH[3].1	Axis with index ·1· in the channel ·2·.
V.SP.LOOPCH[3].2	Spindle with index ·2· in the system.
V.[2].SP.LOOPCH[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].MPA.AXISCH[set].xn
 (V.)[ch].MPA.AXISCH[set].sn
 (V.)[ch].SP.AXISCH[set].sn

Variable that can only be read from the program, PLC and interface.
Variable that may be modified during setup.
Variable valid for rotary and linear axes as well as for spindles.
Variable valid for analog drive, position Sercos and velocity Sercos.
The variable returns the execution value; reading it interrupts block preparation.

Feedback sign change.

Syntax.

·ch· Channel number.
 ·set· Parameter set.
 ·xn· Name, logic number or index of the axis.
 ·sn· Name, logic number or index of the spindle.

V.MPA.AXISCH[3].Z	Z axis.
V.MPA.AXISCH[3].S	Spindle S.
V.SP.AXISCH[3].S	Spindle S.
V.SP.AXISCH[3]	Master spindle.
V.MPA.AXISCH[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.AXISCH[3].1	Axis with index ·1· in the channel ·2·.
V.SP.AXISCH[3].2	Spindle with index ·2· in the system.
V.[2].SP.AXISCH[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

(V.)[ch].MPA.INPOSW[set].xn

(V.)[ch].MPA.INPOSW[set].sn

(V.)[ch].SP.INPOSW[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

In-position zone.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.INPOSW[3].Z	Z axis.
V.MPA.INPOSW[3].S	Spindle S.
V.SP.INPOSW[3].S	Spindle S.
V.SP.INPOSW[3]	Master spindle.
V.MPA.INPOSW[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.INPOSW[3].1	Axis with index -1- in the channel -2-.
V.SP.INPOSW[3].2	Spindle with index -2- in the system.
V.[2].SP.INPOSW[3].1	Spindle with index -1- in the channel -2-.

BACKLASH COMPENSATION IN MOVEMENT REVERSAL.

(V.)[ch].MPA.BACKLASH[set].xn

(V.)[ch].MPA.BACKLASH[set].sn

(V.)[ch].SP.BACKLASH[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Amount of backlash to be compensated.

With linear encoders (scales), this parameter will be set to -0-.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.BACKLASH[3].Z	Z axis.
V.MPA.BACKLASH[3].S	Spindle S.
V.SP.BACKLASH[3].S	Spindle S.
V.SP.BACKLASH[3]	Master spindle.
V.MPA.BACKLASH[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.BACKLASH[3].1	Axis with index -1- in the channel -2-.
V.SP.BACKLASH[3].2	Spindle with index -2- in the system.
V.[2].SP.BACKLASH[3].1	Spindle with index -1- in the channel -2-.



CNC 8060
CNC 8065

(REF: 1402)

BACKLASH COMPENSATION IN MOVEMENT REVERSALS WITH AN ADDITIONAL COMMAND PULSE.

(V.)[ch].MPA.BAKANOUT[set].xn
(V.)[ch].MPA.BAKANOUT[set].sn
(V.)[ch].SP.BAKANOUT[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Additional velocity command pulse.

- With digital drives, the additional velocity command is given in rpm.
- With an analog drive, the additional velocity command is given in units of the D/A converter with an integer within ± 32767 . - 10 V for a value of -32767 and +10V for a value of 32767.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.BAKANOUT[3].Z	Z axis.
V.MPA.BAKANOUT[3].S	Spindle S.
V.SP.BAKANOUT[3].S	Spindle S.
V.SP.BAKANOUT[3]	Master spindle.
V.MPA.BAKANOUT[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.BAKANOUT[3].1	Axis with index ·1· in the channel ·2·.
V.SP.BAKANOUT[3].2	Spindle with index ·2· in the system.
V.[2].SP.BAKANOUT[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.BAKTIME[set].xn
(V.)[ch].MPA.BAKTIME[set].sn
(V.)[ch].SP.BAKTIME[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Duration of the additional velocity command pulse.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.BAKTIME[3].Z	Z axis.
V.MPA.BAKTIME[3].S	Spindle S.
V.SP.BAKTIME[3].S	Spindle S.
V.SP.BAKTIME[3]	Master spindle.
V.MPA.BAKTIME[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.BAKTIME[3].1	Axis with index ·1· in the channel ·2·.
V.SP.BAKTIME[3].2	Spindle with index ·2· in the system.
V.[2].SP.BAKTIME[3].1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

(V.)[ch].MPA.ACTBAKAN[set].xn

(V.)[ch].MPA.ACTBAKAN[set].sn

(V.)[ch].SP.ACTBAKAN[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Application of the additional velocity command pulse.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.ACTBAKAN[3].Z	Z axis.
V.MPA.ACTBAKAN[3].S	Spindle S.
V.SP.ACTBAKAN[3].S	Spindle S.
V.SP.ACTBAKAN[3]	Master spindle.
V.MPA.ACTBAKAN[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.ACTBAKAN[3].1	Axis with index ·1· in the channel ·2·.
V.SP.ACTBAKAN[3].2	Spindle with index ·2· in the system.
V.[2].SP.ACTBAKAN[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	In G02 / G03 positioning.
1	Always.

(V.)[ch].MPA.PEAKDISP[set].xn

(V.)[ch].MPA.PEAKDISP[set].sn

(V.)[ch].SP.PEAKDISP[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Backlash peak cutoff distance.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.PEAKDISP[3].Z	Z axis.
V.MPA.PEAKDISP[3].S	Spindle S.
V.SP.PEAKDISP[3].S	Spindle S.
V.SP.PEAKDISP[3]	Master spindle.
V.MPA.PEAKDISP[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.PEAKDISP[3].1	Axis with index ·1· in the channel ·2·.
V.SP.PEAKDISP[3].2	Spindle with index ·2· in the system.
V.[2].SP.PEAKDISP[3].1	Spindle with index ·1· in the channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.REVEHYST[set].xn**(V.)[ch].MPA.REVEHYST[set].sn****(V.)[ch].SP.REVEHYST[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Hysteresis for applying the additional command pulse in movement reversals.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.REVEHYST[3].Z	Z axis.
V.MPA.REVEHYST[3].S	Spindle S.
V.SP.REVEHYST[3].S	Spindle S.
V.SP.REVEHYST[3]	Master spindle.
V.MPA.REVEHYST[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.REVEHYST[3].1	Axis with index ·1· in the channel ·2·.
V.SP.REVEHYST[3].2	Spindle with index ·2· in the system.
V.[2].SP.REVEHYST[3].1	Spindle with index ·1· in the channel ·2·.

ADJUSTMENT OF RAPID TRAVERSE G00 AND MAXIMUM SPEED.**(V.)[ch].MPA.G00FEED[set].xn****(V.)[ch].MPA.G00FEED[set].sn****(V.)[ch].SP.G00FEED[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified from the environment of the oscilloscope and setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Feedrate in G00.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.G00FEED[3].Z	Z axis.
V.MPA.G00FEED[3].S	Spindle S.
V.SP.G00FEED[3].S	Spindle S.
V.SP.G00FEED[3]	Master spindle.
V.MPA.G00FEED[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.G00FEED[3].1	Axis with index ·1· in the channel ·2·.
V.SP.G00FEED[3].2	Spindle with index ·2· in the system.
V.[2].SP.G00FEED[3].1	Spindle with index ·1· in the channel ·2·.

22.**CNC VARIABLES.**

Variables related to the sets of machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

(V.) [ch].MPA.MAXFEED[set].xn

(V.) [ch].MPA.MAXFEED[set].sn

(V.) [ch].SP.MAXFEED[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Channel [ch]. Maximum axis machining feedrate.

If the variable returns a -0- value, the machining feedrate is not limited, the CNC assumes for all the movements the one set in machine parameter G00FEED as the maximum feedrate.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MAXFEED[3].Z	Z axis.
V.MPA.MAXFEED[3].S	Spindle S.
V.SP.MAXFEED[3].S	Spindle S.
V.SP.MAXFEED[3]	Master spindle.
V.MPA.MAXFEED[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.MAXFEED[3].1	Axis with index -1- in the channel -2-.
V.SP.MAXFEED[3].2	Spindle with index -2- in the system.
V.[2].SP.MAXFEED[3].1	Spindle with index -1- in the channel -2-.

(V.) [ch].MPA.MAXVOLT[set].xn

(V.) [ch].MPA.MAXVOLT[set].sn

(V.) [ch].SP.MAXVOLT[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drives and velocity-Sercos drives.

The variable returns the execution value; reading it interrupts block preparation.

Velocity command to reach G00FEED

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MAXVOLT[3].Z	Z axis.
V.MPA.MAXVOLT[3].S	Spindle S.
V.SP.MAXVOLT[3].S	Spindle S.
V.SP.MAXVOLT[3]	Master spindle.
V.MPA.MAXVOLT[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.MAXVOLT[3].1	Axis with index -1- in the channel -2-.
V.SP.MAXVOLT[3].2	Spindle with index -2- in the system.
V.[2].SP.MAXVOLT[3].1	Spindle with index -1- in the channel -2-.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.MAXFREQ[set].sn**(V.)[ch].SP.MAXFREQ[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified from the environment of the oscilloscope and setup.**Variable valid for spindles.**Variable valid for Mechatrolink drive.**The variable returns the execution value; reading it interrupts block preparation.*

This is the frequency the CNC must provide for the spindle to reach the maximum speed set by parameter G00FEED.

Syntax.

- ch· Channel number.
- set· Parameter set.
- sn· Name, logic number or index of the spindle.

V.MPA.MAXFREQ[3].S	Spindle S.
V.SP.MAXFREQ[3].S	Spindle S.
V.SP.MAXFREQ[3]	Master spindle.
V.MPA.MAXFREQ[3].4	Spindle with logic number ·4·.
V.SP.MAXFREQ[3].2	Spindle with index ·2· in the system.
V.[2].SP.MAXFREQ[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.MAXRPM[set].xn**(V.)[ch].MPA.MAXRPM[set].sn****(V.)[ch].SP.MAXRPM[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for Mechatrolink drive.**The variable returns the execution value; reading it interrupts block preparation.*

Maximum motor rpm.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MAXRPM[3].Z	Z axis.
V.MPA.MAXRPM[3].S	Spindle S.
V.SP.MAXRPM[3].S	Spindle S.
V.SP.MAXRPM[3]	Master spindle.
V.MPA.MAXRPM[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.MAXRPM[3].1	Axis with index ·1· in the channel ·2·.
V.SP.MAXRPM[3].2	Spindle with index ·2· in the system.
V.[2].SP.MAXRPM[3].1	Spindle with index ·1· in the channel ·2·.

RAPID TRAVERSE FOR THE AUTOMATIC MODE.**(V.)[ch].MPA.FRAPIDEN[set].xn****(V.)[ch].MPA.FRAPIDEN[set].sn****(V.)[ch].SP.FRAPIDEN[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Rapid traverse of the axis when the rapid traverse for the automatic mode is active (parameter RAPIDEN). If the parameter is set with a "0" value, the feedrate is not limited.

22.**CNC VARIABLES.**

Variables related to the sets of machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

This parameter does not the movements programmed in G00 or the threads. Movements in G0 are carried out at the feedrate set in parameter G00FEED. The threads are executed at the programmed feedrate.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.FRAPIDEN[3].Z	Z axis.
V.MPA.FRAPIDEN[3].S	Spindle S.
V.SP.FRAPIDEN[3].S	Spindle S.
V.SP.FRAPIDEN[3]	Master spindle.
V.MPA.FRAPIDEN[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.FRAPIDEN[3].1	Axis with index ·1· in the channel ·2·.
V.SP.FRAPIDEN[3].2	Spindle with index ·2· in the system.
V.[2].SP.FRAPIDEN[3].1	Spindle with index ·1· in the channel ·2·.

Remarks.

The rapid traverse cannot exceed the value set in axis parameters G00FEED or FRAPIDEN of the channel or the maximum feedrate set by PLC (variable (V.)PLC.G00FEED). Rapid traverse cannot exceed the value set in axis parameter MAXFEED of the channel and the active feedrate set by PLC (variable (V.)PLC.F).

Variable.	Machine parameters.		Rapid feed.	
	(V.)PLC.G00FEED	G00FEED (axis)	FRAPIDEN (axis)	G00
0	10000	0	10000	10000
0	10000	6000	10000	6000
4000	10000	6000	4000	4000
7000	10000	6000	7000	6000
12000	10000	6000	10000	6000

GAIN SETTING.

(V.)[ch].MPA.PROGAIN[set].xn

(V.)[ch].MPA.PROGAIN[set].sn

(V.)[ch].SP.PROGAIN[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Proportional gain.

It sets the following error (difference between the theoretical instantaneous position and the actual - real - axis position) for a particular feedrate.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

·sn· Name, logic number or index of the spindle.

V.MPA.PROGAIN[3].Z	Z axis.
V.MPA.PROGAIN[3].S	Spindle S.
V.SP.PROGAIN[3].S	Spindle S.
V.SP.PROGAIN[3]	Master spindle.
V.MPA.PROGAIN[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.PROGAIN[3].1	Axis with index ·1· in the channel ·2·.
V.SP.PROGAIN[3].2	Spindle with index ·2· in the system.
V.[2].SP.PROGAIN[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.FFWTYPE[set].xn

(V.)[ch].MPA.FFWTYPE[set].sn

(V.)[ch].SP.FFWTYPE[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Pre-control type

Syntax.

·ch· Channel number.

·set· Parameter set.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.MPA.FFWTYPE[3].Z	Z axis.
V.MPA.FFWTYPE[3].S	Spindle S.
V.SP.FFWTYPE[3].S	Spindle S.
V.SP.FFWTYPE[3]	Master spindle.
V.MPA.FFWTYPE[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.FFWTYPE[3].1	Axis with index ·1· in the channel ·2·.
V.SP.FFWTYPE[3].2	Spindle with index ·2· in the system.
V.[2].SP.FFWTYPE[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Without pre-control.
1	Feed forward.
2	AC-forward
3	Feed-Forward and AC-Forward.

(V.)[ch].MPA.FFGAIN[set].xn

(V.)[ch].MPA.FFGAIN[set].sn

(V.)[ch].SP.FFGAIN[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Percentage of Feed-Forward in automatic.

It sets the part of the velocity command that is proportional to the programmed feedrate. The rest will be proportional to the following error.



22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.FFGAIN[3].Z	Z axis.
V.MPA.FFGAIN[3].S	Spindle S.
V.SP.FFGAIN[3].S	Spindle S.
V.SP.FFGAIN[3]	Master spindle.
V.MPA.FFGAIN[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.FFGAIN[3].1	Axis with index -1- in the channel -2-.
V.SP.FFGAIN[3].2	Spindle with index -2- in the system.
V.[2].SP.FFGAIN[3].1	Spindle with index -1- in the channel -2-.

Remarks.

The reading from the PLC will be given in hundredths (x100); i.e. if the parameter value is -10-, the PLC reading will return a value of -1000-.

Although the parameter may be set with up to 4 decimals, only 2 decimals will be used when reading the variable.

(V.)[ch].MPA.MANFFGAIN[set].xn

(V.)[ch].MPA.MANFFGAIN[set].sn

(V.)[ch].SP.MANFFGAIN[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Percentage of Feed-Forward in JOG

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MANFFGAIN[3].Z	Z axis.
V.MPA.MANFFGAIN[3].S	Spindle S.
V.SP.MANFFGAIN[3].S	Spindle S.
V.SP.MANFFGAIN[3]	Master spindle.
V.MPA.MANFFGAIN[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.MANFFGAIN[3].1	Axis with index -1- in the channel -2-.
V.SP.MANFFGAIN[3].2	Spindle with index -2- in the system.
V.[2].SP.MANFFGAIN[3].1	Spindle with index -1- in the channel -2-.

Remarks.

The reading from the PLC will be given in hundredths (x100); i.e. if the parameter value is -10-, the PLC reading will return a value of -1000-.

Although the parameter may be set with up to 4 decimals, only 2 decimals will be used when reading the variable.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.ACFWFACTOR[set].xn**(V.)[ch].MPA.ACFWFACTOR[set].sn****(V.)[ch].SP.ACFWFACTOR[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified from the environment of the oscilloscope and setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drives and velocity-Sercos drives.**The variable returns the execution value; reading it interrupts block preparation.*

Acceleration time constant.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.ACFWFACTOR[3].Z	Z axis.
V.MPA.ACFWFACTOR[3].S	Spindle S.
V.SP.ACFWFACTOR[3].S	Spindle S.
V.SP.ACFWFACTOR[3]	Master spindle.
V.MPA.ACFWFACTOR[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.ACFWFACTOR[3].1	Axis with index ·1· in the channel ·2·.
V.SP.ACFWFACTOR[3].2	Spindle with index ·2· in the system.
V.[2].SP.ACFWFACTOR[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.ACFGAIN[set].xn**(V.)[ch].MPA.ACFGAIN[set].sn****(V.)[ch].SP.ACFGAIN[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified from the environment of the oscilloscope and setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Percentage of Feed-Forward in automatic.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.ACFGAIN[3].Z	Z axis.
V.MPA.ACFGAIN[3].S	Spindle S.
V.SP.ACFGAIN[3].S	Spindle S.
V.SP.ACFGAIN[3]	Master spindle.
V.MPA.ACFGAIN[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.ACFGAIN[3].1	Axis with index ·1· in the channel ·2·.
V.SP.ACFGAIN[3].2	Spindle with index ·2· in the system.
V.[2].SP.ACFGAIN[3].1	Spindle with index ·1· in the channel ·2·.

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·100·.

Although the parameter may be set with up to 4 decimals, only the first decimal will be used when reading the variable.

22.**CNC VARIABLES.**

Variables related to the sets of machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

(V.)[ch].MPA.MANACFGAIN[set].xn

(V.)[ch].MPA.MANACFGAIN[set].sn

(V.)[ch].SP.MANACFGAIN[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Percentage of Feed-Forward in jog mode.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MANACFGAIN[3].Z	Z axis.
V.MPA.MANACFGAIN[3].S	Spindle S.
V.SP.MANACFGAIN[3].S	Spindle S.
V.SP.MANACFGAIN[3]	Master spindle.
V.MPA.MANACFGAIN[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.MANACFGAIN[3].1	Axis with index -1- in the channel -2-.
V.SP.MANACFGAIN[3].2	Spindle with index -2- in the system.
V.[2].SP.MANACFGAIN[3].1	Spindle with index -1- in the channel -2-.

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is -10-, the PLC reading will return a value of -100-.

Although the parameter may be set with up to 4 decimals, only the first decimal will be used when reading the variable.

LINEAR ACCELERATION.

(V.)[ch].MPA.LACC1[set].xn

(V.)[ch].MPA.LACC1[set].sn

(V.)[ch].SP.LACC1[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Acceleration of the first section.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.LACC1[3].Z	Z axis.
V.MPA.LACC1[3].S	Spindle S.
V.SP.LACC1[3].S	Spindle S.
V.SP.LACC1[3]	Master spindle.
V.MPA.LACC1[3].4	Axis or spindle with logic number -4-.



CNC 8060
CNC 8065

(REF: 1402)

V.[2].MPA.LACC1[3].1	Axis with index ·1· in the channel ·2·.
V.SP.LACC1[3].2	Spindle with index ·2· in the system.
V.[2].SP.LACC1[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.LACC2[set].xn
(V.)[ch].MPA.LACC2[set].sn
(V.)[ch].SP.LACC2[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Acceleration of the second section.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.LACC2[3].Z	Z axis.
V.MPA.LACC2[3].S	Spindle S.
V.SP.LACC2[3].S	Spindle S.
V.SP.LACC2[3]	Master spindle.
V.MPA.LACC2[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.LACC2[3].1	Axis with index ·1· in the channel ·2·.
V.SP.LACC2[3].2	Spindle with index ·2· in the system.
V.[2].SP.LACC2[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.LFEED[set].xn
(V.)[ch].SP.LFEED[set].sn
(V.)[ch].SP.LFEED[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Change speed.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.LFEED[3].Z	Z axis.
V.MPA.LFEED[3].S	Spindle S.
V.SP.LFEED[3].S	Spindle S.
V.SP.LFEED[3]	Master spindle.
V.MPA.LFEED[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.LFEED[3].1	Axis with index ·1· in the channel ·2·.
V.SP.LFEED[3].2	Spindle with index ·2· in the system.
V.[2].SP.LFEED[3].1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

TRAPEZOIDAL AND SQUARE SINE ACCELERATION.

(V.) [ch].MPA.ACCEL[set].xn**(V.) [ch].MPA.ACCEL[set].sn****(V.) [ch].SP.ACCEL[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified from the environment of the oscilloscope and setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Acceleration.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.ACCEL[3].Z	Z axis.
V.MPA.ACCEL[3].S	Spindle S.
V.SP.ACCEL[3].S	Spindle S.
V.SP.ACCEL[3]	Master spindle.
V.MPA.ACCEL[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.ACCEL[3].1	Axis with index ·1· in the channel ·2·.
V.SP.ACCEL[3].2	Spindle with index ·2· in the system.
V.[2].SP.ACCEL[3].1	Spindle with index ·1· in the channel ·2·.

(V.) [ch].MPA.DECCEL[set].xn**(V.) [ch].MPA.DECCEL[set].sn****(V.) [ch].SP.DECCEL[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified from the environment of the oscilloscope and setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Deceleration.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.DECCEL[3].Z	Z axis.
V.MPA.DECCEL[3].S	Spindle S.
V.SP.DECCEL[3].S	Spindle S.
V.SP.DECCEL[3]	Master spindle.
V.MPA.DECCEL[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.DECCEL[3].1	Axis with index ·1· in the channel ·2·.
V.SP.DECCEL[3].2	Spindle with index ·2· in the system.
V.[2].SP.DECCEL[3].1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.ACCJERK[set].xn**(V.)[ch].MPA.ACCJERK[set].sn****(V.)[ch].SP.ACCJERK[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified from the environment of the oscilloscope and setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Acceleration Jerk.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.ACCJERK[3].Z	Z axis.
V.MPA.ACCJERK[3].S	Spindle S.
V.SP.ACCJERK[3].S	Spindle S.
V.SP.ACCJERK[3]	Master spindle.
V.MPA.ACCJERK[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.ACCJERK[3].1	Axis with index ·1· in the channel ·2·.
V.SP.ACCJERK[3].2	Spindle with index ·2· in the system.
V.[2].SP.ACCJERK[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.DECJERK[set].xn**(V.)[ch].MPA.DECJERK[set].sn****(V.)[ch].SP.DECJERK[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified from the environment of the oscilloscope and setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Deceleration Jerk.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.DECJERK[3].Z	Z axis.
V.MPA.DECJERK[3].S	Spindle S.
V.SP.DECJERK[3].S	Spindle S.
V.SP.DECJERK[3]	Master spindle.
V.MPA.DECJERK[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.DECJERK[3].1	Axis with index ·1· in the channel ·2·.
V.SP.DECJERK[3].2	Spindle with index ·2· in the system.
V.[2].SP.DECJERK[3].1	Spindle with index ·1· in the channel ·2·.

22.**CNC VARIABLES.**

Variables related to the sets of machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

ENABLE SPECIFIC ACCELERATION VALUES FOR MOVEMENTS IN G0.

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.

(V.) [ch].MPA.GOACDCJERK[set].xn

(V.) [ch].MPA.GOACDCJERK[set].sn

(V.) [ch].SP.GOACDCJERK[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

More parameters if G0.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.GOACDCJERK[3].Z	Z axis.
V.MPA.GOACDCJERK[3].S	Spindle S.
V.SP.GOACDCJERK[3].S	Spindle S.
V.SP.GOACDCJERK[3]	Master spindle.
V.MPA.GOACDCJERK[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.GOACDCJERK[3].1	Axis with index ·1· in the channel ·2·.
V.SP.GOACDCJERK[3].2	Spindle with index ·2· in the system.
V.[2].SP.GOACDCJERK[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

LINEAR ACCELERATION (G0 MOVEMENTS).

(V.) [ch].MPA.LACC1G0[set].xn

(V.) [ch].MPA.LACC1G0[set].sn

(V.) [ch].SP.LACC1G0[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Acceleration of the first section (G0 movements).

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.LACC1G0[3].Z	Z axis.
V.MPA.LACC1G0[3].S	Spindle S.
V.SP.LACC1G0[3].S	Spindle S.



CNC 8060
CNC 8065

(REF: 1402)

V.SP.LACC1G0[3]	Master spindle.
V.MPA.LACC1G0[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.LACC1G0[3].1	Axis with index -1- in the channel -2-.
V.SP.LACC1G0[3].2	Spindle with index -2- in the system.
V.[2].SP.LACC1G0[3].1	Spindle with index -1- in the channel -2-.

(V.)[ch].MPA.LACC2G0[set].xn
(V.)[ch].MPA.LACC2G0[set].sn
(V.)[ch].SP.LACC2G0[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Acceleration of the second section (G0 movements).

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.LACC2G0[3].Z	Z axis.
V.MPA.LACC2G0[3].S	Spindle S.
V.SP.LACC2G0[3].S	Spindle S.
V.SP.LACC2G0[3]	Master spindle.
V.MPA.LACC2G0[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.LACC2G0[3].1	Axis with index -1- in the channel -2-.
V.SP.LACC2G0[3].2	Spindle with index -2- in the system.
V.[2].SP.LACC2G0[3].1	Spindle with index -1- in the channel -2-.

(V.)[ch].MPA.LFEEDG0[set].xn
(V.)[ch].SP.LFEEDG0[set].sn
(V.)[ch].SP.LFEEDG0[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Acceleration changing speed (G0 movements).

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.LFEEDG0[3].Z	Z axis.
V.MPA.LFEEDG0[3].S	Spindle S.
V.SP.LFEEDG0[3].S	Spindle S.
V.SP.LFEEDG0[3]	Master spindle.
V.MPA.LFEEDG0[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.LFEEDG0[3].1	Axis with index -1- in the channel -2-.
V.SP.LFEEDG0[3].2	Spindle with index -2- in the system.
V.[2].SP.LFEEDG0[3].1	Spindle with index -1- in the channel -2-.

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.



CNC 8060
 CNC 8065

(REF: 1402)

TRAPEZOIDAL AND SQUARE SINE ACCELERATION (G0 MOVEMENTS).

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.

(V.) [ch].MPA.ACCELGO[set].xn

(V.) [ch].MPA.ACCELGO[set].sn

(V.) [ch].SP.ACCELGO[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Acceleration (G0 movements).

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.ACCELGO[3].Z	Z axis.
V.MPA.ACCELGO[3].S	Spindle S.
V.SP.ACCELGO[3].S	Spindle S.
V.SP.ACCELGO[3]	Master spindle.
V.MPA.ACCELGO[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.ACCELGO[3].1	Axis with index ·1· in the channel ·2·.
V.SP.ACCELGO[3].2	Spindle with index ·2· in the system.
V.[2].SP.ACCELGO[3].1	Spindle with index ·1· in the channel ·2·.

(V.) [ch].MPA.DECELGO[set].xn

(V.) [ch].MPA.DECELGO[set].sn

(V.) [ch].SP.DECELGO[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Deceleration (G0 movements).

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.DECELGO[3].Z	Z axis.
V.MPA.DECELGO[3].S	Spindle S.
V.SP.DECELGO[3].S	Spindle S.
V.SP.DECELGO[3]	Master spindle.
V.MPA.DECELGO[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.DECELGO[3].1	Axis with index ·1· in the channel ·2·.
V.SP.DECELGO[3].2	Spindle with index ·2· in the system.
V.[2].SP.DECELGO[3].1	Spindle with index ·1· in the channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.ACCJERKGO[set].xn**(V.)[ch].MPA.ACCJERKGO[set].sn****(V.)[ch].SP.ACCJERKGO[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified from the environment of the oscilloscope and setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Acceleration jerk (G0 movements).

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.ACCJERKGO[3].Z	Z axis.
V.MPA.ACCJERKGO[3].S	Spindle S.
V.SP.ACCJERKGO[3].S	Spindle S.
V.SP.ACCJERKGO[3]	Master spindle.
V.MPA.ACCJERKGO[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.ACCJERKGO[3].1	Axis with index ·1· in the channel ·2·.
V.SP.ACCJERKGO[3].2	Spindle with index ·2· in the system.
V.[2].SP.ACCJERKGO[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.DECJERKGO[set].xn**(V.)[ch].MPA.DECJERKGO[set].sn****(V.)[ch].SP.DECJERKGO[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified from the environment of the oscilloscope and setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Deceleration jerk (G0 movements).

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.DECJERKGO[3].Z	Z axis.
V.MPA.DECJERKGO[3].S	Spindle S.
V.SP.DECJERKGO[3].S	Spindle S.
V.SP.DECJERKGO[3]	Master spindle.
V.MPA.DECJERKGO[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.DECJERKGO[3].1	Axis with index ·1· in the channel ·2·.
V.SP.DECJERKGO[3].2	Spindle with index ·2· in the system.
V.[2].SP.DECJERKGO[3].1	Spindle with index ·1· in the channel ·2·.

22.**CNC VARIABLES.**

Variables related to the sets of machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

CONFIGURATION OF THE HSC MODE.

(V.)[ch].MPA.CORNERACC[set].xn**(V.)[ch].MPA.CORNERACC[set].sn****(V.)[ch].SP.CORNERACC[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Maximum acceleration permitted at the corners.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.CORNERACC[3].Z	Z axis.
V.MPA.CORNERACC[3].S	Spindle S.
V.SP.CORNERACC[3].S	Spindle S.
V.SP.CORNERACC[3]	Master spindle.
V.MPA.CORNERACC[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.CORNERACC[3].1	Axis with index ·1· in the channel ·2·.
V.SP.CORNERACC[3].2	Spindle with index ·2· in the system.
V.[2].SP.CORNERACC[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.CURVACC[set].xn**(V.)[ch].MPA.CURVACC[set].sn****(V.)[ch].SP.CURVACC[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Maximum contouring acceleration permitted.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.CURVACC[3].Z	Z axis.
V.MPA.CURVACC[3].S	Spindle S.
V.SP.CURVACC[3].S	Spindle S.
V.SP.CURVACC[3]	Master spindle.
V.MPA.CURVACC[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.CURVACC[3].1	Axis with index ·1· in the channel ·2·.
V.SP.CURVACC[3].2	Spindle with index ·2· in the system.
V.[2].SP.CURVACC[3].1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.CORNERJERK[set].xn**(V.)[ch].MPA.CORNERJERK[set].sn****(V.)[ch].SP.CORNERJERK[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Maximum Jerk permitted at the corners.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.CORNERJERK[3].Z	Z axis.
V.MPA.CORNERJERK[3].S	Spindle S.
V.SP.CORNERJERK[3].S	Spindle S.
V.SP.CORNERJERK[3]	Master spindle.
V.MPA.CORNERJERK[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.CORNERJERK[3].1	Axis with index ·1· in the channel ·2·.
V.SP.CORNERJERK[3].2	Spindle with index ·2· in the system.
V.[2].SP.CORNERJERK[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.CURVJERK[set].xn**(V.)[ch].MPA.CURVJERK[set].sn****(V.)[ch].SP.CURVJERK[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Maximum contouring Jerk permitted.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.CURVJERK[3].Z	Z axis.
V.MPA.CURVJERK[3].S	Spindle S.
V.SP.CURVJERK[3].S	Spindle S.
V.SP.CURVJERK[3]	Master spindle.
V.MPA.CURVJERK[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.CURVJERK[3].1	Axis with index ·1· in the channel ·2·.
V.SP.CURVJERK[3].2	Spindle with index ·2· in the system.
V.[2].SP.CURVJERK[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.FASTACC[set].xn**(V.)[ch].MPA.FASTACC[set].sn****(V.)[ch].SP.FASTACC[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Maximum acceleration permitted (FAST mode).

22.**CNC VARIABLES.**

Variables related to the sets of machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.FASTACC[3].Z	Z axis.
V.MPA.FASTACC[3].S	Spindle S.
V.SP.FASTACC[3].S	Spindle S.
V.SP.FASTACC[3]	Master spindle.
V.MPA.FASTACC[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.FASTACC[3].1	Axis with index -1- in the channel -2-.
V.SP.FASTACC[3].2	Spindle with index -2- in the system.
V.[2].SP.FASTACC[3].1	Spindle with index -1- in the channel -2-.

(V.)[ch].MPA.MAXERROR[set].xn**(V.)[ch].MPA.MAXERROR[set].sn****(V.)[ch].SP.MAXERROR[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Maximum axis position error in HSC when working outside the plane/trihedron.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MAXERROR[3].Z	Z axis.
V.MPA.MAXERROR[3].S	Spindle S.
V.SP.MAXERROR[3].S	Spindle S.
V.SP.MAXERROR[3]	Master spindle.
V.MPA.MAXERROR[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.MAXERROR[3].1	Axis with index -1- in the channel -2-.
V.SP.MAXERROR[3].2	Spindle with index -2- in the system.
V.[2].SP.MAXERROR[3].1	Spindle with index -1- in the channel -2-.

(V.)[ch].MPA.CONTERROR[set].xn**(V.)[ch].MPA.CONTERROR[set].sn****(V.)[ch].SP.CONTERROR[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Tolerance per axis for smoothing the generated n-dimensional path.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.



CNC 8060
CNC 8065

(REF: 1402)

·sn· Name, logic number or index of the spindle.

V.MPA.CONTERROR[3].Z	Z axis.
V.MPA.CONTERROR[3].S	Spindle S.
V.SP.CONTERROR[3].S	Spindle S.
V.SP.CONTERROR[3]	Master spindle.
V.MPA.CONTERROR[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.CONTERROR[3].1	Axis with index ·1· in the channel ·2·.
V.SP.CONTERROR[3].2	Spindle with index ·2· in the system.
V.[2].SP.CONTERROR[3].1	Spindle with index ·1· in the channel ·2·.

HOME SEARCH.

(V.)[ch].MPA.IOTYPE[set].xn

(V.)[ch].MPA.IOTYPE[set].sn

(V.)[ch].SP.IOTYPE[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Reference mark (I0) type

Syntax.

·ch· Channel number.

·set· Parameter set.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.MPA.IOTYPE[3].Z	Z axis.
V.MPA.IOTYPE[3].S	Spindle S.
V.SP.IOTYPE[3].S	Spindle S.
V.SP.IOTYPE[3]	Master spindle.
V.MPA.IOTYPE[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.IOTYPE[3].1	Axis with index ·1· in the channel ·2·.
V.SP.IOTYPE[3].2	Spindle with index ·2· in the system.
V.[2].SP.IOTYPE[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Incremental (Not distance-coded)
1	Increasing distance-coded reference mark.
2	Decreasing distance-coded reference mark.

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.

(V.)[ch].MPA.REFVALUE[set].xn**(V.)[ch].MPA.REFVALUE[set].sn****(V.)[ch].SP.REFVALUE[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified during setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Home position.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.REFVALUE[3].Z	Z axis.
V.MPA.REFVALUE[3].S	Spindle S.
V.SP.REFVALUE[3].S	Spindle S.
V.SP.REFVALUE[3]	Master spindle.
V.MPA.REFVALUE[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.REFVALUE[3].1	Axis with index -1- in the channel -2-.
V.SP.REFVALUE[3].2	Spindle with index -2- in the system.
V.[2].SP.REFVALUE[3].1	Spindle with index -1- in the channel -2-.

(V.)[ch].MPA.REFSHIFT[set].xn**(V.)[ch].MPA.REFSHIFT[set].sn****(V.)[ch].SP.REFSHIFT[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified during setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Offset of the reference point (home)

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.REFSHIFT[3].Z	Z axis.
V.MPA.REFSHIFT[3].S	Spindle S.
V.SP.REFSHIFT[3].S	Spindle S.
V.SP.REFSHIFT[3]	Master spindle.
V.MPA.REFSHIFT[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.REFSHIFT[3].1	Axis with index -1- in the channel -2-.
V.SP.REFSHIFT[3].2	Spindle with index -2- in the system.
V.[2].SP.REFSHIFT[3].1	Spindle with index -1- in the channel -2-.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.REFFFEED1[set].xn**(V.)[ch].MPA.REFFFEED1[set].sn****(V.)[ch].SP.REFFFEED1[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified during setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Fast home searching feedrate.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.REFFFEED1[3].Z	Z axis.
V.MPA.REFFFEED1[3].S	Spindle S.
V.SP.REFFFEED1[3].S	Spindle S.
V.SP.REFFFEED1[3]	Master spindle.
V.MPA.REFFFEED1[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.REFFFEED1[3].1	Axis with index ·1· in the channel ·2·.
V.SP.REFFFEED1[3].2	Spindle with index ·2· in the system.
V.[2].SP.REFFFEED1[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.REFFFEED2[set].xn**(V.)[ch].MPA.REFFFEED2[set].sn****(V.)[ch].SP.REFFFEED2[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified during setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Slow home searching feedrate.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.REFFFEED2[3].Z	Z axis.
V.MPA.REFFFEED2[3].S	Spindle S.
V.SP.REFFFEED2[3].S	Spindle S.
V.SP.REFFFEED2[3]	Master spindle.
V.MPA.REFFFEED2[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.REFFFEED2[3].1	Axis with index ·1· in the channel ·2·.
V.SP.REFFFEED2[3].2	Spindle with index ·2· in the system.
V.[2].SP.REFFFEED2[3].1	Spindle with index ·1· in the channel ·2·.

22.**CNC VARIABLES.**

Variables related to the sets of machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

(V.)[ch].MPA.REFPULSE[set].xn

(V.)[ch].MPA.REFPULSE[set].sn

(V.)[ch].SP.REFPULSE[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Type of I0 pulse.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.REFPULSE[3].Z	Z axis.
V.MPA.REFPULSE[3].S	Spindle S.
V.SP.REFPULSE[3].S	Spindle S.
V.SP.REFPULSE[3]	Master spindle.
V.MPA.REFPULSE[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.REFPULSE[3].1	Axis with index -1- in the channel -2-.
V.SP.REFPULSE[3].2	Spindle with index -2- in the system.
V.[2].SP.REFPULSE[3].1	Spindle with index -1- in the channel -2-.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Negative pulse.
1	Positive pulse.

(V.)[ch].MPA.POSINREF[set].xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Home search moving the axis to the reference point.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.

V.MPA.POSINREF[3].Z	Z axis.
V.MPA.POSINREF[3].4	Axis with logic number -4-.
V.[2].MPA.POSINREF[3].1	Axis with index -1- in the channel -2-.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.ABSOFF[set].xn**(V.)[ch].MPA.ABSOFF[set].sn****(V.)[ch].SP.ABSOFF[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Offset with respect to the distance-coded ref. mark.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.ABSOFF[3].Z	Z axis.
V.MPA.ABSOFF[3].S	Spindle S.
V.SP.ABSOFF[3].S	Spindle S.
V.SP.ABSOFF[3]	Master spindle.
V.MPA.ABSOFF[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.ABSOFF[3].1	Axis with index ·1· in the channel ·2·.
V.SP.ABSOFF[3].2	Spindle with index ·2· in the system.
V.[2].SP.ABSOFF[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.EXTMULT[set].xn**(V.)[ch].MPA.EXTMULT[set].sn****(V.)[ch].SP.EXTMULT[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drives.**The variable returns the value of block preparation.*

External factor for distance-coded reference marks.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.EXTMULT[3].Z	Z axis.
V.MPA.EXTMULT[3].S	Spindle S.
V.SP.EXTMULT[3].S	Spindle S.
V.SP.EXTMULT[3]	Master spindle.
V.MPA.EXTMULT[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.EXTMULT[3].1	Axis with index ·1· in the channel ·2·.
V.SP.EXTMULT[3].2	Spindle with index ·2· in the system.
V.[2].SP.EXTMULT[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.IOCODDI1[set].xn**(V.)[ch].MPA.IOCODDI1[set].sn****(V.)[ch].SP.IOCODDI1[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the value of block preparation.*

Pitch between 2 fixed distance-coded reference marks.

22.**CNC VARIABLES.**

Variables related to the sets of machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.IOCODDI1[3].Z	Z axis.
V.MPA.IOCODDI1[3].S	Spindle S.
V.SP.IOCODDI1[3].S	Spindle S.
V.SP.IOCODDI1[3]	Master spindle.
V.MPA.IOCODDI1[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.IOCODDI1[3].1	Axis with index -1- in the channel -2-.
V.SP.IOCODDI1[3].2	Spindle with index -2- in the system.
V.[2].SP.IOCODDI1[3].1	Spindle with index -1- in the channel -2-.

(V.)[ch].MPA.IOCODDI2[set].xn

(V.)[ch].MPA.IOCODDI2[set].sn

(V.)[ch].SP.IOCODDI2[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Pitch between 2 variable distance-coded reference marks.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.IOCODDI2[3].Z	Z axis.
V.MPA.IOCODDI2[3].S	Spindle S.
V.SP.IOCODDI2[3].S	Spindle S.
V.SP.IOCODDI2[3]	Master spindle.
V.MPA.IOCODDI2[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.IOCODDI2[3].1	Axis with index -1- in the channel -2-.
V.SP.IOCODDI2[3].2	Spindle with index -2- in the system.
V.[2].SP.IOCODDI2[3].1	Spindle with index -1- in the channel -2-.

FOLLOWING ERROR.

(V.)[ch].MPA.FLWEMONITOR[set].xn

(V.)[ch].MPA.FLWEMONITOR[set].sn

(V.)[ch].SP.FLWEMONITOR[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Type of monitoring of the following error (axis lag).

The CNC offers two types of monitoring of the following error (axis lag). The "standard" monitoring type performs a continuous supervision of the following error whereas the "linear" monitoring performs a dynamic supervision

Syntax.

- ch· Channel number.



CNC 8060
CNC 8065

(REF: 1402)

- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.FLWEMONITOR[3].Z	Z axis.
V.MPA.FLWEMONITOR[3].S	Spindle S.
V.SP.FLWEMONITOR[3].S	Spindle S.
V.SP.FLWEMONITOR[3]	Master spindle.
V.MPA.FLWEMONITOR[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.FLWEMONITOR[3].1	Axis with index ·1· in the channel ·2·.
V.SP.FLWEMONITOR[3].2	Spindle with index ·2· in the system.
V.[2].SP.FLWEMONITOR[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No monitoring.
1	Standard monitoring.
2	Linear monitoring.

(V.)[ch].MPA.MINFLWE[set].xn
 (V.)[ch].MPA.MINFLWE[set].sn
 (V.)[ch].SP.MINFLWE[set].sn

Variable that can only be read from the program, PLC and interface.
Variable that may be modified during setup.
Variable valid for rotary and linear axes as well as for spindles.
Variable valid for analog drive, position Sercos and velocity Sercos.
The variable returns the execution value; reading it interrupts block preparation.

Maximum following error when stopped.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MINFLWE[3].Z	Z axis.
V.MPA.MINFLWE[3].S	Spindle S.
V.SP.MINFLWE[3].S	Spindle S.
V.SP.MINFLWE[3]	Master spindle.
V.MPA.MINFLWE[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.MINFLWE[3].1	Axis with index ·1· in the channel ·2·.
V.SP.MINFLWE[3].2	Spindle with index ·2· in the system.
V.[2].SP.MINFLWE[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.MAXFLWE[set].xn
 (V.)[ch].MPA.MAXFLWE[set].sn
 (V.)[ch].SP.MAXFLWE[set].sn

Variable that can only be read from the program, PLC and interface.
Variable that may be modified during setup.
Variable valid for rotary and linear axes as well as for spindles.
Variable valid for analog drive, position Sercos and velocity Sercos.
The variable returns the execution value; reading it interrupts block preparation.

Maximum following error when moving.



With "standard" monitoring, this variable indicates the maximum amount of following error allowed when the axis is moving; with "linear" monitoring, it indicates when dynamic supervision begins (starting at which amount of following error).

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MAXFLWE[3].Z	Z axis.
V.MPA.MAXFLWE[3].S	Spindle S.
V.SP.MAXFLWE[3].S	Spindle S.
V.SP.MAXFLWE[3]	Master spindle.
V.MPA.MAXFLWE[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.MAXFLWE[3].1	Axis with index -1- in the channel -2-.
V.SP.MAXFLWE[3].2	Spindle with index -2- in the system.
V.[2].SP.MAXFLWE[3].1	Spindle with index -1- in the channel -2-.

(V.)[ch].MPA.FEDYNFAC[set].xn

(V.)[ch].MPA.FEDYNFAC[set].sn

(V.)[ch].SP.FEDYNFAC[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Percentage of following error deviation allowed.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.FEDYNFAC[3].Z	Z axis.
V.MPA.FEDYNFAC[3].S	Spindle S.
V.SP.FEDYNFAC[3].S	Spindle S.
V.SP.FEDYNFAC[3]	Master spindle.
V.MPA.FEDYNFAC[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.FEDYNFAC[3].1	Axis with index -1- in the channel -2-.
V.SP.FEDYNFAC[3].2	Spindle with index -2- in the system.
V.[2].SP.FEDYNFAC[3].1	Spindle with index -1- in the channel -2-.

(V.)[ch].MPA.ESTDELAY[set].xn

(V.)[ch].MPA.ESTDELAY[set].sn

(V.)[ch].SP.ESTDELAY[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the environment of the oscilloscope and setup.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Following error delay.

Syntax.

- ch· Channel number.
- set· Parameter set.

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.ESTDELAY[3].Z	Z axis.
V.MPA.ESTDELAY[3].S	Spindle S.
V.SP.ESTDELAY[3].S	Spindle S.
V.SP.ESTDELAY[3]	Master spindle.
V.MPA.ESTDELAY[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.ESTDELAY[3].1	Axis with index ·1· in the channel ·2·.
V.SP.ESTDELAY[3].2	Spindle with index ·2· in the system.
V.[2].SP.ESTDELAY[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.INPOMAX[set].xn
(V.)[ch].MPA.INPOMAX[set].sn
(V.)[ch].SP.INPOMAX[set].sn

Variable that can only be read from the program, PLC and interface.
Variable that may be modified during setup.
Variable valid for rotary and linear axes as well as for spindles.
Variable valid for analog drive, position Sercos and velocity Sercos.
The variable returns the execution value; reading it interrupts block preparation.

Time to get in position.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.INPOMAX[3].Z	Z axis.
V.MPA.INPOMAX[3].S	Spindle S.
V.SP.INPOMAX[3].S	Spindle S.
V.SP.INPOMAX[3]	Master spindle.
V.MPA.INPOMAX[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.INPOMAX[3].1	Axis with index ·1· in the channel ·2·.
V.SP.INPOMAX[3].2	Spindle with index ·2· in the system.
V.[2].SP.INPOMAX[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.INPOTIME[set].xn
(V.)[ch].MPA.INPOTIME[set].sn
(V.)[ch].SP.INPOTIME[set].sn

Variable that can only be read from the program, PLC and interface.
Variable that may be modified during setup.
Variable valid for rotary and linear axes as well as for spindles.
Variable valid for analog drive, position Sercos and velocity Sercos.
The variable returns the execution value; reading it interrupts block preparation.

Minimum time to stay in position.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.INPOTIME[3].Z	Z axis.
V.MPA.INPOTIME[3].S	Spindle S.
V.SP.INPOTIME[3].S	Spindle S.



22.

CNC VARIABLES.

Variables related to the sets of machine parameters.

V.SP.INPOTIME[3]	Master spindle.
V.MPA.INPOTIME[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.INPOTIME[3].1	Axis with index -1- in the channel -2-.
V.SP.INPOTIME[3].2	Spindle with index -2- in the system.
V.[2].SP.INPOTIME[3].1	Spindle with index -1- in the channel -2-.

AXIS LUBRICATION.

(V.) [ch].MPA.DISTLUBRI [set].xn

(V.) [ch].MPA.DISTLUBRI [set].sn

(V.) [ch].SP.DISTLUBRI [set].sn

*Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drive, position Sercos and velocity Sercos.**The variable returns the execution value; reading it interrupts block preparation.*

Distance to move to lubricate the axis.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.DISTLUBRI[3].Z	Z axis.
V.MPA.DISTLUBRI[3].S	Spindle S.
V.SP.DISTLUBRI[3].S	Spindle S.
V.SP.DISTLUBRI[3]	Master spindle.
V.MPA.DISTLUBRI[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.DISTLUBRI[3].1	Axis with index -1- in the channel -2-.
V.SP.DISTLUBRI[3].2	Spindle with index -2- in the system.
V.[2].SP.DISTLUBRI[3].1	Spindle with index -1- in the channel -2-.

MODULE CONFIGURATION (ROTARY AXES AND SPINDLE).

(V.) [ch].MPA.MODUPLIM [set].xn

(V.) [ch].MPA.MODUPLIM [set].sn

(V.) [ch].SP.MODUPLIM [set].sn

*Variable that can only be read from the program, PLC and interface.**Variable that may be modified during setup.**Variable valid for rotary axes and spindles.**Variable valid for analog drives and velocity-Sercos drives.**The variable returns the execution value; reading it interrupts block preparation.*

Upper limit of the module.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MODUPLIM[3].Z	Z axis.
V.MPA.MODUPLIM[3].S	Spindle S.
V.SP.MODUPLIM[3].S	Spindle S.
V.SP.MODUPLIM[3]	Master spindle.

CNC 8060
CNC 8065

(REF: 1402)

V.MPA.MODUPLIM[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.MODUPLIM[3].1	Axis with index ·1· in the channel ·2·.
V.SP.MODUPLIM[3].2	Spindle with index ·2· in the system.
V.[2].SP.MODUPLIM[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.MODLOWLIM[set].xn
(V.)[ch].MPA.MODLOWLIM[set].sn
(V.)[ch].SP.MODLOWLIM[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary axes and spindles.

Variable valid for analog drives and velocity-Sercos drives.

The variable returns the execution value; reading it interrupts block preparation.

Lower limit of the module.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MODLOWLIM[3].Z	Z axis.
V.MPA.MODLOWLIM[3].S	Spindle S.
V.SP.MODLOWLIM[3].S	Spindle S.
V.SP.MODLOWLIM[3]	Master spindle.
V.MPA.MODLOWLIM[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.MODLOWLIM[3].1	Axis with index ·1· in the channel ·2·.
V.SP.MODLOWLIM[3].2	Spindle with index ·2· in the system.
V.[2].SP.MODLOWLIM[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.MODNROT[set].xn
(V.)[ch].MPA.MODNROT[set].sn
(V.)[ch].SP.MODNROT[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary axes and spindles.

Variable valid for analog drives and velocity-Sercos drives.

The variable returns the execution value; reading it interrupts block preparation.

Module error. Number of turns.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MODNROT[3].Z	Z axis.
V.MPA.MODNROT[3].S	Spindle S.
V.SP.MODNROT[3].S	Spindle S.
V.SP.MODNROT[3]	Master spindle.
V.MPA.MODNROT[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.MODNROT[3].1	Axis with index ·1· in the channel ·2·.
V.SP.MODNROT[3].2	Spindle with index ·2· in the system.
V.[2].SP.MODNROT[3].1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.



CNC 8060
 CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

(V.)[ch].MPA.MODERR[set].xn

(V.)[ch].MPA.MODERR[set].sn

(V.)[ch].SP.MODERR[set].sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified during setup.

Variable valid for rotary axes and spindles.

Variable valid for analog drives and velocity-Sercos drives.

The variable returns the execution value; reading it interrupts block preparation.

Module error. Number of increments.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MODERR[3].Z	Z axis.
V.MPA.MODERR[3].S	Spindle S.
V.SP.MODERR[3].S	Spindle S.
V.SP.MODERR[3]	Master spindle.
V.MPA.MODERR[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.MODERR[3].1	Axis with index -1- in the channel -2-.
V.SP.MODERR[3].2	Spindle with index -2- in the system.
V.[2].SP.MODERR[3].1	Spindle with index -1- in the channel -2-.

SPINDLE SPEED.

(V.)[ch].MPA.SZERO[set].sn

(V.)[ch].SP.SZERO[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for position-Sercos drives and velocity-Sercos drives.

The variable returns the value of block preparation.

Speed considered "0 rpm".

Syntax.

- ch· Channel number.
- set· Parameter set.
- sn· Name, logic number or index of the spindle.

V.MPA.SZERO[3].S	Spindle S.
V.SP.SZERO[3].S	Spindle S.
V.SP.SZERO[3]	Master spindle.
V.MPA.SZERO[3].4	Spindle with logic number -4-.
V.SP.SZERO[3].2	Spindle with index -2- in the system.
V.[2].SP.SZERO[3].1	Spindle with index -1- in the channel -2-.

(V.)[ch].MPA.POLARM3[set].sn

(V.)[ch].SP.POLARM3[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for velocity-Sercos drives.

The variable returns the value of block preparation.

Sign of the velocity command for M3.



CNC 8060
CNC 8065

(REF: 1402)

Syntax.

- ch· Channel number.
- set· Parameter set.
- sn· Name, logic number or index of the spindle.

V.MPA.POLARM3[3].S	Spindle S.
V.SP.POLARM3[3].S	Spindle S.
V.SP.POLARM3[3]	Master spindle.
V.MPA.POLARM3[3].4	Spindle with logic number ·4·.
V.SP.POLARM3[3].2	Spindle with index ·2· in the system.
V.[2].SP.POLARM3[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Negative.
1	Positive.

(V.)[ch].MPA.POLARM4[set].sn

(V.)[ch].SP.POLARM4[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for spindles.

Variable valid for velocity-Sercos drives.

The variable returns the value of block preparation.

Sign of the velocity command for M4.

Syntax.

- ch· Channel number.
- set· Parameter set.
- sn· Name, logic number or index of the spindle.

V.MPA.POLARM4[3].S	Spindle S.
V.SP.POLARM4[3].S	Spindle S.
V.SP.POLARM4[3]	Master spindle.
V.MPA.POLARM4[3].4	Spindle with logic number ·4·.
V.SP.POLARM4[3].2	Spindle with index ·2· in the system.
V.[2].SP.POLARM4[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Negative.
1	Positive.



CNC 8060
CNC 8065

(REF: 1402)

ANALOG COMMAND SETTING.

(V.) [ch].MPA.SERVOOFF[set].xn**(V.) [ch].MPA.SERVOOFF[set].sn****(V.) [ch].SP.SERVOOFF[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified from the environment of the oscilloscope and setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drives.**The variable returns the execution value; reading it interrupts block preparation.*

Offset compensation.

The velocity command is given in D/A converter units which could be any integer within ± 32767 where a value of ± 32767 corresponds to $\pm 10V$.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.SERVOOFF[3].Z	Z axis.
V.MPA.SERVOOFF[3].S	Spindle S.
V.SP.SERVOOFF[3].S	Spindle S.
V.SP.SERVOOFF[3]	Master spindle.
V.MPA.SERVOOFF[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.SERVOOFF[3].1	Axis with index -1- in the channel -2-.
V.SP.SERVOOFF[3].2	Spindle with index -2- in the system.
V.[2].SP.SERVOOFF[3].1	Spindle with index -1- in the channel -2-.

(V.) [ch].MPA.MINANOUT[set].xn**(V.) [ch].MPA.MINANOUT[set].sn****(V.) [ch].SP.MINANOUT[set].sn***Variable that can only be read from the program, PLC and interface.**Variable that may be modified from the environment of the oscilloscope and setup.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for analog drives.**The variable returns the execution value; reading it interrupts block preparation.*

Minimum velocity command.

The velocity command is given in D/A converter units which could be any integer within ± 32767 where a value of ± 32767 corresponds to $\pm 10V$.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MINANOUT[3].Z	Z axis.
V.MPA.MINANOUT[3].S	Spindle S.
V.SP.MINANOUT[3].S	Spindle S.
V.SP.MINANOUT[3]	Master spindle.
V.MPA.MINANOUT[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.MINANOUT[3].1	Axis with index -1- in the channel -2-.
V.SP.MINANOUT[3].2	Spindle with index -2- in the system.
V.[2].SP.MINANOUT[3].1	Spindle with index -1- in the channel -2-.

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

NUMBER OF THE ANALOG OUTPUT AND OF THE FEEDBACK INPUT ASSOCIATED WITH THE AXIS.

(V.)[ch].MPA.ANAOUTTYPE[set].xn
(V.)[ch].MPA.ANAOUTTYPE[set].sn
(V.)[ch].SP.ANAOUTTYPE[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drives.

The variable returns the value of block preparation.

Type of analog output associated with the axis.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.ANAOUTTYPE[3].Z	Z axis.
V.MPA.ANAOUTTYPE[3].S	Spindle S.
V.SP.ANAOUTTYPE[3].S	Spindle S.
V.SP.ANAOUTTYPE[3]	Master spindle.
V.MPA.ANAOUTTYPE[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.ANAOUTTYPE[3].1	Axis with index ·1· in the channel ·2·.
V.SP.ANAOUTTYPE[3].2	Spindle with index ·2· in the system.
V.[2].SP.ANAOUTTYPE[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	The analog output is in the remote modules.
1	The analog output is in a Sercos drive.
2	The analog output is in a RCS-S module.

(V.)[ch].MPA.ANAOUTID[set].xn
(V.)[ch].MPA.ANAOUTID[set].sn
(V.)[ch].SP.ANAOUTID[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drives.

The variable returns the value of block preparation.

Number of the analog output associated with the axis.

The velocity command for an analog axis may be taken from an analog output of the remote modules or of the Sercos drive.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.ANAOUTID[3].Z	Z axis.
V.MPA.ANAOUTID[3].S	Spindle S.
V.SP.ANAOUTID[3].S	Spindle S.

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

V.SP.ANAOUTID[3]	Master spindle.
V.MPA.ANAOUTID[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.ANAOUTID[3].1	Axis with index -1- in the channel -2-.
V.SP.ANAOUTID[3].2	Spindle with index -2- in the system.
V.[2].SP.ANAOUTID[3].1	Spindle with index -1- in the channel -2-.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1 - 16	The analog command is taken from the remote modules. The variable returns the number of the analog output.
101 - 132 201 - 232	The analog command is taken from a Sercos drive. The first digit indicates the analog output to be used (1 or 2) and the other two digits indicate the logic address of the drive (1 through 32).
1 - 32	The analog command is taken from an RCS-S module.. The variable returns the number of the analog output.

(V.)[ch].MPA.COUNTERTYPE[set].xn

(V.)[ch].MPA.COUNTERTYPE[set].sn

(V.)[ch].SP.COUNTERTYPE[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drives and velocity-Sercos drives.

The variable returns the value of block preparation.

Type of feedback input for the axis.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.COUNTERTYPE[3].Z	Z axis.
V.MPA.COUNTERTYPE[3].S	Spindle S.
V.SP.COUNTERTYPE[3].S	Spindle S.
V.SP.COUNTERTYPE[3]	Master spindle.
V.MPA.COUNTERTYPE[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.COUNTERTYPE[3].1	Axis with index -1- in the channel -2-.
V.SP.COUNTERTYPE[3].2	Spindle with index -2- in the system.
V.[2].SP.COUNTERTYPE[3].1	Spindle with index -1- in the channel -2-.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Remote feedback input.
1	Local feedback input.
2	Feedback input of a Sercos drive.
3	Feedback input of an RCS-S module.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.COUNTERID[set].xn
 (V.)[ch].MPA.COUNTERID[set].sn
 (V.)[ch].SP.COUNTERID[set].sn

Variable that can only be read from the program, PLC and interface.
 Variable valid for rotary and linear axes as well as for spindles.
 Variable valid for analog drives and velocity-Sercos drives.
 The variable returns the value of block preparation.

Number of the feedback input associated with the axis.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.COUNTERID[3].Z	Z axis.
V.MPA.COUNTERID[3].S	Spindle S.
V.SP.COUNTERID[3].S	Spindle S.
V.SP.COUNTERID[3]	Master spindle.
V.MPA.COUNTERID[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.COUNTERID[3].1	Axis with index ·1· in the channel ·2·.
V.SP.COUNTERID[3].2	Spindle with index ·2· in the system.
V.[2].SP.COUNTERID[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1 - 40	Number of the remote feedback input.
1 - 32	Address of the Sercos drive (always the second feedback input).
1 - 2	Number of the local feedback input.
1 - 32	Number of the RCS-S module input.

SET THE DRIVE ASSOCIATED WITH THE AXES OF A MULTI-AXIS GROUP.

(V.)[ch].MPA.DRIVESET[set].xn
 (V.)[ch].MPA.DRIVESET[set].sn
 (V.)[ch].SP.DRIVESET[set].sn

Variable that can only be read from the program, PLC and interface.
 Variable valid for rotary and linear axes as well as for spindles.
 Variable valid for position-Sercos drives and velocity-Sercos drives.
 The variable returns the value of block preparation.

At the axes that belong to a multi-axis group, parameter DRIVESET indicates the set to be activated at the drive after a set or gear change at the CNC (G112 and M41 to M44).

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.DRIVESET[3].Z	Z axis.
V.MPA.DRIVESET[3].S	Spindle S.
V.SP.DRIVESET[3].S	Spindle S.



CNC 8060
 CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

V.SP.DRIVESET[3]	Master spindle.
V.MPA.DRIVESET[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.DRIVESET[3].1	Axis with index -1- in the channel -2-.
V.SP.DRIVESET[3].2	Spindle with index -2- in the system.
V.[2].SP.DRIVESET[3].1	Spindle with index -1- in the channel -2-.

FEEDBACK TYPE OF THE RCS-S MODULE.

(V.) [ch].MPA.FEEDBACKTYPE[set].xn

(V.) [ch].MPA.FEEDBACKTYPE[set].sn

(V.) [ch].SP.FEEDBACKTYPE[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for velocity-Sercos drives.

The variable returns the value of block preparation.

Feedback type of the RCS-S module.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.FEEDBACKTYPE[3].Z	Z axis.
V.MPA.FEEDBACKTYPE[3].S	Spindle S.
V.SP.FEEDBACKTYPE[3].S	Spindle S.
V.SP.FEEDBACKTYPE[3]	Master spindle.
V.MPA.FEEDBACKTYPE[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.FEEDBACKTYPE[3].1	Axis with index -1- in the channel -2-.
V.SP.FEEDBACKTYPE[3].2	Spindle with index -2- in the system.
V.[2].SP.FEEDBACKTYPE[3].1	Spindle with index -1- in the channel -2-.

(V.) [ch].MPA.SSITYPE[set].xn

(V.) [ch].MPA.SSITYPE[set].sn

(V.) [ch].SP.SSITYPE[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for velocity-Sercos drives.

The variable returns the value of block preparation.

Feedback type connected to the corresponding feedback.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.SSITYPE[3].Z	Z axis.
V.MPA.SSITYPE[3].S	Spindle S.
V.SP.SSITYPE[3].S	Spindle S.
V.SP.SSITYPE[3]	Master spindle.
V.MPA.SSITYPE[3].4	Axis or spindle with logic number -4-.
V.[2].MPA.SSITYPE[3].1	Axis with index -1- in the channel -2-.
V.SP.SSITYPE[3].2	Spindle with index -2- in the system.
V.[2].SP.SSITYPE[3].1	Spindle with index -1- in the channel -2-.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Fagor LA
1	Fagor GA SA SVA
2	Fagor HA-27-D200
3	Fagor HA-23-D90 SA-23-D90 SA-23-D170
4	ABSIND (inductosyn LIN+ABS)
5	ABSIND (inductosyn ROT+ABS)
6	ABSIND (resolver)
7	ABSIND (inductosyn LIN)
8	ABSIND (inductosyn ROT)
9	G user
10	User

(V.)[ch].MPA.SSICLKREQ[set].xn

(V.)[ch].MPA.SSICLKREQ[set].sn

(V.)[ch].SP.SSICLKREQ[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for velocity-Sercos drives.

The variable returns the value of block preparation.

SSI communication frequency.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.SSICLKREQ[3].Z	Z axis.
V.MPA.SSICLKREQ[3].S	Spindle S.
V.SP.SSICLKREQ[3].S	Spindle S.
V.SP.SSICLKREQ[3]	Master spindle.
V.MPA.SSICLKREQ[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.SSICLKREQ[3].1	Axis with index ·1· in the channel ·2·.
V.SP.SSICLKREQ[3].2	Spindle with index ·2· in the system.
V.[2].SP.SSICLKREQ[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.SSIDATALENGTH[set].xn

(V.)[ch].MPA.SSIDATALENGTH[set].sn

(V.)[ch].SP.SSIDATALENGTH[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for velocity-Sercos drives.

The variable returns the value of block preparation.

Number of bits of the SSI transmission that make up the position value (coordinate).

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

·sn· Name, logic number or index of the spindle.

V.MPA.SSIDATALENGTH[3].Z	Z axis.
V.MPA.SSIDATALENGTH[3].S	Spindle S.
V.SP.SSIDATALENGTH[3].S	Spindle S.
V.SP.SSIDATALENGTH[3]	Master spindle.
V.MPA.SSIDATALENGTH[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.SSIDATALENGTH[3].1	Axis with index ·1· in the channel ·2·.
V.SP.SSIDATALENGTH[3].2	Spindle with index ·2· in the system.
V.[2].SP.SSIDATALENGTH[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.SSIPACKFORMAT[set].xn

(V.)[ch].MPA.SSIPACKFORMAT[set].sn

(V.)[ch].SP.SSIPACKFORMAT[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for velocity-Sercos drives.

The variable returns the value of block preparation.

Syntax.

·ch· Channel number.

·set· Parameter set.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.MPA.SSIPACKFORMAT[3].Z	Z axis.
V.MPA.SSIPACKFORMAT[3].S	Spindle S.
V.SP.SSIPACKFORMAT[3].S	Spindle S.
V.SP.SSIPACKFORMAT[3]	Master spindle.
V.MPA.SSIPACKFORMAT[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.SSIPACKFORMAT[3].1	Axis with index ·1· in the channel ·2·.
V.SP.SSIPACKFORMAT[3].2	Spindle with index ·2· in the system.
V.[2].SP.SSIPACKFORMAT[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Data.
1	Data-CRC.
2	CRC-Data.
3	Data-Alarm.
4	Alarm-Data.
5	Data-CRC-Alarm
6	Alarm-Data-CRC.
7	Data-Alarm-CRC.
8	Alarm-CRC-Data.
9	CRC-Data-Alarm.
10	CRC-Alarm-Data.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].MPA.SSICRCTYPE[set].xn

(V.)[ch].MPA.SSICRCTYPE[set].sn

(V.)[ch].SP.SSICRCTYPE[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for velocity-Sercos drives.

The variable returns the value of block preparation.

Tipo de CRC.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.SSICRCTYPE[3].Z	Z axis.
V.MPA.SSICRCTYPE[3].S	Spindle S.
V.SP.SSICRCTYPE[3].S	Spindle S.
V.SP.SSICRCTYPE[3]	Master spindle.
V.MPA.SSICRCTYPE[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.SSICRCTYPE[3].1	Axis with index ·1· in the channel ·2·.
V.SP.SSICRCTYPE[3].2	Spindle with index ·2· in the system.
V.[2].SP.SSICRCTYPE[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	"Do not calculate CRC"
1	Fagor checksum.
2	INDUCTOSYN checksum.

(V.)[ch].MPA.SSICRCBITS[set].xn

(V.)[ch].MPA.SSICRCBITS[set].sn

(V.)[ch].SP.SSICRCBITS[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for velocity-Sercos drives.

The variable returns the value of block preparation.

Number of CRC bits of the SSI transmission to check that the transmission is valid.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.SSICRCBITS[3].Z	Z axis.
V.MPA.SSICRCBITS[3].S	Spindle S.
V.SP.SSICRCBITS[3].S	Spindle S.
V.SP.SSICRCBITS[3]	Master spindle.
V.MPA.SSICRCBITS[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.SSICRCBITS[3].1	Axis with index ·1· in the channel ·2·.
V.SP.SSICRCBITS[3].2	Spindle with index ·2· in the system.
V.[2].SP.SSICRCBITS[3].1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

(V.) [ch].MPA.SSI STARTBITS[set].xn

(V.) [ch].MPA.SSI STARTBITS[set].sn

(V.) [ch].SP.SSI STARTBITS[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for velocity-Sercos drives.

The variable returns the value of block preparation.

Número de bits de start.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.SSI STARTBITS[3].Z	Z axis.
V.MPA.SSI STARTBITS[3].S	Spindle S.
V.SP.SSI STARTBITS[3].S	Spindle S.
V.SP.SSI STARTBITS[3]	Master spindle.
V.MPA.SSI STARTBITS[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.SSI STARTBITS[3].1	Axis with index ·1· in the channel ·2·.
V.SP.SSI STARTBITS[3].2	Spindle with index ·2· in the system.
V.[2].SP.SSI STARTBITS[3].1	Spindle with index ·1· in the channel ·2·.

(V.) [ch].MPA.SSI ALARMBITS[set].xn

(V.) [ch].MPA.SSI ALARMBITS[set].sn

(V.) [ch].SP.SSI ALARMBITS[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for velocity-Sercos drives.

The variable returns the value of block preparation.

Number of alarm bits.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.SSI ALARMBITS[3].Z	Z axis.
V.MPA.SSI ALARMBITS[3].S	Spindle S.
V.SP.SSI ALARMBITS[3].S	Spindle S.
V.SP.SSI ALARMBITS[3]	Master spindle.
V.MPA.SSI ALARMBITS[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.SSI ALARMBITS[3].1	Axis with index ·1· in the channel ·2·.
V.SP.SSI ALARMBITS[3].2	Spindle with index ·2· in the system.
V.[2].SP.SSI ALARMBITS[3].1	Spindle with index ·1· in the channel ·2·.

(V.) [ch].MPA.SSI ALARMLEVEL[set].xn

(V.) [ch].MPA.SSI ALARMLEVEL[set].sn

(V.) [ch].SP.SSI ALARMLEVEL[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for velocity-Sercos drives.

The variable returns the value of block preparation.

Level of the alarm bits to issue an error.



CNC 8060
CNC 8065

(REF: 1402)

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.SSIALARMLEVEL[3].Z	Z axis.
V.MPA.SSIALARMLEVEL[3].S	Spindle S.
V.SP.SSIALARMLEVEL[3].S	Spindle S.
V.SP.SSIALARMLEVEL[3]	Master spindle.
V.MPA.SSIALARMLEVEL[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.SSIALARMLEVEL[3].1	Axis with index ·1· in the channel ·2·.
V.SP.SSIALARMLEVEL[3].2	Spindle with index ·2· in the system.
V.[2].SP.SSIALARMLEVEL[3].1	Spindle with index ·1· in the channel ·2·.

- (V.)[ch].MPA.SSIDATAMODE[set].xn
- (V.)[ch].MPA.SSIDATAMODE[set].sn
- (V.)[ch].SP.SSIDATAMODE[set].sn

Variable that can only be read from the program, PLC and interface.
 Variable valid for rotary and linear axes as well as for spindles.
 Variable valid for velocity-Sercos drives.
 The variable returns the value of block preparation.

Transmission mode.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.SSIDATAMODE[3].Z	Z axis.
V.MPA.SSIDATAMODE[3].S	Spindle S.
V.SP.SSIDATAMODE[3].S	Spindle S.
V.SP.SSIDATAMODE[3]	Master spindle.
V.MPA.SSIDATAMODE[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.SSIDATAMODE[3].1	Axis with index ·1· in the channel ·2·.
V.SP.SSIDATAMODE[3].2	Spindle with index ·2· in the system.
V.[2].SP.SSIDATAMODE[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	The LSB (Least Significant Bit) is the first one.
1	The MSB (Most Significant Bit) is the first one.

- (V.)[ch].MPA.STARTDELAY[set].xn
- (V.)[ch].MPA.STARTDELAY[set].sn
- (V.)[ch].SP.STARTDELAY[set].sn

Variable that can only be read from the program, PLC and interface.
 Variable valid for rotary and linear axes as well as for spindles.
 Variable valid for velocity-Sercos drives.
 The variable returns the value of block preparation.

Number of clocks to wait between the first down flank and the first up flank.



CNC 8060
 CNC 8065

(REF: 1402)

22.

CNC VARIABLES.

Variables related to the sets of machine parameters.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.STARTDELAY[3].Z	Z axis.
V.MPA.STARTDELAY[3].S	Spindle S.
V.SP.STARTDELAY[3].S	Spindle S.
V.SP.STARTDELAY[3]	Master spindle.
V.MPA.STARTDELAY[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.STARTDELAY[3].1	Axis with index ·1· in the channel ·2·.
V.SP.STARTDELAY[3].2	Spindle with index ·2· in the system.
V.[2].SP.STARTDELAY[3].1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].MPA.SSIREVOL[set].xn**(V.)[ch].MPA.SSIREVOL[set].sn****(V.)[ch].SP.SSIREVOL[set].sn***Variable that can only be read from the program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**Variable valid for velocity-Sercos drives.**The variable returns the value of block preparation.*

Digital counting (feedback) resolution.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.SSIREVOL[3].Z	Z axis.
V.MPA.SSIREVOL[3].S	Spindle S.
V.SP.SSIREVOL[3].S	Spindle S.
V.SP.SSIREVOL[3]	Master spindle.
V.MPA.SSIREVOL[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.SSIREVOL[3].1	Axis with index ·1· in the channel ·2·.
V.SP.SSIREVOL[3].2	Spindle with index ·2· in the system.
V.[2].SP.SSIREVOL[3].1	Spindle with index ·1· in the channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

(V.)MPMAN.COUNTERTYPE[hw]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Handwheel [hw]. Type of handwheel feedback input.

Syntax.

.hw· Handwheel number.

V.MPMAN.COUNTERTYPE[1] Handwheel ·1·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Handwheel connected to the remote feedback modules.
1	Handwheel connected to the keyboards.
2	Handwheel connected to the local feedback inputs.
3	Handwheel connected to an RCS-S module.

(V.)MPMAN.COUNTERID[hw]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Handwheel [hw]. Feedback input associated with the handwheel.

Syntax.

.hw· Handwheel number.

V.MPMAN.COUNTERID[1] Handwheel ·1·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
-1 -2 -3	Handwheel connected to the first keyboard.
-4 -5 -6	Handwheel connected to the second keyboard.
-7 -8 -9	Handwheel connected to the third keyboard.
1 .. 40	Feedback input of the remote modules.
1 .. 2	Local feedback input.
1 .. 32	Feedback input of the RCS-S module.

DELAY ESTIMATE AT THE DRIVE.

(V.)[ch].MPA.AXDELAY[set].xn

(V.)[ch].MPA.AXDELAY[set].sn

(V.)[ch].SP.AXDELAY[set].sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the value of block preparation.

Parameter AXDELAY is an estimate of the drive's delay when applying the velocity command sent by the CNC.

22.

CNC VARIABLES.
Variables related to the sets of machine parameters.

Syntax.

- ch· Channel number.
- set· Parameter set.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.AXDELAY[3].Z	Z axis.
V.MPA.AXDELAY[3].S	Spindle S.
V.SP.AXDELAY[3].S	Spindle S.
V.SP.AXDELAY[3]	Master spindle.
V.MPA.AXDELAY[3].4	Axis or spindle with logic number ·4·.
V.[2].MPA.AXDELAY[3].1	Axis with index ·1· in the channel ·2·.
V.SP.AXDELAY[3].2	Spindle with index ·2· in the system.
V.[2].SP.AXDELAY[3].1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
Standard.	For Fagor drives, that do not require any delay compensation.
0.	The CNC does not apply delay compensation at the drive.
1 to 127.	Delay cycles at the drive. The CNC automatically compensates for the delay differences between the axes of the channel.



CNC 8060
CNC 8065

(REF: 1402)

22.8 Variables related to machine parameters for JOG mode.

HANDWHEEL CONFIGURATION.

(V.)MPMAN.NMPG

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of handwheels connected to the CNC.

V.MPMAN.NMPG

(V.)MPMAN.COUNTERTYPE[hw]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Handwheel [hw]. Type of handwheel feedback input.

Syntax.

·hw· Handwheel number.

V.MPMAN.COUNTERTYPE[1] Handwheel ·1·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Handwheel connected to the remote feedback modules.
1	Handwheel connected to the keyboards.
2	Handwheel connected to the local feedback inputs.
3	Handwheel connected to an RCS-S module.

(V.)MPMAN.COUNTERID[hw]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Handwheel [hw]. Feedback input associated with the handwheel.

Syntax.

·hw· Handwheel number.

V.MPMAN.COUNTERID[1] Handwheel ·1·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
-1 -2 -3	Handwheel connected to the first keyboard.
-4 -5 -6	Handwheel connected to the second keyboard.
-7 -8 -9	Handwheel connected to the third keyboard.
1 .. 40	Feedback input of the remote modules.
1 .. 2	Local feedback input.
1 .. 32	Feedback input of the RCS-S module.

22.

CNC VARIABLES.

Variables related to machine parameters for JOG mode.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

(V.)MPMAN.MPGAXIS[hw]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Handwheel [hw]. Logic number of the axis associated with the handwheel.

If the variable returns a -0- value, it means that it is a general handwheel that may be used to move any axis.

Syntax.

·hw· Handwheel number.

V.MPMAN.MPGAXIS[1]	Handwheel -1-
--------------------	---------------

Remarks.

The logic number of the axes is determined by the order in which the axes have been defined in the machine parameter table. The first axis of the table will be logic axis -1- and so on.

CONFIGURE THE JOG KEYS.

(V.)MPMAN.JOGKEYDEF[jk]

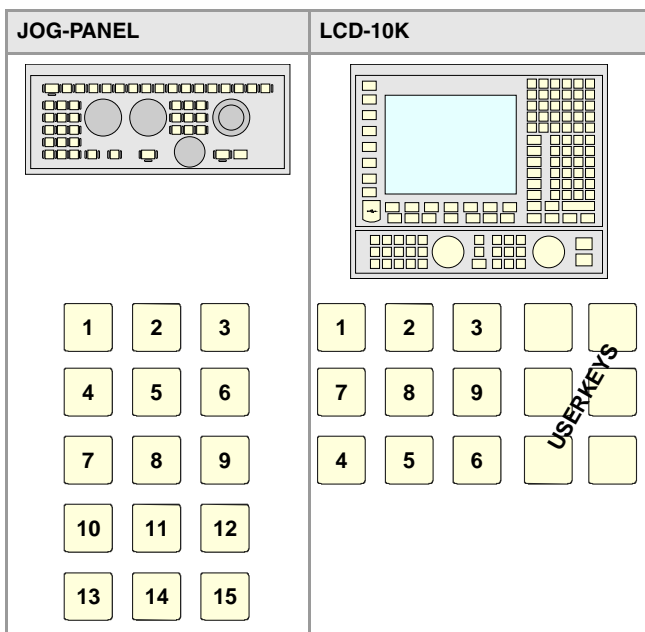
(V.)MPMAN.JOGKEY2DEF[jk]

..

(V.)MPMAN.JOGKEY8DEF[jk]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Jog key [jk]. Axis and moving direction.



Syntax.

·jk· Jog key number.

V.MPMAN.JOGKEYDEF[11]	Jog key -11- of first keyboard.
V.MPMAN.JOGKEY3DEF[11]	Jog key -11- of third keyboard.

22.

CNC VARIABLES.
Variables related to machine parameters for JOG mode.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	The key has no function assigned to it.
1 .. 16	The key is assigned to the logic axis 1, 2, .., 16 for movements in the positive direction.
-1 .. -16	The key is assigned to the logic axis 1, 2, .., 16 for movements in the negative direction.
101 .. 116	The key is assigned to the logic axis 1, 2, .., 16.
300	The key is assigned to the rapid movement.
301	The key is assigned to the movement in the positive direction.
302	The key is assigned to the movement in the negative direction.

Remarks.

The logic number of the axes is determined by the order in which the axes have been defined in the machine parameter table. The first axis of the table will be logic axis -1- and so on.

(V.)MPMAN.JOGTYPE

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Behavior of the JOG keys.

```
V.MPMAN.JOGTYPE
```

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Pressed axis. The axis will move while both keys are pressed, the axis key and the direction key.
1	Selected axis. Pressing the axis key selects the axis. The axis will move while the direction key is kept pressed.

CONFIGURE THE USER KEYS AS JOG KEYS

(V.)MPMAN.USERKEYDEF[uk]

(V.)MPMAN.USERKEY2DEF[uk]

..

(V.)MPMAN.USERKEY8DEF[uk]

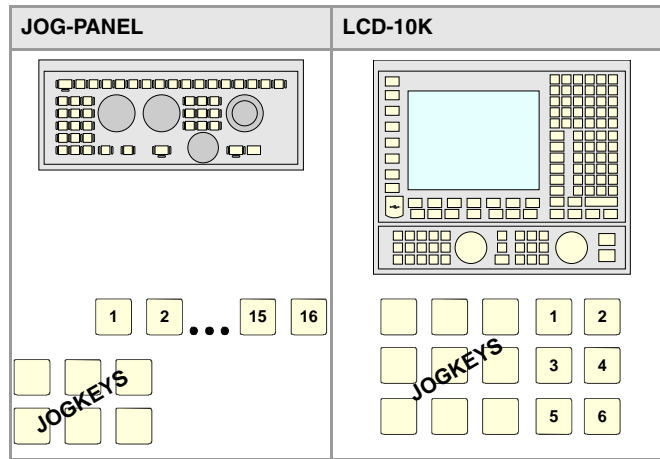
*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

User key [uk] as jog key.



CNC 8060
CNC 8065

(REF: 1402)



Syntax.

·uk· User key number.

V.MPMAN.USERKEYDEF[7]	User key ·7· of first keyboard.
V.MPMAN.USERKEY3DEF[7]	User key ·7· of third keyboard.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	The key has no function assigned to it.
1 .. 16	The key is assigned to the logic axis 1, 2, .., 16 for movements in the positive direction.
-1 .. -16	The key is assigned to the logic axis 1, 2, .., 16 for movements in the negative direction.
101 .. 116	The key is assigned to the logic axis 1, 2, .., 16.
300	The key is assigned to the rapid movement.
301	The key is assigned to the movement in the positive direction.
302	The key is assigned to the movement in the negative direction.

Remarks.

The logic number of the axes is determined by the order in which the axes have been defined in the machine parameter table. The first axis of the table will be logic axis -1- and so on.

(V.)MPMAN.HBLS

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

This parameter indicates whether there is a portable panel HBLS connected to the CNC through the serial line or not.

V.MPMAN.HBLS

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

22.

CNC VARIABLES.
Variables related to machine parameters for JOG mode.



CNC 8060
CNC 8065

(REF: 1402)

22.9 Variables related to machine parameters for M functions.

(V.)MPM.MTABLESIZE

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

"M" function table. Number of table elements.

V.MPM.MTABLESIZE

(V.)MPM.MNUM[pos]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Position [pos] of the "M" function table. "M" function number.

If the variable returns a -1 value, it means that no "M" function has been defined in that position.

Syntax.

·pos· Position inside the "M" function table.

V.MPM.MNUM[12]

Position ·12· of the "M" function table.

(V.)MPM.SYNCHTYPE[pos]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Position [pos] of the "M" function table. Type of synchronism.

Since the M functions may be programmed together with the movement of the axes, in the same block, it must be indicated when the function is to be sent out to the PLC and when it will be checked that it has been already executed (synchronization). The M functions may be sent and/or synchronized before or after the movement.

Syntax.

·pos· Position inside the "M" function table.

V.MPM.SYNCHTYPE[12]

Position ·12· of the "M" function table.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Without synchronization.
2	The M function is both sent to the PLC and synchronized before the movement.
4	The M function is sent to the PLC before the movement and synchronized after the movement.
8	The M function is both sent to the PLC and synchronized after the movement.

(V.)MPM.MPROGNAME[pos]

Variable to be read via interface.

Position [pos] of the "M" function table. Name of the associated subroutine.

MPM.MPROGNAME[12]

Position ·12· of the "M" function table.

(V.)MPM.MTIME[pos]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Position [pos] of the "M" function table. Estimated execution time (in milliseconds).

22.

CNC VARIABLES.

Variables related to machine parameters for M functions.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

Syntax.

·pos· Position inside the "M" function table.

V.MPM.MTIME[12] Position ·12· of the "M" function table.

(V.)MPM.MPLC[pos]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Position [pos] of the "M" function table. Send the M function to the PLC during block search.

Syntax.

·pos· Position inside the "M" function table.

V.MPM.MPLC[12] Position ·12· of the "M" function table.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

22.

CNC VARIABLES.
Variables related to machine parameters for M functions.



CNC 8060
CNC 8065

(REF: 1402)

22.10 Variables related to kinematic machine parameters.

KINEMATICS CONFIGURATION.

(V.)MPK.NKIN

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Kinematics table. Number of kinematics defined.

V.MPK.NKIN

(V.)MPK.TYPE[kin]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Kinematics [kin]. Kinematics type.

Syntax.

.kin· Number of the kinematics.

V.MPK.TYPE[3] Kinematics ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Orthogonal or spherical spindle head YX.
2	Orthogonal or spherical spindle head ZX.
3	Orthogonal or spherical spindle head XY.
4	Orthogonal or spherical spindle head ZY.
5	Swivel (angular) spindle XZ.
6	Swivel (angular) spindle YZ.
7	Swivel (angular) spindle ZX.
8	Swivel (angular) spindle ZY.
9	Rotary table AB.
10	Rotary table AC.
11	Rotary table BA.
12	Rotary table BC.
13	Spindle - table AB.
14	Spindle - table AC.
15	Spindle - table BA.
16	Spindle - table BC.
17	Orthogonal spindle with three rotary axes ABA.
18	Orthogonal spindle with three rotary axes ACA.
19	Orthogonal spindle with three rotary axes ACB.
20	Orthogonal spindle with three rotary axes BAB.
21	Orthogonal spindle with three rotary axes BCA.
22	Orthogonal spindle with three rotary axes BCB.
23	Orthogonal spindle with three rotary axes CAB.
24	Orthogonal spindle with three rotary axes CBA.
41	"C" axis. Machining on the face of the part when ALIGNC = YES.

22.

CNC VARIABLES.

Variables related to kinematic machine parameters.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

Value.	Meaning.
42	"C" axis. Machining on the face of the part when ALIGNC = NO.
43	"C" axis. Machining of the turning side of the part.
100 .. 105	OEM Kinematics.

(V.)MPK.TDATAkin[nb]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Kinematics [kin]. Value of parameter TDATA [nb].

Syntax.

·kin· Number of the kinematics.

·nb· Parameter number.

V.MPK.TDATA2[34] Kinematics ·2·. Value of parameter TDATA34.

(V.)MPK.TDATA_Ikin[nb]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Kinematics [kin]. Value of parameter TDATA_I [nb].

Syntax.

·kin· Number of the kinematics.

·nb· Parameter number.

V.MPK.TDATA2[23] Kinematics ·2·. Value of parameter TDATA_I23.

(V.)MPK.NKINAX[kin]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Kinematics [kin]. Number of axes of the kinematics.

Syntax.

·kin· Number of the kinematics.

V.MPK.NKINAX[2] Kinematics ·2·.

(V.)MPK.PARAM_D_SIZE[kin]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Kinematics [kin]. Number of parameters in decimal format

Syntax.

·kin· Number of the kinematics.

V.MPK.PARAM_D_SIZE[2] Kinematics ·2·.

(V.)MPK.PARAM_I_SIZE[kin]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Kinematics [kin]. Number of parameters in integer format.

22.

CNC VARIABLES.
Variables related to kinematic machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

Syntax.

·kin· Number of the kinematics.

V.MPK.PARAM_I_SIZE[2] Kinematics ·2·.

(V.)MPK.AUXCTE_SIZE[kin]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Kinematics [kin]. Size of the area of auxiliary variables.

Syntax.

·kin· Number of the kinematics.

V.MPK.AUXCTE_SIZE[2] Kinematics ·2·.

(V.)MPK.KINDATA_SIZE[kin]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Kinematics [kin]. Size of the area for general purpose data.

Syntax.

·kin· Number of the kinematics.

V.MPK.KINDATA_SIZE[2] Kinematics ·2·.

CONFIGURATION OF ANGULAR TRANSFORMATIONS.**(V.)MPK.NANG**

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of angular transformations defined.

V.MPK.NANG

(V.)MPK.ANGAXNA[ang]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Angular transformation [ang]. Logic number of the angular axis.

Syntax.

·ang· Number of the angular transformation.

V.MPK.ANGAXNA[2] Angular transformation ·2·.

Remarks.

The logic number of the axes is determined by the order in which the axes have been defined in the machine parameter table. The first axis of the table will be logic axis -1- and so on.

(V.)MPK.ORTAXNA[ang]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Angular transformation [ang]. Logic number of the orthogonal axis.

22.**CNC VARIABLES.**

Variables related to kinematic machine parameters.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to kinematic machine parameters.

Syntax.

·ang· Number of the angular transformation.

V.MPK.ORTAXNA[2]	Angular transformation ·2·.
------------------	-----------------------------

Remarks.

The logic number of the axes is determined by the order in which the axes have been defined in the machine parameter table. The first axis of the table will be logic axis -1- and so on.

(V.)MPK.ANGANTR[ang]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Angular transformation [ang]. Angle between the Cartesian axis and the incline axis.

Syntax.

·ang· Number of the angular transformation.

V.MPK.ORTAXNA[2]	Angular transformation ·2·.
------------------	-----------------------------

(V.)MPK.OFFANGAX[ang]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Angular transformation [ang]. Offset of the origin of the angular transformation.

Syntax.

·ang· Number of the angular transformation.

V.MPK.OFFANGAX[2]	Angular transformation ·2·.
-------------------	-----------------------------

22.11 Variables related to machine parameters for the tool magazine.

(V.)TM.NTOOLMZ

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of tool magazines.

V.TM.NTOOLMZ

(V.)TM.MZGROUND

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Ground tools are permitted (manual load).

V.TM.MZGROUND

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)TM.MZSIZE[mz]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Tool magazine [mz]. Size of the tool magazine (number of pockets).

Syntax.

·mz· Type of tool magazine.

V.TM.MZSIZE[2] Magazine ·2·.

(V.)TM.MZRANDOM[mz]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Tool magazine [mz]. Random magazine.

In a random tool magazine, the tools can occupy any position. In a non-random tool magazine, the tools always occupy the same position.

Syntax.

·mz· Type of tool magazine.

V.TM.MZRANDOM[2] Magazine ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	It is not a random tool magazine.
1	It is a random tool magazine.

22.

CNC VARIABLES.

Variables related to machine parameters for the tool magazine.



CNC 8060
CNC 8065

(REF: 1402)

(V.)TM.MZTYPE[mz]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Tool magazine [mz]. Type of magazine.

Syntax.

·mz· Type of tool magazine.

V.TM.MZTYPE[2] Magazine -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Asynchronous.
2	Synchronous.
3	Turret
4	Synchronous with 2 arms.
5	Synchronous with 1 arm.

(V.)TM.MZCYCLIC[mz]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Tool magazine [mz]. Cyclic tool changer.

A cyclic tool changer requires a tool change command (function M06) after searching a tool and before searching the next one. In a non-cyclic magazine, it is possible to perform several consecutive tool searches without necessarily having to change the tool.

Syntax.

·mz· Type of tool magazine.

V.TM.MZCYCLIC[2] Magazine -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	It is not a cyclic tool magazine.
1	It is a cyclic tool magazine.

(V.)TM.MZOPTIMIZED[mz]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Tool magazine [mz]. Tool management.

When programming several T's in a row without M06, a magazine with optimized management only looks for the tool to be changed; a magazine without optimized management looks for all the tools.

Syntax.

·mz· Type of tool magazine.

V.TM.MZOPTIMIZED[2] Magazine -2.

22.

CNC VARIABLES.
Variables related to machine parameters for the tool magazine.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	The tool magazine has no optimized management.
1	The tool magazine has optimized management.

(V.)TM.MZRESPECTSIZE[mz]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Tool magazine [mz]. In the random magazine, look for pockets of the same size.

Syntax.

·mz· Type of tool magazine.

V.TM.MZRESPECTSIZE[2] Magazine ·2·.

(V.)TM.MZM6ALONE[mz]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Tool magazine [mz]. Action after executing an M6 without having selected a tool.

Syntax.

·mz· Type of tool magazine.

V.TM.MZM6ALONE[2] Magazine ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No action is carried out.
1	The CNC issues a warning.
2	The CNC issues an error message.

22.

CNC VARIABLES.
Variables related to machine parameters for the tool magazine.



CNC 8060
CNC 8065

(REF: 1402)

22.12 Variables related to OEM machine parameters.

GENERIC OEM PARAMETERS.

(V.)MTB.SIZE

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of OEM parameters.

V.MPB.SIZE

(V.)MTB.P[i]

Variable that can be read and written via program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Value of the OEM parameter [nb]

Syntax.

·nb· Parameter number.

V.MTB.P[10] Value of the OEM parameter P10.

Remarks.

When reading this variable from the PLC, it truncates the decimal side. If the value of the parameter is 54.9876, when read from the PLC, it will return the value 54.

Machine parameter.	Reading from the PLC. MTB.P[0]	Reading from the PLC. V.MTB.PF[0]
P0 = 54.9876	54	549876
P0 = -34.1234	-34	-341234

It must be borne in mind that reading and writing these variables interrupts block preparation affecting program execution time. If the value of the parameter is not going to be changed during execution, it is recommended to read the MTB variables at the beginning of the program using arithmetic parameters (local or global) and use the global ones throughout the program.

(V.)MTB.PF[i]

Variable that can be read and written via program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Value of the OEM parameter [nb] Value per 10000.

Syntax.

·nb· Parameter number.

V.MTB.PF[10] Value of the OEM parameter P10.

Remarks.

When reading this variable from the PLC, it returns the value in ten-thousandths. If the value of the parameter is 54.9876, when read from the PLC, it will return the value 549876.

Machine parameter.	Reading from the PLC. MTB.P[0]	Reading from the PLC. V.MTB.PF[0]
P0 = 54.9876	54	549876
P0 = -34.1234	-34	-341234

It must be borne in mind that reading and writing these variables interrupts block preparation affecting program execution time. If the value of the parameter is not going to be changed during execution, it is recommended to read the MTB variables at the beginning of the

22.

CNC VARIABLES.
Variables related to OEM machine parameters.



CNC 8060
CNC 8065

(REF: 1402)

program using arithmetic parameters (local or global) and use the global ones throughout the program.

READING DRIVE VARIABLES.

(V.)DRV.SIZE

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of variables to be consulted at the drive.

V.DRV.SIZE

(V.)DRV.name

(V.)DRV.name.xn

(V.)DRV.name.sn

Variable that can be read and written via program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Variables defined in parameter DRIVEVAR that grant access to the variables and/or parameters of the digital drives.

- Sercos. DRV variables can access the variables of the drive. These variables may be read-only or read and written.
- Mechatrolink. DRV variables can access the variables and/or parameters of the drive. The variables of the drive can only be read, whereas the parameters may be read or written.

Syntax.

·name· Mnemonic name set at the machine parameters.

·xn· Axis name.

·sn· Spindle name.

V.DRV.FEED	Value of the variable defined as FEED.
V.DRV.AXISFEED.Z	Value of the variable defined as AXISFEED for the Z axis.
V.DRV.AXISFEED.S	Value of the variable defined as AXISFEED for the spindle S.

22.

CNC VARIABLES.

Variables related to OEM machine parameters.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.13 Variables associated with the status and resources of the PLC.

22.

CNC VARIABLES.
Variables associated with the status and resources of the PLC.

PLC STATUS.

(V.)PLC.STATUS

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

PLC status.

V.PLC.STATUS

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	PLC stopped.
1	PLC running.

PLC RESOURCES.

(V.)PLC.I[nb]

Variable that can be read and written via program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Status of PLC digital input [nb].

Syntax.

·nb· Number of the digital input.

V.PLC.I[122] Status of PLC digital input ·122·.

(V.)PLC.O[nb]

Variable that can be read and written via program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Status of PLC digital output [nb].

Syntax.

·nb· Number of the digital output.

V.PLC.O[243] Status of PLC digital output ·243·.

(V.)PLC.LI[nb]

Variable that can be read and written via program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Status of PLC local digital input [nb].

Syntax.

·nb· Number of the digital input.

V.PLC.LI[2] Status of PLC local digital input ·2·.



CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.LO[nb]

Variable that can be read and written via program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Status of PLC local digital output [nb].

Syntax.

.nb- Number of the digital output.

V.PLC.LO[3]

Status of PLC local digital output ·3·.

(V.)PLC.M[nb]

Variable that can be read and written via program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Status of PLC mark [nb].

Syntax.

.nb- Mark number.

V.PLC.M[111]

Status of PLC mark ·111·.

(V.)PLC.R[nb]

Variable that can be read and written via program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Status of PLC register [nb].

Syntax.

.nb- Register number.

V.PLC.R[200]

Status of PLC register ·200·.

(V.)PLC.T[nb]

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Status of PLC timer [nb]

Syntax.

.nb- Timer number.

V.PLC.T[8]

Status of PLC timer ·8·.

(V.)PLC.C[nb]

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Status of PLC counter [nb].

Syntax.

.nb- Counter number.

V.PLC.C[16]

Status of PLC counter ·16·.

PLC MESSAGES.**(V.)PLC.MSG[msg]**

Variable that can be read and written via program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Status of PLC message [msg].

22.**CNC VARIABLES.**

Variables associated with the status and resources of the PLC.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

Syntax.

·msg· Message number.

V.PLC.MSG[87] Status of message 87.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Message off.
1	Message on.

(V.)PLC.PRIORMSG

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Active message with the highest priority (the one with the lowest number among the active ones).

V.PLC.PRIORMSG

(V.)PLC.EMERGMSG

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Active emerging message (the one shown at full screen).

V.PLC.EMERGMSG

PLC ERRORS.**(V.)PLC.ERR[err]**

Variable that can be read and written via program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Status of PLC error [err].

Syntax.

·err· Error number.

V.PLC.ERR[62] Status of error 62.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Error off.
1	Error on.

(V.)PLC.PRIORERR

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Active error with the highest priority (the one with the lowest number among the active ones).

V.PLC.PRIORERR

22.

CNC VARIABLES.
Variables associated with the status and resources of the PLC.



CNC 8060
CNC 8065

(REF: 1402)

PLC CLOCKS.

(V.)PLC.TIMER

Variable that can be read and written via program, PLC and interface.
 The variable returns the execution value; reading it interrupts block preparation.
 Report variable (to be used from the scripts).

Value of the freely available clock (in seconds).

V.PLC.TIMER

This variable may be used to consult and/or modify the timing of the clock. Value in seconds.

Remarks.

The PLC "TIMER" is enabled or disabled with the PLC mark TIMERON. This clock is timing when TIMERON=1.

(V.)PLC.CLKnb

Variable that can be read and written via program and interface.
 The variable returns the execution value; reading it interrupts block preparation.

Status of PLC clock nb.

V.PLC.CLK128

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Clock off.
1	Clock on.

22.

CNC VARIABLES.

Variables associated with the status and resources of the PLC.

FAGOR 

CNC 8060
 CNC 8065

(REF: 1402)

22.14 PLC consulting logic signals; general.

22.

CNC VARIABLES.
PLC consulting logic signals; general.

(V.)PLC.CNCREADY

Variable that can only be read from the program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The CNC activates the mark when it is not in an error state.

V.PLC.CNCREADY

(V.)PLC.READY

Variable that can only be read from the program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The channel activates the mark when it is not in an error state.

V.PLC.READYC1	Channel -1.
V.PLC.READYC2	Channel -2.
V.PLC.READYC3	Channel -3.
V.PLC.READYC4	Channel -4.

(V.)PLC.SERCO SRDY

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The CNC activates the mark when the Sercos ring has been initialized properly.

V.PLC.SERCO SRDY

(V.)PLC.START

Variable that can only be read from the program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The channel activates this mark when the [START] key has been pressed.

V.PLC.START	Channel -1.
V.PLC.STARTC1	Channel -1.
V.PLC.STARTC2	Channel -2.
V.PLC.STARTC3	Channel -3.
V.PLC.STARTC4	Channel -4.

(V.)PLC.RESETOUT

Variable that can only be read from the program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

When pressing the [RESET] key or when the PLC activates the RESETIN mark, the CNC assumes the initial conditions and activates the RESETOUT mark.

V.PLC.RESETOUT	Channel -1.
V.PLC.RESETOUTC1	Channel -1.
V.PLC.RESETOUTC2	Channel -2.
V.PLC.RESETOUTC3	Channel -3.
V.PLC.RESETOUTC4	Channel -4.



CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.FHOUT

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC channel activates this mark when the execution of the part-program is interrupted.

V.PLC.FHOUT	Channel -1-.
V.PLC.FHOUTC1	Channel -1-.
V.PLC.FHOUTC2	Channel -2-.
V.PLC.FHOUTC3	Channel -3-.
V.PLC.FHOUTC4	Channel -4-.

(V.)PLC._ALARM

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel turns this mark off when there is an alarm or an emergency in the channel.

V.PLC._ALARM	Channel -1-.
V.PLC._ALARMC1	Channel -1-.
V.PLC._ALARMC2	Channel -2-.
V.PLC._ALARMC3	Channel -3-.
V.PLC._ALARMC4	Channel -4-.

(V.)PLC.MANUAL

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when the jog mode is selected.

V.PLC.MANUAL	Channel -1-.
V.PLC.MANUALC1	Channel -1-.
V.PLC.MANUALC2	Channel -2-.
V.PLC.MANUALC3	Channel -3-.
V.PLC.MANUALC4	Channel -4-.

(V.)PLC.AUTOMAT

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when the automatic mode is selected.

V.PLC.AUTOMAT	Channel -1-.
V.PLC.AUTOMATC1	Channel -1-.
V.PLC.AUTOMATC2	Channel -2-.
V.PLC.AUTOMATC3	Channel -3-.
V.PLC.AUTOMATC4	Channel -4-.

22.**CNC VARIABLES.**

PLC consulting logic signals; general.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

(V.)PLC.MDI

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when the MDI/MDA mode is selected.

V.PLC.MDI	Channel -1-.
V.PLC.MDIC1	Channel -1-.
V.PLC.MDIC2	Channel -2-.
V.PLC.MDIC3	Channel -3-.
V.PLC.MDIC4	Channel -4-.

(V.)PLC.SBOUT

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when the single-block mode is selected.

V.PLC.SBOUT	Channel -1-.
V.PLC.SBOUTC1	Channel -1-.
V.PLC.SBOUTC2	Channel -2-.
V.PLC.SBOUTC3	Channel -3-.
V.PLC.SBOUTC4	Channel -4-.

(V.)PLC.INCYCLE

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when it is executing a block or moving an axis.

V.PLC.INCYCLE	Channel -1-.
V.PLC.INCYCLEC1	Channel -1-.
V.PLC.INCYCLEC2	Channel -2-.
V.PLC.INCYCLEC3	Channel -3-.
V.PLC.INCYCLEC4	Channel -4-.

(V.)PLC.RAPID

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when it is executing a rapid positioning (G00).

V.PLC.RAPID	Channel -1-.
V.PLC.RAPIDC1	Channel -1-.
V.PLC.RAPIDC2	Channel -2-.
V.PLC.RAPIDC3	Channel -3-.
V.PLC.RAPIDC4	Channel -4-.

(V.)PLC.ZERO

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates this mark when searching home (G74).

V.PLC.ZERO	Channel -1-.
V.PLC.ZEROC1	Channel -1-.
V.PLC.ZEROC2	Channel -2-.
V.PLC.ZEROC3	Channel -3-.
V.PLC.ZEROC4	Channel -4-.

(V.)PLC.PROBE

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when it is executing a probing movement (G100).

V.PLC.PROBE	Channel -1-.
V.PLC.PROBEC1	Channel -1-.
V.PLC.PROBEC2	Channel -2-.
V.PLC.PROBEC3	Channel -3-.
V.PLC.PROBEC4	Channel -4-.

(V.)PLC.THREAD

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when it is executing an electronic threading (G33).

V.PLC.THREAD	Channel -1-.
V.PLC.THREADC1	Channel -1-.
V.PLC.THREADC2	Channel -2-.
V.PLC.THREADC3	Channel -3-.
V.PLC.THREADC4	Channel -4-.

(V.)PLC.TAPPING

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when it is executing a tapping canned cycle.

V.PLC.TAPPING	Channel -1-.
V.PLC.TAPPINGC1	Channel -1-.
V.PLC.TAPPINGC2	Channel -2-.
V.PLC.TAPPINGC3	Channel -3-.
V.PLC.TAPPINGC4	Channel -4-.

22.

CNC VARIABLES.
PLC consulting logic signals; general.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

(V.)PLC.RIGID

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when it is executing a rigid tapping cycle (G63).

V.PLC.RIGID	Channel -1-.
V.PLC.RIGIDC1	Channel -1-.
V.PLC.RIGIDC2	Channel -2-.
V.PLC.RIGIDC3	Channel -3-.
V.PLC.RIGIDC4	Channel -4-.

(V.)PLC.CSS

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates this mark when constant surface speed (G96) is active.

V.PLC.CSS	Channel -1-.
V.PLC.CSSC1	Channel -1-.
V.PLC.CSSC2	Channel -2-.
V.PLC.CSSC3	Channel -3-.
V.PLC.CSSC4	Channel -4-.

(V.)PLC.INTEREND

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when the theoretical movement of the axes is completed.

V.PLC.INTEREND	Channel -1-.
V.PLC.INTERENDC1	Channel -1-.
V.PLC.INTERENDC2	Channel -2-.
V.PLC.INTERENDC3	Channel -3-.
V.PLC.INTERENDC4	Channel -4-.

(V.)PLC.INPOSI

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when the axes are in position. This mark also remains active during the movement of independent axes.

V.PLC.INPOSI	Channel -1-.
V.PLC.INPOSIC1	Channel -1-.
V.PLC.INPOSIC2	Channel -2-.
V.PLC.INPOSIC3	Channel -3-.
V.PLC.INPOSIC4	Channel -4-.

(V.)PLC.SPN1**(V.)PLC.SPN2****(V.)PLC.SPN3****(V.)PLC.SPN4****(V.)PLC.SPN5**

(V.)PLC.SPN6

(V.)PLC.SPN7

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel indicates in these registers which spindle of the channel the M functions pending execution are addressed to.

There is a register for each channel. The mnemonics for each channel are the following. Here is an example of the mnemonics for SPN1; it is the same for the rest of the registers.

V.PLC.SPN1C1	Channel ·1·.
V.PLC.SPN1C2	Channel ·2·.
V.PLC.SPN1C3	Channel ·3·.
V.PLC.SPN1C4	Channel ·4·.

(V.)PLC.MFUN1

(V.)PLC.MFUN2

(V.)PLC.MFUN3

(V.)PLC.MFUN4

(V.)PLC.MFUN5

(V.)PLC.MFUN6

(V.)PLC.MFUN7

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel indicates in these registers the H functions pending execution.

There is a register for each channel. The mnemonics for each channel are the following. Here is an example of the mnemonics for MFUN1; it is the same for the rest of the registers.

V.PLC.MFUN1	Channel ·1·.
V.PLC.MFUN1C1	Channel ·1·.
V.PLC.MFUN1C2	Channel ·2·.
V.PLC.MFUN1C3	Channel ·3·.
V.PLC.MFUN1C4	Channel ·4·.

(V.)PLC.HFUN1

(V.)PLC.HFUN2

(V.)PLC.HFUN3

(V.)PLC.HFUN4

(V.)PLC.HFUN5

(V.)PLC.HFUN6

(V.)PLC.HFUN7

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel indicates in these registers the H functions pending execution.

There is a register for each channel. The mnemonics for each channel are the following. Here is an example of the mnemonics for HFUN1; it is the same for the rest of the registers.

V.PLC.HFUN1	Channel ·1·.
V.PLC.HFUN1C1	Channel ·1·.
V.PLC.HFUN1C2	Channel ·2·.
V.PLC.HFUN1C3	Channel ·3·.
V.PLC.HFUN1C4	Channel ·4·.



**CNC 8060
CNC 8065**

(REF: 1402)

22.

(V.)PLC.MSTROBE

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates this mark to indicate to the PLC that it must execute the M functions indicated in MFUN1 through MFUN7.

V.PLC.MSTROBE	Channel -1.
V.PLC.MSTROBEC1	Channel -1.
V.PLC.MSTROBEC2	Channel -2.
V.PLC.MSTROBEC3	Channel -3.
V.PLC.MSTROBEC4	Channel -4.

(V.)PLC.HSTROBE

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates this mark to indicate to the PLC that it must execute the H functions indicated in HFUN1 through HFUN7.

V.PLC.HSTROBE	Channel -1.
V.PLC.HSTROBEC1	Channel -1.
V.PLC.HSTROBEC2	Channel -2.
V.PLC.HSTROBEC3	Channel -3.
V.PLC.HSTROBEC4	Channel -4.

(V.)PLC.SFUN1**(V.)PLC.SFUN2****(V.)PLC.SFUN3****(V.)PLC.SFUN4**

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel indicates in these registers the speed programmed in each spindle.

V.PLC.SFUN1	Spindle -1.
V.PLC.SFUN2	Spindle -2.
V.PLC.SFUN3	Spindle -3.
V.PLC.SFUN4	Spindle -4.

(V.)PLC.SSTROBE

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates this mark to indicate to the PLC that a new spindle speed has been selected in registers SFUN1 through SFUN4.

V.PLC.SSTROBE	Channel -1.
V.PLC.SSTROBE1	Channel -1.
V.PLC.SSTROBE2	Channel -2.
V.PLC.SSTROBE3	Channel -3.
V.PLC.SSTROBE4	Channel -4.

(V.)PLC.DM00**(V.)PLC.DM01****(V.)PLC.DM02****(V.)PLC.DM06****(V.)PLC.DM08**

(V.)PLC.DM09**(V.)PLC.DM30**

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC indicates in these marks the status of the M functions. This mark is active if the M function is active.

Each one of functions M00, M01, M02, M06, M08, M09, M30 has a mark for each channel. Here is an example of the mnemonics for DM00; it is the same for the rest of the marks (DM01, DM02, DM06, DM08, DM09, DM30).

V.PLC.DM00	Channel ·1·.
V.PLC.DM00C1	Channel ·1·.
V.PLC.DM00C2	Channel ·2·.
V.PLC.DM00C3	Channel ·3·.
V.PLC.DM00C4	Channel ·4·.

(V.)PLC.DM03**(V.)PLC.DM04****(V.)PLC.DM05****(V.)PLC.DM19****(V.)PLC.DM41****(V.)PLC.DM42****(V.)PLC.DM43****(V.)PLC.DM44**

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC indicates in these marks the status of the spindle M functions. This mark is active if the M function is active.

Each one of functions M03, M04, M05, M19, M41, M42, M43, M44 has a mark for each spindle. Here is an example of the mnemonics for DM03; it is the same for the rest of the marks (DM04, DM05, DM19, DM41, DM42, DM43, DM44).

V.PLC.DM03	Spindle ·1·.
V.PLC.DM03SP1	Spindle ·1·.
V.PLC.DM03SP2	Spindle ·2·.
V.PLC.DM03SP3	Spindle ·3·.
V.PLC.DM03SP4	Spindle ·4·.

(V.)PLC.BLKSEARCH

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when the block-search mode is active.

V.PLC.BLKSEARCH	Channel ·1·.
V.PLC.BLKSEARCHC1	Channel ·1·.
V.PLC.BLKSEARCHC2	Channel ·2·.
V.PLC.BLKSEARCHC3	Channel ·3·.
V.PLC.BLKSEARCHC4	Channel ·4·.

22.**CNC VARIABLES.**

PLC consulting logic signals; general.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

(V.)PLC.ADVINPOS

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates a certain amount of time before the axes reach position. The amount of time is set by parameter ANTIME.

V.PLC.ADVINPOS	Channel -1.
V.PLC.ADVINPOSC1	Channel -1.
V.PLC.ADVINPOSC2	Channel -2.
V.PLC.ADVINPOSC3	Channel -3.
V.PLC.ADVINPOSC4	Channel -4.

(V.)PLC.CAXIS

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC channel sets this mark when there is a spindle working as C axis. This mark is kept active while any of the functions #CAX, #FACE or #CYL are kept active.

V.PLC.CAXIS	Channel -1.
V.PLC.CAXISC1	Channel -1.
V.PLC.CAXISC2	Channel -2.
V.PLC.CAXISC3	Channel -3.
V.PLC.CAXISC4	Channel -4.

(V.)PLC.FREE

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when it can accept a block sent with CNCEX.

V.PLC.FREEC1	Channel -1.
V.PLC.FREEC2	Channel -2.
V.PLC.FREEC3	Channel -3.
V.PLC.FREEC4	Channel -4.

(V.)PLC.WAITOUT

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when it is waiting for a synchronism signal.

V.PLC.WAITOUTC1	Channel -1.
V.PLC.WAITOUTC2	Channel -2.
V.PLC.WAITOUTC3	Channel -3.
V.PLC.WAITOUTC4	Channel -4.

(V.)PLC.MMCWDG

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC activates this mark when the operating system is locked up.

V.PLC.MMCWDG

(V.)PLC.RETRAEND

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates this mark to cancel the retrace function.

V.PLC.RETRAENDC1	Channel ·1·.
V.PLC.RETRAENDC2	Channel ·2·.
V.PLC.RETRAENDC3	Channel ·3·.
V.PLC.RETRAENDC4	Channel ·4·.

(V.)PLC.TANGACTIV

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The channel activates the mark when a tangential control is active.

V.PLC.TANGACTIVC1	Channel ·1·.
V.PLC.TANGACTIVC2	Channel ·2·.
V.PLC.TANGACTIVC3	Channel ·3·.
V.PLC.TANGACTIVC4	Channel ·4·.

(V.)PLC.PSWSET

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC activates this mark when there is an OEM password.

V.PLC.PSWSET

(V.)PLC.DINDISTC1**(V.)PLC.DINDISTC2****(V.)PLC.DINDISTC3****(V.)PLC.DINDISTC4**

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

These marks are related to the dynamic machining distribution between channels (#DINDIST instruction), for the option to distribute passes between channels. During the roughing operation of the cycle, the CNC channel activates these marks to indicate which channel has the cycle been programmed in and which are the channels involved in the distribution of the passes. During the finishing operation, the CNC channel cancels all these marks.

(V.)PLC.DINDISTC1
(V.)PLC.DINDISTC2
(V.)PLC.DINDISTC3
(V.)PLC.DINDISTC4

(V.)PLC.SERPLCAC

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

This mark is related to the change of the work set or of the parameter set of a Sercos drive (variable (V.)[ch].A.SETGE.xn). The CNC activates the mark to indicate that the requested change is being carried out.

V.PLC.SERPLCAC

22.

CNC VARIABLES.
PLC consulting logic signals; general.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.OVERTEMP

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

This mark indicates the temperature status of the CNC. This mark is deactivated while the CNC temperature is correct. When the CNC temperature exceeds the maximum allowed (60 °C, 140 °F), the CNC activates this mark and issues a warning to indicate this situation. The CNC will deactivate this mark when the temperature of the unit goes below the maximum allowed. The CNC checks its temperature every minute.

V.PLC.OVERTEMP

(V.)PLC.MLINKRDY

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC activates the mark when the Mechatrolink bus has been initialized properly.

V.PLC.MLINKRDY

22.

CNC VARIABLES.
PLC consulting logic signals; general.



CNC 8060
CNC 8065

(REF: 1402)

22.15 PLC consulting logic signals; axes and spindles.

(V.)PLC.ENABLExn

(V.)PLC.ENABLEsn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC activates this mark to allow moving the axis or the spindle.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.ENABLEX	X axis.
V.PLC.ENABLES	Spindle S.
V.PLC.ENABLE3	Axis or spindle with logic number -3-.

(V.)PLC.DIRxn

(V.)PLC.DIRsn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC turns this mark on when moving the axis in the negative direction and turns it off when moving the axis in the positive direction. When the axis is stopped, the mark keeps its last value.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.DIRX	X axis.
V.PLC.DIRS	Spindle S.
V.PLC.DIR3	Axis or spindle with logic number -3-.

(V.)PLC.REFPOINxn

(V.)PLC.REFPOINsn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC activates this mark after a home search.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.REFPOINX	X axis.
V.PLC.REFPOINS	Spindle S.
V.PLC.REFPOIN3	Axis or spindle with logic number -3-.

22.

CNC VARIABLES.

PLC consulting logic signals; axes and spindles.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
PLC consulting logic signals; axes and spindles.

(V.)PLC.DRSTAFxn**(V.)PLC.DRSTAFsn**

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC uses these marks to indicate the drive status.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.DRSTAFX	X axis.
V.PLC.DRSTAFS	Spindle S.
V.PLC.DRSTAF3	Axis or spindle with logic number ·3·.

(V.)PLC.DRSTASxn**(V.)PLC.DRSTASsn**

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC uses these marks to indicate the drive status.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.DRSTASX	X axis.
V.PLC.DRSTASS	Spindle S.
V.PLC.DRSTAS3	Axis or spindle with logic number ·3·.

(V.)PLC.INPOSxn**(V.)PLC.INPOSSn**

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC activates the mark when the axis or the spindle is in position.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.INPOSX	X axis.
V.PLC.INPOSS	Spindle S.
V.PLC.INPOS3	Axis or spindle with logic number ·3·.

(V.)PLC.LUBRxn**(V.)PLC.LUBRsn**

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC activates the mark when the axis or the spindle must be lubricated.

Syntax.

·xn· Name or logic number of the axis.



CNC 8060
CNC 8065

(REF: 1402)

·sn· Name or logic number of the spindle.

V.PLC.LUBRX	X axis.
V.PLC.LUBRS	Spindle S.
V.PLC.LUBR3	Axis or spindle with logic number ·3·.

(V.)PLC.HIRTHONxn
(V.)PLC.HIRTHONsn

Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The CNC activates the mark when the axis or the spindle works as Hirth axis.

Syntax.

·xn· Name or logic number of the axis.
 ·sn· Name or logic number of the spindle.

V.PLC.HIRTHX	X axis.
V.PLC.HIRTHS	Spindle S.
V.PLC.HIRTH3	Axis or spindle with logic number ·3·.

(V.)PLC.MATCHxn
(V.)PLC.MATCHsn

Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The CNC activates the mark when the Hirth axis or spindle is positioned properly.

Syntax.

·xn· Name or logic number of the axis.
 ·sn· Name or logic number of the spindle.

V.PLC.MATCHX	X axis.
V.PLC.MATCHS	Spindle S.
V.PLC.MATCH3	Axis or spindle with logic number ·3·.

(V.)PLC.PARKxn
(V.)PLC.PARKsn

Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The CNC activates the mark when it is parking the axis or the spindle.

Syntax.

·xn· Name or logic number of the axis.
 ·sn· Name or logic number of the spindle.

V.PLC.PARKX	X axis.
V.PLC.PARKS	Spindle S.
V.PLC.PARK3	Axis or spindle with logic number ·3·.



CNC 8060
 CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
PLC consulting logic signals; axes and spindles.

(V.)PLC.UNPARKxn
(V.)PLC.UNPARKsn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC activates the mark when it is unparking the axis or the spindle.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.UNPARKX	X axis.
V.PLC.UNPARKS	Spindle S.
V.PLC.UNPARK3	Axis or spindle with logic number ·3·.

(V.)PLC.ACTFBACKxn
(V.)PLC.ACTFBACKsn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

In systems with external+internal feedback, the CNC turns this mark on when it is using the external feedback and turns it off when it is using the internal feedback.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.ACTFBACKX	X axis.
V.PLC.ACTFBACKS	Spindle S.
V.PLC.ACTFBACK3	Axis or spindle with logic number ·3·.

(V.)PLC.TANGACTxn
(V.)PLC.TANGACTsn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC activates the mark when tangential control is active on the axis or spindle.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.TANGACTX	X axis.
V.PLC.TANGACTS	Spindle S.
V.PLC.TANGACT3	Axis or spindle with logic number ·3·.

(V.)PLC.LOPENxn
(V.)PLC.LOPENsn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The CNC activates this mark to inform the PLC that the axis position loop is open.



CNC 8060
CNC 8065

(REF: 1402)

Syntax.

- xn· Name or logic number of the axis.
- sn· Name or logic number of the spindle.

V.PLC.LOPENX	X axis.
V.PLC.LOPENS	Spindle S.
V.PLC.LOPEN3	Axis or spindle with logic number -3-.

(V.)PLC.MAXDIFFxn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Gantry axis [nb]. The CNC cannot correct the position difference between the master axis and the slave axis.

22.16 PLC consulting logic signals; spindles.

22.

CNC VARIABLES.
PLC consulting logic signals; spindles.

(V.)PLC.REVOK

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The spindle turns this mark on when it reaches the programmed rpm.

The mark is also on when the spindle is stopped (M05) or in position (M19, G63).

V.PLC.REVOK	Spindle ·1·.
V.PLC.REVOK1	Spindle ·1·.
V.PLC.REVOK2	Spindle ·2·.
V.PLC.REVOK3	Spindle ·3·.
V.PLC.REVOK4	Spindle ·4·.

(V.)PLC.SYNCMASTER

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The master spindle activates this mark when it has a spindle synchronized using #SYNC.

V.PLC.SYNCHRON1	Spindle ·1·.
V.PLC.SYNCHRON2	Spindle ·2·.
V.PLC.SYNCHRON3	Spindle ·3·.
V.PLC.SYNCHRON4	Spindle ·4·.

(V.)PLC.SYNCHRON

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The slave spindle activates this mark at the beginning of a synchronization using #SYNC.

V.PLC.SYNCHRON1	Spindle ·1·.
V.PLC.SYNCHRON2	Spindle ·2·.
V.PLC.SYNCHRON3	Spindle ·3·.
V.PLC.SYNCHRON4	Spindle ·4·.

(V.)PLC.SYNCHRONP

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The slave spindle activates this mark at the beginning of a synchronization in position.

V.PLC.SYNCHRONP1	Spindle ·1·.
V.PLC.SYNCHRONP2	Spindle ·2·.
V.PLC.SYNCHRONP3	Spindle ·3·.
V.PLC.SYNCHRONP4	Spindle ·4·.



CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.SYNCSPEED

*Variable that can only be read from the program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).*

The slave spindle activates this mark when it is synchronized in velocity.

V.PLC.SYNCSPEED1	Spindle -1-
V.PLC.SYNCSPEED2	Spindle -2-
V.PLC.SYNCSPEED3	Spindle -3-
V.PLC.SYNCSPEED4	Spindle -4-

(V.)PLC.SYNCPOSI

*Variable that can only be read from the program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).*

The slave spindle activates this mark when it is synchronized in position.

V.PLC.SYNCPOSI1	Spindle -1-
V.PLC.SYNCPOSI2	Spindle -2-
V.PLC.SYNCPOSI3	Spindle -3-
V.PLC.SYNCPOSI4	Spindle -4-

(V.)PLC.GEAROK

*Variable that can only be read from the program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).*

The spindle activates this mark when the parameter set selected at the CNC and at the PLC are the same.

V.PLC.GEAROK	Spindle -1-
V.PLC.GEAROK1	Spindle -1-
V.PLC.GEAROK2	Spindle -2-
V.PLC.GEAROK3	Spindle -3-
V.PLC.GEAROK4	Spindle -4-

22.17 PLC consulting logic signals; independent interpolator.

22.

CNC VARIABLES.

PLC consulting logic signals; independent interpolator.

(V.)PLC.IBUSYxn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The interpolator is active when there is an instruction pending execution.

Syntax.

·xn· Name or logic number of the axis.

V.PLC.IBUSYX	X axis.
V.PLC.IBUSY3	Axis with logic number ·3·.

(V.)PLC.IFREExn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The interpolator turns this mark on when it is ready to accept a motion block.

Syntax.

·xn· Name or logic number of the axis.

V.PLC.IFREEX	X axis.
V.PLC.IFREE3	Axis with logic number ·3·.

(V.)PLC.IFHOUTxn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The interpolator turns this mark on when the execution is interrupted.

Syntax.

·xn· Name or logic number of the axis.

V.PLC.IFHOUTX	X axis.
V.PLC.IFHOUT3	Axis with logic number ·3·.

(V.)PLC.IENDxn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The interpolator turns this mark on when the axis movement has finished and it has reached its the final position.

Syntax.

·xn· Name or logic number of the axis.

V.PLC.IENDX	X axis.
V.PLC.IEND3	Axis with logic number ·3·.



CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.ISYNCxn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The interpolator turns this mark on when the axis or the cam has reached synchronism.

Syntax.

·xn· Name or logic number of the axis.

V.PLC.ISYNCX	X axis.
V.PLC.ISYNC3	Axis with logic number ·3·.

22.

CNC VARIABLES.

PLC consulting logic signals; independent interpolator.



CNC 8060
CNC 8065

(REF: 1402)

22.18 PLC consulting logic signals; tool manager.

(V.)PLC.TMOPERATION

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The manager indicates in this register the type of operation to be carried out by the PLC.

V.PLC.TMOPERATION	Channel -1-.
V.PLC.TMOPERATIONC1	Channel -1-.
V.PLC.TMOPERATIONC2	Channel -2-.
V.PLC.TMOPERATIONC3	Channel -3-.
V.PLC.TMOPERATIONC4	Channel -4-.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Do nothing.
1	Take a tool from the magazine and insert it in the spindle.
2	Leave the spindle tool in the magazine.
3	Insert a ground tool in the spindle.
4	Leave the spindle tool on the ground.
5	Leave the spindle tool in the magazine and take another one from the magazine.
6	Leave the spindle tool in the magazine and take another one from ground.
7	Leave the spindle tool on the ground and take another one from the magazine.
8	Leave the spindle tool on the ground and take another one from ground.
9	Take a ground tool to the magazine going through the spindle.
10	Take a tool from the magazine and leave on the ground going through the spindle.
11	Orient the magazine.
12	Leave the spindle tool in the magazine and take another one from the same magazine. Special for a synchronous magazine in the following cases: <ul style="list-style-type: none"> • Non-random having a tool changer arm with two claws. • Random when having special tools.
13	Orienting two magazines
14	Leave the spindle tool in a magazine and take another one from another magazine.

(V.)PLC.TMOPSTROBE

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The manager turns this mark on to let the PLC know that it must execute the operation indicated by TMOPERATION.

V.PLC.TMOPSTROBE	Channel -1-.
V.PLC.TMOPSTROBEC1	Channel -1-.
V.PLC.TMOPSTROBEC2	Channel -2-.
V.PLC.TMOPSTROBEC3	Channel -3-.
V.PLC.TMOPSTROBEC4	Channel -4-.

22.

CNC VARIABLES.
PLC consulting logic signals; tool manager.



CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.LEAVEPOS

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The manager indicates in this register the magazine position to leave the tool.

V.PLC.LEAVEPOS	Magazine ·1·.
V.PLC.LEAVEPOSMZ1	Magazine ·1·.
V.PLC.LEAVEPOSMZ2	Magazine ·2·.
V.PLC.LEAVEPOSMZ3	Magazine ·3·.
V.PLC.LEAVEPOSMZ4	Magazine ·4·.

(V.)PLC.TAKEPOS

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The manager indicates in this register the magazine position of the tool to be picked up.

V.PLC.TAKEPOS	Magazine ·1·.
V.PLC.TAKEPOSMZ1	Magazine ·1·.
V.PLC.TAKEPOSMZ2	Magazine ·2·.
V.PLC.TAKEPOSMZ3	Magazine ·3·.
V.PLC.TAKEPOSMZ4	Magazine ·4·.

(V.)PLC.NEXTPOS

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The manager indicates in this register the magazine position of the next tool.

V.PLC.NEXTPOS	Magazine ·1·.
V.PLC.NEXTPOSMZ1	Magazine ·1·.
V.PLC.NEXTPOSMZ2	Magazine ·2·.
V.PLC.NEXTPOSMZ3	Magazine ·3·.
V.PLC.NEXTPOSMZ4	Magazine ·4·.

(V.)PLC.TWORNOUT

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The manager turns this mark on when it has rejected the tool.

V.PLC.TWONRNOUT	Channel ·1·.
V.PLC.TWONRNOUTC1	Channel ·1·.
V.PLC.TWONRNOUTC2	Channel ·2·.
V.PLC.TWONRNOUTC3	Channel ·3·.
V.PLC.TWONRNOUTC4	Channel ·4·.

22.**CNC VARIABLES.**

PLC consulting logic signals; tool manager.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

(V.)PLC.TMINEM

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The manager activates the mark when it is in an error state.

V.PLC.TMINEM	Magazine -1.
V.PLC.TMINEMZ1	Magazine -1.
V.PLC.TMINEMZ2	Magazine -2.
V.PLC.TMINEMZ3	Magazine -3.
V.PLC.TMINEMZ4	Magazine -4.

(V.)PLC.MZID

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The manager indicates in this register the magazine that has the requested tool. When two magazines are involved in a tool change, the lower portion of this register indicates the destination magazine for the tool and the higher portion the source magazine for the tool.

V.PLC.MZID	Channel -1.
V.PLC.MZIDC1	Channel -1.
V.PLC.MZIDC2	Channel -2.
V.PLC.MZIDC3	Channel -3.
V.PLC.MZIDC4	Channel -4.

22.

CNC VARIABLES.
PLC consulting logic signals; tool manager.



CNC 8060
CNC 8065

(REF: 1402)

22.19 PLC consulting logic signals; keys.

(V.)PLC.KEYBD1

(V.)PLC.KEYBD2

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

These registers are a copy of the map of the keys pressed on the last keyboard used. These registers indicate which key has been pressed (bit=1).

V.PLC.KEYBD1

V.PLC.KEYBD2

(V.)PLC.KEYBD1_1

(V.)PLC.KEYBD2_1

..

(V.)PLC.KEYBD1_8

(V.)PLC.KEYBD2_8

Variable that can only be read from the program and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

These registers indicate (bit=1) which key has been pressed on each operator panel. Registers KEYBD1_1 and KEYBD2_1 correspond to the first jog panel, KEYBD1_2 and KEYBD2_2 to the second one and so on.

V.PLC.KEYBD1_1

V.PLC.KEYBD2_1

22.

CNC VARIABLES.

PLC consulting logic signals; keys.

FAGOR 

**CNC 8060
CNC 8065**

(REF: 1402)

22.20 PLC modifiable logic signals; general.

22.

CNC VARIABLES.
PLC modifiable logic signals; general.

(V.)PLC._EMERGEN

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

If the PLC turns this mark off, the channel stops the axes and spindles and issues an error message.

While the mark is off, the channel does not allow executing programs and aborts attempt to move the axes or start the spindle.

V.PLC._EMERGEN	Channel -1.
V.PLC._EMERGENC1	Channel -1.
V.PLC._EMERGENC2	Channel -2.
V.PLC._EMERGENC3	Channel -3.
V.PLC._EMERGENC4	Channel -4.

(V.)PLC._STOP

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

If the PLC turns this mark off, the channel stops the execution of the program, but keeps the spindles turning. The status of this mark does not affect the independent axes.

V.PLC._STOP	Channel -1.
V.PLC._STOPC1	Channel -1.
V.PLC._STOPC2	Channel -2.
V.PLC._STOPC3	Channel -3.
V.PLC._STOPC4	Channel -4.

(V.)PLC._XFERINH

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

If the PLC turns this mark off, the prevents the execution of the next block, but it lets complete the execution of the current block.

V.PLC._XFERINH	Channel -1.
V.PLC._XFERINHC1	Channel -1.
V.PLC._XFERINHC2	Channel -2.
V.PLC._XFERINHC3	Channel -3.
V.PLC._XFERINHC4	Channel -4.

(V.)PLC._FEEDHOL

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

If the PLC turns this mark off, the channel stops the axes, but keeps the spindles turning. The status of this mark does not affect the independent axes.

V.PLC._FEEDHOL	Channel -1.
V.PLC._FEEDHOLC1	Channel -1.
V.PLC._FEEDHOLC2	Channel -2.
V.PLC._FEEDHOLC3	Channel -3.
V.PLC._FEEDHOLC4	Channel -4.



CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.CYSTART

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

If the PLC turns this mark on, the execution of the part-program begins.

V.PLC.CYSTART	Channel -1-.
V.PLC.CYSTARTC1	Channel -1-.
V.PLC.CYSTARTC2	Channel -2-.
V.PLC.CYSTARTC3	Channel -3-.
V.PLC.CYSTARTC4	Channel -4-.

(V.)PLC.SBLOCK

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

If the PLC turns this mark on, the channel activates the single-block mode.

V.PLC.SBLOCK	Channel -1-.
V.PLC.SBLOCKC1	Channel -1-.
V.PLC.SBLOCKC2	Channel -2-.
V.PLC.SBLOCKC3	Channel -3-.
V.PLC.SBLOCKC4	Channel -4-.

(V.)PLC.MANRAPID

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

If the PLC turns this mark on, the CNC selects the rapid traverse for jog movements.

V.PLC.MANRAPID	Channel -1-.
V.PLC.MANRAPIDC1	Channel -1-.
V.PLC.MANRAPIDC2	Channel -2-.
V.PLC.MANRAPIDC3	Channel -3-.
V.PLC.MANRAPIDC4	Channel -4-.

(V.)PLC.OVRCAN

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

If the PLC turns this mark on, the channel applies 100% of the feedrate in all work mode.

V.PLC.OVRCAN	Channel -1-.
V.PLC.OVRCANC1	Channel -1-.
V.PLC.OVRCANC2	Channel -2-.
V.PLC.OVRCANC3	Channel -3-.
V.PLC.OVRCANC4	Channel -4-.

(V.)PLC.LATCHM

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

With this mark, it is possible to select how the JOG keys will work in JOG mode.

If this mark is off, the axes will move while the corresponding JOG key kept pressed. If this mark is on, the axes will move from the instant the JOG key is pressed until the software limits

are reached or the [STOP] key is pressed or another JOG key is pressed (in this case the new axis will start moving).

V.PLC.LATCHM

(V.)PLC.RESETIN

*Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).*

If the PLC turns this mark on, the channel assumes the initial conditions.

V.PLC.RESETIN	Channel -1-
V.PLC.RESETINC1	Channel -1-
V.PLC.RESETINC2	Channel -2-
V.PLC.RESETINC3	Channel -3-
V.PLC.RESETINC4	Channel -4-

(V.)PLC.AUXEND

*Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).*

The PLC uses this mark when executing functions S and M with synchronization.

V.PLC.AUXEND	Channel -1-
V.PLC.AUXENDC1	Channel -1-
V.PLC.AUXENDC2	Channel -2-
V.PLC.AUXENDC3	Channel -3-
V.PLC.AUXENDC4	Channel -4-

(V.)PLC.BLKSKIP1

*Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).*

If the PLC turns this mark on, the channel considers the block-skip condition.

V.PLC.BLKSKIP1	Channel -1-
V.PLC.BLKSKIP1C1	Channel -1-
V.PLC.BLKSKIP1C2	Channel -2-
V.PLC.BLKSKIP1C3	Channel -3-
V.PLC.BLKSKIP1C4	Channel -4-

(V.)PLC.M01STOP

*Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).*

If the PLC turns this mark on, the channel considers the block-skips.

V.PLC.M01STOP	Channel -1-
V.PLC.M01STOPC1	Channel -1-
V.PLC.M01STOPC2	Channel -2-
V.PLC.M01STOPC3	Channel -3-
V.PLC.M01STOPC4	Channel -4-

22.

CNC VARIABLES.
PLC modifiable logic signals; general.



CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.TIMERON

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

If the PLC turns this mark on, the CNC enables the freely available clock.

V.PLC.TIMERON

(V.)PLC.PLCREADY

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

If the PLC turns this mark off, it interrupts the execution of the PLC program and issues an error message.

V.PLC.PLCREADY

(V.)PLC.NOWAIT

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The PLC turns this mark on to cancel the channel synchronizations programmed with #WAIT.

V.PLC.NOWAITC1	Channel -1.
V.PLC.NOWAITC2	Channel -2.
V.PLC.NOWAITC3	Channel -3.
V.PLC.NOWAITC4	Channel -4.

- (V.)PLC.DISCROSS1**
- (V.)PLC.DISCROSS2**
- (V.)PLC.DISCROSS3**
- (V.)PLC.DISCROSS4**
- (V.)PLC.DISCROSS5**
- (V.)PLC.DISCROSS6**
- (V.)PLC.DISCROSS7**
- (V.)PLC.DISCROSS8**
- (V.)PLC.DISCROSS9**

Variable that can only be read from the program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The PLC this mark on to disable the cross compensation table.

V.PLC.DISCROSS1	Cross compensation table -1.
V.PLC.DISCROSS2	Cross compensation table -2.

(V.)PLC.PLCABORT

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

If the PLC turns this mark on, the channel aborts the CNCEX command launched from the PLC. This mark does not set the initial conditions in the channel and keeps the history.

V.PLC.PLCABORT	Channel -1.
V.PLC.PLCABORTC1	Channel -1.
V.PLC.PLCABORTC2	Channel -2.
V.PLC.PLCABORTC3	Channel -3.
V.PLC.PLCABORTC4	Channel -4.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
PLC modifiable logic signals; general.

(V.)PLC.NEXTMPGAXIS

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Every time the PLC turns this mark on, the CNC selects an axis to be moved with the handwheel.

V.PLC.NEXTMPGAXIS

(V.)PLC.PANELOFF1**(V.)PLC.PANELOFF2****(V.)PLC.PANELOFF3****(V.)PLC.PANELOFF4****(V.)PLC.PANELOFF5****(V.)PLC.PANELOFF6****(V.)PLC.PANELOFF7****(V.)PLC.PANELOFF8**

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC activates one of these marks the CNC disables the corresponding jog panel.

V.PLC.PANELOFF1	Disable operator panel ·1·.
V.PLC.PANELOFF2	Disable operator panel ·2·.

(V.)PLC.SYNC

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC indicates in this register the spindle to be used for synchronization.

The channel uses this spindle with function G33 to thread a particular spindle and with G95 to program the feedrate according to a particular spindle.

V.PLC.SYNC1	Channel ·1·.
V.PLC.SYNC2	Channel ·2·.
V.PLC.SYNC3	Channel ·3·.
V.PLC.SYNC4	Channel ·4·.

(V.)PLC.RETRACE

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC turns this mark on while a program is being executed, the channel turns the retrace function on.

V.PLC.RETRACEC1	Channel ·1·.
V.PLC.RETRACEC2	Channel ·2·.
V.PLC.RETRACEC3	Channel ·3·.
V.PLC.RETRACEC4	Channel ·4·.



CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.PRGABORT

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC turns this mark on, the channel aborts the execution of the program but without affecting the spindle, initializes the program history and resumes the execution in the point indicated by the #ABORT instruction active in the part-program.

V.PLC.PRGABORT	Channel -1-.
V.PLC.PRGABORTC1	Channel -1-.
V.PLC.PRGABORTC2	Channel -2-.
V.PLC.PRGABORTC3	Channel -3-.
V.PLC.PRGABORTC4	Channel -4-.

(V.)PLC.CNCOFF

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC turns this mark on, the CNC starts the shut-down sequence.

V.PLC.CNCOFF

(V.)PLC.INHIBITMPG1

...

(V.)PLC.INHIBITMPG12

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC turns one of these marks on, it disables the corresponding handwheel. The PLC has a mark for each handwheel, the INHIBITMPG1 mark disables the first handwheel, the INHIBITMPG2 mark the second one and so on.

V.PLC.INHIBITMPG1	Handwheel -1-.
V.PLC.INHIBITMPG2	Handwheel -2-.

(V.)PLC.EXRAPID

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC activates this mark, the CNC channel enables rapid traverse during the execution of a program for the programmed movements. The behavior of this mark depends on how parameter RAPIDEN has been set.

V.PLC.EXRAPIDC1	Channel -1-.
V.PLC.EXRAPIDC2	Channel -2-.
V.PLC.EXRAPIDC3	Channel -3-.
V.PLC.EXRAPIDC4	Channel -4-.

(V.)PLC.KEYBD1CH

..

(V.)PLC.KEYBD8CH

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

These registers may be used to change the default behavior of the keyboards with respect to the channels, set by machine parameters.

V.PLC.KEYBD1CH	Operator panel -1-.
V.PLC.KEYBD2CH	Operator panel -2-.

22.

CNC VARIABLES.

PLC modifiable logic signals; general.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Configuration defined in the machine parameters.
1	Jog panel assigned to channel 1.
2	Jog panel assigned to channel 2.
3	Jog panel assigned to channel 3.
4	Jog panel assigned to channel 4.
FF	Jog panel assigned to the active channel.

(V.)PLC.VOLCOMP1

..

(V.)PLC.VOLCOMP4

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC activates one of these marks, the CNC activates the corresponding volumetric compensation.

V.PLC.VOLCOMP1	Volumetric compensation ·1·.
----------------	------------------------------

(V.)PLC.QWERTYOFF1

..

(V.)PLC.QWERTYOFF8

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC turns one of these marks on, it disables the corresponding alphanumeric keyboard.

V.PLC.QWERTYOFF1	Disable keyboard ·1·.
V.PLC.QWERTYOFF2	Disable keyboard ·2·.

(V.)PLC.FLIMITAC

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC activates this mark, the CNC activates the feedrate safety limits (parameter FLIMIT) on all the axes of the system.

V.PLC.FLIMITAC

(V.)PLC.FLIMITACCH

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC activates this mark, the CNC activates the feedrate safety limits (parameter FLIMIT) on all the axes of the channel.

V.PLC.FLIMITAC	Channel ·1·.
V.PLC.FLIMITACCH1	Channel ·1·.
V.PLC.FLIMITACCH2	Channel ·2·.
V.PLC.FLIMITACCH3	Channel ·3·.
V.PLC.FLIMITACCH4	Channel ·4·.

22.

CNC VARIABLES.
PLC modifiable logic signals; general.



CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.SLIMITAC

*Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).*

If the PLC activates this mark, the CNC activates the speed safety limits (parameter SLIMIT) on all the spindles of the system.

V.PLC.SLIMITAC

(V.)PLC.INT1

..

(V.)PLC.INT4

*Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).*

If the PLC activates one of these marks, the channel executes the corresponding interruption subroutine.

V.PLC.INT1	Channel -1.
V.PLC.INT1C1	Channel -1.
V.PLC.INT1C2	Channel -2.
V.PLC.INT1C3	Channel -3.
V.PLC.INT1C4	Channel -4.

22.21 PLC modifiable logic signals; axes and spindles.

22.

CNC VARIABLES.
PLC modifiable logic signals; axes and spindles.

(V.)PLC.LIMITPOSxn

(V.)PLC.LIMITPOSsn

Variable that can be read and written from the interface and read from the program and from the PLC. Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC must turn this mark on to indicate that the axis or the spindle has exceeded the positive travel limit.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.LIMITPOSX	X axis.
V.PLC.LIMITPOSS	Spindle S.
V.PLC.LIMITPOS3	Axis or spindle with logic number ·3·.

(V.)PLC.LIMITNEGxn

(V.)PLC.LIMITNEGsn

Variable that can be read and written from the interface and read from the program and from the PLC. Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC must turn this mark on to indicate that the axis or the spindle has exceeded the negative travel limit.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.LIMITNEGX	X axis.
V.PLC.LIMITNEGS	Spindle S.
V.PLC.LIMITNEG3	Axis or spindle with logic number ·3·.

(V.)PLC.DECELxn

(V.)PLC.DECELsn

Variable that can be read and written from the interface and read from the program and from the PLC. Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC must turn this mark on to indicate that the home switch is pressed.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.DECELX	X axis.
V.PLC.DECELS	Spindle S.
V.PLC.DECEL3	Axis or spindle with logic number ·3·.



CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.INHIBITxn**(V.)PLC.INHIBITsn**

Variable that can be read and written from the interface and read from the program and from the PLC.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC turns this mark on, the CNC inhibits any movement of the axis or spindle.

For independent axes and electronic cam, if the PLC turns this mark on, it interrupts the synchronization movement switching to zero speed. The system waits for the signal to deactivate before resuming the execution and the movement from the interruption point.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.INHIBITX	X axis.
V.PLC.INHIBITS	Spindle S.
V.PLC.INHIBIT3	Axis or spindle with logic number ·3·.

(V.)PLC.AXISPOSxn**(V.)PLC.AXISPOSsn**

Variable that can be read and written from the interface and read from the program and from the PLC.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC turns this mark on while the CNC is in jog mode, the CNC moves the axis or the spindle in the positive direction.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.AXISPOSX	X axis.
V.PLC.AXISPOSS	Spindle S.
V.PLC.AXISPOS3	Axis or spindle with logic number ·3·.

(V.)PLC.AXISNEGxn**(V.)PLC.AXISNEGsn**

Variable that can be read and written from the interface and read from the program and from the PLC.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC turns this mark on while the CNC is in jog mode, the CNC moves the axis or the spindle in the positive direction.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.AXISNEGX	X axis.
V.PLC.AXISNEGS	Spindle S.
V.PLC.AXISNEG3	Axis or spindle with logic number ·3·.

22.**CNC VARIABLES.**

PLC modifiable logic signals; axes and spindles.

FAGOR **CNC 8060**
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
PLC modifiable logic signals; axes and spindles.

(V.)PLC.SERVOxnON
(V.)PLC.SERVOsnON

Variable that can be read and written from the interface and read from the program and from the PLC.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The PLC must turn this mark on to allow moving the axis or the spindle.

Syntax.

- xn· Name or logic number of the axis.
- sn· Name or logic number of the spindle.

V.PLC.SERVOXON	X axis.
V.PLC.SERVOSON	Spindle S.
V.PLC.SERVO3ON	Axis or spindle with logic number ·3·.

(V.)PLC.DROxn
(V.)PLC.DROsn

Variable that can be read and written from the interface and read from the program and from the PLC.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The PLC must turn this mark on for the axis or the spindle to work in DRO mode.

Syntax.

- xn· Name or logic number of the axis.
- sn· Name or logic number of the spindle.

V.PLC.DROX	X axis.
V.PLC.DROS	Spindle S.
V.PLC.DRO3	Axis or spindle with logic number ·3·.

(V.)PLC.SPENAxn
(V.)PLC.SPENAsn

Variable that can be read and written from the interface and read from the program and from the PLC.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The PLC must turn this mark on to enable the speed enable signal of the drive.

Syntax.

- xn· Name or logic number of the axis.
- sn· Name or logic number of the spindle.

V.PLC.SPENAX	X axis.
V.PLC.SPENAS	Spindle S.
V.PLC.SPENA3	Axis or spindle with logic number ·3·.

(V.)PLC.DRENAxn
(V.)PLC.DRENAsn

Variable that can be read and written from the interface and read from the program and from the PLC.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The PLC must turn this mark on to enable the drive enable signal of the drive.

Syntax.

- xn· Name or logic number of the axis.



CNC 8060
CNC 8065

(REF: 1402)

·sn· Name or logic number of the spindle.

V.PLC.DRENAX	X axis.
V.PLC.DRENAS	Spindle S.
V.PLC.DRENA3	Axis or spindle with logic number ·3·.

(V.)PLC.LIMxnOFF

(V.)PLC.LIMsnOFF

Variable that can be read and written from the interface and read from the program and from the PLC.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC turns this mark on, the CNC ignores the software travel limits.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.LIMXOFF	X axis.
V.PLC.LIMSOFF	Spindle S.
V.PLC.LIM3OFF	Axis or spindle with logic number ·3·.

(V.)PLC.PARKEDxn

(V.)PLC.PARKEDsn

Variable that can be read and written from the interface and read from the program and from the PLC.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC turns this mark on when the axis or the spindle is parked.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.PARKEDX	X axis.
V.PLC.PARKEDS	Spindle S.
V.PLC.PARKED3	Axis or spindle with logic number ·3·.

(V.)PLC.LUBRENAXn

(V.)PLC.LUBRENAsn

Variable that can be read and written from the interface and read from the program and from the PLC.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC turns this mark on to enable the lubrication of the axis or the spindle.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.LUBRENAX	X axis.
V.PLC.LUBRENAS	Spindle S.
V.PLC.LUBRENA3	Axis or spindle with logic number ·3·.

22.

CNC VARIABLES.
PLC modifiable logic signals; axes and spindles.

(V.)PLC.LUBROKxn**(V.)PLC.LUBROKsn**

Variable that can be read and written from the interface and read from the program and from the PLC.
Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC turns this mark on to indicate that the axis lubrication is done.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.LUBROKX	X axis.
V.PLC.LUBROKS	Spindle S.
V.PLC.LUBROK3	Axis or spindle with logic number ·3·.

(V.)PLC.DIFFCOMPxn**(V.)PLC.DIFFCOMPSn**

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC uses this mark on Gantry axes to correct the position difference between the two axes.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.DIFFCOMPX	X axis.
V.PLC.DIFFCOMPS	Spindle S.
V.PLC.DIFFCOMP3	Axis or spindle with logic number ·3·.

(V.)PLC.FBACKSELxn**(V.)PLC.FBACKSELsn**

Variable that can be read and written from the interface and read from the program and from the PLC.
Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

In systems with external-internal feedback, the PLC turns this mark on to use the external feedback and turns it off to use the internal feedback.

Syntax.

·xn· Name or logic number of the axis.

·sn· Name or logic number of the spindle.

V.PLC.FBACKSELX	X axis.
V.PLC.FBACKSELS	Spindle S.
V.PLC.FBACKSEL3	Axis or spindle with logic number ·3·.

(V.)PLC.DEADxn**(V.)PLC.DEADsn**

Variable that can be read and written from the program and interface and read from the PLC.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

In systems with dead axes, the PLC uses this mark to indicate to the CNC how to handle the path joints when a dead axis is involved.

Syntax.

- xn· Name or logic number of the axis.
- sn· Name or logic number of the spindle.

V.PLC.DEADX	X axis.
V.PLC.DEADS	Spindle S.
V.PLC.DEAD3	Axis or spindle with logic number -3-.

(V.)PLC.SWITCHxn**(V.)PLC.SWITCHsn**

Variable that can be read and written from the program and interface and read from the PLC.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

When the system has multi-axis groups, this mark may be used to toggle between the various axes or spindles of the group.

Syntax.

- xn· Name or logic number of the axis.
- sn· Name or logic number of the spindle.

V.PLC.SWITCHX	X axis.
V.PLC.SWITCHS	Spindle S.
V.PLC.SWITCH3	Axis or spindle with logic number -3-.

(V.)PLC.TANDEMOFFxn**(V.)PLC.TANDEMOFFsn**

Variable that can be read and written from the program and interface and read from the PLC.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

This mark may be used to temporarily decouple (unslave) the loop of the axes or spindles involved in the tandem so they can be move separately.

Syntax.

- xn· Name or logic number of the axis.
- sn· Name or logic number of the spindle.

V.PLC.TANDEMOFFX	X axis.
V.PLC.TANDEMOFFS	Spindle S.
V.PLC.TANDEMOFF3	Axis or spindle with logic number -3-.

22.**CNC VARIABLES.**

PLC modifiable logic signals; axes and spindles.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.22 PLC modifiable logic signals; spindles.

22.

CNC VARIABLES.
PLC modifiable logic signals; spindles.

(V.)PLC.GEAR1
(V.)PLC.GEAR2
(V.)PLC.GEAR3
(V.)PLC.GEAR4

*Variable that can be read and written from the interface and read from the program.
 The variable returns the execution value; reading it interrupts block preparation.
 Report variable (to be used from the scripts).*

The PLC must turn on the mark corresponding to the selected speed range.

There is a mark for each spindle. The mnemonics for each channel are the following. Here is an example of the mnemonics for GEAR1; it is the same for the rest of the registers.

V.PLC.GEAR1	Spindle ·1·.
V.PLC.GEAR1SP1	Spindle ·1·.
V.PLC.GEAR1SP2	Spindle ·2·.
V.PLC.GEAR1SP3	Spindle ·3·.
V.PLC.GEAR1SP4	Spindle ·4·.

(V.)PLC.PLCCNTL

*Variable that can be read and written from the interface and read from the program.
 The variable returns the execution value; reading it interrupts block preparation.
 Report variable (to be used from the scripts).*

The PLC must turn this mark on when the spindle is controlled by the PLC.

V.PLC.PLCCNTL	Spindle ·1·.
V.PLC.PLCCNTL1	Spindle ·1·.
V.PLC.PLCCNTL2	Spindle ·2·.
V.PLC.PLCCNTL3	Spindle ·3·.
V.PLC.PLCCNTL4	Spindle ·4·.

(V.)PLC.SANALOG

*Variable that can be read and written from the interface and read from the program.
 The variable returns the execution value; reading it interrupts block preparation.
 Report variable (to be used from the scripts).*

When the spindle is controlled by the PLC, the PLC must indicate in this register the velocity command to be applied to the spindle.

V.PLC.SANALOG	Spindle ·1·.
V.PLC.SANALOG1	Spindle ·1·.
V.PLC.SANALOG2	Spindle ·2·.
V.PLC.SANALOG3	Spindle ·3·.
V.PLC.SANALOG4	Spindle ·4·.

(V.)PLC.SPDLEREV

*Variable that can be read and written from the interface and read from the program.
 The variable returns the execution value; reading it interrupts block preparation.
 Report variable (to be used from the scripts).*

If the PLC turns this mark on, the CNC inverts the spindle turning direction.

V.PLC.SPDLEREV	Spindle ·1·.
V.PLC.SPDLEREV1	Spindle ·1·.
V.PLC.SPDLEREV2	Spindle ·2·.
V.PLC.SPDLEREV3	Spindle ·3·.
V.PLC.SPDLEREV4	Spindle ·4·.



CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.PLCM3

(V.)PLC.PLCM4

(V.)PLC.PLCM5

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC turns this mark on to let the CNC know that it must execute the corresponding M function at the indicated spindle.

V.PLC.PLCM3	Spindle ·1·.
V.PLC.PLCM3SP1	Spindle ·1·.
V.PLC.PLCM3SP2	Spindle ·2·.
V.PLC.PLCM3SP3	Spindle ·3·.
V.PLC.PLCM3SP4	Spindle ·4·.

(V.)PLC.SLIMITACSPDL

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

There is a mark for each spindle. The mnemonics for each spindle are the following:

SLIMITACSPDL1 (can also be programmed as SLIMITACSPDL)

SLIMITACSPDL2 SLIMITACSPDL3 SLIMITACSPDL4

If the PLC activates this mark, the CNC activates the speed safety limits (parameter SLIMIT) on the indicated spindle.

V.PLC.SLIMITACSPDL	Spindle ·1·.
V.PLC.SLIMITACSPDL1	Spindle ·1·.
V.PLC.SLIMITACSPDL2	Spindle ·2·.
V.PLC.SLIMITACSPDL3	Spindle ·3·.
V.PLC.SLIMITACSPDL4	Spindle ·4·.

22.23 PLC modifiable logic signals; independent interpolator.

(V.)PLC._IXFERINHxn

If the PLC deactivates this mark, the movements of the independent axis remain waiting for the PLC to activate this mark again.

Syntax.

·xn· Name or logic number of the axis.

V.PLC._IXFERINHx	X axis.
V.PLC._IXFERINH3	Axis with logic number -3-.

(V.)PLC.IRESETxn

Variable that can be read and written from the program and interface and read from the PLC.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC turns this mark on, the independent interpolator interrupts the instruction being executed and eliminates the instructions pending execution.

Syntax.

·xn· Name or logic number of the axis.

V.PLC.IRESETX	X axis.
V.PLC.IRESET3	Axis with logic number -3-.

(V.)PLC.IABORTxn

Variable that can be read and written from the program and interface and read from the PLC.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

If the PLC turns this mark on, the independent interpolator interrupts the positioning block being executed (if any) eliminating also the rest of the positioning blocks pending execution.

Syntax.

·xn· Name or logic number of the axis.

V.PLC.IABORTX	X axis.
V.PLC.IABORT3	Axis with logic number -3-.

22.

CNC VARIABLES.
PLC modifiable logic signals; independent interpolator.



CNC 8060
CNC 8065

(REF: 1402)

22.24 PLC modifiable logic signals; tool manager.

(V.)PLC.SETTMEM

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The PLC must turn this mark on to activate the emergency of the tool manager.

V.PLC.SETTMEM	Magazine ·1·.
V.PLC.SETTMEMZ1	Magazine ·1·.
V.PLC.SETTMEMZ2	Magazine ·2·.
V.PLC.SETTMEMZ3	Magazine ·3·.
V.PLC.SETTMEMZ4	Magazine ·4·.

(V.)PLC.RESTMEM

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The PLC must turn this mark on to cancel the emergency of the tool manager.

V.PLC.RESTMEM	Magazine ·1·.
V.PLC.RESTMEMZ1	Magazine ·1·.
V.PLC.RESTMEMZ2	Magazine ·2·.
V.PLC.RESTMEMZ3	Magazine ·3·.
V.PLC.RESTMEMZ4	Magazine ·4·.

(V.)PLC.CUTTINGON

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The PLC must turn this mark on to indicate that the tool is machining (cutting).

V.PLC.CUTTINGON	Channel ·1·.
V.PLC.CUTTINGON1	Channel ·1·.
V.PLC.CUTTINGON2	Channel ·2·.
V.PLC.CUTTINGON3	Channel ·3·.
V.PLC.CUTTINGON4	Channel ·4·.

(V.)PLC.TREJECT

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

The PLC must turn this mark on to reject the tool.

V.PLC.TREJECT	Channel ·1·.
V.PLC.TREJECTC1	Channel ·1·.
V.PLC.TREJECTC2	Channel ·2·.
V.PLC.TREJECTC3	Channel ·3·.
V.PLC.TREJECTC4	Channel ·4·.

22.

CNC VARIABLES.

PLC modifiable logic signals; tool manager.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.MZTOCH1

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC must turn this mark on after taking the tool from the magazine to the claw 1 of the tool changer arm.

V.PLC.MZTOCH1	Magazine ·1·.
V.PLC.MZTOCH1MZ1	Magazine ·1·.
V.PLC.MZTOCH1MZ2	Magazine ·2·.
V.PLC.MZTOCH1MZ3	Magazine ·3·.
V.PLC.MZTOCH1MZ4	Magazine ·4·.

(V.)PLC.CH1TOSPDL

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC must turn this mark on after taking the tool from the claw 1 of the tool changer arm to the spindle.

V.PLC.CH1TOSPDL	Magazine ·1·.
V.PLC.CH1TOSPDLMZ1	Magazine ·1·.
V.PLC.CH1TOSPDLMZ2	Magazine ·2·.
V.PLC.CH1TOSPDLMZ3	Magazine ·3·.
V.PLC.CH1TOSPDLMZ4	Magazine ·4·.

(V.)PLC.SPDLTOCH1

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC must turn this mark on after taking the tool from the spindle to the claw 1 of the tool changer arm.

V.PLC.SPDLTOCH1	Magazine ·1·.
V.PLC.SPDLTOCH1MZ1	Magazine ·1·.
V.PLC.SPDLTOCH1MZ2	Magazine ·2·.
V.PLC.SPDLTOCH1MZ3	Magazine ·3·.
V.PLC.SPDLTOCH1MZ4	Magazine ·4·.

(V.)PLC.SPDLTOCH2

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC must turn this mark on after taking the tool from the spindle to the claw 2 of the tool changer arm.

V.PLC.SPDLTOCH1	Magazine ·1·.
V.PLC.SPDLTOCH2MZ1	Magazine ·1·.
V.PLC.SPDLTOCH2MZ2	Magazine ·2·.
V.PLC.SPDLTOCH2MZ3	Magazine ·3·.
V.PLC.SPDLTOCH2MZ4	Magazine ·4·.

22.

CNC VARIABLES.
PLC modifiable logic signals; tool manager.



CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.CH1TOMZ

Variable that can be read and written from the interface and read from the program.
 The variable returns the execution value; reading it interrupts block preparation.
 Report variable (to be used from the scripts).

The PLC must turn this mark on after taking the tool from the claw 1 of the tool changer arm to the magazine.

V.PLC.CH1TOMZ	Magazine ·1·.
V.PLC.CH1TOMZ1	Magazine ·1·.
V.PLC.CH1TOMZ2	Magazine ·2·.
V.PLC.CH1TOMZ3	Magazine ·3·.
V.PLC.CH1TOMZ4	Magazine ·4·.

(V.)PLC.CH2TOMZ

Variable that can be read and written from the interface and read from the program.
 The variable returns the execution value; reading it interrupts block preparation.
 Report variable (to be used from the scripts).

The PLC must turn this mark on after taking the tool from the claw 2 of the tool changer arm to the magazine.

V.PLC.CH2TOMZ	Magazine ·1·.
V.PLC.CH2TOMZ1	Magazine ·1·.
V.PLC.CH2TOMZ2	Magazine ·2·.
V.PLC.CH2TOMZ3	Magazine ·3·.
V.PLC.CH2TOMZ4	Magazine ·4·.

(V.)PLC.SPDLTOGR

Variable that can be read and written from the interface and read from the program.
 The variable returns the execution value; reading it interrupts block preparation.
 Report variable (to be used from the scripts).

The PLC must turn this mark on after taking the tool from the spindle to ground.

V.PLC.SPDLTOGR	Channel ·1·.
V.PLC.SPDLTOGRC1	Channel ·1·.
V.PLC.SPDLTOGRC2	Channel ·2·.
V.PLC.SPDLTOGRC3	Channel ·3·.
V.PLC.SPDLTOGRC4	Channel ·4·.

(V.)PLC.GRTOSPDL

Variable that can be read and written from the interface and read from the program.
 The variable returns the execution value; reading it interrupts block preparation.
 Report variable (to be used from the scripts).

The PLC must turn this mark on after taking the tool from the ground to the spindle.

V.PLC.GRTOSPDL	Channel ·1·.
V.PLC.GRTOSPDLC1	Channel ·1·.
V.PLC.GRTOSPDLC2	Channel ·2·.
V.PLC.GRTOSPDLC3	Channel ·3·.
V.PLC.GRTOSPDLC4	Channel ·4·.

(V.)PLC.MZTOSPD

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC must turn this mark on after taking the tool from the magazine to the spindle.

V.PLC.MZTOSPD	Magazine ·1·.
V.PLC.MZTOSPDLMZ1	Magazine ·1·.
V.PLC.MZTOSPDLMZ2	Magazine ·2·.
V.PLC.MZTOSPDLMZ3	Magazine ·3·.
V.PLC.MZTOSPDLMZ4	Magazine ·4·.

(V.)PLC.SPDLTOMZ

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC must turn this mark on after taking the tool from the spindle to the magazine.

V.PLC.SPDLTOMZ	Magazine ·1·.
V.PLC.SPDLTOMZ1	Magazine ·1·.
V.PLC.SPDLTOMZ2	Magazine ·2·.
V.PLC.SPDLTOMZ3	Magazine ·3·.
V.PLC.SPDLTOMZ4	Magazine ·4·.

(V.)PLC.MZROT

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC must turn this mark on to rotate the turret.

V.PLC.MZROT	Magazine ·1·.
V.PLC.MZROTMZ1	Magazine ·1·.
V.PLC.MZROTMZ2	Magazine ·2·.
V.PLC.MZROTMZ3	Magazine ·3·.
V.PLC.MZROTMZ4	Magazine ·4·.

(V.)PLC.TCHANGEOK

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

The PLC must turn this mark on when the tool change is done.

V.PLC.TCHANGEOK	Magazine ·1·.
V.PLC.TCHANGEOKMZ1	Magazine ·1·.
V.PLC.TCHANGEOKMZ2	Magazine ·2·.
V.PLC.TCHANGEOKMZ3	Magazine ·3·.
V.PLC.TCHANGEOKMZ4	Magazine ·4·.

22.**CNC VARIABLES.**

PLC modifiable logic signals; tool manager.



CNC 8060
CNC 8065

(REF: 1402)

(V.)PLC.MZPOS

*Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).*

The PLC must indicate the current magazine position in this register.

V.PLC.MZPOS	Magazine ·1·.
V.PLC.MZPOSMZ1	Magazine ·1·.
V.PLC.MZPOSMZ2	Magazine ·2·.
V.PLC.MZPOSMZ3	Magazine ·3·.
V.PLC.MZPOSMZ4	Magazine ·4·.



CNC 8060
CNC 8065

(REF: 1402)

22.25 PLC modifiable logic signals; keys.

22.

CNC VARIABLES.
PLC modifiable logic signals; keys.

(V.)PLC.KEYLED1

(V.)PLC.KEYLED2

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

These registers control the LED's (lamps) of the keys of all the operator panels at the same time.

V.PLC.KEYLED1

V.PLC.KEYLED2

(V.)PLC.KEYLED1_1

(V.)PLC.KEYLED2_1

..

(V.)PLC.KEYLED1_8

(V.)PLC.KEYLED2_8

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

These registers control the LED's (lamps) of the keys of each operator panel. Registers KEYLED1_1 and KEYLED2_1 correspond to the first jog panel, KEYLED1_2 and KEYLED2_2 to the second one and so on.

V.PLC.KEYLED1_1

V.PLC.KEYLED2_1

(V.)PLC.KEYDIS1

(V.)PLC.KEYDIS2

(V.)PLC.KEYDIS3

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

These registers inhibit (bit=1) the keys and the switches on all operator panels at the same time.

V.PLC.KEYDIS1

V.PLC.KEYDIS2

V.PLC.KEYDIS3

(V.)PLC.KEYDIS1_1

(V.)PLC.KEYDIS2_1

(V.)PLC.KEYDIS3_1

..

(V.)PLC.KEYDIS1_8

(V.)PLC.KEYDIS2_8

(V.)PLC.KEYDIS3_8

Variable that can be read and written from the interface and read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

These registers inhibit (bit=1) the keys and the switches on each operator panel. Registers KEYDIS1_1 to KEYDIS3_1 correspond to the first jog panel, KEYDIS1_2 to KEYDIS3_2 to the second one and so on.

V.PLC.KEYDIS1_1

V.PLC.KEYDIS2_1

V.PLC.KEYDIS3_1



CNC 8060
CNC 8065

(REF: 1402)

22.26 Variables related to the machine configuration.

AXES AND SPINDLES OF THE SYSTEM.

(V.)G.GAXISNAME_n

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Name of logic axis "n".

Syntax.

Replace the "n" letter with the axis logic number.

V.G.GAXISNAME2	Axis with logic number :2.
----------------	----------------------------

Values of the variable.

The values returned by this variable are encoded as follows.

X=10	X1=11	X2=12	X3=13	X4=14	... X9=19
Y=20	Y1=21	Y2=22	Y3=23	Y4=24	... Y9=29
Z=30	Z1=31	Z2=32	Z3=33	Z4=34	... Z9=39
U=40	U1=41	U2=42	U3=43	U4=44	... U9=49
V=50	V1=51	V2=52	V3=53	V4=54	... V9=59
W=60	W1=61	W2=62	W3=63	W4=64	... W9=69
A=70	A1=71	A2=72	A3=73	A4=74	... A9=79
B=80	B1=81	B2=82	B3=83	B4=84	... B9=89
C=90	C1=91	C2=92	C3=93	C4=94	... C9=99

Remarks.

The logic number of the axes is determined by the order in which the axes have been defined in the machine parameter table. The first axis of the table will be logic axis -1- and so on.

When some axes are parked, it is a good idea to know which axes are available. This variable indicates which axes are available, if an axis is not available, this variable returns a "?".

(V.)G.GSPDLNAME_n

Variable that can only be read from the program, PLC and interface.

The variable returns the value of execution or preparation depending on the spindle.

Report variable (to be used from the scripts).

Name of logic spindle "n".

Syntax.

Replace the "n" letter with the spindle logic number.

V.G.GSPDLNAME2	Spindle with logic number :2.
----------------	-------------------------------

Values of the variable.

The values returned by this variable are encoded as follows.

S=100	S1=101	S2=102	S3=103	S4=104	... S9=109
-------	--------	--------	--------	--------	------------

Remarks.

The variable returns the value of execution or preparation as follows. If the spindle belongs to the channel requesting the variable, it returns the preparation value; if the spindle belongs to a different channel, the variable returns the execution value and interrupts block preparation.

22.

CNC VARIABLES.
Variables related to the machine configuration.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

The logic number of the spindles is determined by the order in which they have been defined in the machine parameter table. The logic numbering of the spindles continues from the last logic axis; hence, in a 5-axis system, the first spindle of the table will be logic spindle -6- and so on.

(V.)[ch].A.ACTCH.xn
(V.)[ch].A.ACTCH.sn
(V.)[ch].SP.ACTCH.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the value of execution or preparation depending on the axis or spindle.

Current channel of the axis or of the spindle.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.ACTCH.Z	Z axis.
V.A.ACTCH.S	Spindle S.
V.SP.ACTCH.S	Spindle S.
V.SP.ACTCH	Master spindle.
V.A.ACTCH.4	Axis or spindle with logic number -4-.
V.[2].A.ACTCH.1	Axis with index -1- in the channel -2-.
V.SP.ACTCH.2	Spindle with index -2- in the system.
V.[2].SP.ACTCH.1	Spindle with index -1- in the channel -2-.

Remarks.

The variable returns the value of execution or preparation as follows. If the spindle belongs to the channel requesting the variable, it returns the preparation value; if the axis or spindle belongs to a different channel, the variable returns the execution value and interrupts block preparation.

(V.)[ch].A.ACTIVSET.xn
(V.)[ch].A.ACTIVSET.sn
(V.)[ch].SP.ACTIVSET.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the value of execution or preparation depending on the axis or spindle.

Parameter set active at the axis or spindle.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.ACTIVSET.Z	Z axis.
V.A.ACTIVSET.S	Spindle S.
V.SP.ACTIVSET.S	Spindle S.
V.SP.ACTIVSET	Master spindle.
V.A.ACTIVSET.4	Axis or spindle with logic number -4-.
V.[2].A.ACTIVSET.1	Axis with index -1- in the channel -2-.
V.SP.ACTIVSET.2	Spindle with index -2- in the system.
V.[2].SP.ACTIVSET.1	Spindle with index -1- in the channel -2-.

22.

CNC VARIABLES.
Variables related to the machine configuration.



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

The variable returns the value of execution or preparation as follows. If the spindle belongs to the channel requesting the variable, it returns the preparation value; if the axis or spindle belongs to a different channel, the variable returns the execution value and interrupts block preparation.

CHANNELS, AXES AND SPINDLES.**(V.)G.NUMCH**

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of channels.

V.G.NUMCH

(V.)[ch].G.AXIS

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.
Report variable (to be used from the scripts).

Number of axes of the channel.

Syntax.

·ch· Channel number.

V.[2].G.AXIS Channel ·2·.

(V.)[ch].G.NAXIS

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.
Report variable (to be used from the scripts).

Number of axes of the channel including the empty positions of the yielded axes.

Syntax.

·ch· Channel number.

V.[2].G.NAXIS Channel ·2·.

(V.)[ch].G.NSPDL

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.
Report variable (to be used from the scripts).

Number of spindles of the channel.

Syntax.

·ch· Channel number.

V.[2].G.NSPDL Channel ·2·.

(V.)[ch].G.AXISCH

Variable to be read via interface.
Report variable (to be used from the scripts).

Axes of the system that belong to the channel.

22.**CNC VARIABLES.**

Variables related to the machine configuration.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the machine configuration.

Syntax.

·ch· Channel number.

[2].G.NSPDL Channel ·2·.

Values of the variable.

The variable returns a 32-bit value where each bit represents one axis; the least significant bit corresponds to the axis with the lowest logic number. Each bit indicates whether the axis belongs to the channel (bit = 1) or not (bit = 0).

AXISNAME.	Channel ·1·.	Channel ·2·.	Reading the variables.
X (Logic axis ·1·.)	X	X2	[1].G.AXISCH = \$7
Y (Logic axis ·2·.)	Y	Y2	[2].G.AXISCH = \$38
Z (Logic axis ·3·.)	Z	Z2	
X2 (Logic axis ·4·.)			
Y2 (Logic axis ·5·.)			
Z2 (Logic axis ·6·.)			

(V.) [ch].G.AXISNAME_n

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Report variable (to be used from the scripts).

Name of the axis with n index in the channel.

Syntax.

·ch· Channel number.

V.[2].G.AXISNAME1 Channel ·2·.

Values of the variable.

The values returned by this variable are encoded as follows.

X=10	X1=11	X2=12	X3=13	X4=14	... X9=19
Y=20	Y1=21	Y2=22	Y3=23	Y4=24	... Y9=29
Z=30	Z1=31	Z2=32	Z3=33	Z4=34	... Z9=39
U=40	U1=41	U2=42	U3=43	U4=44	... U9=49
V=50	V1=51	V2=52	V3=53	V4=54	... V9=59
W=60	W1=61	W2=62	W3=63	W4=64	... W9=69
A=70	A1=71	A2=72	A3=73	A4=74	... A9=79
B=80	B1=81	B2=82	B3=83	B4=84	... B9=89
C=90	C1=91	C2=92	C3=93	C4=94	... C9=99

Remarks.

When some axes are parked, it is a good idea to know which axes are available. This variable indicates which axes are available, if an axis is not available, this variable returns a "?".

(V.) [ch].G.SPDLNAME_n

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Report variable (to be used from the scripts).

Name of the spindle with n index in the channel.

Syntax.

·ch· Channel number.

V.[2].G.SPDLNAME1 Channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

The values returned by this variable are encoded as follows.

S=100 S1=101 S2=102 S3=103 S4=104 ... S9=109

(V.)[ch].G.MASTERSP

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Logic number of the master spindle of the channel.

Syntax.

·ch· Channel number.

V.[2].G.MASTERSP	Channel ·2·.
------------------	--------------

TRAVEL LIMITS OF LINEAR AND ROTARY AXES.**(V.)[ch].G.SOFTLIMIT**

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Software limits reached.

Syntax.

·ch· Channel number.

V.[2].G.SOFTLIMIT	Channel ·2·.
-------------------	--------------

(V.)[ch].A.POSLIMIT.xn**(V.)[ch].A.NEGLIMIT.xn**

Variable that can be read and written from the program and read from the PLC and from the interface.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

Positive and negative software limit.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.POSLIMIT.Z	Z axis.
V.A.POSLIMIT.4	Axis with logic number ·4·.
V.[2].A.POSLIMIT.1	Axis with index ·1· in the channel ·2·.

Remarks.

These variables correspond to the limits set by machine parameters. When modifying these variables, the CNC assumes those values as the new limits from then on.

These variables keep their value after a Reset, they are initialized with the values of the machine parameters when validating them and when turning the CNC on.

(V.)[ch].A.RTPOSLIMIT.xn**(V.)[ch].A.RTNEGLIMIT.xn**

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Second positive and negative software limit.

22.

CNC VARIABLES.
Variables related to the machine configuration.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the machine configuration.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

V.A.RTPOSLIMIT.Z	Z axis.
V.A.RTPOSLIMIT.4	Axis with logic number -4-.
V.[2].A.RTPOSLIMIT.1	Axis with index -1- in the channel -2-.

Remarks.

There are two software limits; the CNC always applies the most restrictive one.

These variables keep their value after a Reset, they are initialized with the maximum value possible when turning the CNC on.

KINETIMATICS DIMENSIONS.

(V.)[ch].A.HEADOF.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution or preparation value.

Dimension in each axis of the kinematics.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

V.A.HEADOF.Z	Z axis.
V.A.HEADOF.4	Axis with logic number -4-.
V.[2].A.HEADOF.1	Axis with index -1- in the channel -2-.

Remarks.

The variable returns the value of execution or preparation as follows. If the axis is consulted through its logic number, the variable always returns the preparation value. In the rest of the cases, if the axis belongs to the channel requesting the variable, it returns the preparation value; if the axis belongs to a different channel, the variable returns the execution value and interrupts block preparation.

This variable returns the resulting measurement of the active kinematics on that axis. It may be a particular value of TDATA (kinematics table) or the combination of several of them depending on the type of kinematics.

CHANGE THE TURNING DIRECTION ASSIGNED TO M3 AND M4.

(V.)[ch].A.POLARITY.sn**(V.)[ch].SP.POLARITY.sn**

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

This variable may be used to reverse the meaning of spindle parameters POLARM3 and POLARM4 that set the spindle turning direction for M3 and M4. The variable does not modify the values of the machine parameters.

If the spindle is turning in open loop (M3/M4), the CNC does not apply the changes of this variable immediately; the CNC applies the changes the next time a speed or an M3/M4 is programmed.



CNC 8060
CNC 8065

(REF: 1402)

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.A.POLARITY.S	Spindle S.
V.SP.POLARITY.S	Spindle S.
V.SP.POLARITY	Master spindle.
V.A.POLARITY.4	Spindle with logic number ·4·.
V.SP.POLARITY.2	Spindle with index ·2· in the system.
V.[2].SP.POLARITY.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

Value.	Meaning.
0	For the M3/M4 functions, the CNC applies parameters POLARM3 and POLARM4.
1	For the M3/M4 functions, the CNC reverses the meaning of parameters POLARM3 and POLARM4.

Remarks.

The variable keeps its value between programs and after a reset; it is initialized to ·0· when turning the CNC on.

Every time the part-program writes or reads this variable, the CNC synchronizes the block preparation and the execution. If it is the PLC who modifies the variable, it must be borne in mind that block preparation takes its value into account every time it detects a change in the spindle movement (Function G63, change of speed or of the turning direction, etc.); therefore, before preparing those blocks, the PLC must have written this variable.

NUMBER OF PULSES SENT BY THE HANDWHEEL.

(V.)G.HANDP[hw]

*Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.*

Number of pulses sent by the handwheel since the system was started up. While the handwheel is disabled from the PLC (marks INHIBITMPG1 through INHIBITMPG12), the variable does not save the pulses sent by the handwheel.

Syntax.

- hw· Handwheel number.

(V.)G.HANDP[1]	Handwheel ·1·.
----------------	----------------

Remarks.

The PLC can use these variables to control the machining feedrate depending on how fast the handwheel is turned, for example when machining a part for the first time or to move back in the machining operation when the retrace function is active. Depending on the handwheel pulses read by the variable, it is possible to calculate from the PLC the right feedrate override percentage and set it for the machining operation. This handwheel mode is known as "feed handwheel".



CNC 8060
CNC 8065

CHANGE THE SIMULATION SPEED FROM THE PLC.

(V.)PLC.SIMUSPEED

Variable that can be read and written from the PLC and read from the program and from the interface. The variable returns the execution value; reading it interrupts block preparation. Report variable (to be used from the scripts).

Percentage of simulation speed (between 0% and 100%).

V.PLC.SIMUSPEED

Currently, the simulation displays a horizontal bar on the screen to change the simulation speed. This variable may be used to change its value via PLC. In some cases, the OEM can transfer via PLC the feedrate override value to this variable to control the simulation speed.



The execution and simulation channels can work simultaneously in which case, the feedrate override switch will affect both. It is up to the OEM to use this variable in the right contexts so it only affects one of the channels.

22.

CNC VARIABLES.
Variables related to the machine configuration.



CNC 8060
CNC 8065

(REF: 1402)

22.27 Variables related to volumetric compensation.

(V.)[ch].A.VOLCOMP.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Value that the volumetric compensation is adding to the axis.

Reading this variable from the oscilloscope allows watching the evolution of the volumetric compensation.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.VOLCOMP.Z	Z axis.
V.A.VOLCOMP.3	Axis with logic number ·3·.
V.[2].A.VOLCOMP.3	Axis with index ·3· in the channel ·2·.

(V.)[ch].A.PIVOT.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

Variable valid for analog drive, position Sercos and velocity Sercos.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Distance from the pivot center to the tool tip in machine coordinates. This variable returns the sum of the spindle offsets (parameters TDATA) and the tool offset.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.PIVOT.Z	Z axis.
V.A.PIVOT.3	Axis with logic number ·3·.
V.[2].A.PIVOT.3	Axis with index ·3· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to volumetric compensation.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.28 Variables associated with the Mechatrolink bus.

STATUS OF COMMUNICATION AND OF THE MECHATROLINK DEVICES.

(V.) [ch].A.MSTATUS.xn

(V.) [ch].A.MSTATUS.sn

(V.) [ch].SP.MSTATUS.sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the oscilloscope environment.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for servo and Mechatrolink inverter.

The variable returns the execution value; reading it interrupts block preparation.

Information on the status of the execution of the command and of the device according to Mechatrolink specifications. Refer to the documentations of the device.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MSTATUS.Z	Z axis.
V.MPA.MSTATUS.S	Spindle S.
V.SP.MSTATUS.S	Spindle S.
V.SP.MSTATUS	Master spindle.
V.MPA.MSTATUS.4	Axis or spindle with logic number ·4·.
V.[2].MPA.MSTATUS.1	Axis with index ·1· in the channel ·2·.
V.SP.MSTATUS.2	Spindle with index ·2· in the system.
V.[2].SP.MSTATUS.1	Spindle with index ·1· in the channel ·2·.

(V.) [ch].A.MSUBSTAT.xn

(V.) [ch].A.MSUBSTAT.sn

(V.) [ch].SP.MSUBSTAT.sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the oscilloscope environment.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for servo and Mechatrolink inverter.

The variable returns the execution value; reading it interrupts block preparation.

Information on the status of the execution of the subcommand and of the device according to Mechatrolink specifications. Refer to the documentations of the device.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MSUBSTAT.Z	Z axis.
V.MPA.MSUBSTAT.S	Spindle S.
V.SP.MSUBSTAT.S	Spindle S.
V.SP.MSUBSTAT	Master spindle.
V.MPA.MSUBSTAT.4	Axis or spindle with logic number ·4·.
V.[2].MPA.MSUBSTAT.1	Axis with index ·1· in the channel ·2·.
V.SP.MSUBSTAT.2	Spindle with index ·2· in the system.
V.[2].SP.MSUBSTAT.1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.
Variables associated with the Mechatrolink bus.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].A.MALARM.xn
 (V.)[ch].A.MALARM.sn
 (V.)[ch].SP.MALARM.sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the oscilloscope environment.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for servo and Mechatrolink inverter.

The variable returns the execution value; reading it interrupts block preparation.

Alarm or error code of the device. Refer to the documentations of the device.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MALARM.Z	Z axis.
V.MPA.MALARM.S	Spindle S.
V.SP.MALARM.S	Spindle S.
V.SP.MALARM	Master spindle.
V.MPA.MALARM.4	Axis or spindle with logic number ·4·.
V.[2].MPA.MALARM.1	Axis with index ·1· in the channel ·2·.
V.SP.MALARM.2	Spindle with index ·2· in the system.
V.[2].SP.MALARM.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.MIOMON.xn
 (V.)[ch].A.MIOMON.sn
 (V.)[ch].SP.MIOMON.sn

Variable that can only be read from the program, PLC and interface.

Variable that may be modified from the oscilloscope environment.

Variable valid for rotary and linear axes as well as for spindles.

Variable valid for Mechatrolink servo.

The variable returns the execution value; reading it interrupts block preparation.

Status of the device inputs and outputs.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.MPA.MIOMON.Z	Z axis.
V.MPA.MIOMON.S	Spindle S.
V.SP.MIOMON.S	Spindle S.
V.SP.MIOMON	Master spindle.
V.MPA.MIOMON.4	Axis or spindle with logic number ·4·.
V.[2].MPA.MIOMON.1	Axis with index ·1· in the channel ·2·.
V.SP.MIOMON.2	Spindle with index ·2· in the system.
V.[2].SP.MIOMON.1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables associated with the Mechatrolink bus.

FAGOR 

**CNC 8060
CNC 8065**

(REF: 1402)

22.29 Variables related to synchronized switching.

The value of these variables is equivalent to the one programmed in the parameters of the instruction #SWTOUT. When executing the instruction without parameters and the variables have a value assigned to it, the CNC assumes these latter values as active. Likewise, when programming the parameters of the instruction, the variables assume these values as their own.

On CNC power-up, the offset wears are set to zero. After power-up, the values programmed for the offsets (using the instruction #SWTOUT or the variables) are maintained even after an error, a reset or an M30.

These variables interrupt block preparation. To modify the offset values without interrupting block preparation, change these values from the PLC or use the parameters of the instruction #SWTOUT. If these variables are changed from the PLC, the new values are assumed when executing the instruction #SWTOUT. If the instruction is active when changing the values from the PLC, it assumes the new values.

V.G.TON

Variable that can be read and written via program, PLC and interface.

The variable returns the execution value; reading/writing it interrupts block preparation.

Time shift (milliseconds) to anticipate the activation of the digital output.

V.G.TON

V.G.TOF

Variable that can be read and written via program, PLC and interface.

The variable returns the execution value; reading/writing it interrupts block preparation.

Time shift (milliseconds) to anticipate the deactivation of the digital output.

V.G.TOF

V.G.PON

Variable that can be read and written via program, PLC and interface.

The variable returns the execution value; reading/writing it interrupts block preparation.

Distance shift (offset, mm(inches) to anticipate the activation of the digital output.

V.G.PON

V.G.POF

Variable that can be read and written via program, PLC and interface.

The variable returns the execution value; reading/writing it interrupts block preparation.

Distance shift (offset, mm(inches) to anticipate the deactivation of the digital output.

V.G.POF

22.

CNC VARIABLES.
Variables related to synchronized switching.



CNC 8060
CNC 8065

(REF: 1402)

22.30 PWM related variables.

(V.)G.PWMON

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

PWM status, when it has been activated via CNC.

Syntax.

V.G.PWMON

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Laser OFF.
1	Laser ON.

Remarks.

Activating the Laser via PLC has priority over activating it from the CNC.

Laser status from the CNC. Variable (V.)G.PWMON	Laser status from the PLC. PWMON mark	Laser status.
0	0	Laser OFF.
1	0	Laser ON from the CNC.
0	1	Laser ON from the PLC.
1	1	Laser ON from the PLC.

(V.)G.PWMFREQ

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Frequency of the PWM (between 2 and 5000 Hz; by default, 0), when the PWM has been activated via CNC. This variable only works when the Laser is active via CNC.

Syntax.

V.G.PWMFREQ

(V.)G.PWMDUTY

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Duty cycle of the PWM (between 0.1 and 100%; by default, 50%), when the PWM has been activated via CNC. This variable only works when the Laser is active via CNC.

Syntax.

V.G.PWMDUTY

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. for a value ·0.1·, the PLC reading will return a value of ·1·. For a value of ·100·, the readin from the PLC will return a value of ·1000·.

22.

CNC VARIABLES.
PWM related variables.

FAGOR 

CNC 8060
CNC 8065

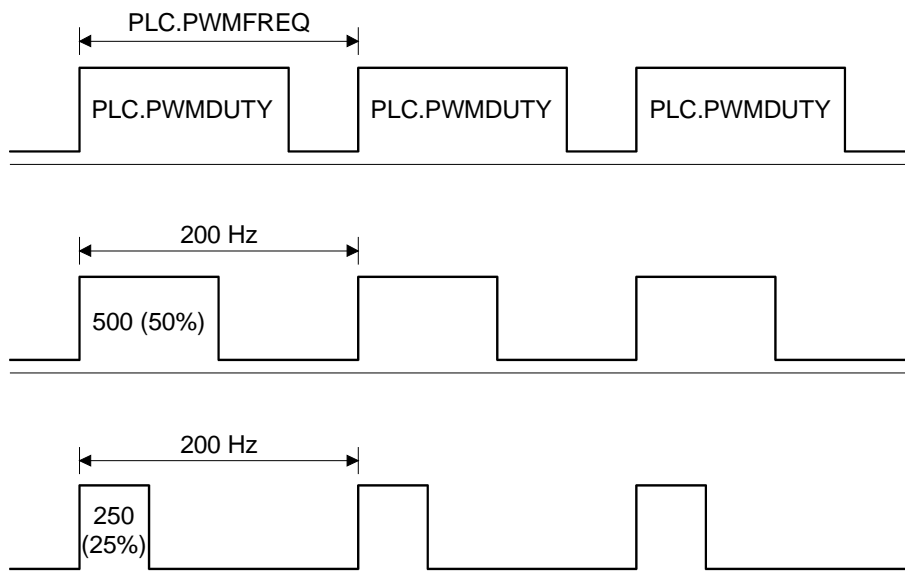
(REF: 1402)

(V.)PLC.PWMFREQ

Variable that can be read and written from the PLC and read from the program and from the interface. The variable returns the execution value; reading it interrupts block preparation.

Frequency of the PWM (between 2 and 5000 Hz; by default, 0), when the PWM has been activated via PLC. Setting variable to $\cdot 0\cdot$ deactivates the output of the PWM. This variable only works when the Laser has been activated via PLC.

PLC.PWMFREQ

**Remarks.**

The changes from the variable are updated (refreshed) in the next loop; the intermediate times between loops will be ignored.

(V.)PLC.PWMDUTY

Variable that can be read and written from the PLC and read from the program and from the interface. The variable returns the execution value; reading it interrupts block preparation.

Duty cycle of the PWM (between 1 and 100%; by default, 50%), when the PWM has been activated via PLC. This variable only works when the Laser has been activated via PLC.

PLC.PWMDUTY

Remarks.

The changes from the variable are updated (refreshed) in the next loop; the intermediate times between loops will be ignored.

The reading from the CNC will be divided by 10; i.e. for the value of $\cdot 1\cdot$, the CNC reading will return a value of $\cdot 0.1\cdot$.

22.

CNC VARIABLES.
PWM related variables.

22.31 Variables related to cycle time.

ANALYSIS OF THE LOOP TIME (CYCLE TIME) AT THE CNC.

(V.)G.NCTIMERATE

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Percentage of loop time (cycle time) used by the real time portion of the CNC.

V.G.NCTIMERATE

This variable serves to evaluate the load of the system and the time that the periodic interruption that controls the movement of the axes leaves the operating time to manage other applications that may be executed in parallel with the CNC. Applications that need time of the operating system are, for example, displaying the screens, user interface applications, refreshing variables on the screen, file management (subroutines or programs that are opened and closed while machining), etc.

If there isn't enough free time for the applications, it is possible to decrease parameter PREPFREQ, increase parameter LOOPTIME, group the subroutines in a the same file or reduce the number of external applications

(V.)G.LOOPTIMERATE

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Percentage of loop time (cycle time) used by the position loop.

V.G.LOOPTIMERATE

This variable serves as reference to know whether the consumption of interruption time is due to the number of axes or to the path preparation process itself.

If the position loop uses most of the interruption time, it means that the system is overloaded by the axes and, therefore, you will have to evaluate the possibility to increase the LOOPTIME parameter.

ANALYSIS OF THE LOOP TIME (CYCLE TIME) IN THE CHANNEL.

(V.)[ch].G.CHTIMERATE

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Percentage of loop time (cycle time) used by the channel.

This variable helps determine whether the particular execution of a channel is taking up too much time.

Syntax.

.ch· Channel number.

V.[2].G.CHTIMERATE Channel -2.

(V.)[ch].G.PREPTIMERATE

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Percentage of loop time (cycle time) used by the channel for block preparation.

22.

CNC VARIABLES.
Variables related to cycle time.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

This variable serves to evaluate the load in path preparation and to know whether it is possible to increase the PREPFREQ parameter or not.

Syntax.

·ch· Channel number.

V.[2].G.PREPTIMERATE Channel -2.

(V.)[ch].G.IPOTIMERATE

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Percentage of loop time (cycle time) used by the interpolator of the channel.

This variable serves to evaluate the overload in the path generating algorithm and the smoothing algorithm.

Syntax.

·ch· Channel number.

V.[2].G.IPOTIMERATE Channel -2.

22.

CNC VARIABLES.
Variables related to cycle time.



CNC 8060
CNC 8065

(REF: 1402)

22.32 Variables associated with the feedback inputs for analog axes.

(V.)[ch].A.COUNTERST.xn
 (V.)[ch].A.COUNTERST.sn
 (V.)[ch].SP.COUNTERST.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Status of the feedback input.

For a feedback input to be active, it must have an analog axis associated with it.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.COUNTERST.Z	Z axis.
V.A.COUNTERST.S	Spindle S.
V.SP.COUNTERST.S	Spindle S.
V.SP.COUNTERST	Master spindle.
V.A.COUNTERST.4	Axis or spindle with logic number ·4·.
V.[2].A.COUNTERST.1	Axis with index ·1· in the channel ·2·.
V.SP.COUNTERST.2	Spindle with index ·2· in the system.
V.[2].SP.COUNTERST.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.COUNTER.xn
 (V.)[ch].A.COUNTER.sn
 (V.)[ch].SP.COUNTER.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Feedback input pulses (integer side + fraction side).

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.COUNTER.Z	Z axis.
V.A.COUNTER.S	Spindle S.
V.SP.COUNTER.S	Spindle S.
V.SP.COUNTER	Master spindle.
V.A.COUNTER.4	Axis or spindle with logic number ·4·.
V.[2].A.COUNTER.1	Axis with index ·1· in the channel ·2·.
V.SP.COUNTER.2	Spindle with index ·2· in the system.
V.[2].SP.COUNTER.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.ASINUS.xn
 (V.)[ch].A.ASINUS.sn
 (V.)[ch].SP.ASINUS.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Fraction side of the A signal.

22.

CNC VARIABLES.

Variables associated with the feedback inputs for analog axes.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables associated with the feedback inputs for analog axes.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.ASINUS.Z	Z axis.
V.A.ASINUS.S	Spindle S.
V.SP.ASINUS.S	Spindle S.
V.SP.ASINUS	Master spindle.
V.A.ASINUS.4	Axis or spindle with logic number ·4·.
V.[2].A.ASINUS.1	Axis with index ·1· in the channel ·2·.
V.SP.ASINUS.2	Spindle with index ·2· in the system.
V.[2].SP.ASINUS.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.BSINUS.xn**(V.)[ch].A.BSINUS.sn****(V.)[ch].SP.BSINUS.sn**

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Fraction side of the B signal.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.BSINUS.Z	Z axis.
V.A.BSINUS.S	Spindle S.
V.SP.BSINUS.S	Spindle S.
V.SP.BSINUS	Master spindle.
V.A.BSINUS.4	Axis or spindle with logic number ·4·.
V.[2].A.BSINUS.1	Axis with index ·1· in the channel ·2·.
V.SP.BSINUS.2	Spindle with index ·2· in the system.
V.[2].SP.BSINUS.1	Spindle with index ·1· in the channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

22.33 Variables associated with the analog inputs and outputs.

(V.)G.ANAI [n]

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

[n] input voltage (in volts).

Syntax.

·n· Number of the analog input.

V.G.ANAI[3] Voltage of the analog input ·3·.

(V.)G.ANAO [n]

Variable that can be read from the interface and written from the program and from the PLC.

The variable returns the execution value; reading it interrupts block preparation.

[n] output voltage (in volts).

Syntax.

·n· Number of the analog input.

V.G.ANAO[3] Voltage of the analog output ·3·.

22.

CNC VARIABLES.

Variables associated with the analog inputs and outputs.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.34 Variables associated with the velocity command and the feedback of the drive.

VELOCITY COMMAND AND TORQUE FOR SERCOS AXES.

22.

CNC VARIABLES.

Variables associated with the velocity command and the feedback of the drive.

(V.)[ch].A.FTEO.xn
(V.)[ch].A.FTEO.sn
(V.)[ch].SP.FTEO.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Velocity command for Sercos (in rpm).

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.FTEO.Z	Z axis.
V.A.FTEO.S	Spindle S.
V.SP.FTEO.S	Spindle S.
V.SP.FTEO	Master spindle.
V.A.FTEO.4	Axis or spindle with logic number -4-.
V.[2].A.FTEO.1	Axis with index -1- in the channel -2-.
V.SP.FTEO.2	Spindle with index -2- in the system.
V.[2].SP.FTEO.1	Spindle with index -1- in the channel -2-.

(V.)[ch].A.POSCMD.xn
(V.)[ch].A.POSCMD.sn
(V.)[ch].SP.POSCMD.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Position command for Sercos.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.POSCMD.Z	Z axis.
V.A.POSCMD.S	Spindle S.
V.SP.POSCMD.S	Spindle S.
V.SP.POSCMD	Master spindle.
V.A.POSCMD.4	Axis or spindle with logic number -4-.
V.[2].A.POSCMD.1	Axis with index -1- in the channel -2-.
V.SP.POSCMD.2	Spindle with index -2- in the system.
V.[2].SP.POSCMD.1	Spindle with index -1- in the channel -2-.

(V.)[ch].A.TORQUE.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Current torque in Sercos.



CNC 8060
CNC 8065

(REF: 1402)

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.TORQUE.Z	Z axis.
V.A.TORQUE.S	Spindle S.
V.SP.TORQUE.S	Spindle S.
V.SP.TORQUE	Master spindle.
V.A.TORQUE.4	Axis or spindle with logic number ·4·.
V.[2].A.TORQUE.1	Axis with index ·1· in the channel ·2·.
V.SP.TORQUE.2	Spindle with index ·2· in the system.
V.[2].SP.TORQUE.1	Spindle with index ·1· in the channel ·2·.

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·100·.

FEEDBACK OF THE ANALOG OR SERCOS DRIVE.

- (V.)[ch].A.POSNC.xn
- (V.)[ch].A.POSNC.sn
- (V.)[ch].SP.POSNC.sn

*Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.*

Position feedback.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.POSNC.Z	Z axis.
V.A.POSNC.S	Spindle S.
V.SP.POSNC.S	Spindle S.
V.SP.POSNC	Master spindle.
V.A.POSNC.4	Axis or spindle with logic number ·4·.
V.[2].A.POSNC.1	Axis with index ·1· in the channel ·2·.
V.SP.POSNC.2	Spindle with index ·2· in the system.
V.[2].SP.POSNC.1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.
Variables associated with the velocity command and the feedback of the drive.



CNC 8060
CNC 8065

(REF: 1402)

22.35 Variables related to the change of gear and set of the Sercos drive.

22.

CNC VARIABLES.

Variables related to the change of gear and set of the Sercos drive.

(V.)[ch].A.SETGE.xn
(V.)[ch].A.SETGE.sn
(V.)[ch].SP.SETGE.sn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Select the set and the gear at the drive. Only one change can be taking place at a time. If other gear or set changes are programmed during this process, even if they are at different drives, the CNC only keeps the one programmed last and ignores the rest of the intermediate changes.

The drive can have 8 work ranges or gears identified from 0 through 7 (drive parameter GP6) and 8 sets of parameters (drive parameter GP4) identified from 0 through 7.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.SETGE.Z	Z axis.
V.A.SETGE.S	Spindle S.
V.SP.SETGE.S	Spindle S.
V.SP.SETGE	Master spindle.
V.A.SETGE.4	Axis or spindle with logic number ·4·.
V.[2].A.SETGE.1	Axis with index ·1· in the channel ·2·.
V.SP.SETGE.2	Spindle with index ·2· in the system.
V.[2].SP.SETGE.1	Spindle with index ·1· in the channel ·2·.

Values of the variable.

The 4 least significant bits indicate the work range and the 4 most significant bits indicate the set of parameters. If the value of any 4-bit set is ·0·, the CNC does not change the active gear or set at the drive. Example of some values of the variable.

Value.	Meaning.
\$21	First range or gear (gear ·0·). Second set of parameters (set ·1·).
\$40	The drive maintains the active gear. Fourth set of parameters (set ·3·).
\$07	Seventh range or gear (gear ·6·). The drive maintains the active parameter set.

Remarks.

The CNC keeps the SERPLCAC mark active during the change process.



CNC 8060
CNC 8065

(REF: 1402)

22.36 Variables related to loop adjustment.

COORDINATE RELATED VARIABLES.

(V.)[ch].A.IPOPOS.xn
 (V.)[ch].A.IPOPOS.sn
 (V.)[ch].SP.IPOPOS.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Theoretical position value (coordinate) at the output of the interpolator, before the transformation; i.e. in part coordinates.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.IPOPOS.Z	Z axis.
V.A.IPOPOS.S	Spindle S.
V.SP.IPOPOS.S	Spindle S.
V.SP.IPOPOS	Master spindle.
V.A.IPOPOS.4	Axis or spindle with logic number ·4·.
V.[2].A.IPOPOS.1	Axis with index ·1· in the channel ·2·.
V.SP.IPOPOS.2	Spindle with index ·2· in the system.
V.[2].SP.IPOPOS.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.FILTERIN.xn
 (V.)[ch].A.FILTERIN.sn
 (V.)[ch].SP.FILTERIN.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Theoretical coordinate of the interpolator before the filter.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.FILTERIN.Z	Z axis.
V.A.FILTERIN.S	Spindle S.
V.SP.FILTERIN.S	Spindle S.
V.SP.FILTERIN	Master spindle.
V.A.FILTERIN.4	Axis or spindle with logic number ·4·.
V.[2].A.FILTERIN.1	Axis with index ·1· in the channel ·2·.
V.SP.FILTERIN.2	Spindle with index ·2· in the system.
V.[2].SP.FILTERIN.1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.
 Variables related to loop adjustment.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to loop adjustment.

(V.)[ch].A.FILTEROUT.xn
(V.)[ch].A.FILTEROUT.sn
(V.)[ch].SP.FILTEROUT.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Theoretical coordinate of the interpolator after the filter.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.FILTEROUT.Z	Z axis.
V.A.FILTEROUT.S	Spindle S.
V.SP.FILTEROUT.S	Spindle S.
V.SP.FILTEROUT	Master spindle.
V.A.FILTEROUT.4	Axis or spindle with logic number ·4·.
V.[2].A.FILTEROUT.1	Axis with index ·1· in the channel ·2·.
V.SP.FILTEROUT.2	Spindle with index ·2· in the system.
V.[2].SP.FILTEROUT.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.LOOPTPOS.xn
(V.)[ch].A.LOOPTPOS.sn
(V.)[ch].SP.LOOPTPOS.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Theoretical coordinate at the input of the position loop.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.LOOPTPOS.Z	Z axis.
V.A.LOOPTPOS.S	Spindle S.
V.SP.LOOPTPOS.S	Spindle S.
V.SP.LOOPTPOS	Master spindle.
V.A.LOOPTPOS.4	Axis or spindle with logic number ·4·.
V.[2].A.LOOPTPOS.1	Axis with index ·1· in the channel ·2·.
V.SP.LOOPTPOS.2	Spindle with index ·2· in the system.
V.[2].SP.LOOPTPOS.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.LOOPPOS.xn
(V.)[ch].A.LOOPPOS.sn
(V.)[ch].SP.LOOPPOS.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Real coordinate at the input of the position loop.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.



CNC 8060
CNC 8065

(REF: 1402)

.sn Name, logic number or index of the spindle.

V.A.LOOPPOS.Z	Z axis.
V.A.LOOPPOS.S	Spindle S.
V.SP.LOOPPOS.S	Spindle S.
V.SP.LOOPPOS	Master spindle.
V.A.LOOPPOS.4	Axis or spindle with logic number .4 .
V.[2].A.LOOPPOS.1	Axis with index .1 in the channel .2 .
V.SP.LOOPPOS.2	Spindle with index .2 in the system.
V.[2].SP.LOOPPOS.1	Spindle with index .1 in the channel .2 .

POSITION INCREMENT AND SAMPLING PERIOD.

(V.)[ch].A.POSINC.xn
(V.)[ch].A.POSINC.sn
(V.)[ch].SP.POSINC.sn

Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.

Real position increment of the current sampling period.

Syntax.

.ch Channel number.
.xn Name, logic number or index of the axis.
.sn Name, logic number or index of the spindle.

V.A.POSINC.Z	Z axis.
V.A.POSINC.S	Spindle S.
V.SP.POSINC.S	Spindle S.
V.SP.POSINC	Master spindle.
V.A.POSINC.4	Axis or spindle with logic number .4 .
V.[2].A.POSINC.1	Axis with index .1 in the channel .2 .
V.SP.POSINC.2	Spindle with index .2 in the system.
V.[2].SP.POSINC.1	Spindle with index .1 in the channel .2 .

(V.)[ch].A.TPOSINC.xn
(V.)[ch].A.TPOSINC.sn
(V.)[ch].SP.TPOSINC.sn

Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.

Theoretical position increment of the current sampling period.

Syntax.

.ch Channel number.
.xn Name, logic number or index of the axis.
.sn Name, logic number or index of the spindle.

V.A.TPOSINC.Z	Z axis.
V.A.TPOSINC.S	Spindle S.
V.SP.TPOSINC.S	Spindle S.
V.SP.TPOSINC	Master spindle.
V.A.TPOSINC.4	Axis or spindle with logic number .4 .

22.

CNC VARIABLES.

Variables related to loop adjustment.



CNC 8060
 CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to loop adjustment.

V.[2].A.TPOSINC.1	Axis with index -1- in the channel -2-.
V.SP.TPOSINC.2	Spindle with index -2- in the system.
V.[2].SP.TPOSINC.1	Spindle with index -1- in the channel -2-.

(V.)[ch].A.PREVPOSINC.xn
(V.)[ch].A.PREVPOSINC.sn
(V.)[ch].SP.PREVPOSINC.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Real position increment of the previous sampling period.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.PREVPOSINC.Z	Z axis.
V.A.PREVPOSINC.S	Spindle S.
V.SP.PREVPOSINC.S	Spindle S.
V.SP.PREVPOSINC	Master spindle.
V.A.PREVPOSINC.4	Axis or spindle with logic number -4-.
V.[2].A.PREVPOSINC.1	Axis with index -1- in the channel -2-.
V.SP.PREVPOSINC.2	Spindle with index -2- in the system.
V.[2].SP.PREVPOSINC.1	Spindle with index -1- in the channel -2-.

FINE ADJUSTMENT OF FEEDRATE, ACCELERATION AND JERK.

(V.)[ch].A.TFEED.xn
(V.)[ch].A.TFEED.sn
(V.)[ch].SP.TFEED.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Instantaneous theoretical speed value at the input of the position loop.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.TFEED.Z	Z axis.
V.A.TFEED.S	Spindle S.
V.SP.TFEED.S	Spindle S.
V.SP.TFEED	Master spindle.
V.A.TFEED.4	Axis or spindle with logic number -4-.
V.[2].A.TFEED.1	Axis with index -1- in the channel -2-.
V.SP.TFEED.2	Spindle with index -2- in the system.
V.[2].SP.TFEED.1	Spindle with index -1- in the channel -2-.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].A.FEED.xn
(V.)[ch].A.FEED.sn
(V.)[ch].SP.FEED.sn

Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.

Instantaneous real speed value at the input of the position loop.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.FEED.Z	Z axis.
V.A.FEED.S	Spindle S.
V.SP.FEED.S	Spindle S.
V.SP.FEED	Master spindle.
V.A.FEED.4	Axis or spindle with logic number ·4·.
V.[2].A.FEED.1	Axis with index ·1· in the channel ·2·.
V.SP.FEED.2	Spindle with index ·2· in the system.
V.[2].SP.FEED.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.TACCEL.xn
(V.)[ch].A.TACCEL.sn
(V.)[ch].SP.TACCEL.sn

Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.

Theoretical instantaneous acceleration value.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.TACCEL.Z	Z axis.
V.A.TACCEL.S	Spindle S.
V.SP.TACCEL.S	Spindle S.
V.SP.TACCEL	Master spindle.
V.A.TACCEL.4	Axis or spindle with logic number ·4·.
V.[2].A.TACCEL.1	Axis with index ·1· in the channel ·2·.
V.SP.TACCEL.2	Spindle with index ·2· in the system.
V.[2].SP.TACCEL.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.ACCEL.xn
(V.)[ch].A.ACCEL.sn
(V.)[ch].SP.ACCEL.sn

Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.

Real instantaneous acceleration value.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

22.

CNC VARIABLES.
 Variables related to loop adjustment.



CNC 8060
 CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to loop adjustment.

·sn· Name, logic number or index of the spindle.

V.A.ACCEL.Z	Z axis.
V.A.ACCEL.S	Spindle S.
V.SP.ACCEL.S	Spindle S.
V.SP.ACCEL	Master spindle.
V.A.ACCEL.4	Axis or spindle with logic number ·4·.
V.[2].A.ACCEL.1	Axis with index ·1· in the channel ·2·.
V.SP.ACCEL.2	Spindle with index ·2· in the system.
V.[2].SP.ACCEL.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.TJERK.xn
(V.)[ch].A.TJERK.sn
(V.)[ch].SP.TJERK.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Theoretical instantaneous jerk value.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.A.TJERK.Z	Z axis.
V.A.TJERK.S	Spindle S.
V.SP.TJERK.S	Spindle S.
V.SP.TJERK	Master spindle.
V.A.TJERK.4	Axis or spindle with logic number ·4·.
V.[2].A.TJERK.1	Axis with index ·1· in the channel ·2·.
V.SP.TJERK.2	Spindle with index ·2· in the system.
V.[2].SP.TJERK.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.JERK.xn
(V.)[ch].A.JERK.sn
(V.)[ch].SP.JERK.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Real instantaneous jerk value.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.A.JERK.Z	Z axis.
V.A.JERK.S	Spindle S.
V.SP.JERK.S	Spindle S.
V.SP.JERK	Master spindle.
V.A.JERK.4	Axis or spindle with logic number ·4·.
V.[2].A.JERK.1	Axis with index ·1· in the channel ·2·.
V.SP.JERK.2	Spindle with index ·2· in the system.
V.[2].SP.JERK.1	Spindle with index ·1· in the channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

GAIN SETTING VIA PLC.

(V.)[ch].A.PLCFFGAIN.xn
 (V.)[ch].A.PLCFFGAIN.sn
 (V.)[ch].SP.PLCFFGAIN.sn

Variable that can be read and written from the PLC and read from the program and from the interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Percentage of feed-forward programmed from the PLC.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.PLCFFGAIN.Z	Z axis.
V.A.PLCFFGAIN.S	Spindle S.
V.SP.PLCFFGAIN.S	Spindle S.
V.SP.PLCFFGAIN	Master spindle.
V.A.PLCFFGAIN.4	Axis or spindle with logic number ·4·.
V.[2].A.PLCFFGAIN.1	Axis with index ·1· in the channel ·2·.
V.SP.PLCFFGAIN.2	Spindle with index ·2· in the system.
V.[2].SP.PLCFFGAIN.1	Spindle with index ·1· in the channel ·2·.

Remarks.

In order for the feed-forward and the AC-forward defined this way to be taken into account, they must be active by machine parameter; i.e. by means of machine parameter FFWTYPE if it is an analog drive or a simulated drive or parameter OPMODEP if it is a Sercos drive.

The values defined by these variables prevail over the ones defined by machine parameters or by program. Setting the variables with a negative value cancels their effect ("0" is a valid value). These variables are initialized neither by a reset nor when validating the parameters.

The reading from the PLC will be given in hundredths (x100); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·1000·.

(V.)[ch].A.PLCACFGAIN.xn
 (V.)[ch].A.PLCACFGAIN.sn
 (V.)[ch].SP.PLCACFGAIN.sn

Variable that can be read and written from the PLC and read from the program and from the interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Percentage of AC-forward programmed from the PLC.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.PLCACFGAIN.Z	Z axis.
V.A.PLCACFGAIN.S	Spindle S.
V.SP.PLCACFGAIN.S	Spindle S.
V.SP.PLCACFGAIN	Master spindle.
V.A.PLCACFGAIN.4	Axis or spindle with logic number ·4·.
V.[2].A.PLCACFGAIN.1	Axis with index ·1· in the channel ·2·.
V.SP.PLCACFGAIN.2	Spindle with index ·2· in the system.
V.[2].SP.PLCACFGAIN.1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.
Variables related to loop adjustment.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to loop adjustment.

Remarks.

In order for the feed-forward and the AC-forward defined this way to be taken into account, they must be active by machine parameter; i.e. by means of machine parameter FFWTYPE if it is an analog drive or a simulated drive or parameter OPMODEP if it is a Sercos drive.

The values defined by these variables prevail over the ones defined by machine parameters or by program. Setting the variables with a negative value cancels their effect ("0" is a valid value). These variables are initialized neither by a reset nor when validating the parameters.

The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is -10, the PLC reading will return a value of -100.

(V.)[ch].A.PLCPROGAIN.xn
(V.)[ch].A.PLCPROGAIN.sn
(V.)[ch].SP.PLCPROGAIN.sn

Variable that can be read and written from the PLC and read from the program and from the interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Proportional gain programmed from the PLC.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.PLCPROGAIN.Z	Z axis.
V.A.PLCPROGAIN.S	Spindle S.
V.SP.PLCPROGAIN.S	Spindle S.
V.SP.PLCPROGAIN	Master spindle.
V.A.PLCPROGAIN.4	Axis or spindle with logic number -4-.
V.[2].A.PLCPROGAIN.1	Axis with index -1- in the channel -2-.
V.SP.PLCPROGAIN.2	Spindle with index -2- in the system.
V.[2].SP.PLCPROGAIN.1	Spindle with index -1- in the channel -2-.

Remarks.

The values defined by these variables prevail over the ones defined by machine parameters or by program. Setting the variables with a negative value cancels their effect ("0" is a valid value). These variables are initialized neither by a reset nor when validating the parameters.



CNC 8060
CNC 8065

(REF: 1402)

22.37 Variables related to the loop of the axis or of the tandem spindle.

(V.)[ch].A.TPIIN.xn
 (V.)[ch].A.TPIIN.sn
 (V.)[ch].SP.TPIIN.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Input of the PI of the master axis of the tandem (in rpm).

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.TPIIN.Z	Z axis.
V.A.TPIIN.S	Spindle S.
V.SP.TPIIN.S	Spindle S.
V.SP.TPIIN	Master spindle.
V.A.TPIIN.4	Axis or spindle with logic number ·4·.
V.[2].A.TPIIN.1	Axis with index ·1· in the channel ·2·.
V.SP.TPIIN.2	Spindle with index ·2· in the system.
V.[2].SP.TPIIN.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.TPIOUT.xn
 (V.)[ch].A.TPIOUT.sn
 (V.)[ch].SP.TPIOUT.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Output of the PI of the master axis of the tandem (in rpm).

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.TPIOUT.Z	Z axis.
V.A.TPIOUT.S	Spindle S.
V.SP.TPIOUT.S	Spindle S.
V.SP.TPIOUT	Master spindle.
V.A.TPIOUT.4	Axis or spindle with logic number ·4·.
V.[2].A.TPIOUT.1	Axis with index ·1· in the channel ·2·.
V.SP.TPIOUT.2	Spindle with index ·2· in the system.
V.[2].SP.TPIOUT.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.TFILTOUT.xn
 (V.)[ch].A.TFILTOUT.sn
 (V.)[ch].SP.TFILTOUT.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Output of the pre-load filter of the tandem.

Syntax.

- ch· Channel number.

22.

CNC VARIABLES.

Variables related to the loop of the axis or of the tandem spindle.

FAGOR 

CNC 8060
 CNC 8065

(REF: 1402)

22.

CNC VARIABLES.

Variables related to the loop of the axis or of the tandem spindle.

- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.TFILTOUT.Z	Z axis.
V.A.TFILTOUT.S	Spindle S.
V.SP.TFILTOUT.S	Spindle S.
V.SP.TFILTOUT	Master spindle.
V.A.TFILTOUT.4	Axis or spindle with logic number ·4·.
V.[2].A.TFILTOUT.1	Axis with index ·1· in the channel ·2·.
V.SP.TFILTOUT.2	Spindle with index ·2· in the system.
V.[2].SP.TFILTOUT.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.PRELOAD.xn**(V.)[ch].A.PRELOAD.sn****(V.)[ch].SP.PRELOAD.sn***Variable that can be read and written via program, PLC and interface.**Variable valid for rotary and linear axes as well as for spindles.**The variable returns the execution value; reading it interrupts block preparation.*

Preload in the tandem.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.PRELOAD.Z	Z axis.
V.A.PRELOAD.S	Spindle S.
V.SP.PRELOAD.S	Spindle S.
V.SP.PRELOAD	Master spindle.
V.A.PRELOAD.4	Axis or spindle with logic number ·4·.
V.[2].A.PRELOAD.1	Axis with index ·1· in the channel ·2·.
V.SP.PRELOAD.2	Spindle with index ·2· in the system.
V.[2].SP.PRELOAD.1	Spindle with index ·1· in the channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

22.38 Variables related to user tables (zero offset table).

ZERO'S OFFSETS			
Origin	X (mm)	Y (mm)	Z (mm)
PLCOF	00000.000	00000.000	00000.000
G54	00000.000	00000.000	00000.000
G55	00000.000	00000.000	00000.000
G56	00000.000	00000.000	00000.000
G57	00000.000	00000.000	00000.000
G58	00000.000	00000.000	00000.000
G59	00000.000	00000.000	00000.000
G159=7	00000.000	00000.000	00000.000

V.A.PLCOF.X

V.G.FORG

(V.)G.FORG

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

First zero offset in the table.

V.G.FORG

(V.)G.NUMORG

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of zero offsets in the table.

V.G.FORG

(V.)[ch].A.PLCOF.xn

(V.)[ch].A.PLCOF.sn

(V.)[ch].SP.PLCOF.sn

Variable that can be read and written from the program and from the PLC and read from the interface.
Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

xn axis. Offset of the zero offset set by PLC.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.PLCOF.Z	Z axis.
V.A.PLCOF.S	Spindle S.
V.SP.PLCOF.S	Spindle S.
V.SP.PLCOF	Master spindle.
V.A.PLCOF.4	Axis or spindle with logic number -4-.
V.[2].A.PLCOF.1	Axis with index -1- in the channel -2-.
V.SP.PLCOF.2	Spindle with index -2- in the system.
V.[2].SP.PLCOF.1	Spindle with index -1- in the channel -2-.

22.

CNC VARIABLES.

Variables related to user tables (zero offset table).



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.

Variables related to user tables (zero offset table).

(V.)[ch].A.ACTPLCOF.xn
 (V.)[ch].A.ACTPLCOF.sn
 (V.)[ch].SP.ACTPLCOF.sn

Variable that can only be read from the program, PLC and interface.
 Variable valid for rotary and linear axes as well as for spindles.
 The variable returns the execution value; reading it interrupts block preparation.
 Report variable (to be used from the scripts).

xn axis. Accumulated PLC offset.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.ACTPLCOF.Z	Z axis.
V.A.ACTPLCOF.S	Spindle S.
V.SP.ACTPLCOF.S	Spindle S.
V.SP.ACTPLCOF	Master spindle.
V.A.ACTPLCOF.4	Axis or spindle with logic number -4-.
V.[2].A.ACTPLCOF.1	Axis with index -1- in the channel -2-.
V.SP.ACTPLCOF.2	Spindle with index -2- in the system.
V.[2].SP.ACTPLCOF.1	Spindle with index -1- in the channel -2-.

ZERO OFFSET TABLE (WITHOUT FINE SETTING OF THE ABSOLUTE ZERO OFFSET).

The fine setting of the absolute zero offset is enabled via machine parameters (parameter FINEORG)

ZERO'S OFFSETS			
Origin	X (mm)	Y (mm)	Z (mm)
PLCOF	00000.000	00000.000	00000.000
G54	00000.000	00000.000	00000.000
G55	00000.000	00000.000	00000.000
G56	00000.000	00000.000	00000.000
G57	00000.000	00000.000	00000.000
G58	00000.000	00000.000	00000.000
G59	00000.000	00000.000	00000.000
G159=7	00000.000	00000.000	00000.000

V.A.ORG[6].Y

(V.)[ch].A.ORG.xn

Variable that can only be read from the program, PLC and interface.
 Variable valid for rotary and linear axes.
 The variable returns the value of block preparation.

xn axis. Value of the active zero offset (absolute G159 + incremental G158).

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

V.A.ORG.Z	Z axis.
V.A.ORG.3	Axis with logic number -3-.
V.[2].A.ORG.3	Axis with index -3- in the channel -2-.



CNC 8060
 CNC 8065

(REF: 1402)

(V.)[ch].A.ADDORG.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

xn axis. Value of the active incremental zero offset (G158).

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.ADDORG.Z	Z axis.
V.A.ADDORG.3	Axis with logic number ·3·.
V.[2].A.ADDORG.3	Axis with index ·3· in the channel ·2·.

(V.)[ch].A.ORG[nb].xn

Variable that can be read and written from the program and from the PLC and read from the interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

xn axis. Offset set in the zero offset [nb].

Syntax.

·ch· Channel number.

·nb· Zero offset number.

·xn· Name, logic number or index of the axis.

V.A.ORG[1].Z	Zero offset G54 (G159=1). Z axis.
V.A.ORG[4]0.3	Zero offset G57 (G159=4). Axis with logic number ·3·.
V.[2].A.ORG[9].3	Zero offset G159=9. Axis with index ·3· in the channel ·2·.

Remarks.

The numbering of zero offsets G54 through G59 is always the same; G54=1, G55=2, G56=3, G57=4, G58=5, G59=6.

ZERO OFFSET TABLE (WITH FINE SETTING OF THE ABSOLUTE ZERO OFFSET).

The fine setting of the absolute zero offset is enabled via machine parameters (parameter FINEORG)

(V.)[ch].A.ORG.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

xn axis. Value of the active zero offset (coarse absolute G159 + fine absolute G159 + incremental G158).

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.ORG.Z	Z axis.
V.A.ORG.3	Axis with logic number ·3·.
V.[2].A.ORG.3	Axis with index ·3· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to user tables (zero offset table).

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to user tables (zero offset table).

(V.)[ch].A.ADDORG.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

xn axis. Value of the active incremental zero offset (G158).

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.ADDORG.Z	Z axis.
V.A.ADDORG.3	Axis with logic number ·3·.
V.[2].A.ADDORG.3	Axis with index ·3· in the channel ·2·.

(V.)[ch].A.COARSEORG.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

xn axis. Value of the active absolute zero offset (G159), coarse part.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.COARSEORG.Z	Z axis.
V.A.COARSEORG.3	Axis with logic number ·3·.
V.[2].A.COARSEORG.3	Axis with index ·3· in the channel ·2·.

(V.)[ch].A.FINEORG.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

xn axis. Value of the active absolute zero offset (G159), fine part.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.FINEORG.Z	Z axis.
V.A.FINEORG.3	Axis with logic number ·3·.
V.[2].A.FINEORG.3	Axis with index ·3· in the channel ·2·.

(V.)[ch].A.ORG[nb].xn

Variable that can be read and written from the program and from the PLC and read from the interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

xn axis. Offset set in the zero offset [nb]; coarse part plus fine part.

When writing this variable, the value is assigned to the coarse part deleting the fine part.

Syntax.

·ch· Channel number.

·nb· Zero offset number.



CNC 8060
CNC 8065

(REF: 1402)

·xn· Name, logic number or index of the axis.

V.A. ORGT[1].Z	Zero offset G54 (G159=1). Z axis.
V.A. ORGT[4]0.3	Zero offset G57 (G159=4). Axis with logic number ·3·.
V.[2].A. ORGT[9].3	Zero offset G159=9. Axis with index ·3· in the channel ·2·.

Remarks.

The numbering of zero offsets G54 through G59 is always the same; G54=1, G55=2, G56=3, G57=4, G58=5, G59=6.

(V.)[ch].A.COARSEORGT[nb].xn

Variable that can be read and written from the program and from the PLC and read from the interface. Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

xn axis. Offset set in the zero offset [nb]; coarse part.

Syntax.

·ch· Channel number.

·nb· Zero offset number.

·xn· Name, logic number or index of the axis.

V.A. COARSEORGT[1].Z	Zero offset G54 (G159=1). Z axis.
V.A. COARSEORGT[4].3	Zero offset G57 (G159=4). Axis with logic number ·3·.
V.[2].A. COARSEORGT[9].3	Zero offset G159=9. Axis with index ·3· in the channel ·2·.

(V.)[ch].A.FINEORGT[nb].xn

Variable that can be read and written from the program and from the PLC and read from the interface. Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

xn axis. Offset set in the zero offset [nb]; fine part.

Syntax.

·ch· Channel number.

·nb· Zero offset number.

·xn· Name, logic number or index of the axis.

V.A. FINEORGT[1].Z	Zero offset G54 (G159=1). Z axis.
V.A. FINEORGT[4].3	Zero offset G57 (G159=4). Axis with logic number ·3·.
V.[2].A. FINEORGT[9].3	Zero offset G159=9. Axis with index ·3· in the channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

22.39 Variables related to user tables (fixture table).

22.

CNC VARIABLES.

Variables related to user tables (fixture table).

FIXTURE'S OFFSETS			
Fixture	X (mm)	Y (mm)	Z (mm)
1	00000.000	00000.000	00000.000
2	00000.000	00000.000	00000.000
3	00000.000	00000.000	00000.000
4	00000.000	00000.000	00000.000
5	00000.000	00000.000	00000.000
6	00000.000	00000.000	00000.000
7	00000.000	00000.000	00000.000
8	00000.000	00000.000	00000.000

V.G.FFIX points to row 1.

V.A.FIXT[4].Y points to row 4, column Y.

V.G.NUMFIX points to row 8.

(V.)G.FFIX

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

First fixture of the table.

V.G.FFIX

(V.)G.NUMFIX

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of fixtures in the table.

V.G.NUMFIX

(V.)[ch].G.FIX

Variable that can be read and written from the program and read from the PLC and from the interface.
The variable returns the value of block preparation.
Report variable (to be used from the scripts).

Number of current fixture.

Syntax.

·ch· Channel number.

V.[2].G.FIX

Channel ·2·. Number of current fixture.

(V.)[ch].A.FIX.xn

Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes.
The variable returns the value of block preparation.

xn axis. Offset set in the current fixture.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.FIX.Z

Z axis.

V.A.FIX.3

Axis with logic number ·3·.

V.[2].A.FIX.3

Axis with index ·3· in the channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].A.FIXT[nb].xn

Variable that can be read and written from the program and from the PLC and read from the interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

xn axis. Offset set in the [nb] fixture.

Syntax.

- ch· Channel number.
- nb· Fixture offset number.
- xn· Name, logic number or index of the axis.

V.A.FIXT[1].Z	First zero offset. Z axis.
V.A.FIXT[4].3	Fourth zero offset. Axis with logic number ·3·.
V.[2].A.FIXT[9].3	Ninth zero offset. Axis with index ·3· in the channel ·2·.

22.40 Variables related to user tables (arithmetic parameters table).

LOCAL ARITHMETIC PARAMETERS.

(V.)[ch].G.LUPACT[nb]

Variable that can be read and written via PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Value of local arithmetic parameter [nb] of the current nesting level.

Syntax.

·ch· Channel number.

·nb· Parameter number.

[2].G.LUPACT[14] Channel ·2·. Value of parameter ·14·.

(V.)[ch].G.LUP1[nb]

(V.)[ch].G.LUP2[nb]

(V.)[ch].G.LUP3[nb]

(V.)[ch].G.LUP4[nb]

(V.)[ch].G.LUP5[nb]

(V.)[ch].G.LUP6[nb]

(V.)[ch].G.LUP7[nb]

Variable that can be read and written via PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Value of local arithmetic parameter [nb] of nesting level 1 through 7.

Syntax.

·ch· Channel number.

·nb· Parameter number.

[2].G.LUP1[14] Channel ·2·. Value of parameter ·14· of nesting level ·1·.

[2].G.LUP7[6] Channel ·2·. Value of parameter ·6· of nesting level ·7·.

Remarks.

When reading this variable from the PLC, it truncates the decimal side. If the value of the parameter is 54.9876, when read from the PLC, it will return the value 54.

Value of the parameter.	Reading from the PLC.
P14 = 23.1234	G.LUP1[14] = 23 G.LUP1F[14] = 231234
P22 = -12.0987	G.LUP1[22] = -12 G.LUP1F[22] = -120987



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].G.LUP1F[nb]
 (V.)[ch].G.LUP2F[nb]
 (V.)[ch].G.LUP3F[nb]
 (V.)[ch].G.LUP4F[nb]
 (V.)[ch].G.LUP5F[nb]
 (V.)[ch].G.LUP6F[nb]
 (V.)[ch].G.LUP7F[nb]

Variable that can be read and written via PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Value of local arithmetic parameter [nb] of nesting level 1 through 7 (value x10000).

Syntax.

- ch· Channel number.
- nb· Parameter number.

[2].G.LUP1F[14] Channel ·2·. Value of parameter ·14· of nesting level ·1·.
 [2].G.LUP7F[6] Channel ·2·. Value of parameter ·6· of nesting level ·7·.

Remarks.

Reading these variables from the PLC returns the parameter value multiplied by 10000. If the value of the parameter is 54.9876, when read from the PLC, it will return the value 549876.

Value of the parameter.	Reading from the PLC.
P14 = 23.1234	G.LUP1[14] = 23 G.LUP1F[14] = 231234
P22 = -12.0987	G.LUP1[22] = -12 G.LUP1F[22] = -120987

GLOBAL ARITHMETIC PARAMETERS.

(V.)[ch].G.GUP[nb]

Variable that can be read and written via PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Value of the global arithmetic parameter [nb].

Syntax.

- ch· Channel number.
- nb· Parameter number.

[2].G.GUP[114] Channel ·2·. Value of parameter ·114·.

Remarks.

When reading this variable from the PLC, it truncates the decimal side. If the value of the parameter is 54.9876, when read from the PLC, it will return the value 54.

Value of the parameter.	Reading the variable.
P114 = 124.4567	G.GUP[114] = 124 G.GUPF[114] = 1244567
P200 = -12.0987	G.GUP[200] = -12 G.GUPF[200] = -120987



CNC 8060
 CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to user tables (arithmetic parameters table).

(V.)[ch].G.GUPF[nb]

Variable that can be read and written via PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Value of the global arithmetic parameter [nb] (value x10000).

Syntax.

- ch· Channel number.
- nb· Parameter number.

[2].G.GUP[114] Channel -2-. Value of parameter -114-.

Remarks.

Reading these variables from the PLC returns the parameter value multiplied by 10000. If the value of the parameter is 54.9876, when read from the PLC, it will return the value 549876.

Value of the parameter.	Reading the variable.
P114 = 124.4567	G.GUP[114] = 124 G.GUPF[114] = 1244567
P200 = -12.0987	G.GUP[200] = -12 G.GUPF[200] = -120987

COMMON ARITHMETIC PARAMETERS.

(V.)G.CUP[nb]

Variable that can be read and written via PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Value of the common arithmetic parameter [nb].

Syntax.

- nb· Parameter number.

[2].G.CUP[10014] Channel -2-. Value of parameter -10014-.

Remarks.

When reading this variable from the PLC, it truncates the decimal side. If the value of the parameter is 54.9876, when read from the PLC, it will return the value 54.

Value of the parameter.	Reading the variable.
P10014 = 124.4567	G.CUP[10014] = 124 G.CUPF[10014] = 1244567
P10200 = -12.0987	G.CUP[10200] = -12 G.CUPF[10200] = -120987

(V.)G.CUPF[nb]

Variable that can be read and written via PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Value of the common arithmetic parameter [nb] (value x10000).

Syntax.

- nb· Parameter number.

[2].G.CUPF[10014] Channel -2-. Value of parameter -10014-.



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

Reading these variables from the PLC returns the parameter value multiplied by 10000. If the value of the parameter is 54.9876, when read from the PLC, it will return the value 549876.

Value of the parameter.	Reading the variable.
P10014 = 124.4567	G.CUP[10014] = 124 G.CUPF[10014] = 1244567
P10200 = -12.0987	G.CUP[10200] = -12 G.CUPF[10200] = -120987

22.**CNC VARIABLES.**

Variables related to user tables (arithmetic parameters table).



CNC 8060
CNC 8065

(REF: 1402)

22.41 Variables related to the position of the axes.

There are real and theoretical coordinates corresponding to the tool base and tool tip. All of them may be referred to Machine Zero or to the current Part Zero.

A theoretical coordinate is the position that the axis must occupy at all times, a real coordinate is the one it actually occupies and the difference between these two is called "following error".

22.

CNC VARIABLES.
Variables related to the position of the axes.

PROGRAMMED COORDINATES.

(V.)[ch].A.PPOS.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

Channel [ch]. Programmed part coordinates (tool tip).

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.PPOS.Z	Z axis.
V.A.PPOS.3	Axis with logic number ·3·.
V.[2].A.PPOS.3	Axis with index ·3· in the channel ·2·.

Remarks.

This variable returns the target coordinate, in part coordinates and referred to the tool tip, in the current reference system; i.e. taking into consideration the coordinate rotation, scaling factor, active inclined plane, etc.

Programmed movements.	Returned values.
G1 X10	V.A.PPOS.X = 10
#SCALE [2] G1 X10	Scaling factor of ·2·. V.A.PPOS.X = 20
G73 Q90 G1 X10	Coordinate system rotation. V.A.PPOS.Y = 20 The Y axis is the one that moves

The values read from a program or from the PLC and the interface will be different when the coordinate is affected by tool compensation or when machining in round corner mode. The value read by program will be the programmed coordinate whereas the value read from the PLC or interface will be the real (actual) coordinate considering tool radius compensation and corner rounding.

(V.)[ch].G.PLPPOS1

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

Channel [ch]. Coordinate programmed in the first axis of the channel (tool tip).

Syntax.

·ch· Channel number.

V.[2].G.PLPPOS1	Channel ·2·.
-----------------	--------------



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].G.PLPPOS2

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

Channel [ch]. Coordinate programmed in the second axis of the channel (tool tip).

Syntax.

·ch· Channel number.

V.[2].G.PLPPOS2	Channel -2·.
-----------------	--------------

(V.)[ch].G.PLPPOS3

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

Channel [ch]. Coordinate programmed in the third axis of the channel (tool tip).

Syntax.

·ch· Channel number.

V.[2].G.PLPPOS3	Channel -2·.
-----------------	--------------

POSITION IN PART COORDINATES.

(V.)[ch].A.APOS.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Real part coordinates of the tool base.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.APOS.Z	Z axis.
V.A.APOS.3	Axis with logic number -3·.
V.[2].A.APOS.3	Axis with index -3· in the channel -2·.

(V.)[ch].A.ATPOS.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Theoretical part coordinates of the tool base.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.ATPOS.Z	Z axis.
V.A.ATPOS.3	Axis with logic number -3·.
V.[2].A.ATPOS.3	Axis with index -3· in the channel -2·.

22.

CNC VARIABLES.
Variables related to the position of the axes.

(V.)[ch].A.ATIPPOS.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Real part coordinates of the tool tip.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.ATIPPOS.Z	Z axis.
V.A.ATIPPOS.3	Axis with logic number ·3·.
V.[2].A.ATIPPOS.3	Axis with index ·3· in the channel ·2·.

(V.)[ch].A.ATIPTPOS.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Theoretical part coordinates of the tool tip.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.ATIPTPOS.Z	Z axis.
V.A.ATIPTPOS.3	Axis with logic number ·3·.
V.[2].A.ATIPTPOS.3	Axis with index ·3· in the channel ·2·.

POSITION IN MACHINE COORDINATES.**(V.)[ch].A.POS.xn**

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Real machine coordinates of the tool base.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.POS.Z	Z axis.
V.A.POS.3	Axis with logic number ·3·.
V.[2].A.POS.3	Axis with index ·3· in the channel ·2·.

(V.)[ch].A.TPOS.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Theoretical machine coordinates of the tool base.

Syntax.

·ch· Channel number.



CNC 8060
CNC 8065

(REF: 1402)

·xn· Name, logic number or index of the axis.

V.A.TPOS.Z	Z axis.
V.A.TPOS.3	Axis with logic number ·3·.
V.[2].A.TPOS.3	Axis with index ·3· in the channel ·2·.

(V.)[ch].A.TIPPOS.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Real machine coordinates of the tool tip.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.TIPPOS.Z	Z axis.
V.A.TIPPOS.3	Axis with logic number ·3·.
V.[2].A.TIPPOS.3	Axis with index ·3· in the channel ·2·.

(V.)[ch].A.TIPTPOS.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Theoretical machine coordinates of the tool tip.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.TIPTPOS.Z	Z axis.
V.A.TIPTPOS.3	Axis with logic number ·3·.
V.[2].A.TIPTPOS.3	Axis with index ·3· in the channel ·2·.

POSITION READ IN THE INTERNAL FEEDBACK (MOTOR FEEDBACK).

(V.)[ch].A.POSMOTOR.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Value of the internal feedback (motor feedback).

When the combined feedback is active (parameter FB MIXTIME), the variable returns the value of the internal feedback (motor feedback). If there is no feedback combination, the value of the variable coincides with (V.)[ch].A.POS.xn.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.POSMOTOR.Z	Z axis.
V.A.POSMOTOR.3	Axis with logic number ·3·.
V.[2].A.POSMOTOR.3	Axis with index ·3· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to the position of the axes.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

FOLLOWING ERROR OF THE AXIS.

(V.)[ch].A.FLWE.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Following error, lag (difference between the theoretical position value and the actual, real, position value of the axis).

If there is no feedback combination, the CNC reads the actual (real) position from either the internal or the external feedback (parameter FBACKSRC). If feedback combination is active (parameter FB MIXTIME), the CNC calculates the following error (lag) using the combined feedback.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.FLWE.Z	Z axis.
V.A.FLWE.3	Axis with logic number ·3·.
V.[2].A.FLWE.3	Axis with index ·3· in the channel ·2·.

(V.)[ch].A.FLWACT.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Following error (lag) with respect to the external feedback (direct feedback).

When the combined feedback is active (parameter FB MIXTIME), the variable returns the following error (axis lag) with respect to the external feedback (direct feedback). If there is no feedback combination, the variable returns the following error (lag) with respect to the active feedback (parameter FBACKSRC), same as the variable "(V.)[ch].A.FLWE.xn".

Active feedback.	V.A.FLWE.xn	V.A.FLWACT.xn
Internal	Following error (lag) with respect to the internal feedback (motor feedback).	
External	Following error (lag) with respect to the external feedback (direct feedback).	
Feedback combination.	Following error (lag) with respect to the combined feedback.	Following error (lag) with respect to the external feedback.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.FLWACT.Z	Z axis.
V.A.FLWACT.3	Axis with logic number ·3·.
V.[2].A.FLWACT.3	Axis with index ·3· in the channel ·2·.

(V.)[ch].A.FLWEST.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Linear estimate of the following error (lag).

Syntax.

·ch· Channel number.

22.

CNC VARIABLES.
Variables related to the position of the axes.



CNC 8060
CNC 8065

(REF: 1402)

·xn· Name, logic number or index of the axis.

V.A.FLWE.Z	Z axis.
V.A.FLWE.3	Axis with logic number ·3·.
V.[2].A.FLWE.3	Axis with index ·3· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to the position of the axes.



CNC 8060
CNC 8065

(REF: 1402)

22.42 Variables related to spindle position.

The following are used when the spindle is working in closed loop, it behaves like an axis. A theoretical coordinate is the position that the spindle must occupy at all times, a real coordinate is the one it actually occupies and the difference between these two is called "following error".

SPINDLE POSITION.

(V.)[ch].A.PPOS.sn

(V.)[ch].SP.PPOS.sn

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Programmed spindle position.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.PPOS.S	Spindle S.
V.SP.PPOS.S	Spindle S.
V.SP.PPOS	Master spindle.
V.A.PPOS.5	Spindle with logic number ·5·.
V.SP.PPOS.2	Spindle with index ·2· in the system.
V.[2].SP.PPOS.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.POS.sn

(V.)[ch].SP.POS.sn

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Real spindle position.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.POS.S	Spindle S.
V.SP.POS.S	Spindle S.
V.SP.POS	Master spindle.
V.A.POS.5	Spindle with logic number ·5·.
V.SP.POS.2	Spindle with index ·2· in the system.
V.[2].SP.POS.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.TPOS.sn

(V.)[ch].SP.TPOS.sn

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Theoretical spindle position.

Syntax.

·ch· Channel number.

22.

CNC VARIABLES.
Variables related to spindle position.



CNC 8060
CNC 8065

(REF: 1402)

.sn Name, logic number or index of the spindle.

V.A.TPOS.S	Spindle S.
V.SP.TPOS.S	Spindle S.
V.SP.TPOS	Master spindle.
V.A.TPOS.5	Spindle with logic number .5 .
V.SP.TPOS.2	Spindle with index .2 in the system.
V.[2].SP.TPOS.1	Spindle with index .1 in the channel .2 .

SPINDLE FOLLOWING ERROR.

(V.)[ch].A.FLWE.sn
(V.)[ch].SP.FLWE.sn

*Variable that can only be read from the program, PLC and interface.
 The variable returns the execution value; reading it interrupts block preparation.*

Channel [ch]. Spindle following error.

Syntax.

.ch Channel number.
.sn Name, logic number or index of the spindle.

V.A.FLWE.S	Spindle S.
V.SP.FLWE.S	Spindle S.
V.SP.FLWE	Master spindle.
V.A.FLWE.5	Spindle with logic number .5 .
V.SP.FLWE.2	Spindle with index .2 in the system.
V.[2].SP.FLWE.1	Spindle with index .1 in the channel .2 .

(V.)[ch].A.FLWEST.sn
(V.)[ch].SP.FLWEST.sn

*Variable that can only be read from the program, PLC and interface.
 The variable returns the execution value; reading it interrupts block preparation.*

Channel [ch]. Linear estimate of the spindle following error (lag).

Syntax.

.ch Channel number.
.sn Name, logic number or index of the spindle.

V.A.FLWEST.S	Spindle S.
V.SP.FLWEST.S	Spindle S.
V.SP.FLWEST.S	Master spindle.
V.A.FLWEST.5	Spindle with logic number .5 .
V.SP.FLWEST.2	Spindle with index .2 in the system.
V.[2].SP.FLWEST.1	Spindle with index .1 in the channel .2 .



22.43 Feedrate related variables.

FEEDRATE ACTIVE IN THE CHANNEL.

(V.) [ch].G.FREAL

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Real feedrate on the tool path.

Use the oscilloscope to compare the real and theoretical feedrates along the path and detect adjustment problems when both are different at specific points. Also, using variables V.G.LINEN and V.G.BLKN, it is possible to associate these feedrate changes with the program blocks or lines where they occur.

Syntax.

·ch· Channel number.

V.[2].G.FREAL	Channel ·2·.
---------------	--------------

Remarks.

This variable takes into account the accelerations and decelerations of the machine. When the axes are stopped, it returns a value of ·0· and when moving it returns the value corresponding to the feedrate type G94/G95. On laser cutting machines, it is recommended to use this variable so the laser power is proportional to the feedrate.

(V.) [ch].G.PATHFEED

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Theoretical feedrate on the tool path.

Use the oscilloscope to compare the real and theoretical feedrates along the path and detect adjustment problems when both are different at specific points. Also, using variables V.G.LINEN and V.G.BLKN, it is possible to associate these feedrate changes with the program blocks or lines where they occur.

Syntax.

·ch· Channel number.

V.[2].G.PATHFEED	Channel ·2·.
------------------	--------------

(V.) [ch].PLC.G00FEED

Variable that can be read and written from the PLC and read from the program and from the interface.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Maximum feedrate permitted in the channel.

This variable may be used to limit any time, in real time, the maximum feedrate in the channel for any type of movement (G00, G01, etc). The CNC assumes the change immediately and stays active until the variable takes a value of ·0· restoring the limit set by machine parameter.

Syntax.

·ch· Channel number.

V.[2].PLC.G00FEED	Channel ·2·.
-------------------	--------------

22.

CNC VARIABLES.
Feedrate related variables.



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

Regardless of the value assigned to this variable, the feedrate will never exceed the limits set by parameters G00FEED and MAXFEED.

Variable.	Machine parameters.		Feedrate active in the channel.	
	G00FEED	MAXFEED	G00	G01, G02, ...
(V.)[ch].PLC.G00FEED	3000	5000	3000	3000
3000	10000	5000	3000	3000
7000	10000	5000	7000	5000
12000	10000	5000	10000	5000

PROGRAMMING THE FEEDRATE IN G94.

(V.) [ch].G.FEED

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Feedrate active in G94.

Syntax.

.ch- Channel number.

```
V.[2].G.FEED Channel -2.
```

Remarks.

The feedrate in G94 may be set by program or by PLC; the one set by PLC has the highest priority.

Feedrates set.	(V.)[ch].PLC.F	(V.)[ch].G.PRGF	(V.)[ch].G.FEED
By program; F2000. By PLC; none.	0	2000	2000
By program; F2000. By PLC; F4000.	4000	2000	4000
By program; F2000. By PLC; F500.	500	2000	500
By program; F2000. By PLC; none. By MDI; F3000.	0	3000	3000
By program; F2000. By PLC; F6000. By MDI; F3000.	6000	3000	6000
By program; F2000. By PLC; F500. By MDI; F3000.	500	3000	500

(V.) [ch].PLC.F

Variable that can be read and written from the PLC and read from the program and from the interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Feedrate by PLC in G94.

The feedrate programmed by PLC prevails over the one programmed by program or MDI. To cancel the feedrate by PLC, set the variable to -0.; the CNC applies the feedrate active by program.

Syntax.

.ch- Channel number.

```
V.[2].PLC.F Channel -2.
```



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Feedrate related variables.

(V.)[ch].G.PRGF

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Feedrate by program in G94.

Being G94 active, programming a new feedrate in MDI mode updates the value of this variable.

Syntax.

·ch· Channel number.

V.[2].G.PRGF Channel -2.

PROGRAMMING THE FEEDRATE IN G95.

(V.)[ch].G.FPREV

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Feedrate active in G95.

Syntax.

·ch· Channel number.

V.[2].G.FPREV Channel -2.

Remarks.

The feedrate in G95 may be set by program or by PLC; the one set by PLC has the highest priority.

Feedrates set.	(V.)[ch].PLC.FPR	(V.)[ch].G.PRGFPR	(V.)[ch].G.FPREV
By program; F0.5. By PLC; none.	0	0.5	0.5
By program; F0.5. By PLC; F0.7.	0.7	0.5	0.7
By program; F0.5. By PLC; F0.12.	0.12	0.5	0.12
By program; F0.5. By PLC; none. By MDI; F1.8.	0	1.8	1.8
By program; F0.5. By PLC; F2.5. By MDI; F1.8.	2.5	1.8	2.5
By program; F0.5. By PLC; F0.7. By MDI; F1.8.	0.7	1.8	0.7

(V.)[ch].PLC.FPR

Variable that can be read and written from the PLC and read from the program and from the interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Feedrate by PLC in G95.

The feedrate programmed by PLC prevails over the one programmed by program or MDI.
To cancel the feedrate by PLC, set the variable to -0.; the CNC applies the feedrate active by program.

Syntax.

·ch· Channel number.

V.[2].PLC.FPR Channel -2.

(V.)[ch].G.PRGFPR

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Feedrate by program in G95.

Being G95 active, programming a new feedrate in MDI mode updates the value of this variable.

Syntax.

.ch· Channel number.

V.[2].G.PRGFPR Channel -2.

PROGRAMMING THE MACHINING TIME.

(V.)[ch].G.FTIME

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Machining time in G93 (in seconds).

Syntax.

.ch· Channel number.

V.[2].G.FTIME Channel -2.

PERCENTAGE OF FEEDRATE (FEEDRATE OVERRIDE).

(V.)[ch].G.FRO

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

Channel [ch]. Feedrate percentage active in the channel.

Syntax.

.ch· Channel number.

V.[2].G.FRO Channel -2.

Remarks.

The Feedrate override percentage may be set by program, by PLC or with the selector switch; the one set by program has the highest priority and the one selected with the switch the lowest.

(V.)[ch].G.PRGFRO	(V.)[ch].PLC.FRO	(V.)[ch].G.CNCFRO	(V.)[ch].G.FRO
0	0	70 %	70 %
0	40 %	70 %	40 %
85 %	40 %	70 %	85 %
20 %	90 %	70 %	20 %
20 %	0	70 %	20 %

(V.)[ch].G.PRGFRO

Variable that can be read and written from the program and read from the PLC and from the interface.
The variable returns the value of block preparation.

Channel [ch]. Percentage of feedrate by program.

22.

CNC VARIABLES.
Feedrate related variables.



CNC 8060
CNC 8065

(REF: 1402)

The percentage set by program has higher priority than the one set by PLC or by the switch. To cancel the value set by program, set the variable to -0-.

Syntax.

·ch· Channel number.

V.[2].G.PRGFRO Channel -2-.

(V.)[ch].PLC.FRO

Variable that can be read and written from the PLC and read from the program and from the interface. The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Percentage of feedrate by PLC.

The percentage set by PLC has higher priority than the one set by the switch, but lower priority than the one set by program.. To cancel the value set by PLC, set the variable to -0-. To set a 0% value set by PLC, set the variable to -1.

Syntax.

·ch· Channel number.

V.[2].PLC.FRO Channel -2-.

(V.)[ch].G.CNCFRO

Variable that can be read and written (asynchronous writing) from the interface and read from the program and from the PLC.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Feedrate percentage at the switch of the operator panel.

The percentage set by the switch of the operator panel has lower priority than the one set by PLC or by program.

Syntax.

·ch· Channel number.

V.[2].G.CNCFRO Channel -2-.

22.

CNC VARIABLES.
Feedrate related variables.



CNC 8060
CNC 8065

(REF: 1402)

22.44 Variables associated with acceleration and jerk on the tool path.

(V.)[ch].G.MAXACCEL

Variable that can be read and written via program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Modify the maximum acceleration allowed on the machining path.

The maximum acceleration allowed on the machining path is set by parameter MAXACCEL and this variable may be used to change this value temporarily at a particular time and in real time. The CNC assumes the change immediately and stays active until M30 or reset is executed, in which case the CNC restores the value set by the machine parameter. If the variable takes the value of ·0·, the CNC does not apply any acceleration limit on the tool path, not even the one set by machine parameter.

The CNC always respects the dynamics of the axes involved in the path; i.e. regardless of the maximum acceleration allowed on the path, the movement will respect the acceleration set for each axis.

Syntax.

·ch· Channel number.

V.[2].G.MAXACCEL	Channel ·2·
------------------	-------------

(V.)[ch].G.MAXJERK

Variable that can be read and written via program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Modify the maximum jerk on the machining path.

The maximum jerk allowed on the machining path is set by parameter MAXJERK and this variable may be used to change this value temporarily at a particular time and in real time. The CNC assumes the change immediately and stays active until M30 or reset is executed, in which case the CNC restores the value set by the machine parameter. If the variable takes the value of ·0·, the CNC does not apply any jerk limit on the tool path, not even the one set by machine parameter.

The CNC always respects the dynamics of the axes involved in the path; i.e. regardless of the maximum jerk allowed on the path, the movement will respect the jerk set for each axis.

Syntax.

·ch· Channel number.

V.[2].G.MAXJERK	Channel ·2·
-----------------	-------------

22.

CNC VARIABLES.

Variables associated with acceleration and jerk on the tool path.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.45 Variables related to managing the feedrate in HSC mode.

VARIABLES RELATED TO BLOCK PREPARATION.

(V.)[ch].G.PERFRATE

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Percentage of blocks managed by the CNC in block preparation, with respect to the best possible to reach the maximum feedrate in each section.

Syntax.

·ch· Channel number.

V.[2].G.PERFRATE	Channel ·2·.
------------------	--------------

Remarks.

This variable can return a value close to 100; if the percentage of blocks is lower than 100%, the HSC might be losing room to increase the feedrate, respecting the available room to brake. To know if this is the case, it will be necessary to analyze the V.G.DROPRATE variable because the reason the feedrate is not increasing might be geometrical and not due to the number of blocks available.

(V.)[ch].G.DROPRATE

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Percentage of reduction of the maximum feedrate possible.

Feedrate reduction may be due to an improper supply of blocks or because a feedrate lower than possible has been programmed.

Syntax.

·ch· Channel number.

V.[2].G.DROPRATE	Channel ·2·.
------------------	--------------

Remarks.

This variable can return a value close to 100; if the value is lower than 100%, the CNC may increase the feedrate if it has more blocks. To know whether the CNC can supply more blocks or not, it will be necessary to analyze the V.G.PERFRATE variable.

If both variables have a value lower than 100, parameter PREPFREQ may be increased to increase the supply of blocks as long as the system has enough time; in other words, the percentage of cycle time used by the CNC is not too close to the total cycle time (about 50%). This information may be checked in the V.G.NCTIMERATE variable.

ANALYSIS OF THE PROGRAMMED ERROR.

(V.)[ch].G.LIMERROR

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Value of the error that cancels the splines (CONTERROR mode).

This variable may be used to evaluate whether the program is generated with more error than we demand from the HSC mode or not.

22.

CNC VARIABLES.
Variables related to managing the feedrate in HSC mode.



CNC 8060
CNC 8065

(REF: 1402)

Syntax.

.ch· Channel number.

V.[2].G.LIMERROR Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
-1	The generated profile does not exceed the programmed error. The programmed error does not limit the maximum feedrate of the axis.
###	Value of the error that cancels the splines (CONERROR mode).

(V.)[ch].G.ACTROUND

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Error programmed in HSC mode.

This variable may be used to optimize the OEM subroutines by cancelling the HSC mode and activating G5 with another error to save time and restore the previous execution mode later one.

Syntax.

.ch· Channel number.

V.[2].G.ACTROUND Channel -2.

FEEDRATE LIMITATION AT THE BLOCK BEING EXECUTED.**(V.)[ch].G.AXLIMF**

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Logic number of the axis that limits the feedrate at the block being executed.

Together with the V.G.PARLIMF variable, it may be used to evaluate the behavior of the machining operation in a particular section where the feedrate decreases too much or is irregular.

Syntax.

.ch· Channel number.

V.[2].G.AXLIMF Channel -2.

(V.)[ch].G.PARLIMF

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Cause that limits the feedrate at the block being executed.

Syntax.

.ch· Channel number.

V.[2].G.PARLIMF Channel -2.

22.**CNC VARIABLES.**

Variables related to managing the feedrate in HSC mode.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to managing the feedrate in HSC mode.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Maximum feedrate of the axis.
2	Acceleration due to curvature (parameter CURVACC).
3	Jerk due to curvature (parameter CURVJERK).
6	Error committed by the spline.
7	Insufficient memory due to very small blocks and/or buffer full.
10	Maximum axis feedrate in transformations.
11	Maximum axis acceleration in transformations.

LIMITATION OF THE FEEDRATE AT THE CORNER.

(V.)[ch].G.AXLIMC

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Logic number of the axis that limits the feedrate at the corner, in the block being executed.

Together with the V.G.PARLIMC variable, it may be used to evaluate the behavior of the machining operation at a particular corner where the feedrate decreases too much or is irregular.

Syntax.

·ch· Channel number.

V.[2].G.AXLIMC Channel ·2·.

(V.)[ch].G.PARLIMC

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Cause that limits the feedrate at the corner in the block being executed.

Syntax.

·ch· Channel number.

V.[2].G.PARLIMC Channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Maximum feedrate of the axis.
4	The acceleration at the corner (parameter CORNERACC).
5	The jerk at the corner (parameter CORNERJERK).
8	The chordal error at the corner (CONTEERROR).
9	Geometry at the corner (FAST mode).



CNC 8060
CNC 8065

(REF: 1402)

MODIFY THE DYNAMICS OF ALL THE AXES OF THE CHANNEL.

(V.)[ch].G.DYNOVR

Variable that can be read and written via program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Percentage applicable to all the dynamics variables of all the axes of the channel.

This variable may be used to make changes in real time to the machining dynamics (ignoring look-ahead blocks). The dynamics can also be changed from the interface using the dynamic-override bar of the HSC.



The upper and lower limits (range) for varying the value depends on machine parameters MINDYNOVR and MAXDYNOVR.

Syntax.

·ch· Channel number.

V.[2].G.DYNOVR Channel ·2·.

22.

CNC VARIABLES.

Variables related to managing the feedrate in HSC mode.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.46 Variables related to spindle speed.

PROGRAMMING THE SPEED.

(V.)[ch].A.SREAL.sn
(V.)[ch].SP.SREAL.sn

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Real spindle speed.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.SREAL.S	Spindle S.
V.SP.SREAL.S	Spindle S.
V.SP.SREAL	Master spindle.
V.A.SREAL.5	Spindle with logic number ·5·.
V.SP.SREAL.2	Spindle with index ·2· in the system.
V.[2].SP.SREAL.1	Spindle with index ·1· in the channel ·2·.

Remarks.

This variable takes into account the accelerations and decelerations of the machine. It returns a value of ·0· when the spindle is stopped, it returns the value in rpm when the spindle is in G96/G97 and in degrees/minute when the spindle is in M19.

SPINDLE SPEED IN G97.

(V.)[ch].A.SPEED.sn
(V.)[ch].SP.SPEED.sn

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Active spindle speed in G97.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.SPEED.S	Spindle S.
V.SP.SPEED.S	Spindle S.
V.SP.SPEED	Master spindle.
V.A.SPEED.5	Spindle with logic number ·5·.
V.SP.SPEED.2	Spindle with index ·2· in the system.
V.[2].SP.SPEED.1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.
Variables related to spindle speed.



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

The speed may be set by program or by PLC; the one set by PLC has the highest priority.

Speed set.	V.PLC.S.sn	V.A.PRGS.sn	V.A.SPEED.sn
By program; S5000. By PLC; none.	0	5000	5000
By program; S5000. By PLC; S9000.	9000	5000	9000
By program; S5000. By PLC; S3000.	3000	5000	3000
By program; S5000. By PLC; none. By MDI; S8000.	0	8000	8000
By program; S5000. By PLC; S9000. By MDI; S8000.	9000	8000	9000
By program; S5000. By PLC; S3000. By MDI; S8000.	3000	8000	3000

(V.)[ch].PLC.S.sn

Variable that can be read and written from the PLC and read from the program and from the interface. The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Speed active by PLC for G97.

The speed programmed by PLC prevails over the one programmed by program or MDI. To cancel the speed by PLC, set the variable to -0-; the CNC applies the speed active by program.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.PLC.S.S2	Spindle S2.
V.PLC.S.5	Spindle with logic number -5-.
V.[2].PLC.S.1	Spindle with index -1- in the channel -2-.

(V.)[ch].A.PRGS.sn

(V.)[ch].SP.PRGS.sn

Variable that can only be read from the program, PLC and interface. The variable returns the value of block preparation.

Channel [ch]. Speed active by program for G97.

Being G97 active, programming a new speed in MDI mode updates the value of this variable.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.A.PRGS.S	Spindle S.
V.SP.PRGS.S	Spindle S.
V.SP.PRGS	Master spindle.
V.A.PRGS.5	Spindle with logic number -5-.
V.SP.PRGS.2	Spindle with index -2- in the system.
V.[2].SP.PRGS.1	Spindle with index -1- in the channel -2-.

22.

CNC VARIABLES.

Variables related to spindle speed.



CNC 8060
CNC 8065

(REF: 1402)

SPINDLE SPEED IN G96 (CSS).

(V.) [ch].A.CSS.sn
(V.) [ch].SP.CSS.sn

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Active spindle speed in G96.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.A.CSS.S	Spindle S.
V.SP.CSS.S	Spindle S.
V.SP.CSS	Master spindle.
V.A.CSS.5	Spindle with logic number -5-.
V.SP.CSS.2	Spindle with index -2- in the system.
V.[2].SP.CSS.1	Spindle with index -1- in the channel -2-.

Remarks.

The speed may be set by program or by PLC; the one set by PLC has the highest priority.

Speed set.	V.PLC.CSS.sn	V.A.PRGCSS.sn	V.A.CSS.sn
By program; S150. By PLC; none.	0	150	150
By program; S150. By PLC; S250.	250	150	250
By program; S150. By PLC; S100.	100	150	100
By program; S150. By PLC; none. By MDI; S300.	0	300	300
By program; S150. By PLC; S250. By MDI; S200.	250	200	250
By program; S150. By PLC; S100. By MDI; S200.	100	200	100

(V.) [ch].PLC.CSS.sn

Variable that can be read and written from the PLC and read from the program and from the interface.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Speed active by PLC for G96.

The speed programmed by PLC prevails over the one programmed by program or MDI. To cancel the speed by PLC, set the variable to -0-; the CNC applies the speed active by program.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.PLC.CSS.S2	Spindle S2.
V.PLC.CSS.5	Spindle with logic number -5-.
V.[2].PLC.CSS.1	Spindle with index -1- in the channel -2-.

22.

CNC VARIABLES.
Variables related to spindle speed.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].A.PRGCSS.sn
(V.)[ch].SP.PRGCSS.sn

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Speed active by program for G96.

Being G96 active, programming a new speed in MDI mode updates the value of this variable.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.A.PRGCSS.S	Spindle S.
V.SP.PRGCSS.S	Spindle S.
V.SP.PRGCSS	Master spindle.
V.A.PRGCSS.5	Spindle with logic number ·5·.
V.SP.PRGCSS.2	Spindle with index ·2· in the system.
V.[2].SP.PRGCSS.1	Spindle with index ·1· in the channel ·2·.

SPINDLE SPEED IN M19.

(V.)[ch].A.SPOS.sn
(V.)[ch].SP.SPOS.sn

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Active spindle speed in M19 .

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.A.SPOS.S	Spindle S.
V.SP.SPOS.S	Spindle S.
V.SP.SPOS	Master spindle.
V.A.SPOS.5	Spindle with logic number ·5·.
V.SP.SPOS.2	Spindle with index ·2· in the system.
V.[2].SP.SPOS.1	Spindle with index ·1· in the channel ·2·.

Remarks.

The speed may be set by program or by PLC; the one set by PLC has the highest priority.

Speed set.	V.PLC.SPOS.sn	V.A.PRGSPOS.sn	V.A.SPOS.sn
By program; S.POS=180. By PLC; none.	0	180	180
By program; S.POS=180. By PLC; S.POS=250.	250	180	250
By program; S.POS=180. By PLC; S.POS=90.	90	180	90
By program; S.POS=180. By PLC; none. By MDI; S.POS=200.	0	200	200
By program; S.POS=180. By PLC; S.POS=250. By MDI; S.POS=200.	250	200	250
By program; S.POS=180. By PLC; S.POS=100. By MDI; S.POS=200.	100	200	100

22.

CNC VARIABLES.
Variables related to spindle speed.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to spindle speed.

(V.)[ch].PLC.SPOS.sn

Variable that can be read and written from the PLC and read from the program and from the interface. The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Speed active by PLC for M19.

The speed programmed by PLC prevails over the one programmed by program or MDI. To cancel the speed by PLC, set the variable to -0.; the CNC applies the speed active by program.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.PLC.SPOS.S2	Spindle S2.
V.PLC.SPOS.5	Spindle with logic number -5.
V.[2].PLC.SPOS.1	Spindle with index -1 in the channel -2.

(V.)[ch].A.PRGSPOS.sn**(V.)[ch].SP.PRGSPOS.sn**

Variable that can only be read from the program, PLC and interface. The variable returns the value of block preparation.

Channel [ch]. Speed active by program for M19.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.A.PRGSPOS.S	Spindle S.
V.SP.PRGSPOS.S	Spindle S.
V.SP.PRGSPOS	Master spindle.
V.A.PRGSPOS.5	Spindle with logic number -5.
V.SP.PRGSPOS.2	Spindle with index -2 in the system.
V.[2].SP.PRGSPOS.1	Spindle with index -1 in the channel -2.

SPEED LIMIT.**(V.)[ch].A.SLIMIT.sn****(V.)[ch].SP.SLIMIT.sn**

Variable that can only be read from the program, PLC and interface. The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Active spindle speed limit.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.A.SLIMIT.S	Spindle S.
V.SP.SLIMIT.S	Spindle S.
V.SP.SLIMIT	Master spindle.
V.A.SLIMIT.5	Spindle with logic number -5.
V.SP.SLIMIT.2	Spindle with index -2 in the system.
V.[2].SP.SLIMIT.1	Spindle with index -1 in the channel -2.

Remarks.

The maximum speed may be set by program or by PLC; the one set by PLC has the highest priority.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].PLC.SL.sn

Variable that can be read and written from the PLC and read from the program and from the interface. The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Spindle speed limit active by PLC.

The speed programmed by PLC prevails over the one programmed by program or MDI. To cancel the speed by PLC, set the variable to -0.; the CNC applies the speed active by program.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.PLC.SL.S2	Spindle S2.
V.PLC.SL.5	Spindle with logic number -5.
V.[2].PLC.SL.1	Spindle with index -1- in the channel -2.

(V.)[ch].A.PRGS�.sn

(V.)[ch].SP.PRGS�.sn

Variable that can only be read from the program, PLC and interface. The variable returns the value of block preparation.

Channel [ch]. Spindle speed limit active by program.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.A.PRGS�.S	Spindle S.
V.SP.PRGS�.S	Spindle S.
V.SP.PRGS�	Master spindle.
V.A.PRGS�.5	Spindle with logic number -5.
V.SP.PRGS�.2	Spindle with index -2- in the system.
V.[2].SP.PRGS�.1	Spindle with index -1- in the channel -2.

PERCENTAGE OF SPINDLE SPEED (SPINDLE SPEED OVERRIDE).

(V.)[ch].A.SSO.sn

(V.)[ch].SP.SSO.sn

Variable that can only be read from the program, PLC and interface. The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Active spindle speed override.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.A.SSO.S	Spindle S.
V.SP.SSO.S	Spindle S.
V.SP.SSO	Master spindle.
V.A.SSO.5	Spindle with logic number -5.
V.SP.SSO.2	Spindle with index -2- in the system.
V.[2].SP.SSO.1	Spindle with index -1- in the channel -2.

22.

CNC VARIABLES.

Variables related to spindle speed.



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

The spindle speed override percentage may be set by program, by PLC or with the selector switch; the one set by program has the highest priority and the one selected with the switch the lowest.

V.A.PRGSO.sn V.SP.PRGSO.sn	V.PLC.SSO.sn	V.A.CNCSSO.sn V.SP.CNCSSO.sn	V.A.SSO.sn V.SP.SSO.sn
0	0	100 %	100 %
0	80 %	100 %	80 %
110 %	80%	100 %	110 %
70 %	80 %	100 %	70 %
70 %	0	100 %	70 %

(V.)[ch].A.PRGSO.sn
(V.)[ch].SP.PRGSO.sn

Variable that can be read and written from the program and read from the PLC and from the interface. The variable returns the value of block preparation.

Channel [ch]. Percentage of spindle speed by program.

The percentage set by program has higher priority than the one set by PLC or by the switch. To cancel the value set by program, set the variable to ·0·.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.A.PRGSO.S	Spindle S.
V.SP.PRGSO.S	Spindle S.
V.SP.PRGSO	Master spindle.
V.A.PRGSO.5	Spindle with logic number ·5·.
V.SP.PRGSO.2	Spindle with index ·2· in the system.
V.[2].SP.PRGSO.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].PLC.SSO.sn

Variable that can be read and written from the PLC and read from the program and from the interface. The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Percentage of spindle speed by PLC.

The percentage set by PLC has higher priority than the one set by the switch, but lower priority than the one set by program.. To cancel the value set by PLC, set the variable to ·0·.

Syntax.

- ch· Channel number.
- sn· Name, logic number or index of the spindle.

V.PLC.SSO.S2	Spindle S2.
V.PLC.SSO.5	Spindle with logic number ·5·.
V.[2].PLC.SSO.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.CNCSSO.sn
(V.)[ch].SP.CNCSSO.sn

Variable that can be read and written (asynchronous writing) from the interface and read from the program and from the PLC.

The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Spindle speed percentage at the switch of the operator panel.

The percentage set by the switch of the operator panel has lower priority than the one set by PLC or by program.

22.

CNC VARIABLES.
Variables related to spindle speed.



CNC 8060
CNC 8065

(REF: 1402)

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.CNCSSO.S	Spindle S.
V.SP.CNCSSO.S	Spindle S.
V.SP.CNCSSO	Master spindle.
V.A.CNCSSO.5	Spindle with logic number ·5·.
V.SP.CNCSSO.2	Spindle with index ·2· in the system.
V.[2].SP.CNCSSO.1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to spindle speed.



CNC 8060
CNC 8065

(REF: 1402)

22.47 Variables associated with the tool manager.

22.

CNC VARIABLES.
Variables associated with the tool manager.

(V.)[ch].TM.MZSTATUS

Variable that can be read from the PLC and from the interface.

Tool manager status.

Syntax.

·ch· Channel number.

[2].TM.MZSTATUS Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Normal.
1	Tool manager error.
2	Error at the tool manager; waiting to complete the maneuver in progress.
4	Tool manager emergency.

(V.)[ch].TM.MZRUN

Variable that can be read from the PLC and from the interface.

Tool manager running.

Syntax.

·ch· Channel number.

[2].TM.MZRUN Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	There is no maneuver in progress.
1	There is a maneuver in progress.

(V.)[ch].TM.MZWAIT

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Tool manager processing a maneuver.

Syntax.

·ch· Channel number.

V.[2].TM.MZWAIT Channel -2.

Remarks.

From software version V2.01 on, there is no need to use this variable in the subroutine associated with M06. The subroutine itself waits for the manager's maneuvers to finish and it does not interrupt block preparation.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No need to wait.
1	It has to wait.

(V.)[ch].TM.MZMODE

Variable that can be read and written from the program and interface and read from the PLC.
The variable returns the execution value; reading and writing it interrupt block preparation.

Operating mode of the tool manager.

This variable may be used to set the magazine in load or unload mode.

Syntax.

·ch· Channel number.

V.[2].TM.MZMODE Channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Regular work mode.
1	Magazine in load mode.
2	Magazine in unload mode.

22.

CNC VARIABLES.
Variables associated with the tool manager.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.48 Variables related to managing the tool magazine and the tool changer arm.

22.

CNC VARIABLES.

Variables related to managing the tool magazine and the tool changer arm.

RELATIONSHIP BETWEEN THE TOOL MAGAZINE AND THE CHANNEL.

(V.)[ch].TM.ACTUALMZ

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Tool Magazine being used by the [ch] channel.

Syntax.

·ch· Channel number.

V.[2].TM.ACTUALMZ	Channel ·2·.
-------------------	--------------

(V.)TM.MZACTUALCH[mz]

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Channel that is using tool magazine [mz].

Syntax.

·mz· Type of tool magazine.

V.TM.MZACTUALCH[2]	Magazine ·2·.
--------------------	---------------

TOOL LOCATION IN THE MAGAZINE.

(V.)TM.T[mz][pos]

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Tool in the [pos] position of the [mz] magazine.

Syntax.

·mz· Type of tool magazine.

·pos· Tool position in the magazine.

V.TM.T[2][15]	Magazine ·2·. Position ·15·.
---------------	------------------------------

Remarks.

If the variable is missing the offset number, the variable will then refer to the first one. This variable returns 0 if the tool is not in the magazine.

(V.)TM.P[mz][tl]

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Position of the [t] tool in the [mz] magazine.

Syntax.

·mz· Type of tool magazine.

·tl· Tool number.

V.TM.P[2][15]	Magazine ·2·. Position ·15·.
---------------	------------------------------



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

If the variable is missing the offset number, the variable will then refer to the first one.

TOOL LOCATION IN THE TOOL CHANGER ARM.

(V.)TM.TOOLCH1[mz]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

Tool in the first claw of the magazine arm [mz].

Syntax.

·mz· Type of tool magazine.

V.TM.TOOLCH1[2] Magazine ·2·.

(V.)TM.TOOLCH2[mz]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

Tool in the second claw of the magazine arm [mz].

Syntax.

·mz· Type of tool magazine.

V.TM.TOOLCH2[2] Magazine ·2·.

22.

CNC VARIABLES.
Variables related to managing the tool magazine and the tool changer arm.



CNC 8060
CNC 8065

(REF: 1402)

22.49 Variables related to the active tool and to the next one.

Variables related to the active tool are always read synchronously. The writing of these variables is always asynchronous, be it for the active tool or not.

TOOL AND ACTIVE OFFSET.

(V.) [ch]. TM.TOOL

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Number of the active tool.

Syntax.

·ch· Channel number.

V.[2].TM.TOOL	Channel -2-. Active tool.
---------------	---------------------------

(V.) [ch]. TM.TOD

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Number of the active tool offset.

Syntax.

·ch· Channel number.

V.[2].TM.TOD	Channel -2-. Active tool offset.
--------------	----------------------------------

NEXT TOOL AND TOOL OFFSET.

(V.) [ch]. TM.NXTOOL

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Number of the next tool.

The "next tool" is the one already selected but waiting to be activated by executing an M06.

Syntax.

·ch· Channel number.

V.[2].TM.NXTOOL	Channel -2-. Next tool.
-----------------	-------------------------

Remarks.

Although the variable has writing permission, the value of 0 (zero) cannot be written.

(V.) [ch]. TM.NXTOD

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Number of the next tool offset.

Syntax.

·ch· Channel number.

V.[2].TM.NXTOD	Channel -2-. Next tool offset.
----------------	--------------------------------

22.

CNC VARIABLES.

Variables related to the active tool and to the next one.



CNC 8060
CNC 8065

(REF: 1402)

STATUS OF THE ACTIVE TOOL.

(V.)[ch].TM.TSTATUS

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Active tool. Tool status.

Syntax.

·ch· Channel number.

V.[2].TM.TSTATUS Channel ·2·. Active tool.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Tool available.
1	Tool available.
2	Tool worn out.

FAMILY OF THE ACTIVE TOOL.

(V.)[ch].TM.TLFF

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Active tool. Family code.

Syntax.

·ch· Channel number.

V.[2].TM.TLFF Channel ·2·.

ACTIVE TOOL MONITORING.

(V.)[ch].TM.TOMON[ofd]

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the active tool. Type of tool life monitoring.

Syntax.

[ofd] Tool offset; if missing, the active tool offset.

·ch· Channel number.

V.[2].TM.TOMON Channel ·2·. Active tool offset.

V.[2].TM.TOMON[3] Channel ·2·. Tool offset ·3·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Without tool life monitoring.
1	Tool life monitoring; machining time.
2	Tool life monitoring; number of operations.

22.

CNC VARIABLES.

Variables related to the active tool and to the next one.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the active tool and to the next one.

(V.)[ch].TM.TLFN[ofd]

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the active tool. Nominal life.

Syntax.

[ofd] Tool offset; if missing, the active tool offset.

·ch· Channel number.

V.[2].TM.TLFN	Channel ·2·. Active tool offset.
V.[2].TM.TLFN[3]	Channel ·2·. Tool offset ·3·.

(V.)[ch].TM.TLFR[ofd]

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the active tool. Real (actual) life.

Syntax.

[ofd] Tool offset; if missing, the active tool offset.

·ch· Channel number.

V.[2].TM.TLFR	Channel ·2·. Active tool offset.
V.[2].TM.TLFR[3]	Channel ·2·. Tool offset ·3·.

(V.)[ch].TM.REMLIFE

Variable that can be read from the PLC and from the interface.

Active tool. Remaining life.

Syntax.

·ch· Channel number.

[2].TM.REMLIFE	Channel ·2·.
----------------	--------------

"CUSTOM" DATA OF THE ACTIVE TOOL.**(V.)[ch].TM.TOTP1****(V.)[ch].TM.TOTP2****(V.)[ch].TM.TOTP3****(V.)[ch].TM.TOTP4**

Variable that can be read and written via program, PLC(asynchronous writing) and interface.
The variable returns the execution value; reading it interrupts block preparation.

Active tool. Custom parameters.

Syntax.

·ch· Channel number.

V.[2].TM.TOTP1	Channel ·2·. Custom parameter ·1·.
V.[2].TM.TOTP2	Channel ·2·. Custom parameter ·2·.
V.[2].TM.TOTP3	Channel ·2·. Custom parameter ·3·.
V.[2].TM.TOTP4	Channel ·2·. Custom parameter ·4·.



CNC 8060
CNC 8065

(REF: 1402)

TOOL GEOMETRY.

(V.)[ch].TM.NUMOFD

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Active tool. Number of tool offsets.

Syntax.

.ch· Channel number.

V.[2].TM.NUMOFD Channel -2·.

(V.)[ch].TM.DTYPE[ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface.
The variable returns the execution value; reading it interrupts block preparation.

Active tool. Code of the tool offset type.

Syntax.

.ch· Channel number.

[ofd] Tool offset; if missing, the active tool offset.

V.[2].TM.DTYPE Channel -2·. Active tool offset.

V.[2].TM.DTYPE[3] Channel -2·. Tool offset -3·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Milling tool..
2	Drilling tool.
3	Surface milling tool.
4	Reaming tool.
5	Boring tool.
6	Threading tool.
7	Grooving or cut-off tool.
8	Turning tool.
9	Other.
10	Measuring probe.

(V.)[ch].TM.DSUBTYPE[ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface.
The variable returns the execution value; reading it interrupts block preparation.

Active tool. Code of the tool offset subtype.

Syntax.

.ch· Channel number.

[ofd] Tool offset; if missing, the active tool offset.

V.[2].TM.DSUBTYPE Channel -2·. Active tool offset.

V.[2].TM.DSUBTYPE[3] Channel -2·. Tool offset -3·.

22.

CNC VARIABLES.

Variables related to the active tool and to the next one.


















CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.	Value.	Meaning.
0	Without type.	8	Surface milling endmill. 
1	Flat endmill. 	9	Reamer. 
2	Ball endmill. 	10	Boring tool. 
3	Toric endmill. 	11	Diamond shaped cutter for turning. 
4	Tap. 	12	Square cutter for turning, grooving or cut-off. 
5	Thread cutter. 	13	Round cutter for turning. 
6	Disk endmill. 	14	Measuring probe (milling machine). 
7	Drill bit. 	15	Measuring probe (lathe). 

(V.)[ch].TM.TURNCONFIG[ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the active tool. Axis orientation.

Only for turning tools. The orientation of the axes is determined by the type of lathe (horizontal or vertical), the position of the turret and the spindle position (on the right or on the left).

Syntax.

·ch· Channel number.

[ofd] Tool offset; if missing, the active tool offset.

V.[2].TM.TURNCONFIG Channel ·2·. Active tool offset.

V.[2].TM.TURNCONFIG[3] Channel ·2·. Tool offset ·3·.

22.

CNC VARIABLES.

Variables related to the active tool and to the next one.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Axis orientation.	Value.	Axis orientation.
0		4	
1		5	
2		6	
3		7	

(V.)[ch].TM.LOCODE[ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the active tool. Location code (shape) or tool calibration point.

Only for turning tools. The location code indicates which is the calibrated tool tip and, therefore, the point controlled by the CNC to apply radius compensation. The location code depends on the orientation of the machine axes.

Syntax.

·ch· Channel number.

[ofd] Tool offset; if missing, the active tool offset.

V.[2].TM.LOCODE	Channel ·2·. Active tool offset.
V.[2].TM.LOCODE[3]	Channel ·2·. Tool offset ·3·.

Values of the variable.

This variable returns one of the following values.

Value.								
0								
1								
2								
3								
4								
5								
6								

22.

CNC VARIABLES.

Variables related to the active tool and to the next one.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.

Variables related to the active tool and to the next one.

Value.								
7								
8								
9								

(V.)[ch].TM.FIXORI[ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the active tool. Tool-holder orientation.

Syntax.

·ch· Channel number.

[ofd] Tool offset; if missing, the active tool offset.

V.[2].TM.FIXORI	Channel -2-. Active tool offset.
V.[2].TM.FIXORI[3]	Channel -2-. Tool offset -3-.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Face axis.
1	Longitudinal axis.

(V.)[ch].TM.SPDLTURDIR[ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the active tool. Spindle turning direction.

Syntax.

·ch· Channel number.

[ofd] Tool offset; if missing, the active tool offset.

V.[2].TM.SPDLTURDIR	Channel -2-. Active tool offset.
V.[2].TM.SPDLTURDIR[3]	Channel -2-. Tool offset -3-.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Undefined direction.
1	Right-hand machining direction.
2	Left-hand machining direction.

(V.)[ch].TM.TOR[ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the active tool. Radius.



CNC 8060
CNC 8065

(REF: 1402)

This variable is not valid for turning tools.

Syntax.

·ch· Channel number.

[odf] Tool offset; if missing, the active tool offset.

V.[2].TM.TOR	Channel ·2·. Active tool offset.
V.[2].TM.TOR[3]	Channel ·2·. Tool offset ·3·.

(V.)[ch].TM.TOI [odf]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the active tool. Radius wear.

This variable is not valid for turning tools.

Syntax.

·ch· Channel number.

[odf] Tool offset; if missing, the active tool offset.

V.[2].TM.TOI	Channel ·2·. Active tool offset.
V.[2].TM.TOI[3]	Channel ·2·. Tool offset ·3·.

(V.)[ch].TM.TOL [odf]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the active tool. Length.

This variable is not valid for turning tools.

Syntax.

·ch· Channel number.

[odf] Tool offset; if missing, the active tool offset.

V.[2].TM.TOL	Channel ·2·. Active tool offset.
V.[2].TM.TOL[3]	Channel ·2·. Tool offset ·3·.

(V.)[ch].TM.TOK [odf]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the active tool. Length wear.

This variable is not valid for turning tools.

Syntax.

·ch· Channel number.

[odf] Tool offset; if missing, the active tool offset.

V.[2].TM.TOK	Channel ·2·. Active tool offset.
V.[2].TM.TOK[3]	Channel ·2·. Tool offset ·3·.

(V.)[ch].TM.TOAN [odf]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the active tool. Penetration angle.

Syntax.

·ch· Channel number.

22.

CNC VARIABLES.

Variables related to the active tool and to the next one.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

[odf] Tool offset; if missing, the active tool offset.

V.[2].TM.TOAN	Channel -2-. Active tool offset.
V.[2].TM.TOAN[3]	Channel -2-. Tool offset -3-.

(V.) [ch]. TM.TOTIPR[odf]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the active tool. Tip radius.

Syntax.

·ch· Channel number.

[odf] Tool offset; if missing, the active tool offset.

V.[2].TM.TOTIPR	Channel -2-. Active tool offset.
V.[2].TM.TOTIPR[3]	Channel -2-. Tool offset -3-.

(V.) [ch]. TM.TOWTIPR[odf]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the active tool. Tool tip radius wear.

Syntax.

·ch· Channel number.

[odf] Tool offset; if missing, the active tool offset.

V.[2].TM.TOWTIPR	Channel -2-. Active tool offset.
V.[2].TM.TOWTIPR[3]	Channel -2-. Tool offset -3-.

(V.) [ch]. TM.TOCUTL[odf]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the active tool. Cutting length.

Syntax.

·ch· Channel number.

[odf] Tool offset; if missing, the active tool offset.

V.[2].TM.TOCUTL	Channel -2-. Active tool offset.
V.[2].TM.TOCUTL[3]	Channel -2-. Tool offset -3-.

(V.) [ch]. TM.NOSEA[odf]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the active tool. Cutter angle.

Syntax.

·ch· Channel number.

[odf] Tool offset; if missing, the active tool offset.

V.[2].TM.NOSEA	Channel -2-. Active tool offset.
V.[2].TM.NOSEA[3]	Channel -2-. Tool offset -3-.

22.

CNC VARIABLES.
Variables related to the active tool and to the next one.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].TM.NOSEW[ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface.
The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the active tool. Cutter width.

Syntax.

·ch· Channel number.

[ofd] Tool offset; if missing, the active tool offset.

V.[2].TM.NOSEW	Channel ·2·. Active tool offset.
V.[2].TM.NOSEW[3]	Channel ·2·. Tool offset ·3·.

(V.)[ch].TM.CUTA[ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface.
The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the active tool. Cutting angle of the cutter.

Syntax.

·ch· Channel number.

[ofd] Tool offset; if missing, the active tool offset.

V.[2].TM.CUTA	Channel ·2·. Active tool offset.
V.[2].TM.CUTA[3]	Channel ·2·. Tool offset ·3·.

(V.)[ch].TM.TOFL[ofd].xn

Variable that can be read and written via program, PLC(asynchronous writing) and interface.
Variable valid for rotary and linear axes.
The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the active tool. Tool length offset on the xn axis.

The offsets are used to define the tool dimensions in each axis. The dimensions of the turning tools are defined using these offsets; either these offsets or tool length and radius may be used for the dimensions of the rest of the tools.

On tools that are not just for turning, e.g. endmills and drill bits, the offsets may also be used to define the tool position when using a tool holder or an intermediate tool. In this case, the tool dimensions are defined with the radius and the length.

Syntax.

·ch· Channel number.

·ofd· Tool offset.

·xn· Name, logic number or index of the axis.

V.TM.TOFL[3].Z	Tool offset ·3·. Z axis.
V.TM.TOFL[3].4	Tool offset ·3·. Axis with logic number ·4·.
V.[2].TM.TOFL[3].1	Tool offset ·3·. Axis with index ·1· in the channel ·2·.

(V.)[ch].TM.TOFLW[ofd].xn

Variable that can be read and written via program, PLC(asynchronous writing) and interface.
Variable valid for rotary and linear axes.
The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the active tool. Tool length offset wear on the xn axis.

Syntax.

·ch· Channel number.

·ofd· Tool offset.

22.**CNC VARIABLES.**

Variables related to the active tool and to the next one.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

·xn· Name, logic number or index of the axis.

V.TM.TOFLW[3].Z	Tool offset ·3· Z axis.
V.TM.TOFLW[3].4	Tool offset ·3· Axis with logic number ·4·.
V.[2].TM.TOFLW[3].1	Tool offset ·3· Axis with index ·1· in the channel ·2·.

Remarks.

The value of these variables depends on the active units (radius or diameter). In order for these variables to return the value in diameter, these units must be enabled by machine parameter and function G151 must be active.

(V.) [ch].TM.TOFL1

(V.) [ch].TM.TOFL2

(V.) [ch].TM.TOFL3

Variable that can be read and written via program, PLC(asynchronous writing) and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Active offset of the active tool. Tool length offset on the first axes of the channel.

On turning tools, it is used to define the tool length in each axis. On milling tools, it is used to define the tool position when using a tool holder or an intermediate tooling.

Syntax.

·ch· Channel number.

V.[2].TM.TOFL1	Channel ·2·. First axis of the channel.
V.[2].TM.TOFL2	Channel ·2·. Second axis of the channel.
V.[2].TM.TOFL3	Channel ·2·. Third axis of the channel.

(V.) [ch].TM.TOFLW1

(V.) [ch].TM.TOFLW2

(V.) [ch].TM.TOFLW3

Variable that can be read and written via program, PLC(asynchronous writing) and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Active offset of the active tool. Tool length offset wear on the first axes of the channel.

Syntax.

·ch· Channel number.

V.[2].TM.TOFLW1	Channel ·2·. First axis of the channel.
V.[2].TM.TOFLW2	Channel ·2·. Second axis of the channel.
V.[2].TM.TOFLW3	Channel ·2·. Third axis of the channel.

Remarks.

The value of these variables depends on the active units (radius or diameter). In order for these variables to return the value in diameter, these units must be enabled by machine parameter and function G151 must be active.

22.

CNC VARIABLES.

Variables related to the active tool and to the next one.



CNC 8060
CNC 8065

(REF: 1402)

CANCEL THE PRESET TURNING DIRECTION OF THE TOOL.

(V.)G.SPDLTURDIR

Variable that can be read and written from the program and read from the PLC (asynchronous) and from the interface.

The variable returns the execution value; reading it interrupts block preparation.

Modify the preset turning direction for the active tool.

The preset turning speed of the active tool may be temporarily canceled from the part-program. This is done by setting this variable to ·0·. The variable does not modify the tool table. When changing a tool, this variable will take the corresponding value according to what has been set in the tool table.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Without preset turning direction.
1	Turning direction M03.
2	Turning direction M04.

22.

CNC VARIABLES.

Variables related to the active tool and to the next one.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.50 Variables associated with any tool.

The variables referred to a tool other than the active one are for synchronous reading if the tool is in the magazine and for asynchronous reading if otherwise. The writing of these variables is always asynchronous, be it for the active tool or not.

TOOL STATUS.

(V.)TM.TSTATUST[tI]

Variable that can be read from the PLC and from the interface.

Tool [tI]. Tool status.

Syntax.

·tI· Tool number.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Tool available.
1	Tool available.
2	Tool worn out.

FAMILY OF THE TOOL.

(V.)TM.TLFFT[tI]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

Tool [tI]. Family code.

Syntax.

·tI· Tool number.

V.TM.TLFFT[23] Tool ·23·.

TOOL MONITORING.

(V.)TM.TOMONT[tI][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tI] tool. Type of tool life monitoring.

Syntax.

·tI· Tool number.

·ofd· Tool offset.

V.TM.TOMONT[23][3] Tool ·23·. Tool offset ·3·.

22.

CNC VARIABLES.
Variables associated with any tool.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Without tool life monitoring.
1	Tool life monitoring; machining time.
2	Tool life monitoring; number of operations.

(V.)TM.TLFNT[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tl] tool. Nominal life.

Syntax.

·tl· Tool number.

·ofd· Tool offset.

V.TM.TLFNT[23][3] Tool ·23·. Tool offset ·3·.

(V.)TM.TLFRt[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tl] tool. Real (actual) life.

Syntax.

·tl· Tool number.

·ofd· Tool offset.

V.TM.TLFRt[23][3] Tool ·23·. Tool offset ·3·.

"CUSTOM" DATA OF THE TOOL.

(V.)TM.TOTP1T[tl]

(V.)TM.TOTP2T[tl]

(V.)TM.TOTP3T[tl]

(V.)TM.TOTP4T[tl]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

Tool [tl]. Custom parameters.

Syntax.

·tl· Tool number.

V.TM.TOTP1T[23] Tool ·23·. Custom parameter ·1·.

V.TM.TOTP2T[23] Tool ·23·. Custom parameter ·2·.

V.TM.TOTP3T[23] Tool ·23·. Custom parameter ·3·.

V.TM.TOTP4T[23] Tool ·23·. Custom parameter ·4·.

TOOL GEOMETRY.

(V.)TM.NUMOFDT[tl]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

Tool [tl]. Number of tool offsets.



CNC 8060
CNC 8065

(REF: 1402)

Syntax.

·tl· Tool number.

V.TM.NUMOFDT[23] Tool ·23·.

(V.)TM.DTYPET[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tl] tool. Code of the tool offset type.

Syntax.

·tl· Tool number.

[ofd] Tool offset; if missing, the active tool offset.

V.TM.DTYPET[23] Tool ·23·. Active tool offset.

V.TM.DTYPET[23][3] Tool ·23·. Tool offset ·3·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Milling tool..
2	Drilling tool.
3	Surface milling tool.
4	Reaming tool.
5	Boring tool.
6	Threading tool.
7	Grooving or cut-off tool.
8	Turning tool.
9	Other.
10	Measuring probe.

(V.)TM.DSUBTYPET[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tl] tool. Code of the tool offset subtype.

Syntax.

·tl· Tool number.

[ofd] Tool offset; if missing, the active tool offset.

V.TM.DSUBTYPET[23] Tool ·23·. Active tool offset.

V.TM.DSUBTYPET[23][3] Tool ·23·. Tool offset ·3·.











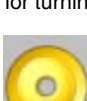






CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.	Value.	Meaning.
0	Without type.	8	Surface milling endmill. 
1	Flat endmill. 	9	Reamer. 
2	Ball endmill. 	10	Boring tool. 
3	Toric endmill. 	11	Diamond shaped cutter for turning. 
4	Tap. 	12	Square cutter for turning, grooving or cut-off. 
5	Thread cutter. 	13	Round cutter for turning. 
6	Disk endmill. 	14	Measuring probe (milling machine). 
7	Drill bit. 	15	Measuring probe (lathe). 

(V.)TM.TURNCONFIG[tI][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tI] tool. Axis orientation.

Only for turning tools. The orientation of the axes is determined by the type of lathe (horizontal or vertical), the position of the turret and the spindle position (on the right or on the left).

Syntax.

·tI· Tool number.

[ofd] Tool offset; if missing, the active tool offset.

V.TM.TURNCONFIG[23] Tool ·23·. Active tool offset.

V.TM.TURNCONFIG[23][3] Tool ·23·. Tool offset ·3·.

22.

CNC VARIABLES.
Variables associated with any tool.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Axis orientation.	Value.	Axis orientation.
0		4	
1		5	
2		6	
3		7	

(V.)TM.LOCODET[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tl] tool. Location code (shape) or tool calibration point.

Only for turning tools. The location code indicates which is the calibrated tool tip and, therefore, the point controlled by the CNC to apply radius compensation. The location code depends on the orientation of the machine axes.

Syntax.

·tl· Tool number.

[ofd] Tool offset; if missing, the active tool offset.

V.TM.LOCODET[23]	Tool ·23·. Active tool offset.
V.TM.LOCODET[23][3]	Tool ·23·. Tool offset ·3·.

Values of the variable.

This variable returns one of the following values.

Value.								
0								
1								
2								
3								
4								
5								
6								
7								

22.

CNC VARIABLES.
Variables associated with any tool.



CNC 8060
CNC 8065

(REF: 1402)

Value.								
8								
9								

(V.)TM.FIXORIT[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tl] tool. Tool-holder orientation.

Syntax.

·tl· Tool number.

[ofd] Tool offset; if missing, the active tool offset.

V.TM.FIXORIT[23]	Tool ·23·. Active tool offset.
V.TM.FIXORIT[23][3]	Tool ·23·. Tool offset ·3·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Face axis.
1	Longitudinal axis.

(V.)TM.SPDLTURDIRT[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tl] tool. Spindle turning direction.

Syntax.

·tl· Tool number.

[ofd] Tool offset; if missing, the active tool offset.

V.TM.SPDLTURDIRT[23]	Tool ·23·. Active tool offset.
V.TM.SPDLTURDIRT[23][3]	Tool ·23·. Tool offset ·3·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Undefined direction.
1	Right-hand machining direction.
2	Left-hand machining direction.

(V.)TM.TORT[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tl] tool. Radius.

This variable is not valid for turning tools.

Syntax.

·tl· Tool number.

[odf] Tool offset; if missing, the active tool offset.

V.TM.TORT[23]	Tool ·23·. Active tool offset.
V.TM.TORT[23][3]	Tool ·23·. Tool offset ·3·.

(V.)TM.TOIT[tI][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the [tI] tool. Radius wear.

This variable is not valid for turning tools.

Syntax.

·tl· Tool number.

[odf] Tool offset; if missing, the active tool offset.

V.TM.TOIT[23]	Tool ·23·. Active tool offset.
V.TM.TOIT[23][3]	Tool ·23·. Tool offset ·3·.

(V.)TM.TOLT[tI][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the [tI] tool. Length.

This variable is not valid for turning tools.

Syntax.

·tl· Tool number.

[odf] Tool offset; if missing, the active tool offset.

V.TM.TOLT[23]	Tool ·23·. Active tool offset.
V.TM.TOLT[23][3]	Tool ·23·. Tool offset ·3·.

(V.)TM.TOKT[tI][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the [tI] tool. Length wear.

This variable is not valid for turning tools.

Syntax.

·tl· Tool number.

[odf] Tool offset; if missing, the active tool offset.

V.TM.TOKT[23]	Tool ·23·. Active tool offset.
V.TM.TOKT[23][3]	Tool ·23·. Tool offset ·3·.

(V.)TM.TOANT[tI][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the [tI] tool. Penetration angle.

Syntax.

·tl· Tool number.

22.

CNC VARIABLES.
Variables associated with any tool.



CNC 8060
CNC 8065

(REF: 1402)

[odf] Tool offset; if missing, the active tool offset.

V.TM.TOANT[23]	Tool ·23·. Active tool offset.
V.TM.TOANT[23][3]	Tool ·23·. Tool offset ·3·.

(V.)TM.TOTIPRT[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the [tl] tool. Tip radius.

Syntax.

·tl· Tool number.

[odf] Tool offset; if missing, the active tool offset.

V.TM.TOTIPRT[23]	Tool ·23·. Active tool offset.
V.TM.TOTIPRT[23][3]	Tool ·23·. Tool offset ·3·.

(V.)TM.TOWTIPRT[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the [tl] tool. Tool tip radius wear.

Syntax.

·tl· Tool number.

[odf] Tool offset; if missing, the active tool offset.

V.TM.TOWTIPRT[23]	Tool ·23·. Active tool offset.
V.TM.TOWTIPRT[23][3]	Tool ·23·. Tool offset ·3·.

(V.)TM.TOCUTLT[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the [tl] tool. Cutting length.

Syntax.

·tl· Tool number.

[odf] Tool offset; if missing, the active tool offset.

V.TM.TOCUTLT[23]	Tool ·23·. Active tool offset.
V.TM.TOCUTLT[23][3]	Tool ·23·. Tool offset ·3·.

(V.)TM.NOSEAT[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface. The variable returns the execution value; reading it interrupts block preparation.

[odf] offset of the [tl] tool. Cutter angle.

Syntax.

·tl· Tool number.

[odf] Tool offset; if missing, the active tool offset.

V.TM.NOSEAT[23]	Tool ·23·. Active tool offset.
V.TM.NOSEAT[23][3]	Tool ·23·. Tool offset ·3·.

22.

CNC VARIABLES.
Variables associated with any tool.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

(V.)TM.NOSEWT[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface.
The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tl] tool. Cutter width.

Syntax.

·tl· Tool number.

[ofd] Tool offset; if missing, the active tool offset.

V.TM.NOSEWT[23]	Tool ·23·. Active tool offset.
V.TM.NOSEWT[23][3]	Tool ·23·. Tool offset ·3·.

(V.)TM.CUTAT[tl][ofd]

Variable that can be read and written via program, PLC(asynchronous writing) and interface.
The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tl] tool. Cutting angle of the cutter.

Syntax.

·tl· Tool number.

[ofd] Tool offset; if missing, the active tool offset.

V.TM.CUTAT[23]	Tool ·23·. Active tool offset.
V.TM.CUTAT[23][3]	Tool ·23·. Tool offset ·3·.

(V.)TM.TOFLT[tl][ofd].xn

Variable that can be read and written via program, PLC(asynchronous writing) and interface.
Variable valid for rotary and linear axes.
The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tl] tool. Tool length offset on the xn axis.

On turning tools, it is used to define the tool length in each axis. On milling tools, it is used to define the tool position when using a tool holder or an intermediate tooling.

Syntax.

·ofd· Tool offset.

·xn· Name or logic number of the axis.

V.TM.TOFL[23].Z	Tool ·23·. Z axis.
V.TM.TOFL[23][3].4	Tool ·23·. Axis with logic number ·4·.

(V.)TM.TOFLWT[tl][ofd].xn

Variable that can be read and written via program, PLC(asynchronous writing) and interface.
Variable valid for rotary and linear axes.
The variable returns the execution value; reading it interrupts block preparation.

[ofd] offset of the [tl] tool. Tool length offset wear on the xn axis.

Syntax.

·ofd· Tool offset.

·xn· Name or logic number of the axis.

V.TM.TOFLWT[23].Z	Tool ·23·. Z axis.
V.TM.TOFLWT[23][3].4	Tool ·23·. Axis with logic number ·4·.

Remarks.

The value of these variables depends on the active units (radius or diameter). In order for these variables to return the value in diameter, these units must be enabled by machine parameter and function G151 must be active.

22.

CNC VARIABLES.
Variables associated with any tool.



CNC 8060
CNC 8065

(REF: 1402)

22.51 Variables associated with the tool being prepared.

The CNC reads several blocks ahead of the one being executed in order to calculate in advance the path to follow.

As can be seen in the following example, the block being prepared is calculated with the tool T6; whereas the tool T1 is the one currently selected.

G1 X100 F200 T1 M6	(Block in execution)
Y200	
G1 X20 F300 T6 M6	
X30 Y60	(Block being prepared)

There are specific variables for consulting and/or modifying the values being used in the preparation. These variables can only be accessed from the program and are evaluated during block preparation. Writing these variables does not modify the tool table; the new value is only used during block preparation.

TOOL AND ACTIVE OFFSET.

(V.)[ch].G.TOOL

Variable that can be read from the program.

The variable returns the value of block preparation.

Number of the tool being prepared.

Syntax.

.ch. Channel number.

V.[2].G.TOOL	Channel -2.
--------------	-------------

(V.)[ch].G.TOD

Variable that can be read from the program.

The variable returns the value of block preparation.

Number of tool offset being prepared.

Syntax.

.ch. Channel number.

V.[2].G.TOD	Channel -2.
-------------	-------------

NEXT TOOL AND TOOL OFFSET.

(V.)[ch].G.NXTOOL

Variable that can be read from the program.

The variable returns the value of block preparation.

Number of the next tool being prepared.

Syntax.

.ch. Channel number.

V.[2].G.NXTOOL	Channel -2.
----------------	-------------

(V.)[ch].G.NXTOD

Variable that can be read from the program.

The variable returns the value of block preparation.

Number of the next tool offset being prepared.

22.

CNC VARIABLES.

Variables associated with the tool being prepared.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

Syntax.

·ch· Channel number.

V.[2].G.NXTOD Channel -2.

TOOL STATUS.**(V.) [ch].G.TSTATUS**

Variable that can be read from the program.

The variable returns the value of block preparation.

Tool being prepared. Tool status.

Syntax.

·ch· Channel number.

V.[2].G.TSTATUS Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Tool available.
1	Tool available.
2	Tool worn out.

FAMILY OF THE TOOL.**(V.) [ch].G.TLFF**

Variable that can be read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Tool being prepared. Family code.

Syntax.

·ch· Channel number.

V.[2].G.TLFF Channel -2.

TOOL MONITORING.**(V.) [ch].G.TOMON**

Variable that can be read and written from the program.

The variable returns the value of block preparation.

Tool offset being prepared. Type of tool life monitoring.

Syntax.

·ch· Channel number.

V.[2].G.TOMON Channel -2.

22.**CNC VARIABLES.**

Variables associated with the tool being prepared.



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Without tool life monitoring.
1	Tool life monitoring; machining time.
2	Tool life monitoring; number of operations.

(V.)[ch].G.TLFN

*Variable that can be read from the program.
The variable returns the value of block preparation.*

Tool offset being prepared. Nominal life.

Syntax.

·ch· Channel number.

V.[2].G.TLFN Channel ·2·.

(V.)[ch].G.TLFR

*Variable that can be read from the program.
The variable returns the value of block preparation.*

Tool offset being prepared. Real (actual) life.

Syntax.

·ch· Channel number.

V.[2].G.TLFR Channel ·2·.

(V.)[ch].G.REMLIFE

*Variable that can be read and written from the program.
The variable returns the value of block preparation.*

Tool offset being prepared. Remaining life.

Syntax.

·ch· Channel number.

V.[2].G.REMLIFE Channel ·2·.

"CUSTOM" DATA OF THE TOOL.

- (V.)[ch].G.TOTP1**
- (V.)[ch].G.TOTP2**
- (V.)[ch].G.TOTP3**
- (V.)[ch].G.TOTP4**

*Variable that can be read and written from the program.
The variable returns the value of block preparation.*

Tool being prepared. Custom parameters.

Syntax.

·ch· Channel number.

V.[2].G.TOTP1 Channel ·2·. Custom parameter ·1·.
 V.[2].G.TOTP2 Channel ·2·. Custom parameter ·2·.
 V.[2].G.TOTP3 Channel ·2·. Custom parameter ·3·.
 V.[2].G.TOTP4 Channel ·2·. Custom parameter ·4·.



**CNC 8060
CNC 8065**

(REF: 1402)

TOOL GEOMETRY.

(V.) [ch].G.DSUBTYPE

Variable that can be read from the program.

The variable returns the value of block preparation.

Tool being prepared. Code of the tool offset subtype.

Syntax.
















·ch· Channel number.

V.[2].G.DSUBTYPE

Channel -2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.	Value.	Meaning.
0	Without type.	8	Surface milling endmill. 
1	Flat endmill. 	9	Reamer. 
2	Ball endmill. 	10	Boring tool. 
3	Toric endmill. 	11	Diamond shaped cutter for turning. 
4	Tap. 	12	Square cutter for turning, grooving or cut-off. 
5	Thread cutter. 	13	Round cutter for turning. 
6	Disk endmill. 	14	Measuring probe (milling machine). 
7	Drill bit. 	15	Measuring probe (lathe). 

22.

CNC VARIABLES.
Variables associated with the tool being prepared.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].G.LOCODE

Variable that can be read from the program.

The variable returns the value of block preparation.

[odf] offset of the tool being prepared. Location code (shape) or tool calibration point.

Only for turning tools. The location code indicates which is the calibrated tool tip and, therefore, the point controlled by the CNC to apply radius compensation. The location code depends on the orientation of the machine axes.

Syntax.

·ch· Channel number.

V.[2].G.LOCODE Channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.								
0								
1								
2								
3								
4								
5								
6								
7								
8								
9								

(V.)[ch].G.TOR

Variable that can be read and written from the program.

The variable returns the value of block preparation.

Tool being prepared. Radius.

This variable is not valid for turning tools.

Syntax.

·ch· Channel number.

V.[2].G.TOR Channel ·2·.

22.

CNC VARIABLES.
Variables associated with the tool being prepared.

(V.) [ch].G.TOI

*Variable that can be read and written from the program.
The variable returns the value of block preparation.*

Tool being prepared. Radius wear.

This variable is not valid for turning tools.

Syntax.

·ch· Channel number.

V.[2].G.TOI Channel ·2·.

(V.) [ch].G.TOL

*Variable that can be read and written from the program.
The variable returns the value of block preparation.*

Tool being prepared. Length.

This variable is not valid for turning tools.

Syntax.

·ch· Channel number.

V.[2].G.TOL Channel ·2·.

(V.) [ch].G.TOK

*Variable that can be read and written from the program.
The variable returns the value of block preparation.*

Tool being prepared. Length wear.

This variable is not valid for turning tools.

Syntax.

·ch· Channel number.

V.[2].G.TOK Channel ·2·.

(V.) [ch].G.TOAN

*Variable that can be read and written from the program.
The variable returns the value of block preparation.*

Tool being prepared. Penetration angle.

Syntax.

·ch· Channel number.

V.[2].G.TOAN Channel ·2·.

(V.) [ch].G.TOTIPR

*Variable that can be read and written from the program.
The variable returns the value of block preparation.*

Tool being prepared. Tip radius.

Syntax.

·ch· Channel number.

V.[2].G.TOTIPR Channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].G.TOWTIPR

Variable that can be read and written from the program.
The variable returns the value of block preparation.

Tool being prepared. Tool tip radius wear.

Syntax.

·ch· Channel number.

V.[2].G.TOWTIPR	Channel ·2·.
-----------------	--------------

(V.)[ch].G.TOCUTL

Variable that can be read and written from the program.
The variable returns the value of block preparation.

Tool being prepared. Cutting length.

Syntax.

·ch· Channel number.

V.[2].G.TOCUTL	Channel ·2·.
----------------	--------------

(V.)[ch].G.NOSEA

Variable that can be read and written from the program.
The variable returns the value of block preparation.

Tool being prepared. Cutter angle.

Syntax.

·ch· Channel number.

V.[2].G.NOSEA	Channel ·2·.
---------------	--------------

(V.)[ch].G.NOSEW

Variable that can be read and written from the program.
The variable returns the value of block preparation.

Tool being prepared. Cutter width.

Syntax.

·ch· Channel number.

V.[2].G.NOSEW	Channel ·2·.
---------------	--------------

(V.)[ch].G.CUTA

Variable that can be read and written from the program.
The variable returns the value of block preparation.

Tool being prepared. Cutting angle of the cutter.

Syntax.

·ch· Channel number.

V.[2].G.CUTA	Channel ·2·.
--------------	--------------

(V.)[ch].A.TOFL.xn

Variable that can be read and written from the program.
Variable valid for rotary and linear axes.
The variable returns the value of block preparation.

Tool being prepared. Tool length offset on the xn axis.

On turning tools, it is used to define the tool length in each axis. On milling tools, it is used to define the tool position when using a tool holder or an intermediate tooling.

22.**CNC VARIABLES.**

Variables associated with the tool being prepared.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables associated with the tool being prepared.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

V.A.TOFL.Z	Z axis.
V.A.TOFL.4	Axis with logic number -4.
V.[2].A.TOFL.1	Axis with index -1 in the channel -2.

(V.)[ch].A.TOFLW.xn

Variable that can be read and written from the program.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

Tool being prepared. Tool length offset wear on the xn axis.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

V.A.TOFLW.Z	Z axis.
V.A.TOFLW.4	Axis with logic number -4.
V.[2].A.TOFLW.1	Axis with index -1 in the channel -2.

(V.)[ch].G.TOFL1**(V.)[ch].G.TOFL2****(V.)[ch].G.TOFL3**

Variable that can be read and written from the program.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

Tool being prepared. Tool length offset on the first axes of the channel.

On turning tools, it is used to define the tool length in each axis. On milling tools, it is used to define the tool position when using a tool holder or an intermediate tooling.

Syntax.

- ch· Channel number.

V.[2].G.TOFL1	Channel -2. First axis of the channel.
V.[2].G.TOFL2	Channel -2. Second axis of the channel.
V.[2].G.TOFL3	Channel -2. Third axis of the channel.

(V.)[ch].G.TOFLW1**(V.)[ch].G.TOFLW2****(V.)[ch].G.TOFLW3**

Variable that can be read and written from the program.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

Tool being prepared. Tool length offset wear on the first axes of the channel.

Syntax.

- ch· Channel number.

V.[2].G.TOFLW1	Channel -2. First axis of the channel.
V.[2].G.TOFLW2	Channel -2. Second axis of the channel.
V.[2].G.TOFLW3	Channel -2. Third axis of the channel.



CNC 8060
CNC 8065

(REF: 1402)

22.52 Variables related to jog mode.

MOVEMENT ALLOWED IN JOG MODE.

(V.)[ch].G.INTMAN

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Movements in jog mode are allowed.

Syntax.

·ch· Channel number.

V.[2].G.INTMAN Channel ·2·.

Remarks.

Jog movements are allowed when the jog mode or the TEACH-IN mode is active, during tool inspection and when functions G200 and G201 are active.

TYPE OF MOVEMENT ACTIVE IN THE CHANNEL.

(V.)G.MANMODE

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Type of movement active for all the axes.

V.G.MANMODE

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Handwheel mode.
2	Continuous jog mode.
3	Incremental jog mode.

Remarks.

The type of movement may be set by the switch of the operator panel or by PLC; the one set by PLC has the highest priority.

(V.)G.CNCMANMODE

Variable that can be read and written from the interface and read from the program and from the PLC.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Type of movement selected at the switch for all the axes.

The value set by the switch of the operator panel has lower priority than the one set by PLC.

V.G.CNCMANMODE

22.

CNC VARIABLES.

Variables related to jog mode.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to jog mode.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Handwheel mode.
2	Continuous jog mode.
3	Incremental jog mode.

(V.)PLC.MANMODE

Variable that can be read and written from the PLC and read from the program and from the interface. The variable returns the execution value; reading it interrupts block preparation. Report variable (to be used from the scripts).

Type of movement selected by PLC for all the axes.

The value selected by PLC prevails over the one selected on the operator panel. To cancel the feedrate set by PLC, set the variable to ·0·.

V.PLC.MANMODE

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Nothing has been selected by PLC.
1	Handwheel mode.
2	Continuous jog mode.
3	Incremental jog mode.

TYPE OF MOVEMENT ACTIVE ON AN AXIS.**(V.)[ch].A.MANMODE.xn**

Variable that can only be read from the program, PLC and interface. Variable valid for rotary and linear axes. The variable returns the execution value; reading it interrupts block preparation. Report variable (to be used from the scripts).

Type of movement active for all the ·xn· axis.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.MANMODE.Z	Z axis.
V.A.MANMODE.4	Axis with logic number ·4·.
V.[2].A.MANMODE.1	Axis with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Handwheel mode.
2	Continuous jog mode.
3	Incremental jog mode.
4	Handwheel mode without selected axis. The handwheel mode has been selected but the axis to be moved has not been selected.

Remarks.

The type of movement may be set by the switch of the operator panel or by PLC; the one set by PLC has the highest priority.

(V.)[ch].A.CNCMMODE.xn

Variable that can be read and written from the interface and read from the program and from the PLC. Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation. Report variable (to be used from the scripts).

Type of movement selected at the switch for the ·xn· axis.

The value set by the switch of the operator panel has lower priority than the one set by PLC.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.CNCMMODE.Z	Z axis.
V.A.CNCMMODE.4	Axis with logic number ·4·.
V.[2].A.CNCMMODE.1	Axis with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Handwheel mode.
2	Continuous jog mode.
3	Incremental jog mode.

(V.)[ch].A.PLCMMODE.xn

Variable that can be read and written from the PLC and read from the program and from the interface. Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation. Report variable (to be used from the scripts).

Type of movement selected by PLC for the ·xn· axis.

The value selected by PLC prevails over the one selected on the operator panel. To cancel the feedrate set by PLC, set the variable to ·0·.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.PLCMMODE.Z	Z axis.
V.A.PLCMMODE.4	Axis with logic number ·4·.
V.[2].A.PLCMMODE.1	Axis with index ·1· in the channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Nothing has been selected by PLC.
1	Handwheel mode.
2	Continuous jog mode.
3	Incremental jog mode.

22.

CNC VARIABLES.
Variables related to jog mode.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

Remarks.

If an axis has been set in handwheel mode from the PLC, it can only be deactivated from the PLC; a reset does not deactivate it.

SWITCH POSITION IN HANDWHEEL MODE.**22.**

CNC VARIABLES.
Variables related to jog mode.

(V.)G.MPGIDX

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Active position for all the handwheels.

V.G.MPGIDX

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Position 1.
2	Position 10.
3	Position 100.

Remarks.

The value may be set by the switch of the operator panel or by PLC; the one set by PLC has the highest priority.

(V.)G.CNCMPGIDX

Variable that can be read and written from the interface and read from the program and from the PLC.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Position selected at the switch.

The value set by the switch of the operator panel has lower priority than the one set by PLC.

V.PLC.CNCMPGIDX

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Position 1.
2	Position 10.
3	Position 100.

(V.)PLC.MPGIDX

Variable that can be read and written from the PLC and read from the program and from the interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Position selected by PLC.

The value selected by PLC prevails over the one selected on the operator panel. To cancel the feedrate set by PLC, set the variable to -0.

V.PLC.MPGIDX



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Position 1.
2	Position 10.
3	Position 100.

SWITCH POSITION IN INCREMENTAL JOG MODE.**(V.)G.INCJOGIDX**

*Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).*

Active position for all the axes.

V.G.INCJOGIDX

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Position 1.
2	Position 10.
3	Position 100.
4	Position 1000.
5	Position 10000.

Remarks.

The value may be set by the switch of the operator panel or by PLC; the one set by PLC has the highest priority.

(V.)G.CNCINCJOGIDX

*Variable that can be read and written from the interface and read from the program and from the PLC.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).*

Position selected by the switch.

The value set by the switch of the operator panel has lower priority than the one set by PLC.

V.G.CNCINCJOGIDX

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Position 1.
2	Position 10.
3	Position 100.
4	Position 1000.
5	Position 10000.

22.

CNC VARIABLES.
Variables related to jog mode.

FAGOR 

**CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to jog mode.

(V.)PLC.INCJOGIDX

Variable that can be read and written from the PLC and read from the program and from the interface. The variable returns the execution value; reading it interrupts block preparation. Report variable (to be used from the scripts).

Position selected by PLC.

The value selected by PLC prevails over the one selected on the operator panel. To cancel the feedrate set by PLC, set the variable to -0.

V.PLC.INCJOGIDX

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	Position 1.
2	Position 10.
3	Position 100.
4	Position 1000.
5	Position 10000.

JOG FEEDRATES.**(V.)[ch].G.FMAN**

Variable that can be read and written from the program and interface and read from the PLC. The variable returns the execution value; reading it interrupts block preparation.

Jogging feedrate in G94.

Syntax.

·ch· Channel number.

V.[2].G.FMAN Channel -2.

Remarks.

This variable also updates its value when setting a new feedrate ("F" field on the screen for jog mode) or MDI/MDA mode and function G94 is active.

(V.)[ch].G.MANFPR

Variable that can be read and written from the program and interface and read from the PLC. The variable returns the execution value; reading it interrupts block preparation.

Jogging feedrate in G95.

Syntax.

·ch· Channel number.

V.[2].G.MANFPR Channel -2.

Remarks.

This variable also updates its value when setting a new feedrate ("F" field on the screen for jog mode) or MDI/MDA mode and function G95 is active.

22.53 Variables related to the programmed functions.

MOVEMENT OF AXES AND SPINDLES.

(V.)[ch].A.INPOS.xn
 (V.)[ch].A.INPOS.sn
 (V.)[ch].SP.INPOS.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Axis or spindle in position.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.INPOS.Z	Z axis.
V.A.INPOS.S	Spindle S.
V.SP.INPOS.S	Spindle S.
V.SP.INPOS	Master spindle.
V.A.INPOS.4	Axis or spindle with logic number ·4·.
V.[2].A.INPOS.1	Axis with index ·1· in the channel ·2·.
V.SP.INPOS.2	Spindle with index ·2· in the system.
V.[2].SP.INPOS.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.DIST.xn
 (V.)[ch].A.DIST.sn
 (V.)[ch].SP.DIST.sn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Distance traveled by the axis or spindle.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.DIST.Z	Z axis.
V.A.DIST.S	Spindle S.
V.SP.DIST.S	Spindle S.
V.SP.DIST	Master spindle.
V.A.DIST.4	Axis or spindle with logic number ·4·.
V.[2].A.DIST.1	Axis with index ·1· in the channel ·2·.
V.SP.DIST.2	Spindle with index ·2· in the system.
V.[2].SP.DIST.1	Spindle with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to the programmed functions.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the programmed functions.

(V.) [ch].A.ACCUDIST.xn
(V.) [ch].A.ACCUDIST.sn
(V.) [ch].SP.ACCUDIST.sn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Distance traveled by the axis or spindle since the last coordinate latching. This variable is initialized to `.0` when a latching event takes place. To add a position offset to this variable at the latch point, just add it from the PLC in a later cycle.

Axis synchronization permits treating a rotary axis as an infinite axis making it possible to increase the feedback count of the axis indefinitely (without limits) regardless of the value of the module. The CNC uses this variable follow up on the axis. This variable may be initialized in a sampling period and keep counting from the initialized value on.

This feature is useful, for example, for a rotary axis or encoder that moves an endless conveyor belt on where the part is. The infinite axis treatment allows synchronizing the coordinate of the conveyor belt with an external event and, this way, count the moving distance of the part beyond the value of the module of the rotary axis that moves the belt.

Syntax.

- `.ch` Channel number.
- `.xn` Name, logic number or index of the axis.
- `.sn` Name, logic number or index of the spindle.

V.A.ACCUDIST.Z	Z axis.
V.A.ACCUDIST.S	Spindle S.
V.SP.ACCUDIST.S	Spindle S.
V.SP.ACCUDIST	Master spindle.
V.A.ACCUDIST.4	Axis or spindle with logic number <code>.4</code> .
V.[2].A.ACCUDIST.1	Axis with index <code>.1</code> in the channel <code>.2</code> .
V.SP.ACCUDIST.2	Spindle with index <code>.2</code> in the system.
V.[2].SP.ACCUDIST.1	Spindle with index <code>.1</code> in the channel <code>.2</code> .

(V.) [ch].A.PREVACCUDIST.xn
(V.) [ch].A.PREVACCUDIST.sn
(V.) [ch].SP.PREVACCUDIST.sn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Distance traveled by the axis or spindle between the last two coordinate latching. This variable updates (refreshes) its value at every latching event; that's why the variable will have a `.0` value until the first one takes place.

Syntax.

- `.ch` Channel number.
- `.xn` Name, logic number or index of the axis.
- `.sn` Name, logic number or index of the spindle.

V.A.PREVACCUDIST.Z	Z axis.
V.A.PREVACCUDIST.S	Spindle S.
V.SP.PREVACCUDIST.S	Spindle S.
V.SP.PREVACCUDIST	Master spindle.
V.A.PREVACCUDIST.4	Axis or spindle with logic number <code>.4</code> .
V.[2].A.PREVACCUDIST.1	Axis with index <code>.1</code> in the channel <code>.2</code> .
V.SP.PREVACCUDIST.2	Spindle with index <code>.2</code> in the system.
V.[2].SP.PREVACCUDIST.1	Spindle with index <code>.1</code> in the channel <code>.2</code> .



CNC 8060
CNC 8065

(REF: 1402)

WORK PLANE AND AXES.

(V.)[ch].G.PLANE

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Axes making up the work plane.

Syntax.

.ch· Channel number.

V.[2].G.PLANE Channel ·2·.

Values of the variable.

The values returned by this variable are encoded as follows.

X=10	X1=11	X2=12	X3=13	X4=14	... X9=19
Y=20	Y1=21	Y2=22	Y3=23	Y4=24	... Y9=29
Z=30	Z1=31	Z2=32	Z3=33	Z4=34	... Z9=39
U=40	U1=41	U2=42	U3=43	U4=44	... U9=49
V=50	V1=51	V2=52	V3=53	V4=54	... V9=59
W=60	W1=61	W2=62	W3=63	W4=64	... W9=69
A=70	A1=71	A2=72	A3=73	A4=74	... A9=79
B=80	B1=81	B2=82	B3=83	B4=84	... B9=89
C=90	C1=91	C2=92	C3=93	C4=94	... C9=99

Main axes.	Main plane.	Reading the variable.
X-Y-Z	G17 (XY)	V.[1].G.PLANE = 1020 V.[1].G.LONGAX = 30 V.[1].G.TOOLDIR = 2
X-Y-Z	G18 (ZX)	V.[1].G.PLANE = 3010 V.[1].G.LONGAX = 20 V.[1].G.TOOLDIR = 2
X-V1-Z3	G17 (X-V1) #TOOL AX [V1-]	V.[1].G.PLANE = 1051 V.[1].G.LONGAX = 33 V.[1].G.TOOLDIR = 1

(V.)[ch].G.LONGAX

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Longitudinal axis.

Syntax.

.ch· Channel number.

V.[2].G.LONGAX Channel ·2·.

Values of the variable.

The values returned by this variable are encoded as follows.

X=10	X1=11	X2=12	X3=13	X4=14	... X9=19
Y=20	Y1=21	Y2=22	Y3=23	Y4=24	... Y9=29
Z=30	Z1=31	Z2=32	Z3=33	Z4=34	... Z9=39
U=40	U1=41	U2=42	U3=43	U4=44	... U9=49
V=50	V1=51	V2=52	V3=53	V4=54	... V9=59
W=60	W1=61	W2=62	W3=63	W4=64	... W9=69

22.

CNC VARIABLES.

Variables related to the programmed functions.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

Syntax.

·ch· Channel number.

[2].G.PLAXNAME1	Channel ·2·. Abscissa axis.
[2].G.PLAXNAME2	Channel ·2·. Ordinate axis.
[2].G.PLAXNAME3	Channel ·2·. Third main axis.

(V.)[ch].G.PLANELONG

Variable to be read via interface.

Index of the tool in the axis channel.

Syntax.

·ch· Channel number.

[2].G.PLANELONG	Channel ·2·.
-----------------	--------------

Remarks.

In this variable, the index in the channel of the first axis will be ·0·, that of the second axis will be ·1· and so on.

"G" AND "M" FUNCTIONS.**(V.)[ch].G.GS[nb]**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Report variable (to be used from the scripts).

Status of the requested "G" function.

Each function has a bit that indicates whether the relevant function is active (=1) or not (=0).

Syntax.

·ch· Channel number.

·nb· Function number.

V.[2].G.GS[3]	Channel ·2·. G3 function.
---------------	---------------------------

(V.)[ch].G.MS[nb]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Report variable (to be used from the scripts).

Status of the requested "M" function.

Each function has a bit that indicates whether the relevant function is active (=1) or not (=0).

Syntax.

·ch· Channel number.

·nb· Function number.

V.[2].G.MS[5]	Channel ·2·. M5 function.
---------------	---------------------------

(V.)[ch].G.HGS1

(V.)[ch].G.HGS2

(V.)[ch].G.HGS3

(V.)[ch].G.HGS4

(V.)[ch].G.HGS5

(V.)[ch].G.HGS6

(V.)[ch].G.HGS7

22.

CNC VARIABLES.
Variables related to the programmed functions.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].G.HGS8
 (V.)[ch].G.HGS9
 (V.)[ch].G.HGS10

Variable that can only be read from the program, PLC and interface.
 The variable returns the value of block preparation.

Status of the requested "G" (32 bit) functions.

Syntax.

·ch· Channel number.

V.[2].G.HGS1 Channel -2.

Remarks.

Every variable corresponds to a range of 32 G functions and returns a 32-bit value; 1 bit per function. Each bit indicates whether the function is active (bit = 1) or not (bit = 0). The least significant bit corresponds to the lowest function of the range.

Variable.	G function range.	
(V.)[ch].G.HGS1	G0 - G31.	Bit 0 corresponds to G0.
(V.)[ch].G.HGS2	G32-G63	Bit 0 corresponds to G32.
(V.)[ch].G.HGS3	G64-G95	Bit 0 corresponds to G64.
(V.)[ch].G.HGS4	G96-G127	Bit 0 corresponds to G96.
(V.)[ch].G.HGS5	G128-G159	Bit 0 corresponds to G128.
(V.)[ch].G.HGS6	G160-G191	Bit 0 corresponds to G160.
(V.)[ch].G.HGS7	G192-G223	Bit 0 corresponds to G192.
(V.)[ch].G.HGS8	G224-G255	Bit 0 corresponds to G224.
(V.)[ch].G.HGS9	G256-G287	Bit 0 corresponds to G256.
(V.)[ch].G.HGS10	G288-G319	Bit 0 corresponds to G288.

To check the status of function G08 from the part-program.

```
$IF [V.[1].G.HGS1 & [2**8]] == 2**8
```

To check the status of function G101 from the part-program.

```
$IF [V.[1].G.HGS4 & [2**5]] == 2**5
```

To check the status of function G08 from the PLC.

```
DFU B0KEYBD1 = CNCRD(G.HGS1, R100, M100)  
B8R100 = ...
```

To check the status of function G101 from the PLC.

```
DFU B0KEYBD1 = CNCRD(G.HGS4, R101, M100)  
B5R101 = ...
```

(V.)[ch].G.HGS

Variable to be read via interface.

Report variable (to be used from the scripts).

"G" functions to be displayed in the history.

[2].G.HGS Channel -2.

Remarks.

This variable returns a binary value. Each function has a bit that indicates whether the relevant variable will be displayed (=1) or not (=0). Bit 0, the least significant bit, corresponds to G0, bit 1 to G1 and so on.

22.

CNC VARIABLES.
Variables related to the programmed functions.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].G.HMS

Variable to be read via interface.

Report variable (to be used from the scripts).

"M" functions of the master spindle to be displayed in the history.

[2].G.HMS	Channel -2-. Master spindle.
-----------	------------------------------

Remarks.

This variable returns a binary value. Each function has a bit that indicates whether the relevant variable will be displayed (=1) or not (=0). Bit 0, the least significant bit, corresponds to M0, bit 1 to M1 and so on.

(V.)[ch].G.HMS1**(V.)[ch].G.HMS2****(V.)[ch].G.HMS3****(V.)[ch].G.HMS4**

Variable to be read via interface.

Report variable (to be used from the scripts).

"M" functions of spindle 1 through 4 to be displayed in the history.

[2].G.HMS2	Channel -2-. Spindle -2-.
------------	---------------------------

Remarks.

This variable returns a binary value. Each function has a bit that indicates whether the relevant variable will be displayed (=1) or not (=0). Bit 0, the least significant bit, corresponds to M0, bit 1 to M1 and so on.

CANNED CYCLES.**(V.)[ch].G.CYCLETYPON**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Type of canned cycle active

Syntax.

·ch· Channel number.

V.[2].G.CYCLETYPON	Channel -2-.
--------------------	--------------

CANNED CYCLE CALLING PARAMETERS.**(V.)C.A·Z**

Variable that can be read and written from the program.

The variable returns the value of block preparation.

Value of the ISO canned cycle calling parameter.

Syntax.

·A·Z· Call parameter.

V.C.F	"F" parameter.
-------	----------------

22.**CNC VARIABLES.**

Variables related to the programmed functions.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

Remarks.

Calling a canned cycle.	Reading the variable.
G90 G81 Z0 I-15	V.C.Z = 0 V.C.Z = -15

(V.)C.name

Variable that can be read and written from the program.

The variable returns the value of block preparation.

Value of the parameters that call upon the canned cycles of the editor.

Syntax.

·name· Call parameter.

V.C.MROUGHIN	Parameter MROUGHIN.
--------------	---------------------

(V.)C.CALLP_A·Z

Variable that can be read from the program.

The variable returns the value of block preparation.

Parameter programmed in the call to a canned cycle.

Syntax.

·A·Z· Call parameter.

V.C.CALLP_F	"F" parameter.
-------------	----------------

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	It has not been programmed.
1	It has been programmed.

Remarks.

Calling a canned cycle.	Reading the variable.
G90 G81 Z0 I-15	V.C.CALLP_Z = 1 V.C.CALLP_I = 1 V.C.CALLP_K = 0

(V.)C.P_A·Z

Variable that can be read and written from the program.

The variable returns the value of block preparation.

Value of the positioning cycle calling parameter.

Syntax.

·A·Z· Call parameter.

V.C.P_F	"F" parameter.
---------	----------------

22.

CNC VARIABLES.
Variables related to the programmed functions.



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

Calling a canned cycle.	Reading the variable.
G160 A30 X100 K10 P6	V.C.P_A = 30 V.C.P_X = 100

(V.)C.P_CALLP_A..Z

Variable that can be read from the program.

The variable returns the value of block preparation.

Parameter programmed in the call to a positioning cycle.

Syntax.

.A..Z: Call parameter.

V.C.P_CALLP_F	"F" parameter.
---------------	----------------

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	It has not been programmed.
1	It has been programmed.

Remarks.

Calling a canned cycle.	Reading the variable.
G160 A30 X100 K10 P6	V.C.P_CALLP_A = 1 V.C.P_CALLP_K = 1 V.C.P_CALLP_R = 0

SUBROUTINE CALLING PARAMETERS.

(V.)C.PCALLP_A..Z

Variable that can be read from the program.

The variable returns the value of block preparation.

Parameter programmed in the call to a subroutine

This variable is valid for OEM subroutines (G18x) and for subroutines called upon using #PCALL or #MCALL..

Syntax.

.A..Z: Call parameter.

V.C.PCALLP_F	"F" parameter.
--------------	----------------

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	It has not been programmed.
1	It has been programmed.



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

Call to a subroutine.	Reading the variable.
#PCALL sub.nc A12.56 D3	V.C.PCALLP_A = 1 V.C.PCALLP_D = 1

ARC RELATED.**(V.)[ch].G.R**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Radius of the arc.

Syntax.

·ch· Channel number.

V.[2].G.R	Channel -2·.
-----------	--------------

(V.)[ch].G.I**(V.)[ch].G.J****(V.)[ch].G.K**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Coordinates referred to the arc center.

Syntax.

·ch· Channel number.

V.[2].G.I	Channel -2·. First axis of the channel.
V.[2].G.J	Channel -2·. Second axis of the channel.
V.[2].G.K	Channel -2·. Third axis of the channel.

Remarks.

Being G20 active, parameters "I", "J" and "K" are associated with the abscissa axis, ordinate axis and the axis perpendicular to the work plane respectively.

(V.)[ch].G.CIRERR[1]**(V.)[ch].G.CIRERR[2]**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Arc center correction in the first axes of the channel.

Syntax.

·ch· Channel number.

V.[2].G.CIRERR[1]	Channel -2·. First axis of the channel.
V.[2].G.CIRERR[2]	Channel -2·. Second axis of the channel.

22.

CNC VARIABLES.
Variables related to the programmed functions.



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

Being function G265 active, the CNC recalculates the center if the arc is not exact but it is within tolerance.

Part-program.	Reading the variable.
G00 X0 Y0 G2 X120 Y120.001 I100 J20	V.G.R = 101.980881 V.G.I = 100.0004 V.G.J = 20.0004 V.G.CIRERR[1] = -0.000417 V.G.CIRERR[2] = -0.000417

Being function G264 active, if the arc is not exact, but it is within tolerances, the CNC executes an arc with the radius calculated from the starting point. The CNC maintains the position of the center.

Part-program.	Reading the variable.
G00 X0 Y0 G2 X120 Y120.001 I100 J20	V.G.R = 101.981371 V.G.I = 100 V.G.J = 20 V.G.CIRERR[1] = 0 V.G.CIRERR[2] = 0

POLAR ORIGIN.**(V.)[ch].G.PORGF**

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Position of the polar origin referred to part zero (abscissa).

Syntax.

.ch. Channel number.

V.[2].G.PORGF Channel -2.

(V.)[ch].G.PORGS

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Position of the polar origin referred to part zero (ordinate).

Syntax.

.ch. Channel number.

V.[2].G.PORGS Channel -2.

GEOMETRIC ASSISTANCE. MIRROR IMAGE.**(V.)[ch].G.MIRROR**

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Active mirror images.

Syntax.

.ch. Channel number.

V.[2].G.MIRROR Channel -2.

22.

CNC VARIABLES.
Variables related to the programmed functions.

FAGOR 

**CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the programmed functions.

Remarks.

This variable returns the result in the least significant bits, one per axis (1= active and 0=not active). The least significant bit is for the first axis, the next one for the second axis and so on.

(V.)[ch].G.MIRROR1**(V.)[ch].G.MIRROR2****(V.)[ch].G.MIRROR3**

Variable that can be read from the program.

The variable returns the value of block preparation.

Mirror image active on the first axes of the channel.

Syntax.

·ch· Channel number.

V.[2].G.MIRROR1	Channel ·2·. First axis of the channel.
V.[2].G.MIRROR2	Channel ·2·. Second axis of the channel.
V.[2].G.MIRROR3	Channel ·2·. Third axis of the channel.

GEOMETRIC ASSISTANCE. SCALING FACTOR.**(V.)[ch].G.SCALE**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

It indicates the active general scaling factor.

Syntax.

·ch· Channel number.

V.[2].G.SCALE	Channel ·2·.
---------------	--------------

GEOMETRIC ASSISTANCE. COORDINATE SYSTEM ROTATION.**(V.)[ch].G.ROTPF**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Position of the rotation center referred to part zero (abscissa).

Syntax.

·ch· Channel number.

V.[2].G.ROTPF	Channel ·2·.
---------------	--------------

(V.)[ch].G.ROTPTS

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Position of the rotation center referred to part zero (ordinate).

Syntax.

·ch· Channel number.

V.[2].G.ROTPTS	Channel ·2·.
----------------	--------------



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].G.ORGROT

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Rotation angle of the coordinate system.

Syntax.

·ch· Channel number.

V.[2].G.ORGROT	Channel -2.
----------------	-------------

BLOCK REPETITION.**(V.)[ch].G.PENDRPT**

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of pending repetitions with #RPT.

Syntax.

·ch· Channel number.

V.[2].G.PENDRPT	Channel -2.
-----------------	-------------

Remarks.

This variable indicates the number of repetitions left to execute. In the first execution, its value is the programmed number of repetitions minus one and in the last one, its value is zero.

(V.)[ch].G.PENDNR

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Number of pending repetitions with NR.

Syntax.

·ch· Channel number.

V.[2].G.PENDNR	Channel -2.
----------------	-------------

Remarks.

This variable indicates the number of repetitions left to execute. In the first execution, its value is the programmed number of repetitions minus one and in the last one, its value is zero.

AXIS SLAVING.**(V.)[ch].G.LINKACTIVE**

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Axis slaving (coupling) status.

Syntax.

·ch· Channel number.

V.[2].G.LINKACTIVE	Channel -2.
--------------------	-------------

22.**CNC VARIABLES.**

Variables related to the programmed functions.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

HSC FUNCTION.

(V.)[ch].G.HSC

Variable that can be read from the program.

The variable returns the value of block preparation.

HSC function active.

Syntax.

·ch· Channel number.

V.[2].G.HSC Channel ·2·.

ACTIVE PROBE.

(V.)[ch].G.ACTIVPROBE

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Number of the active probe.

Syntax.

·ch· Channel number.

V.[2].G.ACTIVPROBE Channel ·2·.

STATUS OF THE LOCAL PROBES.

(V.)G.PRBST1

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Status of probe ·1·.

V.G.PRBST1

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	The probe is not making contact or has not been initialized.
1	The probe is making contact.

(V.)G.PRBST2

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Status of probe ·2·.

V.G.PRBST2

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	The probe is not making contact or has not been initialized.
1	The probe is making contact.

22.

CNC VARIABLES.
Variables related to the programmed functions.



CNC 8060
CNC 8065

(REF: 1402)

PROBING (G100/G101/G102).

(V.)[ch].G.MEASOK

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Probing completed.

Syntax.

·ch· Channel number.

V.[2].G.MEASOK Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].A.MEASOK.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Probing done on the axis.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.A.MEASOK.Z	Z axis.
V.A.MEASOK.S	Spindle S.
V.SP.MEASOK.S	Spindle S.
V.SP.MEASOK	Master spindle.
V.A.MEASOK.4	Axis or spindle with logic number -4.
V.[2].A.MEASOK.1	Axis with index -1 in the channel -2.
V.SP.MEASOK.2	Spindle with index -2 in the system.
V.[2].SP.MEASOK.1	Spindle with index -1 in the channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].G.PLMEASOK1

(V.)[ch].G.PLMEASOK2

(V.)[ch].G.PLMEASOK3

Variable that can be read from the program.

The variable returns the execution value; reading it interrupts block preparation.

Probing on the plane axes completed.

22.

CNC VARIABLES.

Variables related to the programmed functions.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the programmed functions.

Syntax.

·ch· Channel number.

V.[2].G.PLMEASOK1	Channel ·2·. First axis of the plane.
V.[2].G.PLMEASOK2	Channel ·2·. Second axis of the plane.
V.[2].G.PLMEASOK3	Channel ·2·. Third axis of the plane.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

(V.)[ch].A.MEAS.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Measured value. Machine coordinates of the tool base.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.A.MEAS.Z	Z axis.
V.A.MEAS.S	Spindle S.
V.SP.MEAS.S	Spindle S.
V.SP.MEAS	Master spindle.
V.A.MEAS.4	Axis or spindle with logic number ·4·.
V.[2].A.MEAS.1	Axis with index ·1· in the channel ·2·.
V.SP.MEAS.2	Spindle with index ·2· in the system.
V.[2].SP.MEAS.1	Spindle with index ·1· in the channel ·2·.

Remarks.

Part-program.	Reading the variable.
G00 X0 Y0 G100 X100 F100	V.A.MEAS.X = 95 V.A.MEASOF.X = -5 V.A.MEASOK.X = 1

(V.)[ch].A.ATIPMEAS.xn

Variable that can be read from the program.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Measured value. Part coordinates of the tool tip.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.A.ATIPMEAS.Z	Z axis.
V.A.ATIPMEAS.S	Spindle S.
V.SP.ATIPMEAS.S	Spindle S.
V.SP.ATIPMEAS	Master spindle.



CNC 8060
CNC 8065

(REF: 1402)

V.A.ATIPMEAS.4	Axis or spindle with logic number -4-.
V.[2].A.ATIPMEAS.1	Axis with index -1- in the channel -2-.
V.SP.ATIPMEAS.2	Spindle with index -2- in the system.
V.[2].SP.ATIPMEAS.1	Spindle with index -1- in the channel -2-.

(V.)[ch].G.PLMEAS1

(V.)[ch].G.PLMEAS2

(V.)[ch].G.PLMEAS3

Variable that can be read from the program.

The variable returns the value of block preparation.

Value measured on the first axes of the channel. Part coordinates of the tool tip.

Syntax.

.ch- Channel number.

V.[2].G.PLMEAS1	Channel -2-. First axis of the channel.
V.[2].G.PLMEAS2	Channel -2-. Second axis of the channel.
V.[2].G.PLMEAS3	Channel -2-. Third axis of the channel.

(V.)[ch].A.MEASOF.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Difference with respect to programmed point.

Syntax.

.ch- Channel number.

.xn- Name, logic number or index of the axis.

.sn- Name, logic number or index of the spindle.

V.A.MEASOF.Z	Z axis.
V.A.MEASOF.S	Spindle S.
V.SP.MEASOF.S	Spindle S.
V.SP.MEASOF	Master spindle.
V.A.MEASOF.4	Axis or spindle with logic number -4-.
V.[2].A.MEASOF.1	Axis with index -1- in the channel -2-.
V.SP.MEASOF.2	Spindle with index -2- in the system.
V.[2].SP.MEASOF.1	Spindle with index -1- in the channel -2-.

(V.)[ch].A.MEASIN.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Coordinate that includes a measurement offset.

Syntax.

.ch- Channel number.

.xn- Name, logic number or index of the axis.

.sn- Name, logic number or index of the spindle.

V.A.MEASIN.Z	Z axis.
V.A.MEASIN.S	Spindle S.
V.SP.MEASIN.S	Spindle S.
V.SP.MEASIN	Master spindle.
V.A.MEASIN.4	Axis or spindle with logic number -4-.



22.

CNC VARIABLES.
Variables related to the programmed functions.

V.[2].A.MEASIN.1	Axis with index -1- in the channel -2-.
V.SP.MEASIN.2	Spindle with index -2- in the system.
V.[2].SP.MEASIN.1	Spindle with index -1- in the channel -2-.

Remarks.

The CNC only refreshes this variable when probing with G101.

MANUAL INTERVENTION.**(V.)[ch].A.MANOF.xn**

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Distance moved in jog mode or tool inspection.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.MANOF.Z	Z axis.
V.A.MANOF.4	Axis with logic number -4-.
V.[2].A.MANOF.1	Axis with index -1- in the channel -2-.

(V.)[ch].A.ADDMANOF.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Distance moved with G200 or G201.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.ADDMANOF.Z	Z axis.
V.A.ADDMANOF.4	Axis with logic number -4-.
V.[2].A.ADDMANOF.1	Axis with index -1- in the channel -2-.

Remarks.

The value of this variable is maintained during the execution of the program even when canceling manual intervention.

STATUS OF THE ANGULAR TRANSFORMATION.**(V.)[ch].G.ANGAXST**

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Overall status of the angular transformation of the channel.

Syntax.

·ch· Channel number.

V.[2].G.ANGAXST	Channel -2-.
-----------------	--------------



CNC 8060
CNC 8065

(REF: 1402)

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Off.
1	On.
2	Frozen (suspended).

(V.)G.ANGIDST[nb]

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Status of the angular transformation [nb] of the machine parameters.

Syntax.

·nb· Number of the angular transformation in the order defined in the machine parameters.

V.G.ANGIDST[1] Angular transformation ·1·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Off.
1	On.
2	Frozen (suspended).

TANGENTIAL CONTROL STATUS.**(V.)[ch].G.TGCTRLST**

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Status of tangential control in the channel.

Syntax.

·ch· Channel number.

V.[2].G.TGCTRLST Channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Off.
1	On.
2	Frozen (suspended).

22.**CNC VARIABLES.**

Variables related to the programmed functions.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the programmed functions.

(V.)[ch].A.TGCTRLST.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the value of block preparation.

Status of tangential control in the axis.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.TGCTRLST.Z	Z axis.
V.A.TGCTRLST.4	Axis with logic number -4-.
V.[2].A.TGCTRLST.1	Axis with index -1- in the channel -2-.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Off.
1	On.
2	Frozen (suspended).

(V.)[ch].A.TANGAN.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Angle programmed on the axis.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.TANGAN.Z	Z axis.
V.A.TANGAN.4	Axis with logic number -4-.
V.[2].A.TANGAN.1	Axis with index -1- in the channel -2-.

(V.)[ch].G.TANGFEED

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Positioning feedrate for tangential control.

Syntax.

·ch· Channel number.

V.[2].G.TANGFEED	Channel -2-.
------------------	--------------

SYNCHRONIZATION OF CHANNELS.**(V.)[ch].G.MEETST[mk]**

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Status of the MEET type mark [mk] in the [ch] channel.

Syntax.

·ch· Channel number.

`.mk` Number of the synchronization mark.

`V.[2].G.MEETST[4]` Channel `.2.` Mark `.4.`

(V.)[ch].G.WAITST[mk]

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Status of the WAIT type mark [mk] in the [ch] channel.

Syntax.

`.ch` Channel number.

`.mk` Number of the synchronization mark.

`V.[2].G.WAITST[4]` Channel `.2.` Mark `.4.`

(V.)[ch].G.MEETCH[nch]

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

MEET type mark originated in channel [nch] and expected by channel [ch].

Syntax.

`.ch` Channel number.

`.nch` Channel that originates the synchronization mark.

`V.[2].G.MEETCH[4]` Channel `.2.` Channel `.4.`

(V.)[ch].G.WAITCH[nch]

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

WAIT type mark originated in channel [nch] and expected by channel [ch].

Syntax.

`.ch` Channel number.

`.nch` Channel that originates the synchronization mark.

`V.[2].G.WAITCH[4]` Channel `.2.` Channel `.4.`

SELECTION OF THE KINEMATICS.

(V.)[ch].G.KINID

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Number of the active kinematics.

If no kinematics is active, the variable will return a `.0.` value.

Syntax.

`.ch` Channel number.

`V.[2].G.KINID` Channel `.2.`

22.

CNC VARIABLES.

Variables related to the programmed functions.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

POSITION OF THE AXES OF THE KINEMATICS.

(V.) [ch].G.POSROTF**(V.) [ch].G.POSROTS****(V.) [ch].G.POSROTT***Variable that can be read and written via program, PLC and interface.**The variable returns the value of block preparation.*

Current position of the main rotary axes.

Syntax.

·ch· Channel number.

V.[2].G.POSROTF	Current position of the main rotary axis.
V.[2].G.POSROTS	Current position of the secondary rotary axis.
V.[2].G.POSROTT	Current position of the third rotary axis.

(V.) [ch].G.TOOLORIF1**(V.) [ch].G.TOOLORIS1****(V.) [ch].G.TOOLORIT1***Variable that can only be read from the program, PLC and interface.**The variable returns the value of block preparation.*

Target position for the main rotary axes (solution 1).

Syntax.

·ch· Channel number.

V.[2].G.TOOLORIF1	Target position for the main rotary axis.
V.[2].G.TOOLORIS1	Target position for the secondary rotary axis.
V.[2].G.TOOLORIT1	Target position for the third rotary axis.

Remarks.

These variables that indicate the position the main axes must occupy in order for the tool to be perpendicular to the define work plane. They are very useful when the spindle is not fully motorized (mono-rotary or manual spindles).

On angular (swivel) spindle heads, there are 2 possible solutions when calculating this target position. These variables indicate the solution that involves the shortest movement of the main rotary axis with respect to the zero position.

(V.) [ch].G.TOOLORIF2**(V.) [ch].G.TOOLORIS2****(V.) [ch].G.TOOLORIT2***Variable that can only be read from the program, PLC and interface.**The variable returns the value of block preparation.*

Target position for the main rotary axes (solution 2).

Syntax.

·ch· Channel number.

V.[2].G.TOOLORIF2	Target position for the main rotary axis.
V.[2].G.TOOLORIS2	Target position for the secondary rotary axis.
V.[2].G.TOOLORIT2	Target position for the third rotary axis.

Remarks.

These variables that indicate the position the main axes must occupy in order for the tool to be perpendicular to the define work plane. They are very useful when the spindle is not fully motorized (mono-rotary or manual spindles).

22.

CNC VARIABLES.
Variables related to the programmed functions.



CNC 8060
CNC 8065

(REF: 1402)

On angular (swivel) spindle heads, there are 2 possible solutions when calculating this target position. These variables indicate the solution that involves the longest movement of the main rotary axis with respect to the zero position.

INCLINED PLANES.

(V.)[ch].G.CS

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.
Report variable (to be used from the scripts).

Number of the active CS function.

Syntax.

.ch· Channel number.

V.[2].G.CS Channel ·2·.

(V.)[ch].G.ACS

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.
Report variable (to be used from the scripts).

Number of the active ACS function.

Syntax.

.ch· Channel number.

V.[2].G.ACS Channel ·2·.

(V.)[ch].G.TOOLCOMP

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Longitudinal compensation function active.

Syntax.

.ch· Channel number.

V.[2].G.TOOLCOMP Channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
1	RTCP.
2	TLC.
3	None.

DIE RESULTING FROM THE INCLINED PLANE.

(V.)[ch].G.CSMAT1
(V.)[ch].G.CSMAT2
(V.)[ch].G.CSMAT3
(V.)[ch].G.CSMAT4
(V.)[ch].G.CSMAT5
(V.)[ch].G.CSMAT6
(V.)[ch].G.CSMAT7

22.

CNC VARIABLES.

Variables related to the programmed functions.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the programmed functions.

(V.)[ch].G.CSMAT8
(V.)[ch].G.CSMAT9

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Die resulting from the inclined plane.

These variables correspond to the transformation matrix from theoretical reference system to the real reference system.

Syntax.

·ch· Channel number.

(V.)[ch].G.CSMAT1	Channel ·2·. Element row 1 column 1.
(V.)[ch].G.CSMAT2	Channel ·2·. Element row 1 column 2.
(V.)[ch].G.CSMAT3	Channel ·2·. Element row 1 column 3.
(V.)[ch].G.CSMAT4	Channel ·2·. Element row 2 column 1.
(V.)[ch].G.CSMAT5	Channel ·2·. Element row 2 column 2.
(V.)[ch].G.CSMAT6	Channel ·2·. Element row 2 column 3.
(V.)[ch].G.CSMAT7	Channel ·2·. Element row 3 column 1.
(V.)[ch].G.CSMAT8	Channel ·2·. Element row 3 column 2.
(V.)[ch].G.CSMAT9	Channel ·2·. Element row 3 column 3.

(V.)[ch].G.CSMAT10
(V.)[ch].G.CSMAT11
(V.)[ch].G.CSMAT12

*Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.*

Offset of the current coordinate system referred to machine zero on the first axes.

These variables correspond to the transformation matrix from theoretical reference system to the real reference system.

Syntax.

·ch· Channel number.

(V.)[ch].G.CSMAT10	Channel ·2·. Offset on the first axis.
(V.)[ch].G.CSMAT11	Channel ·2·. Offset on the second axis.
(V.)[ch].G.CSMAT12	Channel ·2·. Offset on the third axis.

FEED-FORWARD AND AC-FORWARD.

(V.)[ch].A.FFGAIN.xn
(V.)[ch].A.FFGAIN.sn
(V.)[ch].SP.FFGAIN.sn

*Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.*

Active percentage of feed-forward.

Syntax.

·ch· Channel number.
·xn· Name, logic number or index of the axis.
·sn· Name, logic number or index of the spindle.

V.A.FFGAIN.Z	Z axis.
V.A.FFGAIN.S	Spindle S.
V.SP.FFGAIN.S	Spindle S.
V.SP.FFGAIN	Master spindle.



CNC 8060
CNC 8065

(REF: 1402)

V.A.FFGAIN.4	Axis or spindle with logic number ·4·.
V.[2].A.FFGAIN.1	Axis with index ·1· in the channel ·2·.
V.SP.FFGAIN.2	Spindle with index ·2· in the system.
V.[2].SP.FFGAIN.1	Spindle with index ·1· in the channel ·2·.

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·100·.

(V.)[ch].A.ACFGAIN.xn
(V.)[ch].A.ACFGAIN.sn
(V.)[ch].SP.ACFGAIN.sn

Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.

Active percentage of AC-forward.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.ACFGAIN.Z	Z axis.
V.A.ACFGAIN.S	Spindle S.
V.SP.ACFGAIN.S	Spindle S.
V.SP.ACFGAIN	Master spindle.
V.A.ACFGAIN.4	Axis or spindle with logic number ·4·.
V.[2].A.ACFGAIN.1	Axis with index ·1· in the channel ·2·.
V.SP.ACFGAIN.2	Spindle with index ·2· in the system.
V.[2].SP.ACFGAIN.1	Spindle with index ·1· in the channel ·2·.

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·100·.

(V.)[ch].A.ACTFFW.xn
(V.)[ch].A.ACTFFW.sn
(V.)[ch].SP.ACTFFW.sn

Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the execution value; reading it interrupts block preparation.

Instant feed-forward.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.ACTFFW.Z	Z axis.
V.A.ACTFFW.S	Spindle S.
V.SP.ACTFFW.S	Spindle S.
V.SP.ACTFFW	Master spindle.
V.A.ACTFFW.4	Axis or spindle with logic number ·4·.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the programmed functions.

V.[2].A.ACTFFW.1	Axis with index -1- in the channel -2-.
V.SP.ACTFFW.2	Spindle with index -2- in the system.
V.[2].SP.ACTFFW.1	Spindle with index -1- in the channel -2-.

(V.)[ch].A.ACTACF.xn
(V.)[ch].A.ACTACF.sn
(V.)[ch].SP.ACTACF.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Instant AC-forward.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.ACTACF.Z	Z axis.
V.A.ACTACF.S	Spindle S.
V.SP.ACTACF.S	Spindle S.
V.SP.ACTACF	Master spindle.
V.A.ACTACF.4	Axis or spindle with logic number -4-.
V.[2].A.ACTACF.1	Axis with index -1- in the channel -2-.
V.SP.ACTACF.2	Spindle with index -2- in the system.
V.[2].SP.ACTACF.1	Spindle with index -1- in the channel -2-.

ERRORS AND WARNINGS.

(V.)[ch].G.CNCERR

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Number of the error with highest priority in the indicated channel.

Syntax.

- ch· Channel number.

V.[2].G.CNCERR	Channel -2-.
----------------	--------------

Remarks.

This variable is initialized with a reset. If several channels are in the same group, the reset of a channel means resetting all of them, thus initializing the variables of all the channels of the group.

If several channels are in the same group, an error in a channel causes the same error in all of them; in this case, this variable will have the same value for all the channels of the group.

(V.)[ch].G.CNCWARNING

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Number of warning displayed in the indicated channel.

Syntax.

- ch· Channel number.

V.[2].G.CNCWARNING	Channel -2-.
--------------------	--------------



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

This variable is initialized with a reset. If several channels are in the same group, the reset of a channel means resetting all of them, thus initializing the variables of all the channels of the group.

If there are several warnings, the value of the variable is updated as they are eliminated. When the last warning is eliminated, this variable is initialized to zero.

REPOSITIONING OF AXES AND SPINDLES.**(V.)[ch]G.ENDREP**

Variable that can be read and written via program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

All the axes are repositioned.

Syntax.

.ch. Channel number.

V.[2].G.ENDREP Channel -2.

(V.)[ch].G.SPDLREP

Variable that can be read and written via program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

M function to be used to reposition the spindle after a tool inspection.

Syntax.

.ch. Channel number.

V.[2].G.SPDLREP Channel -2.

ACTIVE ZERO OFFSET.**(V.)[ch]G.EXTORG**

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Function that defines the zero offset (zero offset or coordinate preset).

Syntax.

.ch. Channel number.

V.[2].G.EXTORG Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	G53. There is no zero offset.
1 - 99	G159. Zero offset (99 available). If 1, G159=1; if 2, G159=2 and so on.
1000	G92. Coordinate preset.

Remarks.

The numbering of zero offsets G54 through G59 is always the same; G54=1, G55=2, G56=3, G57=4, G58=5, G59=6.

22.**CNC VARIABLES.**

Variables related to the programmed functions.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.54 Variables related to the electronic cam.

(V.)G.CAMST[cam]

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Cam status.

Use this variable in the PLC program to condition the cam operations to the right status and consequently prevent execution errors. The operations of the cam from the part-program stay in execution, without issuing an error message, waiting for the right cam status.

Syntax.

·cam· Cam number.

V.G.CAMST[2] Cam ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	<p>CAM_NULL</p> <p>The cam does not exist. The cam is not defined in the machine parameters and there is no file cam available, the CNC or the PLC have not selected it (SELECT) or they have deselected it (DESELECT).</p> <ul style="list-style-type: none"> • This state only allows loading one file cam from the CNC or PLC. • This status does not allow activating the cam, either from the CNC or from the PLC.
1	<p>CAM_LOADING</p> <p>The CNC or the PLC is loading the file that defines the cam.</p> <ul style="list-style-type: none"> • This state only allows activating the cam from the CNC, where the CNC will remain in execution until the cam reaches the CAM_READY state. The cam cannot be activated from the PLC.
2	<p>CAM_READY</p> <p>Cam ready. The CNC has validated the cam defined in the machine parameters or is done loading the cam from a file.</p> <ul style="list-style-type: none"> • This state allows any cam operation, either from the CNC or from the PLC.
3	<p>CAM_START</p> <p>The cam is activated, but it has not entered the range of the master axis yet. In this situation, if the CNC or the PLC cancels the cam, the cam will not reach the CAM_READY state as long as it does not go into and out of the range of the master axis at least once.</p> <ul style="list-style-type: none"> • This state allows canceling the cam, either from the CNC or from the PLC. • This state allows deselecting the cam, but only from the CNC.
4	<p>CAM_RUNNING_OUTSIDE</p> <p>The cam is active and in execution out of the range of the master axis. In this state, the cam has been inside the range of the master axis at least once. If the CNC or the PLC cancel the cam, the cam will go into CAM_READY state.</p> <ul style="list-style-type: none"> • This state allows canceling the cam, either from the CNC or from the PLC. • This state allows deselecting the cam, but only from the CNC.
5	<p>CAM_RUNNING_INSIDE</p> <p>The cam is active and in execution inside the range of the master axis. If the CNC or the PLC cancel the cam, the cam will go into CAM_FINISH state.</p> <ul style="list-style-type: none"> • This state allows canceling the cam, either from the CNC or from the PLC. • This state allows deselecting the cam, but only from the CNC.
6	<p>CAM_FINISH</p> <p>The CNC or the PLC has canceled the cam, but it keeps executing until the cam goes out of the range of the master axis. This situation occurs when the CNC or the PLC cancel a cam that was in CAM_RUNNING_INSIDE state; if they cancel a cam that was in CAM_RUNNING_OUTSIDE state, the cam reaches the CAM_READY state.</p> <ul style="list-style-type: none"> • This state only allows deselecting the cam, but only from the CNC.

22.

CNC VARIABLES.
Variables related to the electronic cam.



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

To switch from CAM_READY to CAM_NULL, just deselect the file cam and make sure that the cam is not in the machine parameters. If the cam is defined in the machine parameters, it maintains the CAM_READY state, but the cam data will be defined in the machine parameters.

- To select a file cam, the cam status (state) must be either CAM_NULL or CAM_READY.
- To select a file cam from the PLC, preventing execution errors, the cam status (state) must be CAM_READY. If the cam is activated from the CNC, the channel will remain in execution until the cam reaches the CAM_READY state.
- To deselect a file cam from the PLC, preventing execution errors, the cam status (state) must be CAM_READY. If the cam is deselected from the CNC, the channel will remain in execution until the cam reaches the CAM_READY state.

(V.)G.CAM[cam][index]

Variable that can be read and written via program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Factor to be applied to the range of the slave axis when activating the cam.

Syntax.

·cam· Cam number (between 1 and 16).

·index· Point of the cam. For the cams defined in the machine parameters, a value between 1 and 1024. For the file cams, the line number.

V.G.CAM[2][123]

Cam ·2·. Point or line of file ·123·

22.**CNC VARIABLES.**

Variables related to the electronic cam.



CNC 8060
CNC 8065

(REF: 1402)

22.55 Variables related to the independent axes.

22.

CNC VARIABLES.

Variables related to the independent axes.

INDEPENDENT INTERPOLATOR.

(V.)[ch].A.INDPOS.xn
 (V.)[ch].A.INDPOS.sn
 (V.)[ch].SP.INDPOS.sn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Theoretical coordinate of the independent interpolator.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.INDPOS.Z	Z axis.
V.A.INDPOS.S	Spindle S.
V.SP.INDPOS.S	Spindle S.
V.SP.INDPOS	Master spindle.
V.A.INDPOS.4	Axis or spindle with logic number ·4·.
V.[2].A.INDPOS.1	Axis with index ·1· in the channel ·2·.
V.SP.INDPOS.2	Spindle with index ·2· in the system.
V.[2].SP.INDPOS.1	Spindle with index ·1· in the channel ·2·.

INDEPENDENT AXIS IN EXECUTION.

(V.)[ch].G.IBUSY

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

An independent axis is in execution.

Syntax.

- ch· Channel number.

V.[2].G.IBUSY	Channel ·2·.
---------------	--------------

PERCENTAGE OF FEEDRATE (FEEDRATE OVERRIDE).

(V.)[ch].A.FRO.xn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Channel [ch]. Feedrate percentage active on the axis.

Syntax.

- ch· Channel number.



CNC 8060
CNC 8065

(REF: 1402)

·xn· Name, logic number or index in the channel of the axis.

V.A.FRO.Z	Z axis.
V.A.FRO.4	Axis with logic number ·4·.
V.[2].A.FRO.1	Axis with index ·1· in the channel ·2·.

POSITIONING MOVE.

(V.)[ch].A.IPPOS.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Coordinate programmed for the independent axis.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index in the channel of the axis.

V.A.IPPOS.Z	Z axis.
V.A.IPPOS.4	Axis with logic number ·4·.
V.[2].A.IPPOS.1	Axis with index ·1· in the channel ·2·.

(V.)[ch].A.ITPOS.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Theoretical coordinate of the independent axis.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index in the channel of the axis.

V.A.ITPOS.Z	Z axis.
V.A.ITPOS.4	Axis with logic number ·4·.
V.[2].A.ITPOS.1	Axis with index ·1· in the channel ·2·.

(V.)[ch].A.IPRGF.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Feedrate programmed in the independent axis.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index in the channel of the axis.

V.A.IPRGF.Z	Z axis.
V.A.IPRGF.4	Axis with logic number ·4·.
V.[2].A.IPRGF.1	Axis with index ·1· in the channel ·2·.

(V.)[ch].A.IORG.xn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Offset for the independent axis.

22.

CNC VARIABLES.

Variables related to the independent axes.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the independent axes.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index in the channel of the axis.

V.A.IORG.Z	Z axis.
V.A.IORG.4	Axis with logic number -4-.
V.[2].A.IORG.1	Axis with index -1- in the channel -2-.

SYNCHRONIZATION MOVEMENT.

(V.)[ch].A.SYNCTOUT.xn
(V.)[ch].A.SYNCTOUT.sn
(V.)[ch].SP.SYNCTOUT.sn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Maximum time to establish synchronism.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.SYNCTOUT.Z	Z axis.
V.A.SYNCTOUT.S	Spindle S.
V.SP.SYNCTOUT.S	Spindle S.
V.SP.SYNCTOUT	Master spindle.
V.A.SYNCTOUT.4	Axis or spindle with logic number -4-.
V.[2].A.SYNCTOUT.1	Axis with index -1- in the channel -2-.
V.SP.SYNCTOUT.2	Spindle with index -2- in the system.
V.[2].SP.SYNCTOUT.1	Spindle with index -1- in the channel -2-.

(V.)[ch].A.SYNCVELxn
(V.)[ch].A.SYNCVEL.sn
(V.)[ch].A.SYNCVEL.sn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Synchronization speed.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.SYNCVEL.Z	Z axis.
V.A.SYNCVEL.S	Spindle S.
V.SP.SYNCVEL.S	Spindle S.
V.SP.SYNCVEL	Master spindle.
V.A.SYNCVEL.4	Axis or spindle with logic number -4-.
V.[2].A.SYNCVEL.1	Axis with index -1- in the channel -2-.
V.SP.SYNCVEL.2	Spindle with index -2- in the system.
V.[2].SP.SYNCVEL.1	Spindle with index -1- in the channel -2-.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].A.SYNCPOSW.xn
 (V.)[ch].A.SYNCPOSW.sn
 (V.)[ch].SP.SYNCPOSW.sn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Maximum position difference to start correcting it.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.SYNCPOSW.Z	Z axis.
V.A.SYNCPOSW.S	Spindle S.
V.SP.SYNCPOSW.S	Spindle S.
V.SP.SYNCPOSW	Master spindle.
V.A.SYNCPOSW.4	Axis or spindle with logic number ·4·.
V.[2].A.SYNCPOSW.1	Axis with index ·1· in the channel ·2·.
V.SP.SYNCPOSW.2	Spindle with index ·2· in the system.
V.[2].SP.SYNCPOSW.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.SYNCVELW.xn
 (V.)[ch].A.SYNCVELW.sn
 (V.)[ch].SP.SYNCVELW.sn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Maximum velocity difference to start correcting it.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.SYNCVELW.Z	Z axis.
V.A.SYNCVELW.S	Spindle S.
V.SP.SYNCVELW.S	Spindle S.
V.SP.SYNCVELW	Master spindle.
V.A.SYNCVELW.4	Axis or spindle with logic number ·4·.
V.[2].A.SYNCVELW.1	Axis with index ·1· in the channel ·2·.
V.SP.SYNCVELW.2	Spindle with index ·2· in the system.
V.[2].SP.SYNCVELW.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.SYNCPOSOFF.xn
 (V.)[ch].A.SYNCPOSOFF.sn
 (V.)[ch].SP.SYNCPOSOFF.sn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Position offset for synchronization.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

22.

CNC VARIABLES.
Variables related to the independent axes.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Variables related to the independent axes.

·sn· Name, logic number or index of the spindle.

V.A.SYNCPOSOFF.Z	Z axis.
V.A.SYNCPOSOFF.S	Spindle S.
V.SP.SYNCPOSOFF.S	Spindle S.
V.SP.SYNCPOSOFF	Master spindle.
V.A.SYNCPOSOFF.4	Axis or spindle with logic number ·4·.
V.[2].A.SYNCPOSOFF.1	Axis with index ·1· in the channel ·2·.
V.SP.SYNCPOSOFF.2	Spindle with index ·2· in the system.
V.[2].SP.SYNCPOSOFF.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.SYNCVELOFF.xn
(V.)[ch].A.SYNCVELOFF.sn
(V.)[ch].SP.SYNCVELOFF.sn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Velocity offset for synchronization.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.SYNCVELOFF.Z	Z axis.
V.A.SYNCVELOFF.S	Spindle S.
V.SP.SYNCVELOFF.S	Spindle S.
V.SP.SYNCVELOFF	Master spindle.
V.A.SYNCVELOFF.4	Axis or spindle with logic number ·4·.
V.[2].A.SYNCVELOFF.1	Axis with index ·1· in the channel ·2·.
V.SP.SYNCVELOFF.2	Spindle with index ·2· in the system.
V.[2].SP.SYNCVELOFF.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.GEARADJ.xn
(V.)[ch].A.GEARADJ.sn
(V.)[ch].SP.GEARADJ.sn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Fine adjustment of the gear ratio during synchronization.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.GEARADJ.Z	Z axis.
V.A.GEARADJ.S	Spindle S.
V.SP.GEARADJ.S	Spindle S.
V.SP.GEARADJ	Master spindle.
V.A.GEARADJ.4	Axis or spindle with logic number ·4·.
V.[2].A.GEARADJ.1	Axis with index ·1· in the channel ·2·.
V.SP.GEARADJ.2	Spindle with index ·2· in the system.
V.[2].SP.GEARADJ.1	Spindle with index ·1· in the channel ·2·.



CNC 8060
CNC 8065

(REF: 1402)

Remarks.

The reading from the PLC will be given in hundredths (x100); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·1000·.

(V.)[ch].A.SYNCERR.xn

(V.)[ch].A.SYNCERR.sn

(V.)[ch].SP.SYNCERR.sn

Variable that can be read and written via program, PLC and interface.

Variable valid for rotary and linear axes as well as for spindles.

The variable returns the execution value; reading it interrupts block preparation.

Synchronization error.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.
- sn· Name, logic number or index of the spindle.

V.A.SYNCERR.Z	Z axis.
V.A.SYNCERR.S	Spindle S.
V.SP.SYNCERR.S	Spindle S.
V.SP.SYNCERR	Master spindle.
V.A.SYNCERR.4	Axis or spindle with logic number ·4·.
V.[2].A.SYNCERR.1	Axis with index ·1· in the channel ·2·.
V.SP.SYNCERR.2	Spindle with index ·2· in the system.
V.[2].SP.SYNCERR.1	Spindle with index ·1· in the channel ·2·.

COORDINATE LATCHING WITH THE HELP OF A PROBE OR A DIGITAL INPUT.

(V.)[ch].A.LATCH1.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Machine coordinate obtained from latching probe 1 on the ·xn· axis.

Syntax.

- ch· Channel number.
- xn· Name, logic number or index of the axis.

V.A.LATCH1.Z	Z axis.
V.A.LATCH1.4	Axis with logic number ·4·.
V.[2].A.LATCH1.1	Axis with index ·1· in the channel ·2·.

(V.)[ch].A.LATCH2.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for rotary and linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Machine coordinate obtained from latching probe 2 on the ·xn· axis.

Syntax.

- ch· Channel number.

22.

CNC VARIABLES.

Variables related to the independent axes.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

·xn· Name, logic number or index of the axis.

V.A.LATCH2.Z	Z axis.
V.A.LATCH2.4	Axis with logic number ·4·.
V.[2].A.LATCH2.1	Axis with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables related to the independent axes.



CNC 8060
CNC 8065

(REF: 1402)

22.56 Variables associated with the virtual tool axis.

(V.)[ch].G.VIRTAXIS

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Logic number of the virtual tool axis.

Syntax.

·ch· Channel number.

V.[2].G.VIRTAXIS Channel ·2·.

(V.)[ch].G.VIRTAXST

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Channel [ch]. Status of the virtual tool axis: activated/deactivated.

Syntax.

·ch· Channel number.

V.[2].G.VIRTAXST Channel ·2·.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Virtual tool axis deactivated.
1	Virtual tool axis activated.

(V.)[ch].A.VIRTAXOF.xn

Variable that can only be read from the program, PLC and interface.

Variable valid for linear axes.

The variable returns the execution value; reading it interrupts block preparation.

Distance traveled by the axis, due to the movement of the virtual tool axis.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.A.VIRTAXOF.Z Z axis.
 V.A.VIRTAXOF.4 Axis or spindle with logic number ·4·.
 V.[2].A.VIRTAXOF.1 Axis with index ·1· in the channel ·2·.

22.

CNC VARIABLES.

Variables associated with the virtual tool axis.

FAGOR 

**CNC 8060
CNC 8065**

(REF: 1402)

22.57 Variables defined by the user.

22.

CNC VARIABLES.
Variables defined by the user.

(V.)P.name

Variable that can be read and written from the program.

The variable returns the value of block preparation.

User local variables.

Syntax.

.name·Name of the variable.

V.P.myvar	Variable with the name of "myvar".
-----------	------------------------------------

Remarks.

These variables maintain their value in local and global subroutines called upon from the program.

(V.)S.name

Variable that can be read and written from the program.

The variable returns the value of block preparation.

User global variables.

Syntax.

.name·Name of the variable.

V.S.myvar	Variable with the name of "myvar".
-----------	------------------------------------

Remarks.

These variables maintain their value between programs and after a reset. Variables are deleted when the CNC is turned off and they can also be deleted from the part-program using the #DELETE instruction.

#DELETE V.S.myvar1 V.S.myvar2



CNC 8060
CNC 8065

(REF: 1402)

22.58 General variables of the CNC.

HARDWARE TYPE.

(V.)G.HARDTYPE

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Hardware type.

V.G.HARDTYPE

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Simulator
2	ICU central unit.

EMERGENCY RELAY.

(V.)G.ERELAYST

Variable that can only be read from the program, PLC (asynchronous) and interface.
The variable returns the execution value; reading it interrupts block preparation.

Status of the emergency relay.

V.G.ERELAYST

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	Open relay.
1	Closed relay.

SOFTWARE VERSION.

(V.)G.VERSION

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

CNC version and release number.

V.G.VERSION

Values of the variable.

This variable returns a decimal value; to understand the meaning of the variables, resource to their hexadecimal format. The high portion of this variable contains the software version and its low portion the release.

V.G.VERSION	Hexadecimal value.	Meaning.
778	\$30A	Version 3.10 Software version: \$300 Software release: \$0A

22.

CNC VARIABLES.
General variables of the CNC.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

DATE, TIME AND AMOUNT OF TIME OF TURNED ON.

(V.)G.DATE

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Date in year-month-day format.

V.G.DATE

Values of the variable.

If the date is "April 25th 1999", the returned value will be 990425.

Remarks.

This variable updates its value every hour.

(V.)G.TIME

Variable that can only be read from the program, PLC (asynchronous) and interface.
The variable returns the execution value; reading it interrupts block preparation.

Time in hours-minutes-seconds format.

V.G.TIME

Values of the variable.

If the time is "18hr 22min 34s", the returned value will be 182234.

(V.)G.CLOCK

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Seconds since the CNC was turned on.

V.G.TIME

INFORMATION ABOUT THE CHANNELS.

(V.)[ch].G.CNCHANNEL

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel number.

Syntax.

·ch· Channel number.

V.[2].G.CNCHANNEL Channel ·2·.

(V.)G.FOCUSCHANNEL

Variable that can be read and written from the PLC and from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).

Channel with active focus.

V.G.FOCUSCHANNEL

22.

CNC VARIABLES.
General variables of the CNC.



CNC 8060
CNC 8065

(REF: 1402)

KEY SIMULATION.

(V.)G.KEY

Variable that can be read and written from the PLC and read from the program and from the interface. The variable returns the execution value; reading it interrupts block preparation.

Code of the last key accepted by the CNC.

V.G.KEY

Remarks.

To read the last key that has been accepted by the CNC or simulate the keyboard from the PLC by writing in it the code of the desired key.

22.

CNC VARIABLES.
General variables of the CNC.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.59 Variables related to CNC status.

CNC STATUS.

(V.)[ch].G.STATUS

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

CNC status (brief).

Syntax.

·ch· Channel number.

V.[2].G.STATUS Channel -2.

Values of the variable.

This variable returns a decimal value; to understand the meaning of the variables, resource to their hexadecimal format. The list of codes is the following.

Decimal.	Hexadecimal.	Meaning.
0	\$0	Not Ready
1	\$1	Ready.
2	\$2	In execution.
4	\$4	Interrupted.
8	\$8	In error.

(V.)[ch].G.FULLSTATUS

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

CNC status (detailed).

Syntax.

·ch· Channel number.

V.[2].G.FULLSTATUS Channel -2.

Value of the variable.

This variable returns a decimal value; to understand the meaning of the variables, resource to their hexadecimal format. The high portion of this variable contains the information of the STATUS variable and its low portion provides the substate of the CNC; FULLSTATUS = (STATUS)(substate). Thus, if the variable returns a value of 514, it means the following.

Decimal.	Hexadecimal.	Status.	Substate.
514	\$0202	\$0200 In execution.	\$02 In MDI.

22.

CNC VARIABLES.
Variables related to CNC status.



CNC 8060
CNC 8065

(REF: 1402)

The list of codes for the low portion of FULLSTATUS is: In reset state, the low portion of FULLSTATUS is \$0, in jog mode is \$1, in simulation mode is \$D and so on.

Decimal.	Hexadecimal.	Meaning.
0	\$0	In Reset
1	\$1	In Jog mode
2	\$2	In MDI mode
3	\$3	In program
4	\$4	Stopped by an M0
5	\$5	Stopped with CYCLE STOP
6	\$6	Stopped in Single block mode
9	\$9	Syntax checks.
10	\$A	Block search (without moving the axes)
11	\$B	Block search finished. In stand by
12	\$C	Execution time estimate.
13	\$D	In simulation.

Example of some values of the FULLSTATUS variable.

Decimal.	Hexadecimal.	Status.	Substate.
521	\$209	\$200 In execution.	\$09 Syntax checks.
522	\$20A	\$200 In execution.	\$0A Block search.
524	\$20C	\$200 In execution.	\$0C Execution time estimate.
525	\$20D	\$200 In execution.	\$0D In simulation.
515	\$203	\$200 In execution.	\$03 In program.

SELECTED AXES.

(V.)[ch].G.SELECTEDAXIS

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Axes selected for home search, repositioning, coordinate preset or movement to a coordinate.

Syntax.

.ch- Channel number.

V.[2].G.SELECTEDAXIS Channel -2.

Values of the variable.

The variable returns a 32-bit value where each bit represents one axis; the least significant bit corresponds to the axis with the lowest logic number. Each bit indicates whether the axis is selected (bit = 1) or not (bit = 0).

22.

CNC VARIABLES.
Variables related to CNC status.



CNC 8060
CNC 8065

(REF: 1402)

DETAILED CNC STATUS IN JOG MODE.

(V.) [ch].G.CNCMANSTATUS

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

Detailed CNC status in jog mode.

Syntax.

.ch- Channel number.

V.[2].G.CNCMANSTATUS Channel -2.

Values of the variable.

This variable returns a hexadecimal value with the following meaning.

Value.	Meaning.
\$1	Jog mode.
\$2	Jog or handwheel movements.
\$4	Manual tool calibration.
\$8	Part centering (mill model).
\$10	Home search with subroutine. The variable takes this value when pressing the [ZERO] key and maintains it until the CNC returns to the READY state after the home search.
\$20	Home search on an axis. The variable takes this value when pressing the [ZERO] key with an axis selected and maintains it until the CNC returns to the READY state after the home search.
\$40	"T" field selected. The variable removes the bit when the CNC returns to the READY state after pressing [START].
\$80	"F" field selected. The variable removes the bit when the CNC returns to the READY state after pressing [START].
\$100	"S" field selected. The variable removes the bit when the CNC returns to the READY state after pressing [START].
\$200	Axis ready for coordinate preset or for movement to a coordinate. Axis selected and coordinate set, waiting for [ENTER] or [START].
\$1000000	MDI/MDA mode.
\$2000000	CNCEX.

Example of some values of the variable.

CNC status.	Variables.
Tool calibration screen is active.	V.G.CNCMANSTATUS = \$5
An axis has been selected for home search, but [START] has not been pressed.	V.G.CNCMANSTATUS = \$21 V.G.STATUS = \$1 (READY)
A home search is in progress on an axis.	V.G.CNCMANSTATUS = \$21 V.G.STATUS = \$2 (IN EXECUTION)
Home search on an axis has been interrupted.	V.G.CNCMANSTATUS = \$21 V.G.STATUS = \$4 (INTERRUPTED)
The CNC is executing a block in MDI/MDA from jog mode.	V.G.CNCMANSTATUS=\$1000001 V.G.STATUS = \$2 (IN EXECUTION)
The execution of a block in MDI/MDA from jog mode has been interrupted.	V.G.CNCMANSTATUS=\$1000001 V.G.STATUS = \$4 (INTERRUPTED)

22.

CNC VARIABLES.
Variables related to CNC status.



CNC 8060
CNC 8065

(REF: 1402)

DETAILED CNC STATUS IN AUTOMATIC MODE.

(V.)[ch].G.CNCAUTSTATUS

Variable that can only be read from the program, PLC and interface.
 The variable returns the execution value; reading it interrupts block preparation.
 Report variable (to be used from the scripts).

Detailed CNC status in automatic mode.

Syntax.

·ch· Channel number.

V.[2].G.CNCAUTSTATUS Channel -2.

Values of the variable.

This variable returns a hexadecimal value with the following meaning.

Value.	Meaning.
\$1	Automatic mode or MDI/MDA mode over the automatic mode.
\$2	First block selected.
\$4	Stop condition selected.
\$8	The option to execute the blocks of the program independently (EXBLK softkey) is active.
\$10	Block search is selected.
\$20	The M H F S function display is selected (during block search or tool inspection).
\$40	The M H F S function selection is active (during tool inspection).
\$80	Tool inspection is selected.
\$100	The "cancel and resume" option is active.
\$200	Axis repositioning is selected (during block search or tool inspection).
\$400	An axis has been selected for repositioning.
\$800	The CNC is executing an interruption subroutine.
\$1000	Simulated execution is active. Theoretical travel.
\$2000	Simulated execution is active. G functions.
\$4000	Simulated execution is active. GMST functions.
\$8000	Simulated execution is active. Main plane.
\$10000	Simulated execution is active. Rapid.
\$20000	Simulated execution is active. Rapid S=0.
\$40000	The cycle editor is executing a cycle.
\$80000	The cycle editor is simulating a cycle.
\$10000000	MDI/MDA mode active over the automatic mode.
\$20000000	The PLC is executing a CNCEX command.

Example of some values of the variable.

CNC status.	Variables.
There is a program in execution.	V.G.CNCAUTSTATUS = \$1 V.G.STATUS = \$2 (IN EXECUTION)
The execution has been interrupted and tool inspection is active.	V.G.CNCAUTSTATUS = \$81 V.G.STATUS = \$4 (INTERRUPTED)
Tool inspection is active and there is a jog movement.	V.G.CNCAUTSTATUS = \$81 V.G.CNCMANSTATUS = \$2 V.G.STATUS = \$2 (IN EXECUTION)

22.

CNC VARIABLES.
 Variables related to CNC status.



CNC 8060
 CNC 8065

(REF: 1402)

CNC status.	Variables.
Tool inspection is active and an axis has been selected for repositioning.	V.G.CNCAUTSTATUS = \$681 V.G.STATUS = \$4 (INTERRUPTED)
The CNC is executing a block in MDI/MDA from automatic mode.	V.G.CNCAUTSTATUS = \$10000001 V.G.STATUS = \$2 (IN EXECUTION)
The execution of a block in MDI/MDA from automatic mode has been interrupted.	V.G.CNCAUTSTATUS=\$10000001 V.G.STATUS = \$4 (INTERRUPTED)

22.

CNC VARIABLES.

Variables related to CNC status.



CNC 8060
CNC 8065

(REF: 1402)

22.60 Variables associated with the part-program being executed.**PART-PROGRAM INFORMATION.****(V.)[ch].G.FILENAME**

Variable to be read via interface.

Report variable (to be used from the scripts).

Name of the program in execution.

Syntax.

.ch. Channel number.

[2].G.FILENAME	Channel -2.
----------------	-------------

(V.)[ch].G.PRGPATH

Variable to be read via interface.

Report variable (to be used from the scripts).

Path of the program in execution.

Syntax.

.ch. Channel number.

[2].G.PRGPATH	Channel -2.
---------------	-------------

(V.)[ch].G.FILEOFFSET

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Number of characters up to the line being executed or prepared.

Syntax.

.ch. Channel number.

V.[2].G.FILEOFFSET	Channel -2.
--------------------	-------------

Remarks.

The variable returns the value of execution or preparation as follows. Reading the variable from the PLC or from the interface returns the value of the program being executed; reading it from the part-program or MDI returns the program being prepared.

(V.)[ch].G.BLKN

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Last block executed (number).

If no label has been executed, the variable will return a -1 value.

Syntax.

.ch. Channel number.

V.[2].G.BLKN	Channel -2.
--------------	-------------

(V.)[ch].G.LINEN

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Number of program blocks executed or prepared.

22.**CNC VARIABLES.**

Variables associated with the part-program being executed.

FAGOR **CNC 8060
CNC 8065**

(REF: 1402)

22.

CNC VARIABLES.
Variables associated with the part-program being executed.

Syntax.

.ch· Channel number.

V.[2].G.LINEN Channel -2.

Remarks.

The variable returns the value of execution or preparation as follows. Reading the variable from the PLC or from the interface returns the number of blocks executed; reading it from the part-program or MDI returns the number of blocks prepared.

INFORMATION ON PROGRAM EXECUTION.**(V.)[ch].G.FIRST**

Variable that can be read and written from the interface and read from the program and from the PLC. The variable returns the execution value; reading it interrupts block preparation.

First time a program is executed.

Syntax.

.ch· Channel number.

V.[2].G.FIRST Channel -2.

Values of the variable.

This variable returns one of the following values.

Value.	Meaning.
0	No.
1	Yes.

Remarks.

First time execution means every time a new program is selected. This variable is initialized when changing the program being executed in the channel, even with the #EXEC instruction.

When the instruction #EXEC is involved in a program, the variable behaves as follows:

- 1 The variable is initialized when selecting and executing the program.
- 2 When executing the #EXEC instruction, the variable is re-initialized because the program in execution has changed.
- 3 If then, this program is executed again, the program in execution changes again and the variable is updated.

(V.)[ch].G.PARTC

Variable that can be read and written via program, PLC and interface. The variable returns the execution value; reading it interrupts block preparation. Report variable (to be used from the scripts).

Parts counter.

Syntax.

.ch· Channel number.

V.[2].G.PARTC Channel -2.

Remarks.

This variable is initialized when changing the program being executed and it increments its value every time an M02 or M30 is executed. This variable is initialized when changing the program being executed in the channel, even with the #EXEC instruction.

When the instruction #EXEC is involved in a program, the variable behaves as follows:

- 1 The variable is initialized when selecting and executing the program.
- 2 When executing the #EXEC instruction, the variable is re-initialized because the program in execution has changed.
- 3 If then, this program is executed again, the program in execution changes again and the variable is updated.

In this case, to keep track of how many times the program has been executed, it is recommended to use an arithmetic parameter at the end of the program like a counter.

(V.)[ch].G.CYTIME

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Part-program execution time (in hundredths of a second).

Syntax.

.ch· Channel number.

V.[2].G.CYTIME	Channel -2.
----------------	-------------

Remarks.

This variable is set to 0 at every new execution even of the same program. It does not measure the time that execution has been stopped.

EXECUTING OPTIONS; SINGLE BLOCK, RAPID, ETC..

(V.)[ch].G.SBOUT

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Report variable (to be used from the scripts).

"Single block" function activated.

Syntax.

.ch· Channel number.

V.[2].G.SBOUT	Channel -2.
---------------	-------------

Remarks.

The single block function may be activated or canceled from the keyboard or from the PLC (SBLOCK mark). To activate it, just set one of them high (=1), but to cancel it both must be low (=0).

(V.)[ch].G.SBLOCK

Variable that can be read and written from the interface and read from the program and from the PLC.

The variable returns the execution value; reading it interrupts block preparation.

"Single block" function requested via keyboard.

Syntax.

.ch· Channel number.

V.[2].G.SBLOCK	Channel -2.
----------------	-------------

22.

CNC VARIABLES.

Variables associated with the part-program being executed.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].G.BLKSKIP

Variable that can be read and written from the interface and read from the program and from the PLC. The variable returns the execution value; reading it interrupts block preparation.

Block skip function (\) activated.

Syntax.

·ch· Channel number.

V.[2].G.BLKSKIP Channel ·2·.

Remarks.

The "block skip" function is selected from the PLC using the BLKSKIP1 mark.

(V.)[ch].G.M01STOP

Variable that can be read and written from the interface and read from the program and from the PLC. The variable returns the execution value; reading it interrupts block preparation.

Conditional stop function (M01) activated.

Syntax.

·ch· Channel number.

V.[2].G.M01STOP Channel ·2·.

Remarks.

The "conditional stop" function is activated from the PLC using the M01STOP mark.

(V.)[ch].G.RAPID

Variable that can be read and written from the interface and read from the program and from the PLC. The variable returns the execution value; reading it interrupts block preparation. Report variable (to be used from the scripts).

"Rapid traverse" function activated.

Syntax.

·ch· Channel number.

V.[2].G.RAPID Channel ·2·.

Remarks.

The "rapid traverse" function is selected from the PLC using the MANRAPID mark.

22.

CNC VARIABLES.
Variables associated with the part-program being executed.



CNC 8060
CNC 8065

(REF: 1402)

22.61 Interface related variables.

(V.)[ch].E.PROGSELECT

Variable that can be read and written via program, PLC and interface.

Report variable (to be used from the scripts).

Name of the program selected in automatic.

This variable can only be written with the value of `.0` to eliminate the program selected in automatic mode. When simulating the program, writing this variable has no effect.

Syntax.

`.ch` Channel number.

[2].E.PROGSELECT Channel `.2`.

(V.)E.COMPONENTNAME

Variable to be read via interface.

Report variable (to be used from the scripts).

Component name.

G.COMPONENTNAME

(V.)[ch].E.MMCMODE

Variable to be read via interface.

Report variable (to be used from the scripts).

Syntax.

`.ch` Channel number.

[2].E.MMCMODE Channel `.2`.

(V.)E.NERRORS

Variable to be read via interface.

Report variable (to be used from the scripts).

Number of errors at the CNC.

G.NERRORS

(V.)E.ERRORPRI

Variable to be read via interface.

Report variable (to be used from the scripts).

Error priority.

G.ERRORPRI

(V.)[ch].E.MANUALMODE

Variable to be read via interface.

Report variable (to be used from the scripts).

Exclusive jog mode active.

Syntax.

`.ch` Channel number.

[2].E.MANUALMODE Channel `.2`.

22.

CNC VARIABLES.
Interface related variables.

FAGOR 

CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.
Interface related variables.

(V.)E.CALCRESUL

Variable to be read via interface.
Report variable (to be used from the scripts).

Result of the calculator.

G.CALCRESULT

(V.)E.NUMJOGPANELS

Variable to be read via interface.

Number of jog panels.

G.NUMJOGPANELS

(V.)E.HELpdata

Variable that can be read and written from the interface.
Report variable (to be used from the scripts).

Mark of the help file corresponding to the help page.

G.HELpdata

(V.)E.CNCMSG

Variable to be read via interface.
Report variable (to be used from the scripts).

Error text.

G.CNCMSG

(V.)[ch].E.PATHSELECT

Variable to be read via interface.
Report variable (to be used from the scripts).

Path of the program selected in automatic.

Syntax.

·ch· Channel number.

[2].E.PATHSELECT Channel -2.

(V.)[ch].E.CMPNTNUMBER

Variable to be read via interface.
Report variable (to be used from the scripts).

Component where the channel is.

Syntax.

·ch· Channel number.

[2].E.CMPNTNUMBER Channel -2.

(V.)[ch].E.PAGENUMBER

Variable to be read via interface.
Report variable (to be used from the scripts).

Page where the channel is.

Syntax.

·ch· Channel number.

[2].E.PAGENUMBER Channel -2.



CNC 8060
CNC 8065

(REF: 1402)

(V.)[ch].E.GRAPHTYPE

Variable to be read via interface.

Report variable (to be used from the scripts).

Graphics configuration selected in parameter GRAPHTYPE.

Syntax.

·ch· Channel number.

[2].E.GRAPHTYPE

Channel ·2·.

22.

CNC VARIABLES.

Interface related variables.



CNC 8060
CNC 8065

(REF: 1402)

22.

CNC VARIABLES.

Interface related variables.



CNC 8060
CNC 8065

(REF: 1402)



CNC 8060
CNC 8065

(REF. 1402)



CNC 8060
CNC 8065

(REF. 1402)



CNC 8060
CNC 8065

(REF. 1402)



CNC 8060
CNC 8065

(REF. 1402)