

CNC

8070

Installation manual.

(Ref: 1709)



FAGOR AUTOMATION

TRANSLATION OF THE ORIGINAL MANUAL

This manual is a translation of the original manual. This manual, as well as the documents derived from it, have been drafted in Spanish. In the event of any contradictions between the document in Spanish and its translations, the wording in the Spanish version shall prevail. The original manual will be labeled with the text "ORIGINAL MANUAL".

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 the following 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.

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.

DUAL-USE PRODUCTS

Products manufactured by FAGOR AUTOMATION since April 1st 2014 will include "-MDU" in their identification if they are included on the list of dual-use products according to regulation UE 428/2009 and require an export license depending on destination.



FAGOR AUTOMATION

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 subject to changes due to technical modifications. Fagor Automation reserves the right to change 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, hence no absolute match is guaranteed. However, the contents of this document are regularly checked and updated implementing the necessary corrections in a later 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 - CNC 8070 11
 Declaration of CE conformity and warranty conditions 15
 Version history - CNC 8070 17
 Safety conditions 33
 Returning conditions 37
 CNC maintenance 39

CHAPTER 1 SOFTWARE INSTALLATION.

1.1 Software installation at the CNC 43
 1.1.1 Work modes and software protection at the CNC 45
 1.2 Software installation at the PC (simulator) 48
 1.3 Updating the software version 50
 1.3.1 Software update since previous versions 51
 1.4 Requirements before and after CNC setup 54
 1.5 Installing third-party software 55
 1.6 Software configuration 56
 1.6.1 MTB (Machine Tool Builder) folder 57
 1.6.2 USERS folder 58

CHAPTER 2 MACHINE PARAMETERS.

2.1 Parameter matching between the CNC and the Sercos drive 61
 2.2 Parameters to verify before the startup 63
 2.3 General machine parameters 65
 2.3.1 Channel configuration 65
 2.3.2 Configuring the axes of the system 66
 2.3.3 Configuration of a tandem system 67
 2.3.4 Configuration of a gantry axis 71
 2.3.5 Configuration of a multi-axis group 74
 2.3.6 Configuring the spindles of the system 76
 2.3.7 Time setting (system) 77
 2.3.8 Sercos bus configuration 79
 2.3.9 Mechatrolink bus configuration 80
 2.3.10 CAN bus configuration 81
 2.3.11 Serial line configuration 82
 2.3.12 MODBUS 83
 2.3.13 Default conditions (system) 84
 2.3.14 Arithmetic parameters 85
 2.3.15 Cross compensation table 87
 2.3.16 Volumetric compensation tables 90
 2.3.17 Execution times 93
 2.3.18 Numbering of the digital inputs (CANfagor bus) 94
 2.3.19 Numbering of the digital outputs (CANfagor bus) 95
 2.3.20 Numbering of the digital inputs (CANopen bus) 96
 2.3.21 Numbering of the digital outputs (CANopen bus) 98
 2.3.22 Numbering of analog inputs for temperature sensors PT100 100
 2.3.23 Probe setting 101
 2.3.24 Shared PLC memory 103
 2.3.25 Management of local I/O 103
 2.3.26 Synchronized switching 104
 2.3.27 PWM (Pulse-Width Modulation) 105
 2.3.28 Power control 106
 2.3.29 Gap control 107
 2.3.30 Leapfrog 112
 2.3.31 Compensating the dispersion by the CO2 laser path 113
 2.3.32 Backup of non-volatile data 115
 2.3.33 Tool offset and wear 115
 2.3.34 Spindle synchronization 116
 2.3.35 Define the number of jog panels and their relationship with the channels 116
 2.3.36 PLC type 117
 2.3.37 Rename the axes and the spindles 117
 2.3.38 Zero offsets 118
 2.3.39 Remote module RCS-S (Sercos Counter) 118
 2.3.40 Access to the parameter table of the channel 119
 2.4 General machine parameters. Execution channels 120



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4.1	Channel configuration.....	120
2.4.2	Configuring the axes of the channel.....	121
2.4.3	Configuring the spindles of the channel.....	124
2.4.4	Configuration of the C axis.....	125
2.4.5	Time setting (channel).....	126
2.4.6	Configuration of the HSC mode (channel).....	127
2.4.7	Virtual tool axis.....	131
2.4.8	Default conditions (channel).....	132
2.4.9	Arc center correction.....	139
2.4.10	Behavior of the feedrate and the feedrate override.....	140
2.4.11	Override of the dynamics for HSC.....	141
2.4.12	Movement of the independent axes.....	141
2.4.13	Definition of the subroutines.....	142
2.4.14	Tabletop probe position.....	145
2.4.15	Block search.....	147
2.4.16	Interruption subroutines.....	147
2.4.17	Machining feedrate.....	148
2.4.18	Rapid traverse for the automatic mode.....	149
2.4.19	Maximum acceleration and jerk on the tool path.....	150
2.4.20	Maximum frequency on the tool path.....	150
2.4.21	Resonance frequency of the machine.....	151
2.4.22	"Retrace" function.....	151
2.4.23	Withdrawal the tool.....	152
2.4.24	Master spindle.....	152
2.5	Machine parameters for the axes and spindles.....	153
2.5.1	Belonging to the channel.....	153
2.5.2	Type of axis and drive.....	154
2.5.3	Configuring a Sercos drive.....	155
2.5.4	Configuring a Mechatrolink drive.....	158
2.5.5	Hirth axis.....	159
2.5.6	Axis configuration for lathe type machines.....	160
2.5.7	Synchronization of axes and spindles.....	161
2.5.8	Configuration of the rotary axes.....	162
2.5.9	Module configuration (rotary axes and spindle).....	164
2.5.10	Configuration of the C axis.....	165
2.5.11	Configuration of the spindle.....	166
2.5.12	Spindle override change while threading.....	168
2.5.13	Software axis limits.....	169
2.5.14	Work zones.....	170
2.5.15	Runaway protection and tendency test.....	170
2.5.16	PLC Offset.....	171
2.5.17	Dwell for dead axes.....	172
2.5.18	Radius / diameter.....	173
2.5.19	Home search.....	174
2.5.20	Configuration of the probing movement.....	176
2.5.21	Repositioning of the axes in tool inspection.....	178
2.5.22	Configuration of the independent axis.....	178
2.5.23	Configure the maximum safety limit for the feedrate and for the speed.....	179
2.5.24	JOG mode.....	180
2.5.25	Leadscrew error compensation.....	185
2.5.26	Filters to eliminate resonance frequency.....	188
2.5.27	Work sets.....	191
2.6	Machine parameters for the axes. Work sets.....	192
2.6.1	Feedback resolution.....	192
2.6.2	Loop setting.....	196
2.6.3	Backlash compensation.....	196
2.6.4	Backlash compensation with an additional command pulse.....	197
2.6.5	Adjustment of rapid traverse G00 and maximum speed.....	200
2.6.6	Rapid traverse for the automatic mode.....	202
2.6.7	Gain setting.....	203
2.6.8	Linear acceleration.....	207
2.6.9	Trapezoidal and square sine acceleration.....	209
2.6.10	Enable specific acceleration values for movements in G0.....	212
2.6.11	Linear acceleration (G0 movements).....	213
2.6.12	Trapezoidal and square sine acceleration (G0 movements).....	214
2.6.13	Configuration of the HSC mode.....	216
2.6.14	Home search.....	218
2.6.15	Following error.....	223
2.6.16	Axis lubrication.....	226
2.6.17	Module configuration (rotary axes and spindle).....	227
2.6.18	Spindle speed.....	228
2.6.19	Analog command setting.....	229
2.6.20	Number of the analog output and of the feedback input associated with the axis... ..	230
2.6.21	Set the drive associated with the axes of a multi-axis group.....	233
2.6.22	Feedback type.....	233



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.23	Encoder Information.....	234
2.6.24	EnDat transmission format.....	235
2.6.25	SSI Feedback type.....	236
2.6.26	Delay estimate at the drive.....	241
2.6.27	Correction of bus and drive delay.....	241
2.7	Machine parameters for JOG mode.....	242
2.7.1	Handwheel configuration.....	242
2.7.2	Configure the jog keys.....	244
2.7.3	Configure the user keys as jog keys.....	246
2.7.4	HBL5 portable operator panel.....	247
2.7.5	Example of how to set the handwheels and jog keys.....	248
2.8	Machine parameters for the M function table.....	252
2.8.1	M function table.....	252
2.9	Machine parameters for kinetics.....	254
2.9.1	Kinematics configuration.....	255
2.9.2	Definition of the spindle kinetics (types 1 through 8).....	259
2.9.3	Definition of the table kinetics (types 9 through 12).....	263
2.9.4	Definition of the kinematics of the spindle - table (types 13 through 16).....	268
2.9.5	Definition of the spindle kinetics (types 17 through 24).....	273
2.9.6	Definition of the -C- axis kinematics (types 41 through 42).....	277
2.9.7	Definition of the -C- axis kinematics (type 43).....	279
2.9.8	Vectorial definition of spindle kinematics (type 50).....	280
2.9.9	Vectorial definition of table kinematics (type 51).....	284
2.9.10	Vectorial definition of spindle-table kinematics (type 52).....	288
2.9.11	Definition of the OEM kinematics (types 100 through 105).....	295
2.9.12	Configuration of angular transformations.....	296
2.9.13	Configuration of angular transformations (parameters).....	297
2.10	Machine parameters for the magazine.....	299
2.10.1	Tool magazine configuration.....	299
2.10.2	Magazine data.....	300
2.10.3	Tool magazine management.....	301
2.10.4	Types of tool magazine.....	303
2.11	Machine parameters for HMI (Interface).....	305
2.11.1	Main window dimensions and resolution.....	305
2.11.2	Customizing the softkeys.....	306
2.11.3	Interface setting.....	307
2.11.4	Keyboard configuration.....	309
2.11.5	Simulated jog keyboard.....	312
2.11.6	CNC shut down.....	312
2.11.7	Graphics configuration.....	313
2.12	OEM machine parameters.....	314
2.12.1	Reading drive (Sercos or Mechatrolink) variables.....	314
2.12.2	Generic OEM parameters.....	316
2.12.3	Cam editor.....	318

CHAPTER 3 INTRODUCTION TO THE PLC.

3.1	PLC program.....	320
3.2	Modular structure of the PLC program.....	321
3.3	PLC program execution.....	322
3.4	PLC resources.....	323
3.4.1	Numbering of the physical inputs and outputs.....	326
3.5	Operation of a timer.....	328
3.5.1	Monostable mode. TG1 input.....	330
3.5.2	Delayed activation mode. TG2 input.....	332
3.5.3	Delayed deactivation mode. TG3 input.....	334
3.5.4	Signal limiting mode. TG4 Input.....	336
3.6	Operation of a counter.....	338

CHAPTER 4 PLC PROGRAMMING.

4.1	Directing instructions.....	343
4.2	Consulting instructions.....	347
4.2.1	Simple consulting instructions.....	347
4.2.2	Flank detection instructions.....	348
4.2.3	Comparing instructions.....	349
4.3	Operators and symbols.....	350
4.4	Action instructions.....	351
4.4.1	Assignment binary instructions.....	352
4.4.2	Conditional binary instructions.....	353
4.4.3	Sequence breaking action instructions.....	354
4.4.4	Arithmetic action instructions.....	355
4.4.5	Logic action instructions.....	357
4.4.6	Specific action instructions.....	359
4.4.7	Action instruction of the electronic cam.....	362



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.4.8	Instructions for independent move: positioning.	364
4.4.9	Instructions for independent move: synchronization.	366
4.4.10	Instructions for coordinate latching using a probe or digital input.	368
4.5	Considerations for some functions.	372
4.5.1	Work zones.	372
4.6	Summary programming commands.	374

CHAPTER 5**CNC-PLC COMMUNICATION.**

5.1	Auxiliary –M– functions.	378
5.1.1	Special considerations with the multi-spindle option and channels.	379
5.2	Auxiliary –H– functions.	380
5.2.1	Special considerations with the multi-spindle option and channels.	381
5.3	Auxiliary –S– function.	382
5.3.1	Special considerations with the multi-spindle option and channels.	383
5.4	Transferring auxiliary functions -M-, -H-, -S-.	384
5.4.1	Synchronized transfer.	385
5.4.2	Non-synchronized transfer.	386
5.5	Displaying PLC errors and messages.	387

CHAPTER 6**LOGIC CNC INPUTS AND OUTPUTS.**

6.1	General consulting signals.	390
6.2	Consulting signals for axes and spindles.	400
6.3	Consulting signals for the spindle.	406
6.4	Consultation signals of the independent interpolator.	408
6.5	Consulting logic signals; laser.	410
6.6	Tool manager consulting signals.	412
6.7	Keystroke consulting signals.	414
6.8	General modifiable signals.	418
6.9	Modifiable signals for axes and spindles.	429
6.10	Spindle modifiable signals.	433
6.11	Modifiable logic signals; laser.	437
6.12	Modifiable signals of the independent interpolator.	439
6.13	Tool manager modifiable signals.	440
6.14	Keystroke modifiable signals.	443

CHAPTER 7**TOOL AND MAGAZINE MANAGEMENT.**

7.1	Types of tool magazine.	451
7.2	Tool table, active tool table and tool magazine table.	453
7.3	Communication between manager and PLC.	454
7.3.1	Manager --> PLC communication.	455
7.3.2	PLC --> Manager communication.	456
7.3.3	Manager Emergency.	458
7.3.4	Tool monitoring.	459
7.4	Variables related to tool magazine management.	460
7.5	Tool loading and unloading from the magazines.	461
7.6	Magazine-less system.	462
7.6.1	Valid operations and marks activated by the PLC with each one of them.	463
7.6.2	Detailed description of the operations of the magazine.	464
7.6.3	Basic PLC programming.	464
7.7	Turret type magazine.	465
7.7.1	Valid operations and marks activated by the PLC with each one of them.	466
7.7.2	Detailed description of the operations of the magazine.	468
7.7.3	Communication between the PLC and the M06 subroutine.	470
7.7.4	Program of the M06 subroutine.	471
7.7.5	Basic PLC programming.	473
7.8	Synchronous magazine without changer arm.	474
7.8.1	Valid operations and marks activated by the PLC with each one of them.	475
7.8.2	Detailed description of the operations of the magazine.	477
7.8.3	Communication between the PLC and the M06 subroutine.	480
7.8.4	Program of the M06 subroutine.	481
7.8.5	Basic PLC programming.	483
7.9	Synchronous magazine with changer arm and 1 claw.	484
7.9.1	Valid operations and marks activated by the PLC with each one of them.	485
7.9.2	Detailed description of the operations of the magazine.	487
7.9.3	Communication between the PLC and the M06 subroutine.	490
7.9.4	Program of the M06 subroutine.	491
7.9.5	Basic PLC programming.	494
7.10	Synchronous magazine with changer arm and 2 claws.	496
7.10.1	Valid operations and marks activated by the PLC with each one of them.	497
7.10.2	Detailed description of the operations of the magazine.	499
7.10.3	Communication between the PLC and the M06 subroutine.	502



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.10.4	Program of the M06 subroutine.....	503
7.10.5	Basic PLC programming.....	506
7.11	Asynchronous magazine with changer arm.....	508
7.11.1	Valid operations and marks activated by the PLC with each one of them.....	509
7.11.2	Detailed description of the operations of the magazine.....	511
7.11.3	Communication between the PLC and the M06 subroutine.....	515
7.11.4	Program of the M06 subroutine.....	516
7.11.5	Basic PLC programming.....	519

CHAPTER 8 KEYBOARD SIMULATION VIA THE PLC. KEY CODES.

8.1	Selecting the language and the keyboard distribution.....	521
8.2	Key codes.....	523
8.2.1	Keyboard shortcuts. CNC's own keys.....	524
8.3	Consulting the last key accepted by the CNC.....	525
8.4	Keyboard simulation via the PLC.....	525
8.5	Key codes depending on keyboard layout.....	527
8.5.1	Scan codes. "Spanish (Spain)" keyboard layout.....	527
8.5.2	Scan codes. Keyboard distribution "English (United States)".	528
8.6	Example. Keyboard simulation via the PLC.....	529

CHAPTER 9 CNC VARIABLES.

CHAPTER 10 GENERAL SETUP.

10.1	Configure the name and number of axes and spindles.....	535
10.1.1	Configure the number of axes and spindles of the system.....	536
10.1.2	Configure the number of axes and spindles of the channels.....	537
10.2	Configuration examples.....	538
10.2.1	Milling machine with 1 channel, 3 axes and 1 spindle.....	539
10.2.2	Milling machine with 1 channel, 5 axes (2 free) and 1 spindle.....	540
10.2.3	Milling machine with 3 channels, 9 axes and 2 spindles.....	541
10.2.4	Lathe with 1 channel, 3 axes and 1 spindle. Configuration of "plane" type axes.....	543
10.2.5	Lathe with 1 channel, 3 axes and 1 spindle. Configuration of trihedron type axes.....	544
10.2.6	Lathe with 1 channel, 3 axes (1 free) and 1 spindle. Configuration of "plane" type axes. 545	
10.2.7	Lathe with 2 channels, 4 axes and 2 spindles. Configuration of "plane" type axes.....	546
10.2.8	Lathe with 3 channels, 6 axes and 2 spindles. Configuration of "plane" type axes.....	547

CHAPTER 11 CONFIGURING AN AXIS.

11.1	Configure an axis as rotary axis.....	549
11.2	Configure two axes as a tandem axis.....	552
11.2.1	Tandem axis configuration. Machine parameters.....	553
11.2.2	Effect of the preload.....	555
11.2.3	Tandem axis configuration. Block diagram.....	557
11.2.4	Tandem related variables.....	559
11.2.5	Tandem adjustment procedure.....	560
11.3	Analog axes.....	561
11.3.1	Configure the number of the analog output and of the feedback input.....	561
11.3.2	Configure 2 axes with the same feedback input and analog output.....	563
11.4	Multi-axis management.....	564
11.4.1	Configuration of a multi-axis group. Machine parameters.....	566
11.4.2	Configuration of a multi-axis group. The PLC routine generates an error.....	569
11.4.3	Changing the set and the gear at the CNC and at the drive.....	570
11.4.4	Configuration examples.....	571
11.5	Home search.....	573
11.5.1	Home search (axes and spindles).....	575
11.5.2	Home search (gantry axes).....	578
11.6	Software limits of the axes.....	580
11.6.1	How to set the software travel limits.....	582
11.6.2	Set the tolerance for an axis located at the software travel limits.....	584

CHAPTER 12 VOLUMETRIC COMPENSATION.

12.1	Setup.....	587
12.1.1	General; machine parameters.....	587
12.1.2	PLC. General modifiable signals.....	589
12.1.3	File containing basic volumetric compensation data.....	590
12.1.4	Sequence to define the values in the file.....	591
12.2	Variables.....	593
12.2.1	Variables. General; machine parameters.....	593
12.2.2	Variables. General modifiable signals.....	594



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

12.2.3	Variables. Volumetric compensation.....	595
12.3	Error messages (cause and solution).....	596

CHAPTER 13 CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).

13.1	Configuration of the HSC mode.....	600
13.2	Influence of the type of acceleration and of the filters in HSC mode.....	604
13.3	Procedure for analysis and adjustment of the HSC.....	605
13.4	Summary of the useful variables to analyze the HSC.....	608
13.4.1	Analysis of the loop time (cycle time) at the CNC.....	608
13.4.2	Analysis of the loop time (cycle time) in the channel.....	608
13.4.3	Variables associated with feedrate limitation.....	609
13.4.4	Variables associated with the FEEDRATE on the tool path.....	610
13.4.5	Variables associated with the block being executed.....	611
13.4.6	Variables associated with the coordinates in the loop.....	611
13.4.7	Variables associated with the velocity in the loop.....	612
13.4.8	Variables associated with the velocity command and the feedback.....	612
13.5	The loops and the variables.....	613

CHAPTER 14 SUBROUTINES.

14.1	Run subroutines from RAM (extension *.fst).....	617
14.2	Subroutine associated with the start.....	618
14.2.1	Configure the subroutines.....	618
14.3	Subroutine associated with the reset.....	619
14.3.1	Configure the subroutines.....	619
14.4	Subroutine associated with 8055-MC and 8055-TC programs (subroutines 9998 and 9999).....	621
14.4.1	Configure the subroutines.....	621
14.5	Subroutines associated with the kinematics calibration cycle.....	622
14.5.1	Configure the subroutines.....	622
14.6	Subroutines associated with the tool calibration cycle.....	623
14.6.1	Configure the subroutines.....	623
14.6.2	Example of a subroutine.....	624
14.7	Subroutines associated with the part measuring cycle.....	625
14.7.1	Configure the subroutines.....	625
14.7.2	Example of a subroutine.....	626
14.8	Subroutine associated with the tool change (function T).....	627
14.8.1	Configure the subroutines.....	627
14.9	Subroutine associated with the tool change (function M6).....	628
14.9.1	Configure the subroutines.....	628
14.10	Subroutine associated with function G74.....	629
14.10.1	Configure the subroutines.....	629
14.11	Subroutines associated with M functions.....	630
14.11.1	Configure the subroutines.....	630
14.11.2	Variables.....	630
14.12	OEM subroutines associated with functions G180 to G189 / G380 to G399.....	631
14.12.1	Configure the subroutines.....	631
14.12.2	Variables.....	631
14.13	Interruption subroutines.....	632
14.13.1	Configure the subroutines.....	632
14.13.2	Variables.....	632
14.14	Subroutine associated with the command #INITIALSUB.....	633
14.14.1	Configure the subroutines.....	633
14.14.2	Variables.....	633
14.15	Subroutine associated with the command #PIERCING.....	634
14.15.1	Configure the subroutines.....	634
14.15.2	Variables.....	635
14.16	Subroutine associated with the command #CUTTING ON and #CUTTING OFF.....	636
14.16.1	Configure the subroutines.....	636
14.16.2	Variables.....	637
14.17	Subroutine associated with the command #FINALSUB.....	638
14.17.1	Configure the subroutines.....	638
14.17.2	Variables.....	638
14.18	User subroutines associated with functions G500 to G599.....	639
14.18.1	Configure the subroutines.....	639
14.19	User subroutines associated with functions G8000 to G8999.....	640
14.19.1	Configure the subroutines.....	640
14.20	Assistance for subroutines.....	641
14.20.1	Subroutine help files.....	641
14.20.2	List of available subroutines.....	643

CHAPTER 15 **HARDWARE.**

15.1	Management of several keyboards.	645
15.1.1	How to configure the feature.	645
15.1.2	Operation of the jog panels.	650
15.2	Configure a handwheel as "feed handwheel".	651
15.3	Assigning a help text to the graphic softkeys and to the CNC status icon.	653
15.4	Remote module RCS-S.	654
15.4.1	Configure the module like as a node of the Sercos bus.	655
15.4.2	Configure the analog outputs.	655
15.4.3	Configuration of the feedback inputs.	656
15.4.4	Configure the feedback input for a handwheel.	656
15.4.5	Disabling the feedback input alarms.	657
15.4.6	Parameter setting example.	658
15.5	Number the digital inputs and outputs of the CANopen bus.	660
15.5.1	Example. I/O numbering via machine parameters.	662
15.6	Configure the PT100 inputs.	666
15.6.1	CNC variables.	667

CHAPTER 16 **LASER.**

16.1	Laser. Gap control.	669
16.1.1	Connecting the gap sensor.	671
16.1.2	Adjusting the axis that controls the gap.	672
16.1.3	Configuring the gap.	673
16.1.4	Configuring the gap sensor.	674
16.1.5	Adjusting the sensor position loop.	675
16.1.6	PLC signals.	676
16.1.7	Gap compensation examples.	677
16.1.8	Operating the PLC to remove the axis associated with the gap control.	678

CHAPTER 17 **CALCULATION OF THE KINEMATICS DIMENSIONS.**

17.1	Angular spindle. Calculation of the dimensions using a probe.	680
17.2	Angular spindle. Calculation of the dimensions using a dial indicator.	686
17.3	Rotary table. Calculation of the dimensions using a probe.	690

CHAPTER 18 **REMOTE OPENPCS.**



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

BLANK PAGE

ABOUT THE PRODUCT - CNC 8070

BASIC CHARACTERISTICS.

Basic characteristics.	.BL.	.OL.	.L.
Number of axes.	3 to 7	3 to 31	3 to 31
Number of spindles.	1	1 to 6	1 to 6
Number of tool magazines.	1	1 to 4	1 to 4
Number of execution channels.	1	1 to 4	1 to 4
Number of interpolated axes (maximum).	4	3 to 31	3 to 31
Number of handwheels.	1 to 12		
Type of servo system.	Analog / Digital Sercos Digital Mechatrolink		Analog Sercos Digital
Communications.	RS485 / RS422 / RS232 Ethernet		
PCI expansion.	No	Option	No
Integrated PLC. PLC execution time. Digital inputs / Digital outputs. Marks / Registers. Timers / Counters. Symbols.	< 1ms/K 1024 / 1024 8192 / 1024 512 / 256 Unlimited		
Block processing time.	< 1 ms		< 1 ms

Remote modules.	RIOW	RIO5	RIO70	RIOR	RCS-S
Valid for CNC.	8070 8065 8060	8070 8065 8060	8070 8065 ---	8070 8065 8060	8070 8065 8060
Communication with the remote modules.	CANopen	CANopen	CANfagor	CANopen	Sercos
Digital inputs per module.	8	24 / 48	16	48	---
Digital outputs per module.	8	16 / 32	16	32	---
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 / FeeDat / EnDat

Customizing.

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

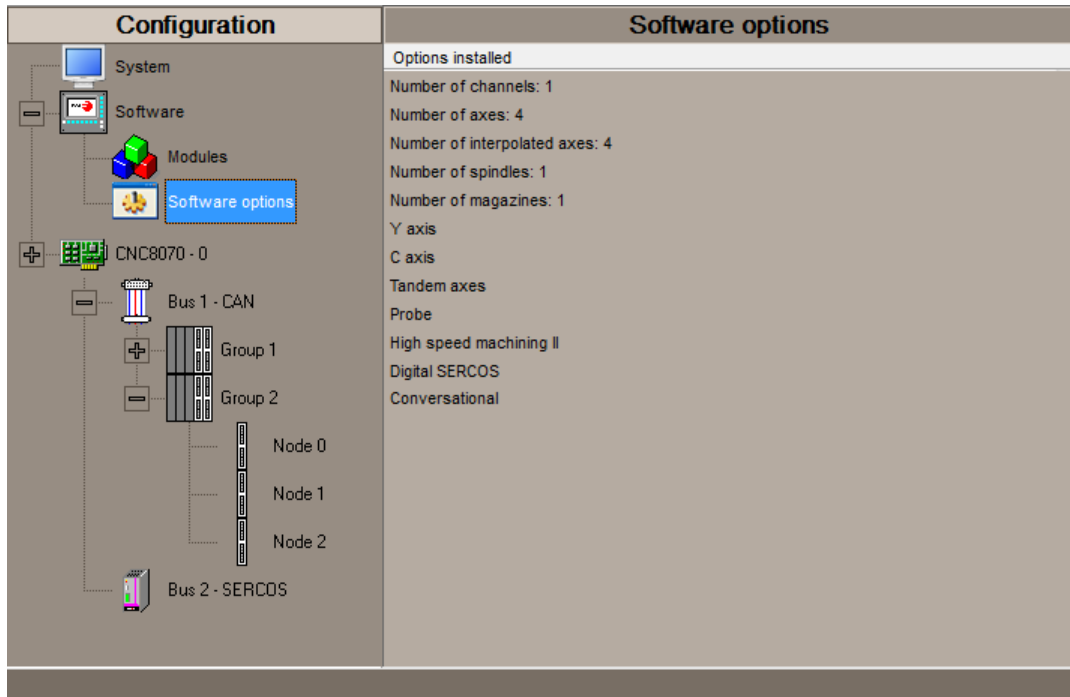
(REF: 1709)

FAGOR 
FAGOR AUTOMATION

CNC 8070

SOFTWARE OPTIONS.

Some of the features described in this manual are dependent on the acquired software options. The active software options for the CNC can be consulted in the diagnostics mode (accessible from the task window by pressing [CTRL] [A]), under software options.



Consult the ordering handbook for information on the software options available for your model.

SOFT ADDIT AXES

Additional shaft.

Add axes to the default configuration.

SOFT ADDIT SPINDLES

Additional spindle.

Add spindles to the default configuration.

SOFT ADDIT TOOL MAGAZ

Additional tool magazine.

Add tool magazines to the default configuration.

SOFT ADDIT CHANNELS

Additional channel.

Add channels to the default configuration.

SOFT 4 AXES INTERPOLATION LIMIT

Limited to 4 interpolated axes.

It limits the number of axes to 4, where the CNC can also interpolate these at the same time.

SOFT OPEN SYSTEM

Open system.

The CNC is a closed system that offers all the features needed to machine parts. Nevertheless, at times there are some customers who use third-party applications to take measurements, perform statistics or other tasks apart from machining a part.

This feature must be active when installing this type of application, even if they are Office files. Once the application has been installed, it is recommended to close the CNC in order to prevent the operators from installing other kinds of applications that could slow the system down and affect the machining operations.

SOFT DIGITAL SERCOS

Sercos digital bus.

Sercos digital bus.

SOFT DIGIT NO FAGOR

Non-Fagor digital servo system.

Mechatrolink digital bus.

SOFT EDIT/SIMUL

EDISIMU mode (editing and simulation).

It allows for the editing, modification and simulation of a part-program.

SOFT IEC 61131 LANGUAGE

IEC 61131 language

IEC 61131 is a PLC programming language that is very popular in alternative markets, which is slowly entering into the machine-tool market. With this feature, the PLC may be programmed either in the usual Fagor language or in IEC 61131 format.

SOFT TOOL RADIUS COMP

Compensación de radio.

Tool compensation allows programming the contour to be machined based on part dimensions of the and without taking into account the dimensions of the tool that will be used later on. This avoids having to calculate and define the tool path based on the tool radius.

SOFT PROFILE EDITOR

Profile editor.

Allows for the part profiles to be edited graphically and to import dxf files.

SOFT RTCP

Dynamic RTCP (Rotating Tool Center Point).

The dynamic RTCP option is required for interpolation machining with 4, 5 or 6 axis.

SOFT C AXIS

C axis.

It activates the kinematics for working with the C axis and the associated canned cycles. The CNC can control several C axes. The parameters of each axis indicate if it will function as a C axis or not, where it will not be necessary to activate another axis for the machine parameters.

SOFT TANDEM AXES

Tandem axes.

A tandem axis consists in two motors mechanically coupled (slaved) and making up a single transmission system (axis or spindle). A tandem axis helps provide the necessary torque to move an axis when a single motor is not capable of supplying enough torque to do it.

When activating this feature, it should be kept in mind that for each tandem axis of the machine, another axis must be added to the entire configuration. For example, on a large 3-axis lathe (X Z and tailstock), if the tailstock is a tandem axis, the final purchase order for the machine must indicate 4 axes.

SOFT SYNCHRONISM

Synchronization of axes and spindles.

The axes and ballscrews may be synchronized in two ways: in terms of speed or position. The CNC configuration takes into consideration the synchronization of 2 axes or 2 spindles. Once synchronized, only the master displays and programs the element.

SOFT HSSA II MACHINING SYSTEM

HSSA-II machining system.

This is the new version of algorithms for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved.

SOFT TANGENTIAL CONTROL

Tangential control.

"Tangential Control" maintains a rotary axis always in the same orientation with respect to the programmed tool path. The machining path is defined on the axes of the active plane and the CNC maintains the orientation of the rotary axis along the entire tool path.

SOFT DRILL CYCL OL

Drilling ISO cycles for the OL model.

Drilling ISO cycles for the OL model (G80, G81, G82, G83).

SOFT PROBE

Probing canned cycles.

The CNC may have two probes; usually a tabletop probe to calibrate tools and a measuring probe to measure the part.

This option activates the functions G100, G103 and G104 (for probe movements); probe canned cycles are not included.

SOFT THIRD PARTY CANOPEN

Third-party CANopen.

Enables the use of non-Fagor CANopen modules.

SOFT FVC UP TO 10m3

SOFT FVC MORE TO 10m3

Medium and large volumetric compensation.

5-axis machines are generally used during the manufacturing of large parts. The accuracy of the parts is limited by the machine manufacturing tolerances and is affected by temperature variations during machining.

In sectors such as the aerospace industry, machining demands mean that classic compensation tools are becoming suboptimal. Volumetric compensation FVC comes in to complement the machine adjusting tools. When mapping the total work volume of the machine, the CNC knows the exact position of the tool at all times. After applying the required compensation, the resulting part is made with the desired precision and tolerance.

There are 2 choices, which depend on the size of the machine, being up to 10 m³ and over 10 m³.

SOFT 60 PWM CONTROL

Pulse-Width Modulation.

This function is only available for Sercos bus controlled systems. It is mostly oriented toward laser machines for the cutting of very thick sheets, where the CNC generates a series of PWM pulses to control the power of the laser when drilling the starting point.

This feature is essential for cutting very thick sheets and it requires two quick digital outputs located on the central unit. With this new feature, the OEM does not need to install or program any external device, which reduces machine costs and installation times. The end user also benefits, since the "Cutting with PWM" feature is much easier to use and program.

SOFT 60 GAP CONTROL

Gap control.

This is mostly oriented toward laser machines. Gap control makes it possible to maintain a set distance between the laser nozzle and the surface of the sheet. This distance is calculated by a sensor connected to the CNC, so that the CNC offsets the sensor variations on the distance programmed with additional movements in the axis programmed for the gap.

BLANK PAGE

DECLARATION OF CE CONFORMITY AND WARRANTY CONDITIONS

DECLARATION OF CONFORMITY

The declaration of conformity for the CNC is available in the downloads section of FAGOR'S corporate website. <http://www.fagorautomation.com>. (Type of file: Declaration of conformity).

WARRANTY TERMS

The warranty conditions for the CNC are available in the downloads section of FAGOR's corporate website. <http://www.fagorautomation.com>. (Type of file: General sales-warranty conditions).



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

BLANK PAGE

VERSION HISTORY - CNC 8070

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

Ref. 0201

Software V01.00

First version. Milling model.

Ref. 0212

Software V01.10

Set the numbering of the digital I/O.	<ul style="list-style-type: none"> Machine parameters: NDIMOD DIMODADRR DIMOD NDOMOD DOMODADRR DOMOD
Probe management from the digital inputs; it is not possible to manage remote feedback inputs ("Counter" module).	<ul style="list-style-type: none"> Machine parameters: PRBDI1 PRBDI2 PRBPULSE1 PRBPULSE2
Tabletop probe configuration.	<ul style="list-style-type: none"> Machine parameters: PROBE PRB1MIN PRB2MIN PRB3MIN, PRB1MAX PRB2MAX PRB3MAX
Define the repositioning feedrate after tool inspection.	<ul style="list-style-type: none"> Machine parameter: REPOSFEED.
New treatment of the JOG keys. Different keys to select the axis and the direction.	<ul style="list-style-type: none"> Machine parameter: JOGKEYDEF.
Kinematics for rotary tables (TYPE9 to TYPE12).	<ul style="list-style-type: none"> Kinematics TYPE9 through TYPE12.
Know the dimensions of the kinematics on an axis.	<ul style="list-style-type: none"> Variable: (V.)A.HEADOF.xn
Keyboard simulation from the PLC.	<ul style="list-style-type: none"> Variable: (V.)G.KEY
Knowing which is the active probe.	<ul style="list-style-type: none"> Variable: (V.)G.ACTIVPROBE
Know the active general scaling factor.	<ul style="list-style-type: none"> Variable: (V.)G.SCALE
Park and unpark Sercos axes from the PLC.	<ul style="list-style-type: none"> PLC command: PARK UNPARK

Ref. 0501

Software V02.01

Windows XP operating system.	
Emergency shutdown with battery (central unit PC104).	
Multi-channel system, up to 4 channels. Swapping of axes and spindles, communication and synchronization between channels, common arithmetic parameters, access variables by channel, etc.	
Multi-spindle system, up to 4 spindles.	
Tool management with up to 4 magazines.	
Parameter matching between the CNC and the Sercos drive	<ul style="list-style-type: none"> Parameter matching.
New main operating mode for the drive; velocity Sercos.	<ul style="list-style-type: none"> Machine parameter: OPMODEP.
New kinematics table-spindle (TYPE13 to TYPE16).	<ul style="list-style-type: none"> Kinematics TYPE13 through TYPE16.
New kinematics for C axis (TYPE41 to TYPE43).	<ul style="list-style-type: none"> Kinematics TYPE41 through TYPE43.
New languages (Basque and Portuguese).	<ul style="list-style-type: none"> Machine parameter: LANGUAGE.
Placing the vertical softkeys on the left or on the right.	<ul style="list-style-type: none"> Machine parameter: VMENU.
Tandem axes.	<ul style="list-style-type: none"> Machine parameters: TANDEM TMASTERAXIS TSLAVEAXIS TORQDIST PRELOAD PRELFITI TPROGAIN TINTTIME TCOMPLIM
Gantry axis. Maximum difference allowed between the following errors of both axes before issuing a warning.	<ul style="list-style-type: none"> Machine parameter: WARNCOUPE.
Apply cross compensation to either theoretical or real coordinates.	<ul style="list-style-type: none"> Machine parameter: TYPCROSS.
Apply leadscrew compensation to either theoretical or real coordinates.	<ul style="list-style-type: none"> Machine parameter: TYPLSCRW.
Tool radius compensation mode (G136/G137) by default	<ul style="list-style-type: none"> Machine parameter: IRCOMP.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Software V02.01	
Defining the type of reference pulse.	• Machine parameter: REFPULSE.
Memory sharing between applications.	• Machine parameter: PLCDATASIZE.
OEM generic machine parameters.	• Machine parameter: MTBPAR.
Reading Sercos variables from the CNC.	• Machine parameter: DRIVEVAR.
Electronic-cam editor.	• Machine parameter: CAM.
Backlash peak compensation.	• Machine parameters: BAKANOUT BAKTIME ACTBAKAN
New behavior for rotary axes.	• Machine parameters: AXISMODE UNIDIR SHORTESTWAY
Sercos transmission at 8 MHz and 16 MHz.	• Machine parameter: SERBRATE.
Define the anticipation time for the axes to be considered to be in position.	• Machine parameters: ANTIME. • PLC marks: ADVINPOS.
PLC. The TMOOPERATION may take the values 13 and 14.	• PLC mark: TMOOPERATION.
PLC. Detect when the operating system locks up.	• PLC marks: MMCWDG.
PLC. Disable the cross compensation tables.	• PLC marks: DISCROSS.
PLC. Correct the parallelism on Gantry axes.	• PLC marks: DIFFCOMP.
PLC. Execute CNC blocks.	• PLC command: CNCEX. • PLC marks: FREE.
PLC. Reading arithmetic parameters and OEM parameters with CNCRD returns the value multiplied by 10000 (reading in float mode).	• PLC command: CNCRD.
PLC. Define external symbols.	• PLC command: PDEF.
PLC. The RESETIN mark is not routine to park/unpark axes or spindles from the PLC.	• PLC marks: RESETIN PARK UNPARK
The "(V.).TM.MZWAIT" variable is not necessary in the subroutine associated with M06.	• Subroutine associated with M6. • Variable: (V.).TM.MZWAIT
Apply filters to eliminate the resonance of the spindle when it works as C axis or during rigid tapping.	• Frequency filters. C axis. Rigid tapping.
Optimize the reading and writing of variables from the PLC. Only the access to the following variables will be asynchronous. <ul style="list-style-type: none"> • 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. • The variables referred to local arithmetic parameters of the active levels will be read and written asynchronously. 	• Reading and writing of variables from the PLC.
Spindle parking and unparking.	• PLC marks: PARK, UNPARK. • Instructions #PARK, #UNPARK.
Know the software version.	• Variable: (V.)G.SOFTWARE
Variables related to loop adjustment. Gain setting via PLC.	• Variables: (V.)A.PLCFFGAIN.xn (V.)A.PLCACFGAIN.xn (V.)A.PLCPROGAIN.xn
Variables related to loop adjustment. Position increment and sampling period.	• Variables: (V.)A.POSINC.xn (V.)A.TPOSINC.xn (V.)A.PREVPOSINC.xn
Variables related to loop adjustment. Fine adjustment of feedrate, acceleration and jerk.	• Variables: (V.)A.FEED.xn (V.)A.TFEED.xn (V.)A.ACCEL.xn (V.)A.TACCEL.xn (V.)A.JERK.xn (V.)A.TJERK.xn
Variables related to the feedback inputs.	• Variables: (V.)A.COUNTER.xn (V.)A.COUNTERST.xn (V.)A.ASINUS.xn (V.)A.BSINUS.xn

Ref. 0504

Software V02.03	
New values of Sercos optic power for the "Sercos II" board.	• Machine parameters: SERPOWSE.
Electronic cam programming commands (real coordinates).	• PLC command: CAM.
Independent axis movement programming commands.	• PLC command: MOVE.
Programming commands for axis synchronization (real coordinates).	• PLC command: FOLLOW.
New signals that may be consulted and changed for the independent interpolator (electronic cam and independent movement)	
The simulated axes are ignored regarding the validation code.	
When matching parameters, it does not send parameters G00FEED and MAXVOLT to the drive.	
Function G112 changes the drive gear set.	• Function G112.

(REF: 1709)



FAGOR AUTOMATION

CNC 8070

Ref. 0509

Software V03.00	
Lathe model. Machining canned cycles, lathe tool calibration, variables to consult the geometry of lathe tools, etc.	
Incline axis.	<ul style="list-style-type: none"> Machine parameter: NANG ANGTR ANGAXNA ORTAXNA ANGANTR OFFANGAX
CAN bus type selection (CANopen/CANfagor).	<ul style="list-style-type: none"> Machine parameter: CANMODE.
Function G95 being active, allow setting the feed per turn in jog mode.	<ul style="list-style-type: none"> Machine parameter: FPRMAN.
Lathe model. Select graphics configuration.	<ul style="list-style-type: none"> Machine parameter: GRAPHTYPE.
Lathe model. Select axis configuration.	<ul style="list-style-type: none"> Machine parameter: GEOCONFIG.
Set of parameters of the master axis or spindle for synchronization.	<ul style="list-style-type: none"> Machine parameter: SYNCSET.
The CNC can keep the C axis active after executing M02, M30 or after an emergency or reset.	<ul style="list-style-type: none"> Machine parameter: PERCAX.
Improved definition of kinematics for the C axis.	
Magazine-less system.	<ul style="list-style-type: none"> Machine parameter: NTOOLMZ.
Ground tools for a turret magazine. The TMOOPERATION register may take the values 3, 4, 9 and 10.	<ul style="list-style-type: none"> PLC register: TMOOPERATION.
Commands CNCRD and CNCWR. The channel number and the indexes may be defined in the variables using an integer, a register or a symbol.	<ul style="list-style-type: none"> PLC commands: CNCRD and CNCWR.
Spindle synchronization.	
Variable to read the accumulated PLC offset.	<ul style="list-style-type: none"> Variable: (V.)A.ACTPLCOF.xn
Variable to obtain a linear estimation of the following error.	<ul style="list-style-type: none"> Variable: (V.)A.FLWEST.xn
Variables to read the instant value of feed-forward or AC-forward.	<ul style="list-style-type: none"> Variables: (V.)A.ACTFFW.xn (V.)A.ACTACF.xn
Variable to know the line number of the file being executed.	<ul style="list-style-type: none"> Variable: (V.)G.LINEN
Variable to know what kind of cycle is active.	<ul style="list-style-type: none"> Variable: (V.)G.CYCLETYPEON
Variable to know the tool orientation.	<ul style="list-style-type: none"> Variable: (V.)G.TOOLDIR
Variable to know the theoretical feedrate on 3D path.	<ul style="list-style-type: none"> Variable: (V.)G.F3D
Variable to know the number of the warning being displayed.	<ul style="list-style-type: none"> Variable: (V.)G.CNCWARNING
The variable (V.)G.CNCERR is now per channel.	<ul style="list-style-type: none"> Variable: (V.)G.CNCERR
Improved coordinate transformation. <ul style="list-style-type: none"> Keep the part zero when deactivating the transformation. Working with 45° spindles. Select between the two choices. Keep the rotation of the plane axes with MODE 6. 	
Function G112 is not valid for the spindle.	<ul style="list-style-type: none"> Function G112.
New criteria when assuming a new master spindle in the channel.	

Ref. 0601

Software V03.01	
Screen test on power-up, if any element is missing, it restores the relevant backup.	
CAN bus. Transmission speed for cable lengths of 110, 120 and 130 m.	<ul style="list-style-type: none"> Machine parameter: CANLENGTH.
The CNC can keep axes slaved (coupled to each other) after executing M02, M30 or after an emergency or reset.	<ul style="list-style-type: none"> Machine parameter: LINKCANCEL.
Configure the way the CNC cancels tool radius compensation.	<ul style="list-style-type: none"> Machine parameter: COMPCANCEL.
Dual-feedback (internal + external) system, they may be swapped via PLC.	<ul style="list-style-type: none"> Machine parameters: FBACKSRC FBACKDIFF PLC marks: FBACKSEL(axis) ACTBACK(axis)
The PLC informs that a position synchronization has begun.	<ul style="list-style-type: none"> PLC mark: SYNCRONP.
The PLC informs about the status of the Sercos ring.	<ul style="list-style-type: none"> PLC mark: SERCOSRDY.
Parameter matching sends parameter MODUPLIM.	
Variables. Geometry of the lathe tools.	
Variables. Number of the tool in the claws of the changer arm.	<ul style="list-style-type: none"> Variables: (V.)TM.TOOLCH1[mz] (V.)TM.TOOLCH2[mz]

Ref. 0606

Software V03.10	
New home searching method for spindles with home switch. The spindle goes through the home switch twice.	
New FAGOR low passing filters.	
Maximum machining feedrate.	<ul style="list-style-type: none"> Machine parameter: MAXFEED.
Default machining feedrate when none has been programmed.	<ul style="list-style-type: none"> Machine parameter: DEFAULTFEED.
The user keys may be configured as jog keys.	<ul style="list-style-type: none"> Machine parameter: USERKEYDEF.
The CNC allows changing the spindle override during electronic threading (G33) and in the threading canned cycles of the ·T· model (G86, G87 and their equivalent of the cycle editor).	<ul style="list-style-type: none"> Machine parameters: THREADOVR OVRFILTER
CANopen protocol.	<ul style="list-style-type: none"> Machine parameter: CANMODE.
The CNC may have several general handwheels.	<ul style="list-style-type: none"> Machine parameter: MPGAXIS.
A general handwheel can move several axes at the same time.	<ul style="list-style-type: none"> Machine parameter: MPGAXIS.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Software V03.10	
New parameter to set whether or not the CNC sends the M, H, S functions to the PLC during block search.	• Machine parameter: FUNPLC.
M function table. New field to define whether the function is sent out to the PLC or not during block search.	• Field: MPLC.
M function table. Each M function may have a different describing comment associated with it.	• Field: COMMENT.
On a handwheel with button, it is possible to configure the button from the PLC to select sequentially the axis to be jogged with the handwheel.	• PLC mark: NEXTMPGAXIS.
Dead axis. Handling the blending of (transition between) blocks.	• PLC mark: DEAD(axis)
Disabling a keyboard or jog panel integrated into the CAN bus.	• PLC mark: PANELOFF.
Aborting CNCEX commands launched from the PLC.	• PLC mark: PLCABORT.
In automatic and jog modes, the CNC shows the status of the _FEEDHOL mark.	• PLC mark: _FEEDHOL.
In automatic and jog modes, the CNC shows the status of the INHIBIT mark of the axes and spindle.	• PLC mark: INHIBIT.
OEM machine parameters. <ul style="list-style-type: none"> • Range of parameters that can be written from the part-program, from the PLC or from the interface. • Range of parameters affected by the change of units. • Each parameter may have a different describing comment associated with it. 	
The CNC displays the warnings generated at the drive.	
Configuring 2 axes with the same feedback input and analog output.	
Up to 8 axes can now get involved in a kinematics (before just 5).	
Kinematics type 41/42. Angular offset for the rotary axis.	• Kinematics TYPE41 and TYPE42.
Kinematics type 41/42. Tool misalignment with the C axis.	• Kinematics TYPE41 and TYPE42.
Kinematics type 43. Angular offset for the rotary axis.	• Kinematics TYPE43.
New kinematics; type 100 to 105. Integrating OEM kinematics through machine parameters.	• Kinematics TYPE100 to TYPE105.
Programming help files for OEM and global subroutines.	
Help file with the list of available subroutines.	
"Retrace" function.	<ul style="list-style-type: none"> • Machine parameter: RETRACAC NRETLK RETMFUNC • PLC marks: RETRAEND RETRACE
Tangential control.	<ul style="list-style-type: none"> • PLC marks: TANGACTIV TANGACT(axis) • Variables: (V.)A.TANGAN.xn (V.)G.TANGFEED (V.)A.TGCTRLST.xn (V.)G.TGCTRLST
Canceling the preset turning direction of a tool.	• Variable: (V.)G.SPDLTURDIR
Change the maximum feedrate allowed in the channel from the PLC.	• Variable: (V.)PLC.PLCG00FEED
Show the status of the emergency relay.	• Variable: (V.)G.ERELAYST

Ref. 0608

Software V03.11	
Simulator Possibility to use the dongle (hardware key) in a network.	
The default value of some machine parameters is different for the CNC and for the simulator installed on a PC.	
"Retrace" function. Improvements to the retrace function.	
Abort the execution of the program and resume it somewhere else.	• PLC mark: PRGABORT.

Ref. 0610

Software V03.12	
In parameter matching, the CNC sends parameters it sends REFSHIFT and FBMIXTIME.	
Time constant for combined feedback.	• Machine parameter: FBMIXTIME.
When the axes are position-Sercos; during parameter matching, the CNC sends the value of parameter REFSHIFT to the drive so it takes it into account; this way the CNC coordinate and that of the drive will be the same.	

Ref. 0704 / Ref. 0706

Software V03.13	
Sign criteria for tool offsets (dimensions) and tool wear.	• Machine parameter: TOOLOFSG.
Variables (V.)TM.TOOLCH1[mz] / (V.)TM.TOOLCH2[mz] may be written from the PLC.	• Variables: (V.)TM.TOOLCH1[mz] (V.)TM.TOOLCH2[mz]

(REF: 1709)



FAGOR AUTOMATION

CNC 8070

Software V03.14	
MCU and ICU central unit. battery powered RAM. Connecting handwheels to the central unit. local I/O. Local feedback inputs. Loca probes.	
The handwheels may be connected to the central unit.	<ul style="list-style-type: none"> Machine parameters: COUNTERTYPE COUNTERID
Local feedback inputs.	<ul style="list-style-type: none"> Machine parameters: COUNTERTYPE COUNTERID
Management of local I/O.	<ul style="list-style-type: none"> Machine parameters: NLOCOUT EXPSCHK.
Number of non-volatile PLC registers.	<ul style="list-style-type: none"> Machine parameter: BKUPREG.
Number of non-volatile PLC counters	<ul style="list-style-type: none"> Machine parameter: BKUPCOUN.
Number of common non-volatile arithmetic parameters.	<ul style="list-style-type: none"> Machine parameter: BKUPCUP.
Connect the probes to the central unit (local probes).	<ul style="list-style-type: none"> Machine parameters: PRBDI1 PROBETYPE1 PRBDI2 PROBETYPE2
Define whether the spindle is homed automatically with the first movement or not.	<ul style="list-style-type: none"> Machine parameter: REFINI.

Ref. 0707

Software V03.15	
In parameter matching, the CNC sends parameter ABSOFF when using absolute feedback.	
On rotary axes or spindles working in velocity-Sercos, the CNC calculates the module of the coordinate. In parameter matching, drive parameter PP76(7) is set to 0.	
Variable to know the type of hardware.	<ul style="list-style-type: none"> Variable: (V.)G.HARDTYPE
Variable to know the theoretical tool feedrate along the path.	<ul style="list-style-type: none"> Variable: (V.)G.PATHFEED
Managing an analog axis through the analog output and the second feedback of a Sercos drive.	
PLC errors may have an additional data file associated with them, same as PLC messages.	
The CNC shows a warning when a channel is expecting a tool that is being used in another channel.	

Ref. 0709

Software V03.16	
Tandem spindles.	
The CNC uses the combined feedback to calculate the velocity command, but it uses the direct feedback to calculate the compensations, circularity test, etc.	
The CNC does not assume any kinematics on power-up.	<ul style="list-style-type: none"> Machine parameters: KINID
The CNC allows modifying the spindle override while threading if it detects that the feed forward (parameter FFWTYPE) is not active in a gear of the axes involved or if the active feed forward is lower than 90%	<ul style="list-style-type: none"> Machine parameters: THREADOVR OVRFILTER

Ref. 0712

Software V03.17	
On rotary axes with module and spindle working in velocity-Sercos mode, with a whole gear ratio and with drive parameter PP76(7)=1, the parameter matching does not redefine parameter PP76(7)=1; the CNC shows a warning so the user recalculates the value of drive parameter PP4 (GC6 command).	
In a tandem system, the master axis or spindle must have external feedback and the slave internal feedback.	
Keep C axis active after executing M02, M30 or after an emergency or reset.	<ul style="list-style-type: none"> Machine parameter: PERCAX.

Ref. 0801

Software V03.20	
The CNC has a different MTB folder for each type of software installed; MTB_T for lathe, MTB_M for mill and MTB_MC for motion control.	
The CNC allows setting the maximum position difference between the two axes of a gantry axis in order to correct it.	<ul style="list-style-type: none"> Machine parameters: MAXDIFF.
Configure the analog inputs for temperature sensors PT100.	<ul style="list-style-type: none"> Machine parameters: NPT100, PT100.
By default, the feedback alarms of the analog axes are activated.	<ul style="list-style-type: none"> Machine parameter: FBACKAL.
At the lathe model, the orientation of the axes in the profile editor is defined by parameter GRAPHTYPE.	<ul style="list-style-type: none"> Machine parameter: GRAPHTYPE.
Kinematics type 17 to 24. 3-axis orthogonal spindles.	<ul style="list-style-type: none"> Kinematics TYPE17 through TYPE24.
When a feedback alarm occurs on the analog axes, the CNC deactivates the REFPOIN(axis) mark.	<ul style="list-style-type: none"> PLC mark: REFPOIN(axis).
Set change. For the CNC to assume the new parameter set, it must wait for the PLC to receive the confirmation of one of the marks GEAR1 to GEAR4.	<ul style="list-style-type: none"> PLC mark: GEAR1 to GEAR4.
The CNC concludes the gear change when the PLC receives the confirmation signal AUXEND.	<ul style="list-style-type: none"> PLC mark: AUXEND.
The PLC informs that there is an OEM password.	<ul style="list-style-type: none"> PLC mark: PSWSET.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Software V03.20	
Gear change on a Sercos spindle. The set change only affects the drive when it implies a change of gear ratio.	
The CNC lets change the set of the slave axis or spindle of a tandem.	
The PLC informs that, for the spindle, the parameter set selected at the CNC does not match that of the PLC.	<ul style="list-style-type: none"> • PLC mark: GEAROK.
The PLC can initiate the CNC shut-down sequence.	<ul style="list-style-type: none"> • PLC mark: CNCOFF.
Park and unpark the main axes.	<ul style="list-style-type: none"> • PLC marks: PARK, UNPARK.
Coordinate latching with the help of a probe or a digital input.	<ul style="list-style-type: none"> • PLC command: TOUCHPROBE • PLC marks: PROBE1ACTIVE PROBE2ACTIVE LATCH1ACTIVE(axis) LATCH2ACTIVE(axis) LATCH1DONE(axis) LATCH2DONE(axis). • Variables: (V.)A.LATCH1.xn (V.)A.LATCH2.xn
Dynamic distribution of the machining operations 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.	<ul style="list-style-type: none"> • PLC marks: DINDISTC1 DINDISTC2 DINDISTC3 DINDISTC4
Electronic cam programming commands (theoretical coordinates).	<ul style="list-style-type: none"> • PLC command: TCAM.
There is no need to use the partition character "\" in the PLC program to divide a logic expression in two lines.	
PLC. The PLC program can have several mnemonic files (extension "plc").	
PLC. Grouping the additional information text files in a single file.	
Check the status of the probes.	<ul style="list-style-type: none"> • Variables: (V.)G.PRBST1 (V.)G.PRBST2.
Axis synchronization. Managing 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.	<ul style="list-style-type: none"> • Variable: (V.)A.ACCUDIST.xn
Interface related variables.	
When defining each PLC error, it is possible to select whether it opens the emergency relay or not.	
Machine parameters tables. Import and export leadscrew compensation tables.	
CNC errors between 10000 and 20000 are reserved for the OEM so he can create his own warning or error texts in different languages.	
The CNC allows modifying the percentage of acceleration and jerk of the spindles via part-program or in MDI/MDA mode	<ul style="list-style-type: none"> • Functions G130 and G132.

Ref. 0809

Software V04.00 (it does not include the features of version V03.21)	
Unicode.	
New language (Chinese).	<ul style="list-style-type: none"> • Machine parameter: LANGUAGE.
When the CNC is installed as a simulator at a PC, the drive may be a simulated type or a Sercos type.	<ul style="list-style-type: none"> • Machine parameter: DRIVETYPE.
In the machine parameter table, an icon indicates which parameters are involved in parameter matching.	
There can now be up to 12 handwheels.	<ul style="list-style-type: none"> • Machine parameter: NMPG.
The CNC applies module compensation throughout the entire revolution of the axis.	<ul style="list-style-type: none"> • Machine parameter: MODCOMP.
HSC mode. Eliminate the first resonance frequency of the machine when generating the velocity command.	<ul style="list-style-type: none"> • Machine parameter: FREQRES.
Home search moving the axis to the reference point.	<ul style="list-style-type: none"> • Machine parameter: POSINREF.
Delay estimate at the drive.	<ul style="list-style-type: none"> • Machine parameter: AXDELAY.
Transfer inhibit for the independent axes.	<ul style="list-style-type: none"> • Machine parameter: XFITOIND. • PLC mark: _XFERINH and _XFERINH(axis).
Status of the axis position loop.	<ul style="list-style-type: none"> • PLC mark: LOPEN(axis).
Positioning a turret magazine whether there is a tool in the indicated position or not (TMOOPERATION=15).	<ul style="list-style-type: none"> • PLC mark: TMOOPERATION.
Detect overtemperature at the CNC.	<ul style="list-style-type: none"> • PLC mark: OVRTEMP.
There can now be up to 1024 PLC messages.	<ul style="list-style-type: none"> • PLC resources: MSG.
There can now be up to 1024 PLC errors.	<ul style="list-style-type: none"> • PLC resources: ERR.
Inhibit the handwheels of the system.	<ul style="list-style-type: none"> • PLC mark: INHIBITMPG1/INHIBITMPG12.
Cancel spindle synchronization after executing M02, M30 or after an error or a reset.	<ul style="list-style-type: none"> • Machine parameter: SYNCANCEL.
A channel can maintain its master spindle after executing M02, M30 or after an emergency or a reset or restarting the CNC.	<ul style="list-style-type: none"> • Machine parameter: MASTERSPDL.
Force the change of gears and/or of the parameter set of a Sercos drive	<ul style="list-style-type: none"> • PLC mark: SERPLCAC. • Variable: (V.)A.SETGE.xn
Set a machine coordinate.	<ul style="list-style-type: none"> • PLC mark: REFPOIN(axis). • Function G174.
Axis synchronization. Managing 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.	<ul style="list-style-type: none"> • Variables: (V.)A.PREVACCUDIST.xn
The variable (V.)E.PROGSELECT can be written via part-program, PLC and interface. This variable can only be written with the value of -0.	<ul style="list-style-type: none"> • Variables: (V.)E.PROGSELECT

Software V04.00 (it does not include the features of version V03.21)	
The following variables are valid for the spindle.	<ul style="list-style-type: none"> Variables: (V.)A.MEAS.sn (V.)A.ATIPMEAS.sn (V.)A.MEASOF.sn (V.)A.MEASOK.sn (V.)A.MEASIN.sn
Number of pulses sent by the handwheel since the system was started up.	Variable: (V.)G.HANDP[hw]
The general handwheels can move axes with an associated individual handwheel.	
Feed handwheel.	

Ref. 0811

Software V03.21 (features not included in version V04.00)	
There can now be up to 1024 PLC messages.	• PLC resources: MSG.
There can now be up to 1024 PLC errors.	• PLC resources: ERR.

Ref. 0907

Software V04.01	
The CNC turns the internal fan on and off as necessary. The CNC turns the fan on when the temperature exceeds 50 °C (122 °F) and turns it off when it gets under 45 °C (113 °F).	
Communication with servos (axis and spindle) and inverters (spindle) through the Mechatrolink bus, in Mlink-I (17 bytes) and Mlink-II (17 or 32 bytes) mode.	• Machine parameter: MLINK.
Multi-axis management. Controlling several Sercos axes or spindles with a single drive.	<ul style="list-style-type: none"> Machine parameter: MULTIAXIS. PLC mark: SWITCH(axis).
Kinematics (types 1 through 8). Spindle position referred to the machine reference point.	• Kinematics TYPE1 through TYPE8.
Define the maximum acceleration and jerk allowed on the tool path.	<ul style="list-style-type: none"> Machine parameters: MAXACCEL MAXJERK Variables: (V.)G.MAXACCEL (V.)G.MAXJERK
New behavior of parameter "PREPFREQ".	• Machine parameter: PREPFREQ
Variable to know the following error (lag) when feedback combination is active.	<ul style="list-style-type: none"> Variables: (V.)A.FLWE.xn (V.)A.FLWACT.xn
Variable to know the position value of the first feedback when feedback combination is active.	• Variable: (V.)A.POSMOTOR.xn

Ref. 1007

Software V04.10 (it does not include the features of version V04.02)	
New languages (Russian and Czech).	• Machine parameter: LANGUAGE.
Volumetric compensation.	<ul style="list-style-type: none"> Machine parameter: VOLCOMP. PLC mark: VOLCOMP1/VOLCOMP4. Variable: (V.)A.COMPVOL.xn (V.)A.PIVOT.xn
Remote OpenPCS.	
Cancel the inclined plane on start-up.	• Machine parameter: CSCANCEL.
Handwheels. Setting a negative resolution reverses the axis moving direction.	• Machine parameter: MPGRESOL.
Activate the rapid traverse for the automatic mode while executing a program.	<ul style="list-style-type: none"> Machine parameters: RAPIDEN FRAPIDEN. PLC mark: EXTRAPID.
Maximum axis machining feedrate.	• Machine parameter: MAXFEED.
Management of several keyboards.	• Machine parameter: NKEYBD.
Configure the serial line as RS232, RS422 or RS485.	• Machine parameter: RSTYPE.
Enable the HBLS handwheel.	• Machine parameter: HBLS.
Selecting the type of PLC (IEC61131 or Fagor).	• Machine parameter: PLCTYPE
Set the feedback system units.	• Machine parameter: POSUNITS.
Cancel the name change for 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.	<ul style="list-style-type: none"> Machine parameter: RENAMECANCEL. #RENAME instruction.
Connection with ACSD Sercos drives	
RTCP. On tilting tables, rotate the part coordinate system when rotating the table.	• Kinematics TYPE9 through TYPE12.
PLC. There are now 512 PLC timers.	• PLC resources: Timers.
PLC. Management of spindle M functions (M3, M4 and M5) from the PLC.	<ul style="list-style-type: none"> PLC marks: PLCM3 PLCM4 PLCM5
PLC. Disable the alphanumeric keyboards.	• PLC marks: QWERTYOFF
PLC. When the keyboard and the operator panel make up a single unit, the PANELOFF mark only disables the jog panel.	• PLC marks: PANELOFF
PLC. Unslave the tandem axis temporarily.	• PLC marks: TANDEMOFF(axis).
PLC. The CNC allows eliminating certain errors by pressing the [ESC] key without having to do a reset.	
M functions with an associated subroutine.	
Detailed CNC status in jog mode.	• Variable: (V.)G.CNCMANSTATUS
Detailed CNC status in automatic mode.	• Variable: (V.)G.CNCAUTSTATUS



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Software V04.10 (it does not include the features of version V04.02)	
Know the axes selected for home search, repositioning, coordinate preset or movement to a coordinate.	• Variable: (V.)G.SELECTEDAXIS
Know the current position of the main rotary axes of the kinematics (third axis).	• Variable: (V.)G.POSROTT
Know the target position of the main rotary axes of the kinematics (third axis).	• Variable: (V.)G.TOOLORIT1 (V.)G.TOOLORIT2

Ref. 1010

Software V04.02 (features not included in version V04.10)	
New language (Russian).	• Machine parameter: LANGUAGE.
Activate the rapid traverse for the automatic mode while executing a program.	• Machine parameters: RAPIDEN FRAPIDEN. • PLC mark: EXTRAPID.
Maximum axis machining feedrate.	• Machine parameter: MAXFEED.
Management of several keyboards.	• Machine parameter: NKEYBD.
Configure the serial line as RS232, RS422 or RS485.	• Machine parameter: RSTYPE.
Set the feedback system units.	• Machine parameter: POSUNITS.
Runaway protection and tendency test.	• Machine parameters: TENDENCY TENDTIME.
Synchronize spindles without forcing a set change.	• Machine parameter: SYNCSET.
Mechatrolink. Activate the drive options.	• Machine parameter: OPTION.
Cancel the name change for 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.	• Machine parameter: RENAMECANCEL. • #RENAME instruction.
Connection with ACS D Sercos drives	
RTCP. On tilting tables, rotate the part coordinate system when rotating the table.	• Kinematics TYPE9 through TYPE12.
PLC. Disable the alphanumeric keyboards.	• PLC marks: QWERTYOFF
PLC. When the keyboard and the operator panel make up a single unit, the PANELOFF mark only disables the jog panel.	• PLC marks: PANELOFF
Load a file cam from the PLC.	• PLC command: CAM SELECT CAM DESELECT
Detailed CNC status in jog mode.	• Variable: (V.)G.CNCMANSTATUS
Detailed CNC status in automatic mode.	• Variable: (V.)G.CNCAUTSTATUS
Know the axes selected for home search, repositioning, coordinate preset or movement to a coordinate.	• Variable: (V.)G.SELECTEDAXIS
Know the current position of the main rotary axes of the kinematics (third axis).	• Variable: (V.)G.POSROTT
Know the target position of the main rotary axes of the kinematics (third axis).	• Variable: (V.)G.TOOLORIT1 (V.)G.TOOLORIT2
Know the status of a cam.	• Variable: (V.)G.CAMST[cam]
Modify the range of the slave axis when activating the cam.	• Variable: (V.)G.CAM[cam][index]
Set 0% feedrate override via PLC.	• Variable: (V.)PLC.FRO

Ref. 1107

Software V04.11	
Synchronized switching.	• Machine parameters: SWTOUTPUT SWTDELAY • Variables: (V.)G.TON (V.)G.TOF (V.)G.PON (V.)G.POF
CANopen bus. Configuring CNC baudrate.	• Machine parameter: CANOPENFREQ.

Ref. 1304

Software V04.20	
Runaway protection and tendency test.	• Machine parameters: TENDENCY TENDTIME.
Synchronize spindles without forcing a set change.	• Machine parameter: SYNCSET.
Mechatrolink. Activate the drive options.	• Machine parameter: OPTION.
Load a file cam from the PLC.	• PLC command: CAM SELECT CAM DESELECT
Interruption subroutines per channel.	• Machine parameters: INT1SUB/INT4SUB SUBINTSTOP • PLC commands: INT1/ INT4.
Set specific acceleration and jerk for G00 movements.	• Machine parameters: G0ACDCJERK LACC1G0 LACC2G0 LFEEDG0 ACCELG0 DECELG0 ACCJERK0 DECJERK0
There may be up to 30 OEM subroutines per channel now (G180-G189/ G380-G399).	• Machine parameters: OEMSUB (G380) - OEMSUB (G399)
Configure how to operate the CNC. Access work modes using hotkeys or from the softkey menu.	• Machine parameter: HMITYPE.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Software V04.20	
Configure how to use the softkey menu, either using menus and submenus (there are different softkey levels within a work mode) or using popup menus (there is only 1 softkey menu, without submenus).	• Machine parameter: SFTYPE.
Maximum safety limit for feedrate.	• Machine parameter: FLIMIT. • PLC commands: FLIMITAC FLIMITACCH.
Maximum safety speed limit.	• Machine parameter: SLIMIT. • PLC commands: SLIMITAC SLIMITACSPDL.
Know the status of a cam.	• Variable: (V.)G.CAMST[cam]
Modify the range of the slave axis when activating the cam.	• Variable: (V.)G.CAM[cam][index]
Set 0% feedrate override via PLC.	• Variable: (V.)PLC.FRO
Detailed CNC status in automatic mode. New values.	• Variable: (V.)G.CNCAUTSTATUS
Active zero offset.	• Variable: (V.)G.EXTORG
Assign a help text to the graphic softkeys.	
Assign a help text to the icons that show the CNC status, at the top of the screen	

Software V04.21	
New model LCD-10K.	• Machine parameters: JOGKEYDEF n USERKEYDEF n • Variables: (V.)MPMAN.JOGKEYDEF[jk] (V.)MPMAN.USERKEYDEF[uk]

Software V04.22	
Set the zero offsets with a coarse part and a fine part.	• Machine parameter: FINEORG • Variables: (V.)A.ADDORG.xn (V.)A.COARSEORG.xn (V.)A.FINEORG.xn (V.)A.COARSEORGT[nb].xn (V.)A.FINEORGT[nb].xn
Cancel mirror image (G11/G12/G13/G14) after M30 and reset.	• Machine parameter: MIRRORCANCEL
Set the feedrate displaying format (integers and decimals).	• Machine parameter: FFORMAT
Set the spindle speed displaying format (integers and decimals).	• Machine parameter: SFORMAT

Backlash peak cutoff distance.	• Machine parameter: PEAKDISP
Hysteresis for applying the additional command pulse in movement reversals.	• Machine parameter: REVEHYST

Software V04.24	
Additional negative command pulse for analog axes.	• Machine parameters: BAKANOUT
The SPDLEREV mark (reverse turning direction) affects the spindle in M19.	• Machine parameters: M19SPDLEREV • PLC signal: SPDLEREV
The default format for displaying the feedrate is 5.2.	• Machine parameter: FFORMAT
The feedrate display format accepts 0.0.	• Machine parameter: FFORMAT
The spindle speed display format accepts 0.0.	• Machine parameter: SFORMAT
Dynamic distribution of the machining operations between channels. Both options use the same PLC marks.	• PLC signal: DINDISTC1 DINDISTC2 DINDISTC3 DINDISTC4
Leadscrew error compensation tables. The machine reference point, whose position is indicated in parameter REFVALUE, may have any value.	

Software V04.25	
Synchronized switching.	• Machine parameters: SWTOUTPUT SWTDELAY • Variables: (V.)G.TON (V.)G.TOF (V.)G.PON (V.)G.POF
HSC mode. The filter used by parameter HSCFILTERFREQ was IIR and it is now FIR.	• Machine parameter: HSCFILTERFREQ.
HSC mode. The units of parameter FTIMELIM were milliseconds and are now percentage.	• Machine parameter: FTIMELIM.
When OEM subroutines have the .fst extension, the CNC will load them into RAM memory on power-up.	
Error programmed in HSC mode.	• Variable: (V.)G.CONTERROR

Ref. 1305

Software V04.26	
New model LCD-10K.	• Machine parameters: JOGKEYDEF n USERKEYDEF n
New LCD-15 model.	• Variables: (V.)MPMAN.JOGKEYDEF[jk] (V.)MPMAN.USERKEYDEF[uk]
New keyboard VERTICAL-KEYB.	
New keyboard HORIZONTAL-KEYB.	
New operator panel OP-PANEL.	
Keep the longitudinal axis when changing planes (G17/G18/G19).	• Machine parameter: LCOMPTYP.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Software V04.26	
Show the softkey for toggling the display format between millimeters and inches.	• Machine parameter: MMINCHSOFTKEY.
Parameter REFSHIFT is only valid for linear and rotary axes.	• Machine parameter: REFSHIFT.
PLC mark to indicate that the CNC has not corrected the position (coordinate) difference between the master axis and the slave axis of a gantry axis.	• PLC signal: MAXDIFF(axis).

Ref. 1308

Software V04.26.10	
The CNC checks every minute the unit temperature; if in three samples in a row the temperature exceeds 60 °C (140 °F), the CNC activates the OVERTEMP mark.	• PLC signal: OVERTEMP.
When the CPU has a fan, during regular operation of the CNC, it monitors and verifies that the fan is running. This test is run every minute, same as the temperature watch.	
Every time [START] is pressed, the CNC checks that the room temperature does not exceed 65 °C (149 °F) and, if it does, the CNC does not let run the program and issues the corresponding error message.	

Ref. 1309

Software V04.27	
Hardware.	
<ul style="list-style-type: none"> The LCD-10K model (front mounting) has been eliminated. The LCD-15 model (front mounting) has been eliminated. The OP-PANEL-H/E module has been eliminated. The JOG-PANEL module has been eliminated. The KB-PANEL-H module has been eliminated. 	
The _EMERGEN signal activates the _ALARM mark.	• PLC signal: _ALARM.
New language (Korean).	• Machine parameter: LANGUAGE.
Virtual tool axis.	<ul style="list-style-type: none"> Machine parameters: VIRTAXISNAME VIRTAXCANCEL Variables: (V.)G.VIRTAXIS (V.)G.VIRTAXST (V.)A.VIRTAXOF.xn
RCS-S module management.	<ul style="list-style-type: none"> Machine parameters: NSERCOUNT SERCOUNTID FEEDBACKTYPE
Select the feedback type of the RCS-S module.	• Machine parameters: FEEDBACKTYPE.
SSI feedback in the RCS-S module.	<ul style="list-style-type: none"> Machine parameters: SSITYPE SSI
Modify the simulation speed via PLC.	• Variable: (V.)PLC.SIMUSPEED
Correction of the delay inserted by the bus and the drive.	• Machine parameters: TRANSDELAY.
PWM (Pulse-Width Modulation)	<ul style="list-style-type: none"> Machine parameters: PWMOUTPUT PWMCANCEL PLC signal: PWMON. Variables: (V.)G.PWMON (V.)G.PWMFREQ (V.)G.PWMDUTY (V.)PLC.PWMFREQ (V.)PLC.PWMDUTY

Ref. 1408

Software V04.27.10	
The axes can also be position-Sercos.	
HSC. Path filter frequency for profiles of linear acceleration.	• Machine parameter: SOFTFREQ
HSC. Default value of the maximum path error in HSC.	• Machine parameter: HSCROUND
HSC. Axis filter frequency (SURFACE mode).	• Machine parameter: SURFFILFREQ
HSC. Default mode when programming #HSC ON.	• Machine parameter: HSCDEFAULMODE
Override of the dynamics for HSC.	<ul style="list-style-type: none"> Machine parameters: MINDYNOVR MAXDYNOVR STEPDYNOVR Variable: (V.)PLC.DYNOVR
New name for the (V.)G.CONTERROR variable	• Variable: (V.)G.ACTROUND
Maximum frequency generated on the machining path.	• Machine parameter: MAXFREQ
HSC. Position error.	• Machine parameter: MAXERROR
HSC. Tolerance per axis for smoothing the generated n-dimensional path.	• Machine parameter: CONTERROR
Software V05.01	
ModBUS server.	<ul style="list-style-type: none"> Machine parameters: MODBUSSVRTCP MODBUSSVRRS MODSVRID MODBRATE
CANopen bus communication frequency.	• Machine parameter: CANOPENFREQ
Feedback type associated with the handwheel input,	• Machine parameter: HWFBTYPE
Detailed CNC status in jog mode. New values.	• Machine parameter: (V.)G.CNCMANSTATUS



(REF: 1709)

Software V05.01	
Activate the Mechatrolink drive options.	• Machine parameter: OPTION
Enable the hardware alarm (alarm pin) of the local feedback.	• Machine parameter: HWFBACKAL
Maximum position difference allowed to consider that there is no need to home again.	• Machine parameter: MAXDIFREF
Software V05.10	
Vectorial definition of spindle kinematics.	• TYPE50 kinematics.
Vectorial definition of table kinematics.	• TYPE51 kinematics.
Vectorial definition of spindle-table kinematics.	• TYPE52 kinematics.
Type of the active kinematics.	• Variable: (V.)G.KINTYPE
Number of axes of the active kinematics.	• Variable: (V.)G.NKINAX
Current position of the fourth rotary axis of the kinematics.	• Variable: (V.)G.POSROTO
Position to be occupied by the fourth rotary axis of the kinematics in order to position the tool perpendicular to the inclined plane (solution 1 and 2).	• Variable: (V.)G.TOOLORIO1 • Variable: (V.)G.TOOLORIO2
Status of the #CSROT function.	• Variable: (V.)G.CSROTST
Position (machine coordinates) calculated for the rotary axis of the kinematics at the beginning of the block, for solution 1 of the #CSROT mode.	• Variables: (V.)G.CSROTF1[1] (V.)G.CSROTS1[1] (V.)G.CSROTT1[1] (V.)G.CSROTO1[1]
Position (machine coordinates) calculated for the rotary axis of the kinematics at the end of the block, for solution 1 of the #CSROT mode.	• Variables: (V.)G.CSROTF1[2] (V.)G.CSROTS1[2] (V.)G.CSROTT1[2] (V.)G.CSROTO1[2]
Position (machine coordinates) calculated for the rotary axis of the kinematics at the beginning of the block, for solution 2 of the #CSROT mode.	• Variables: (V.)G.CSROTF2[1] (V.)G.CSROTS2[1] (V.)G.CSROTT2[1] (V.)G.CSROTO2[1]
Position (machine coordinates) calculated for the rotary axis of the kinematics at the end of the block, for solution 2 of the #CSROT mode.	• Variables: (V.)G.CSROTF2[2] (V.)G.CSROTS2[2] (V.)G.CSROTT2[2] (V.)G.CSROTO2[2]
Position (machine coordinates) to be occupied by the rotary axis of the kinematics at the beginning of the block, for the #CSROT mode.	• Variables: (V.)G.CSROTF[1] (V.)G.CSROTS[1] (V.)G.CSROTT[1] (V.)G.CSROTO[1]
Position (machine coordinates) to be occupied by the rotary axis of the kinematics at the end of the block, for the #CSROT mode.	• Variables: (V.)G.CSROTF[2] (V.)G.CSROTS[2] (V.)G.CSROTT[2] (V.)G.CSROTO[2]
Position of the part zero transformed by the instruction #KINORG, considering the table position, on the first three axes of the channel.	• Variable: (V.)G.KINORG1 (V.)G.KINORG2 (V.)G.KINORG3
Allow the user modify the kinematics parameters.	• Machine parameters: TDATA, TDATA_I • Variable: (V.)MPK.TDATAFkin[nb] (V.)G.OFTDATAkin[nb] (V.)G.OFTDATAFkin[nb] (V.)G.OFTDATA_lkin[nb] (V.)MPK.MAXOFTDATAkin[nb] (V.)MPK.MAXOFTDATAFkin[nb] (V.)MPK.MAXOFTDATA_lkin[nb]

Ref. 1501

Software V05.20	
New language (dutch).	• Machine parameter: LANGUAGE.
Multi-axis management. The deactivated axis maintains its coordinate regardless of the active axis.	• Machine parameter: KEEPPPOS.
Define the feedback type of the handwheel connected to the local feedback.	• Machine parameter: HWFBTYPE.
Define the feedback type of the axis connected to the local feedback.	• Machine parameter: FEEDBACKTYPE.
Distance left (to go) for the axis to reach the programmed coordinate.	• Variable: (V.)A.TOGO.xn

Ref. 1505

Software V05.31	
Enable tool withdrawal (retraction) in threading.	• Machine parameter: RETRACTTHREAD. • PLC marks: RETRACT.
Assume IPLANE as active plane after M30/RESET or keep the active one.	• Machine parameter: PLANECANCEL.
HSC. Orientation smoothing of the rotary axes working with RTCP.	• Machine parameter: ORISMOOTH.
If the [ESC] key is configured as "Previous component", it does not change the main component (Main Menu).	• Machine parameter: FUNCTION.
Maximum frequency recommended for SSI communication, depending on cable length.	• Machine parameter: SSICLKREQ.
The SPDLEREV mark (reverse turning direction in M19) also affects the Sercos-velocity spindles.	• Machine parameter: M19SPDLEREV.
Kinematics (types 9 through 12). Origin for applying the RTCP.	• Machine parameter: TDATA7.
Kinematics (types 13 through 16). Origin for applying the RTCP.	• Machine parameter: TDATA7.
Variables related to CNC status.	
• Detailed CNC status in automatic mode. New value \$100000.	• Variable: (V.)G.CNCAUTSTATUS
Variables associated with the analog inputs and outputs.	
• Volts of output [n] of the RCS-S module.	• Variable: (V.)G.ANASO[n]



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Software V05.31	
Variables related to the programmed functions.	
<ul style="list-style-type: none"> Withdraw the axes after interrupting an electronic threading. 	<ul style="list-style-type: none"> Variable: (V.)G.RETREJ
Variables related to functions.	
<ul style="list-style-type: none"> System spindles involved in the subroutine associated with M3, M4, M5, M19 and M41-M44. 	<ul style="list-style-type: none"> Variable: (V.)G.SUBMSPDL
Variables related to canned cycles.	
<ul style="list-style-type: none"> Active canned cycle. 	<ul style="list-style-type: none"> Variable: (V.)G.ACTIVECYCLE
Variables related to the probe.	
<ul style="list-style-type: none"> Status of probe ·1·. Probing movement. Value measured at the master spindle of the channel. 	<ul style="list-style-type: none"> Variable: (V.)G.PRBST Variable: (V.)G.PLMEAS4
Variables related to axis and spindle repositioning.	
<ul style="list-style-type: none"> End of axis and spindle repositioning at the starting point. End of axis and spindle repositioning at the interruption point. 	<ul style="list-style-type: none"> Variable: (V.)G.ENDREPINI Variable: (V.)G.ENDREPINT
Variables related to synchronized switching.	
<ul style="list-style-type: none"> Time remaining to activate the laser output. Time remaining to deactivate the laser output. 	<ul style="list-style-type: none"> Variable: (V.)G.LASEROTMON Variable: (V.)G.LASEROTMOFF
PWM related variables	
<ul style="list-style-type: none"> Amount of time that PWM stays active in burst mode. Final PWM status once burst mode is over. 	<ul style="list-style-type: none"> Variable: (V.)G.PWMBTIME Variable: (V.)G.PWMBEND
Variables related to cycle time.	
<ul style="list-style-type: none"> Percentage of loop time (cycle time) used by the PLC. Percentage of loop time (cycle time) used by the dynamic preparation of the tool path. 	<ul style="list-style-type: none"> Variable: (V.)G.PLCTIMERATE Variable: (V.)G.TRAYTIMERATE
Variables related to the feedback inputs.	
<ul style="list-style-type: none"> Value of the local count-up 1 input. Value of the local count-up 2 input. 	<ul style="list-style-type: none"> Variable: (V.)G.LCOUNTER1 Variable: (V.)G.LCOUNTER2
Feedrate related variables.	
<ul style="list-style-type: none"> Actual (real) CNC feedrate in G95. Real feedrate on the tool path. Feedrate active in the block. 	<ul style="list-style-type: none"> Variable: (V.)G.FREALPR Variable: (V.)G.ACTFEED Variable: (V.)G.IPOFEED
Variables related to the active tool and to the next one.	
<ul style="list-style-type: none"> Active tool. Code of the tool offset type. 	<ul style="list-style-type: none"> Variable: (V.)TM.TOOLTYP[ofd]
Variables associated with the tool being prepared.	
<ul style="list-style-type: none"> Tool being prepared. Code of the tool offset type. Tool being prepared. Tool-holder orientation. 	<ul style="list-style-type: none"> Variable: (V.)G.TOOLTYP Variable: (V.)G.FIXORI
Variables related to the kinematics and coordinate transformation.	
<ul style="list-style-type: none"> Solution 2 is selected in instruction #CS or #ACS. 	<ul style="list-style-type: none"> Variable: (V.)G.TORISOL2
General variables of the CNC.	
<ul style="list-style-type: none"> CNC model. CNC sub-version number (decimal value). 	<ul style="list-style-type: none"> Variable: (V.)G.CNCMODEL Variable: (V.)G.SUBVERSION
Variables associated with the part-program being executed.	
<ul style="list-style-type: none"> Number of the line of the program where the cursor is. 	<ul style="list-style-type: none"> Variable: (V.)G.CURSORLINE
Variables related to the machine parameters of the channels.	
<ul style="list-style-type: none"> Orientation smoothing of the rotary axes working with RTCP. 	<ul style="list-style-type: none"> Variable: (V.)MPG.ORISMOOTH
Variables related to managing the feedrate in HSC mode.	
<ul style="list-style-type: none"> Amount of error allowed on the axis for the HSC mode. 	<ul style="list-style-type: none"> Variable: (V.)A.ACTROUND.xn

Ref. 1512

Software V05.40	
The maximum number of axes that can be programmed is now 32 instead of 28.	<ul style="list-style-type: none"> Machine parameter: NAXIS.
The maximum number of spindles is now 6 instead of 4.	<ul style="list-style-type: none"> Machine parameter: NSPDL.
The maximum number of axes depends on the number of spindles, the sum of axes plus spindles being 32 at the most.	
Home search mode for position-Sercos axes.	<ul style="list-style-type: none"> Machine parameter: REFMODE.
Tandem axes. Torque control mode.	<ul style="list-style-type: none"> Machine parameter: TORQMODE.
Apply tool radius compensation in G00.	<ul style="list-style-type: none"> Machine parameter: G00COMP.
The default value of parameter CANMODE is now CANopen.	<ul style="list-style-type: none"> Machine parameter: CANMODE.
Work zones.	<ul style="list-style-type: none"> Machine parameter: ZONELIMITTOL. PLC mark: LIM(axis)OFF. Variable: (V.)G.ZONEST[k] (V.)G.ZONETOOLWATCH[k] (V.)G.ZONEWARN[k] (V.)A.ZONELIMITTOL.xn (V.)A.ZONELOWLIM[k].xn (V.)A.ZONEUPLIM[k].xn (V.)G.ZONECIR1[k] (V.)G.ZONECIR2[k] (V.)G.ZONER[k] (V.)G.ZONECIRAX1[k] (V.)G.ZONECIRAX2[k]
PLC. Asynchronous magazine. The TMOOPERATION mark can take the value of 16.	<ul style="list-style-type: none"> PLC mark: TMOOPERATION.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Ref. 1603

Software V05.45	
Safe probing cycles.	<ul style="list-style-type: none"> PLC mark: PROBE1MONIT, PROBE2MONIT.
Probe enabled.	<ul style="list-style-type: none"> PLC mark: PROBE1ENA, PROBE2ENA.

Ref. 1604

Software V05.50	
Number the digital inputs of the CANopen bus based on logic blocks.	<ul style="list-style-type: none"> Machine parameters: NDIMOD DIMODADDR NODE BLOCK ADDRESS NDI
Number the digital outputs of the CANopen bus based on logic blocks.	<ul style="list-style-type: none"> Machine parameters: NDOMOD DOMODADDR NODE BLOCK ADDRESS NDO
Enable/disable a PT100 input from the PLC.	<ul style="list-style-type: none"> PLC mark. PT100OFF1..PT100OFF20
New management of M06 in turret-type magazines	<ul style="list-style-type: none"> Function: T and M06.

Ref. 1702

Software V05.50.50	
Laser model.	
Laser. Laser status.	<ul style="list-style-type: none"> PLC mark. LASERON SHUTTERON LASERREADY
Laser. Default pressure units.	<ul style="list-style-type: none"> Machine parameters: PRESSURE
Laser. Synchronized switching.	<ul style="list-style-type: none"> Machine parameters: SWTOUTPUT SWTDELAY SWTG0FEED Variable: (V.)G.TON (V.)G.TOF (V.)G.PON (V.)G.POF (V.)G.LASEROTMON (V.)G.LASEROTMOFF
Laser. PWM (Pulse-Width Modulation).	<ul style="list-style-type: none"> Machine parameters: PWMOUTPUT PWMCANCEL PLC mark: PWMON PLC register: PWMFREQ PWMDUTY Variable: (V.)G.PWMON (V.)G.PWMFREQ (V.)G.PWMDUTY (V.)G.PWMBTIME (V.)G.PWMBEND
Laser. Power control.	<ul style="list-style-type: none"> Machine parameters: PWRCTRLACT PWRCTRLCANCEL PLC mark: ENABLEPWROUT ENABLEPWRDUTY. Variable: (V.)G.PWROUTON (V.)G.PWROUTOVRMIN (V.)G.PWROUTOVRMAX (V.)G.PWROUTFMIN (V.)G.PWROUTFMAX (V.)G.PWRDUTYON (V.)G.PWRDUTYOVRMIN (V.)G.PWRDUTYOVRMAX (V.)G.PWRDUTYFMIN (V.)G.PWRDUTYFMAX



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Software V05.50.50	
Laser. Gap control.	<ul style="list-style-type: none"> Machine parameters: GAPANAINID GAPDISTLIMIT GAPVOLTLIMIT GAPSENSOROFFSET GAPSENSORCH GAPGAIN GAPSENSORFILTER GAPSENORDELAY GAPERRORCANCEL GAPMIN GAPMAX GAPTOLCANCEL GAPTOL GAPCOLLISIONMODE GAPTONEG PLC mark: INPOSGAP INTOL ENABLEGAP GAPCOLLISION Variable: (V.)G.GAPCTRLON (V.)G.GAPAXISPOS (V.)G.GAPPRG (V.)G.GAP (V.)A.GAPCOMP.xn
Laser. Leapfrog.	<ul style="list-style-type: none"> Machine parameters: LEAPDYNOVR PLC mark: INPOSLIMIT Variable: (V.)G.LEAPON (V.)G.LEAPG91 (V.)G.LEAPPOS (V.)G.LEAPOSLIMIT
Laser. CO2 laser path dispersion compensation.	<ul style="list-style-type: none"> Machine parameters: LASERFOLLOWAXIS LASERFOLLOWOFFSET LASERFOLLOW1 LASERFOLLOW2 LASERFOLLOW3 Variable: (V.)A.FLWMMASTER.xn
Laser. First resonance frequency of the machine in the arcs.	<ul style="list-style-type: none"> Machine parameters: CURVFREQRES
Laser. Subroutines definition. <ul style="list-style-type: none"> Subroutine associated with the command #INITIALSUB.. Subroutine associated with the command #PIERCING. Subroutine associated with the command #CUTTING ON. Subroutine associated with the command #CUTTING OFF. Subroutine associated with the command #FINALSUB. 	<ul style="list-style-type: none"> Machine parameters: INITIALSUB Machine parameters: PIERCING Machine parameters: CUTTINGON Machine parameters: CUTTINGOFF Machine parameters: FINALSUB
Laser. Adjusting the maximum power.	<ul style="list-style-type: none"> Machine parameters: G00FEED MAXVOLT
Laser. Sign of the spindle command for M3/M4.	<ul style="list-style-type: none"> Machine parameters: POLARM3 POLARM4
Laser. Number of the analog output and of the feedback input associated with the spindle.	<ul style="list-style-type: none"> Machine parameters: ANAOUTTYPE COUNTERID
Laser. In the laser model, the parameters related to spindle speed (defined in rpm) define the power (defined in watts).	
Laser. Active material.	<ul style="list-style-type: none"> PLC mark: ACTIVEMATERIALON
Laser. Piercing enabled.	<ul style="list-style-type: none"> PLC mark: PIERCING
Laser. Active cutting.	<ul style="list-style-type: none"> PLC mark: CUTTING
Laser. Technology tables.	<ul style="list-style-type: none"> PLC mark: COMVARACT CUTVARACT PIRVARACT
Laser. Other marks of PLC.	<ul style="list-style-type: none"> PLC mark: _STOP _FEEDHOL
Laser. Name of the active technological table	<ul style="list-style-type: none"> Variable: (V.)G.ACTIVEMATERIAL
Laser. Active cutting type.	<ul style="list-style-type: none"> Variable: (V.)G.CUTTINGTYPE
Laser. Active piercing type.	<ul style="list-style-type: none"> Variable: (V.)G.PIERCINGTYPE
Laser. Technological tables (common parameters).	<ul style="list-style-type: none"> Variable: (V.)TT.name
Laser. Technological tables (piercing).	<ul style="list-style-type: none"> Variable: (V.)TTPIR.name (V.)TTPIR.GAPDIST (V.)TTPIR.POWER (V.)TTPIR.FREQUENCY (V.)TTPIR.DUTY (V.)TTPIR.PIERCINGTIME (V.)TTPIR.GASPRESSURE (V.)TTPIR.GASTYPE (V.)TTPIR.GASTIME (V.)TTPIR.NOZZLEDIAM (V.)TTPIR.FEEDRATE (V.)TTPIR.CUTRADIUS (V.)TTPIR.FOCALPOS

Software V05.50.50	
Laser. TTechnological tables (cutting).	<ul style="list-style-type: none"> Variable: (V.)TTCUT.name (V.)TTCUT.GAPDIST (V.)TTCUT.POWER (V.)TTCUT.FREQUENCY (V.)TTCUT.DUTY (V.)TTCUT.PWROVRMIN (V.)TTCUT.PWROVRMAX (V.)TTCUT.PWRFMIN (V.)TTCUT.PWRFMAX (V.)TTCUT.GASPRESSURE (V.)TTCUT.GASTYPE (V.)TTCUT.GASTIME (V.)TTCUT.NOZZLEDIAM (V.)TTCUT.FEEDRATE (V.)TTCUT.CUTRADIUS (V.)TTCUT.FOCALPOS
Laser. Power programming.	<ul style="list-style-type: none"> Variable: (V.)A.SREAL.sn (V.)SP.SREAL.sn
Laser. Power in G97.	<ul style="list-style-type: none"> Variable: (V.)A.SPEED.sn (V.)SP.SPEED.sn (V.)PLC.S.sn (V.)A.PRGS.sn (V.)SP.PRGS.sn
Laser. Power limit.	<ul style="list-style-type: none"> Variable: (V.)A.SLIMIT.sn (V.)SP.SLIMIT.sn (V.)PLC.SL.sn (V.)A.PRGS�.sn (V.)SP.PRGS�.sn
Laser. Power percentage (override).	<ul style="list-style-type: none"> Variable: (V.)A.SSO.sn (V.)SP.SSO.sn (V.)PLC.SSO.sn (V.)A.PRGSO.sn (V.)SP.PRGSO.sn (V.)A.CNCSSO.sn (V.)SP.CNCSSO.sn
Laser. Subroutine calling parameters.	<ul style="list-style-type: none"> Variable: (V.)C.PCALLP_A...(V.)C.PCALLP_Z (V.)G.CALLP

Ref. 1703

Software V05.50.51	
<p>Laser. Improvements in the gap control.</p> <ul style="list-style-type: none"> The machine parameter GAPSENSORDELAY is no longer functional. Dynamic response during the final part of the movement approaching the plate. If the axis exceeds the GAPMIN-GAPMAX range, the INPOSGAP mark only stops the axis if the machine parameter GAPERRORCANCEL is set to show an error. If the axis exceeds the GAPTOL range, the INTOL mark only stops the axis if the machine parameter GAPTOLCANCEL is set to show an error. 	<ul style="list-style-type: none"> Machine parameters: GAPSENSORDELAY GAPAPPROACHDYN PLC marks: INPOSGAP INTOL
Laser. Adjusting the axis that controls the gap. The compensation of the delays between the CNC and the drive is implemented through the ACFWFACTOR parameter. The TRANSDELAY parameter of the axis associated with the sensor must have a value of 0 (zero).	<ul style="list-style-type: none"> Machine parameters: TRANSDELAY ACFWFACTOR
Laser. Improvements in the leapfrog.	
HMI. Define the initial position of the interface on the screen. The interface can be moved by clicking with the mouse on the icon that shows the program status of the active channel.	<ul style="list-style-type: none"> Machine parameters: POSX POSY
HMI. Define the interface resolution.	<ul style="list-style-type: none"> Machine parameters: RESOLUTION WIDTH HEIGHT
HMI. Customizing the number and the dimensions of the softkeys.	<ul style="list-style-type: none"> Machine parameters: HMENUNSOFTKEY HMENUNSOFTKEY SOFTKEYMETRICS

Ref. 1704

Software V05.50.51	
The "CNC Variables" Chapter has been removed from the manual. New manual.	

Ref. 1707

Software V05.50.53	
Improved leapfrog.	
Enable and disable the leapfrog from the PLC.	<ul style="list-style-type: none"> PLC mark: ENABLELEAP



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Ref. 1709

Software V05.60.00	
Operating Terms.	• Variables: G.CNCDISSTAT
Basic volumetric compensation.	• Machine parameters: VMOVAXIS1, VMOVAXIS2, VMOVAXIS3 NPOINTS1, NPOINTS2, NPOINTS3 INIPOSAX1, INIPOSAX2, INIPOSAX3 INCREAX1, INCREAX2, INCREAX3
Non-modal G02 and G03 functions.	• Machine parameters: G2G3MODAL
Non-modal function G00.	• Machine parameters: G0MODAL IMOVE
Subroutine associated with the reset.	• Subroutine: PROGRAM_RESET
Subroutine associated with the tool calibration cycle.	• Subroutine: KinCal_Begin.nc KinCal_End.nc
Status of the power-up and power-down process.	• Variables: (V.)E.CNCLOGST
Logic number of the axis, according to its order in the channel.	• Variables: (V.)[ch].G.NLOGAXIS1 (V.)[ch].G.NLOGAXIS2 .. (V.)[ch].G.NLOGAXISn • Variables: (V.)[ch].G.NLOGSPDL1 (V.)[ch].G.NLOGSPDL2 .. (V.)[ch].G.NLOGSPDLn
Modify the proportional gain (Kp) of the tandem from the PLC.	• Variables: V.MPG.TPROGAIN[nb]
Disable the feedback input alarms for the RCS-S module.	• Machine parameters: DRIVEVAR.
Disable the local feedback input alarms.	• PLC marks: LCOUNTALARMOFF1 LCOUNTALARMOFF2
FeeDat / EnDat encoder Information.	• Machine parameters: NAME TYPE RESOLUTION SERIALNUMBER DATALENGTH
EnDat transmission format.	• Machine parameters: ENDAT ENDATCLKFREQ



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

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 or material damage originated from not complying with these basic safety rules.



Before start-up, verify that the machine that integrates this CNC meets the 2006/42/EC Directive.

PRECAUTIONS BEFORE CLEANING THE UNIT

- Do not get into the inside of the unit.** Only personnel authorized by Fagor Automation may access the interior of this unit.
- 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 powered.

PRECAUTIONS DURING REPAIRS

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 access the interior of this unit.
- 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 powered.

PRECAUTIONS AGAINST PERSONAL HARM

- Interconnection of modules.** Use the connection cables provided with the unit.
- Use proper cables.** To prevent risks, only use cables and Sercos fiber recommended for this unit.
To prevent a risk of electrical shock at the central unit, use the proper connector (supplied by Fagor); use a three-prong power cable (one of them being ground).
- Avoid electric shocks.** To prevent electrical shock and fire risk, do not apply electrical voltage out of the indicated range.
- Ground connection.** In order to avoid electrical discharges, connect the ground terminals of all the modules to the main ground terminal. Also, before connecting the inputs and outputs of this product, make sure that the ground connection has been done.
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.** In order to avoid electrical discharges, always work with a relative humidity (non-condensing).
- Do not work in explosive environments.** In order to avoid risks, harm or damages, do not work in explosive environments.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

PRECAUTIONS AGAINST DAMAGE TO THE PRODUCT

Work 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, homes, etc.).
Install this unit in the proper place.	It is recommended, whenever possible, to install the CNC away from coolants, chemical product, blows, etc. that could damage it. This unit meets 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 mains as the unit. Nearby portable transmitters (radio-telephones, Ham radio transmitters). Nearby radio / TC transmitters. Nearby arc welding machines. Nearby high voltage lines.
Enclosures.	It is up to the manufacturer to guarantee that the enclosure where the unit has been installed meets all the relevant directives of the European Union.
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, operator panel and the remote modules.
Connecting the power supply to ground.	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.	Maintain the CNC within the recommended temperature range, both when running and not running. See the corresponding chapter in the hardware manual.
Central unit enclosure.	To maintain the right ambient conditions in the enclosure of the central unit, it must meet the requirements indicated by Fagor. See the corresponding chapter in the hardware manual.
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.

SAFETY SYMBOLS

Symbols that may appear in the manual.



Danger or prohibition symbol.
This symbol indicates actions or operations that may hurt people or damage products.



Warning or caution symbol.
This symbol indicates situations that certain operations could cause and the suggested actions to prevent them.



Obligation symbol.
This symbol indicates actions and operations that must be carried out.



Information symbol.
This symbol indicates notes, warnings and advises.



Symbol for additional documentation.
This symbol indicates that there is another document with more detailed and specific information.

Symbols that the product may carry.



*Ground symbol.
This symbol indicates that that point must be under voltage.*



*ESD components.
This symbol identifies the cards as ESD components (sensitive to electrostatic discharges).*



BLANK PAGE

RETURNING CONDITIONS

Pack it in its original package along with its original packaging material. If you do not have the original packaging material, pack it as follows:

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

BLANK PAGE

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. Never use air compressed at high pressure to clean the unit because it could cause the accumulation of electrostatic charges that could result in electrostatic shocks.

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 supplied with power. Before handling these connectors (I/O, feedback, etc.), make sure that the unit is not powered.
- Do not get into the inside of the unit. Only personnel authorized by Fagor Automation may access the interior of this unit.

BLANK PAGE

PART 1. INSTALLATION MANUAL.



BLANK PAGE

SOFTWARE INSTALLATION.

1

1.1 Software installation at the CNC.

Fagor delivers the CNC with the software properly installed in a compact flash type disk. Both at a CNC and at a PC (simulator), the necessary files for the CNC are located in the folder C:\CNC8070 and its relevant subfolders. See "[1.6 Software configuration.](#)" on page 56.



The CNC software must not be re-installed or modified in any way without the express consent from Fagor Automation. Fagor Automation shall not be held responsible for any personal injuries, physical or material damage suffered or caused by the CNC due to software manipulation.

Software protection at the CNC. System with a write-protected (read-only) disk.

Fagor delivers the unit with the disk that is write-protected except for the folders or files that must be unprotected for the normal operation of the CNC. The disk has been pre-configured with three access modes, each offers a different protection level. See "[1.1.1 Work modes and software protection at the CNC.](#)" on page 45.

Software setting.

It is up to manufacturer to set up the CNC and adapt it to his machine. The manufacturer can also customize the CNC's look using the screen customizing program FGUIM. Before using this tool, read the relevant documentation carefully.

CNC setting; the MTB folder.

Both at a CNC and at a PC (simulator), the OEM has the folder ..\MTB to save the modifications done at the CNC; such as the PLC program, the machine parameters, etc. The CNC manages the MTB folders as follows.

- No MTB folder appears when installing the software for the first time. On system startup and depending on the validation code, the system will move the corresponding folder from ..\CONFIGURATION to the CNC8070 and will rename it MTB.
- When changing the validation code, the system will return the MTB folder to the ..\CONFIGURATION folder and will give it its old name (MTB_M, MTB_T, etc); then, it will move the corresponding folder from ..\CONFIGURATION to CNC8070 and will rename it MTB.
- If the OEM has created the MTB folder manually, for example by copying it from a backup, the system will not make any changes on power-up nor when modifying the validation code.

Changing the language of the help files.

Fagor supplies the CNC with the help files installed in English. The CD that comes with the product contains the help files in different languages. You can change the help files installed by default with the ones provided on the CD.

Locate the folder *Help files* inside the CD, select one of the available languages and copy all the files to its CNC location. The help files installed at the CNC (or at the PC, if it is a simulator) are located in the following folder.

C:\Cnc8070\Fagor\MMC\Help

The help files can only be in one language at a time at the CNC. The language of the help files may be different from the one chosen for the interface.

1.

SOFTWARE INSTALLATION.
Software installation at the CNC.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

1.1.1 Work modes and software protection at the CNC.

Fagor delivers the unit with a protected compact flash type disk that is write-protected except for the folders or files that must be unprotected for the normal operation of the CNC. The changes made to protected folders or files will be operative until the unit is turned off and back on, the CNC will then restore the initial configuration. The changes made to unprotected areas of the disk will remain.

The disk has been pre-configured with three access modes, each offers a different protection level. The unit shows the active work mode with an icon on the task bar of the operating system, next to the clock. When the CNC is turned on, the status bar shows the active work mode with icons.

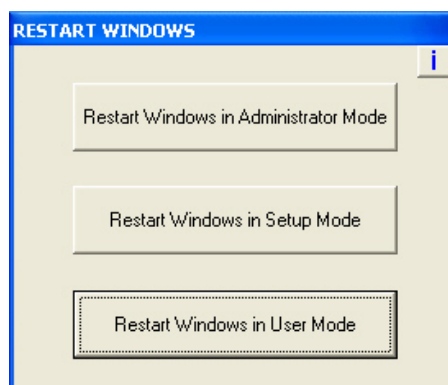
Icon.	Work mode.
	Administrator mode.
	Setup mode.
	User mode.



These work modes will not be available when upgrading the software from a version where the unit has a disk that is not write-protected (not read-only). Since it is not write-protected, the changes made to it will remain.

Work mode change (diskMonitor window).

- Having the CNC application turned off, the work mode change is done using the [ALT][D] or with icon indicating the active work mode, on the task bar of the operative system (next to the clock). In either case, the unit shows the DiskMonitor window to switch from one work mode to another. This process implies restarting the unit and if the manufacturer has set it up this way, it will also be necessary to enter the corresponding password.
- Having the CNC started up in any work mode, the [ALT][D] hotkey shows the DiskMonitor window to change from one work mode to another. This process implies restarting the unit and if the manufacturer has set it up this way, it will also be necessary to enter the corresponding password.
- The desired work mode may be selected from the utilities mode, from the softkey menu. This process implies restarting the unit and if the manufacturer has set it up this way, it will also be necessary to enter the corresponding password.



SOFTWARE INSTALLATION.
Software installation at the CNC.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Protecting or unprotecting the folders.

In the previous window, when selecting the -i- at the top right hand side, the unit shows the list of folders and files that are unprotected at the time. In administrator mode and the list of folders being visible, pressing [CTRL]+[ALT]+[TAB]+[SHIFT] it is possible to protect or unprotect any folder or file of the system that is not key for the proper operation of the CNC.

Administrator mode.

The access to the administrator mode is enabled with the validation code ("Open system" software option). If you don't have this software option, (i.e. you have a "closed system") you will not be able to access the administrator mode and, therefore, you will not be able to install third-party software.



This mode must only be used to install non-Fagor software, to install the CNC (also possible from the setup mode), to update the operative system or change the system configuration. The CNC application does not start up in this mode.



The unit shows the following image on the desk, with red background, indicating the active work mode and warning that it is not a protected mode.

Protection level.

There is not protection level in administrator mode, the whole disk is unprotected.

Protection password.

The access to this mode is protected with the password "administrator mode", defined in the utilities mode. When starting the unit up in this work mode, it will request the access password.

Setup mode.



The setup mode must only be used to update the CNC software and to set up the machine; it does not allow installing third-party software. This mode may be used to access the operative system.



The unit shows the following image on the desk, with yellow background, indicating the active work mode and warning that it is not a protected mode.

Protection level.

The setup mode has an intermediate protection level where everything that may be changed while setting the machine up is unprotected; folders ..\MTB, ..\USERS, ..\DIAGNOSIS and the Windows register.

Protection password.

The access to this mode is protected with the password "machine parameters", defined in the utilities mode. When starting the unit up in this work mode, it will request the access password.

1.

SOFTWARE INSTALLATION.
Software installation at the CNC.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

User mode.



It is the usual work mode for the user, once the setup is completed. This mode does not allow updating the CNC or accessing the operative system. Some utilities of the operative system (task manager, clock) will be available from the diagnosis mode.

Part-programs must be saved in the "..\USERS" folder; the CNC considers the files saved in other folders as temporary files and will be deleted when the CNC is turned off. Files from flash, pendrives, ethernet, etc. can only be managed from the explorer of the utilities mode.

Protection level.

The user mode has the maximum protection level where only the folders and files that may be changed during the normal operation of the machine are unprotected.

Protection password.

The access to this mode is not protected with the password.

Restrictions for working in Setup mode and switching to User mode.

The purpose of the following restrictions is for the OEM to deliver his machines with the CNC in protected work mode (User mode).

SETUP mode

On power-up, the CNC will show a message indicating that it is in an unprotected mode and that the setup has not been completed yet. In this situation, the CNC is no longer under Fagor warranty. After a certain period of time, with the next reset, the CNC shows the message again.

When closing the application, the CNC asks if the setup is completed..

- If YES is selected and there are passwords, the CNC makes a backup copy and switches over to USER mode (process OK).
- If YES is selected and there are no passwords, the CNC issues a warning message and does not close the application.

USER mode (work mode).

On power-up, the CNC checks for passwords and a backup copy. If any of these two is missing, the CNC will show a message indicating that setup has not been completed yet. In this situation, the CNC is no longer under Fagor warranty. The CNC checks this at every reset.

This situation can be reached when accessing the USER mode from SETUP mode through "DiskMonitor".

1.

SOFTWARE INSTALLATION.
Software installation at the CNC.

FAGOR

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

1.2 Software installation at the PC (simulator).

The CNC may be installed in the specific hardware that will later be mounted onto the machine or at a table-top PC that will be used as a simulator for training purposes. The CNC installed at a PC offers all the features and functions but it can only be used in simulator mode and cannot be connected to any type of machine.

The CNC installation CD contains all that is necessary to install the software and the documentation needed to install, set up and operate the CNC. The CNC software must be installed in the hard disk of the PC; it cannot be executed directly from the CD. The installation will start automatically when inserting the CD in the CD drive; if not, double-click on the setup70_Vxx_xx.exe file, where Vxx_xx indicates the version to be installed. Then, follow the instructions displayed on the screen.

Both at a CNC and at a PC (simulator), the necessary files for the CNC are located in the folder C:\CNC8070 and its relevant subfolders. See "[1.6 Software configuration](#)." on page 56.

Installing the licenses.

Local license or network license.

During the installation, you will have to choose the type of license; local license (LOCAL) or network license (NET). The type of license indicates where the dongle (hardware key) is connected. For the dongle (hardware key) of the parallel port, always choose the local license.

- Choose the local license if the dongle (hardware key) will be connected to the PC where the CNC software has been installed.
- Choose the net license if the dongle (hardware key) will be connected to a server. When the CNC simulator needs to be authenticated, it will automatically look for the server throughout the net. The CNC software needs not be installed at the server; however, the software that enables the dongle (hardware key) must be installed at the server. This software is included on the CD.

About the dongle (hardware key).

Fagor supplies two types of dongle (hardware key) to be connected either to the parallel port or to the USB port. Once the software has been installed, in order to use the CNC, the dongle (hardware key) supplied with the CD must be connected to the PC. If the net license was chosen when installing the software, the dongle (hardware key) may be connected to a server.

CNC setting; the MTB folder.

Both at a CNC and at a PC (simulator), the OEM has the folder ..MTB to save the modifications done at the CNC; such as the PLC program, the machine parameters, etc. The CNC manages the MTB folders as follows.

- No MTB folder appears when installing the software for the first time. On system startup and depending on the validation code, the system will move the corresponding folder from ..\CONFIGURATION to the CNC8070 and will rename it MTB.
- When changing the validation code, the system will return the MTB folder to the ..\CONFIGURATION folder and will give it its old name (MTB_M, MTB_T, etc); then, it will move the corresponding folder from ..\CONFIGURATION to CNC8070 and will rename it MTB.
- If the OEM has created the MTB folder manually, for example by copying it from a backup, the system will not make any changes on power-up nor when modifying the validation code.

Changing the language of the help files.

The help files are installed in English. The CD that comes with the product contains the help files in different languages. You can change the help files installed by default with the ones provided on the CD.

1.

SOFTWARE INSTALLATION.
Software installation at the PC (simulator).



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Installation manual.

Locate the folder *Help files* inside the CD, select one of the available languages and copy all the files to its CNC location. The help files installed at the CNC (or at the PC, if it is a simulator) are located in the following folder.

C:\Cnc8070\Fagor\MMC\Help

The help files can only be in one language at a time at the CNC. The language of the help files may be different from the one chosen for the interface.

1.

SOFTWARE INSTALLATION.
Software installation at the PC (simulator).

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

1.3 Updating the software version.

The updates must be carried out using the software supplied by Fagor Automation. Updating the software maintains the set up of the machine parameters, PLC program, tool table and tool magazine data. Before updating the software, check the list of possible incompatibilities between versions. See "[1.3.1 Software update since previous versions.](#)" on page 51.

Before updating the software.

It is recommended to always have a backup copy of the full configuration (ASCII files) such as machine parameter tables, tool tables, active-tools table and tool magazine tables as well as the PLC program. Should any anomaly occur during the installation, these file will help restore the CNC configuration.

Software update.

To update the software, close all the programs that may be running, including the CNC. In a system with a protected disk, the software must be installed in setup mode.

The installation will start automatically when inserting the CD in the CD drive; if not, double-click on the setup70_Vxx_xx.exe file, where Vxx_xx indicates the version to be installed. Then, follow the instructions displayed on the screen.

When starting the process, the CNC offers the possibility of installing from scratch; in this case, the installation process will not keep the CNC configuration like the machine parameters, PLC program, etc.

Updating remote nodes of the CAN bus.

Every time the CNC is powered up, it verifies the versions of the remote nodes detected in the CAN bus and automatically updates all these devices if necessary. When done loading, it goes on with the usual start-up process.

If the loading is not successful, and, consequently, the software coherence between all the elements of the CAN bus cannot be guaranteed, the CNC will display the corresponding error message every time [RESET] is pressed.

1.

SOFTWARE INSTALLATION.
Updating the software version.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

1.3.1 Software update since previous versions.

Version V2.00.

Tool and tool magazine table.

Due to the improvements made to the tool tables and magazine data in version V2.00, these tables must be updated manually.

Before updating the software, save all the data of these tables in ASCII format and once the installation is completed, load this data into the tables. Both operations are carried out from the tool table and magazine table using the "Save" and "Load" softkeys.

Validation code.

After activating the software from a version V1.1x or older, the validation code of this version is no longer valid and a new validation code must be entered.

Version V3.10.

Enabling the spindle.

On power-up, when executing any operation on the spindle, the CNC homes the spindle. In order to avoid "axis locked up" type errors, the PLC maneuver must include the enabling of the mark SERVOSON with the M19 functions, rigid tapping and when activating the mark ENABLES.

Tool table.

The CNC attends to the icon that sets the tool turning direction. If the turning direction is not the same as the programmed one, the CNC will display an error message.

C axis programming.

The #CYL instruction requires programming the radius. To correct the programs of older versions where the radius has not been defined, it must be defined with a value of 0 (zero). It works the same way.

Version V3.15.

Modifications to parameter matching.

In earlier versions, the CNC sent to the drive the value of parameter ABSOFF only when the system had distance-coded reference marks. From this version on, the CNC also sends the value of parameter ABSOFF to the drive (PP177/PP178) when the system uses absolute feedback.

From version V3.15, when updating an earlier version on machines that use absolute feedback, parameter ABSOFF must be set with the value of PP177 or PP178 at the drive.

Version V3.20.

The MTB folder has been changed.

From version V3.20 on, when installing the software for the first time, the CNC8070 folder does not contain the any MTB folder. In the new CONFIGURATION folder, the CNC has a different MTB folder for each type of software installed; MTB_T for lathe and MTB_M for mill. On system startup and depending on the validation code, the system will move the corresponding CONFIGURATION folder to the CNC8070 and will rename it MTB. If the type of machine changes when changing the validation code, the CNC will make the necessary changes to update the MTB folder.

When updating the software, the CNC respects the original MTB folder. In this case, if the type of machine changes when changing the validation code, the CNC will not move the MTB folder.

1.

SOFTWARE INSTALLATION.
Updating the software version.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Version V4.00.

Zero offsets and synchronism marks.

When installing version V4.00, there is no guarantee that the CNC will keep the active zero offsets or the status of the synchronization marks.

Sercos axes.

While updating from a version V3.00 or later, the CNC will show error E9332 for each Sercos axis with external feedback. This error indicates that the values of parameters COUNTERTYPE and COUNTER ID are not compatible. To correct this error, change the value of parameter COUNTERTYPE from `·remote·` to `·drive·` on the axes involved.

Unicode.

The editor of the CNC admits Unicode characters. Programs in Unicode format are not compatible with versions older than V4.0. A program in Unicode format cannot be edited or executed in a version older than V4.0.

A third-party text editor (like Windows notepad) may be used to convert Unicode format programs into ANSI format; but special characters that have no ANSI equivalent will be lost in the process.

Version V4.02.

Machine with several keyboards.

From version V4.02 on, when upgrading an earlier version on machines using several keyboards, they must be configured as follows. If the keyboards are not configured, the CNC sets parameter NKEYBD to 1 (default value) and only enables the first keyboard (the one with the lowest number according to the Address switch).

Parameter.	Value.	Meaning.
NKEYBD	2	Number of jog panels.
KEYBD1CH	Active channel.	Channel to which the first jog panel is assigned.
KEYBD2CH	Active channel.	Channel to which the seconds jog panel is assigned.

The PLC mark MANRAPID is no longer a general mark, it is now a channel mark.

The MANRAPID mark that in earlier versions affected all the axes of the system now only affects the axes of the first channel. In order for the mark to affect all the axes of the system, the PLC program must be modified using marks MANRAPIDC1 through MANRAPIDC4 according to the channels of the system.

Marks PANELOFF and QWERTYOFF to disable the keyboards.

In earlier versions, the PLC mark PANELOFF disabled both the jog part and the alphanumeric part (module OP-PANEL-H/E). From this version, the new QWERTYOFF disables the alphanumeric keypad and the PANELOFF mark only disables the jog panel.

Version V4.10.

Updating the software version.

The version must be updated in administrator mode, not in setup mode as it is usually done, in order to register the new libraries permanently. If the installation is done in setup mode, the register is only temporary until the unit is turned off.

1.

SOFTWARE INSTALLATION.
Updating the software version.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Version V4.20.

Updating the software version.

The version must be updated in administrator mode, not in setup mode as it is usually done, in order to register the new libraries permanently. If the installation is done in setup mode, the register is only temporary until the unit is turned off.

Version V4.25.

Unit change in machine parameter FTIMELIM, related to HSC machining.

In earlier versions, parameter FTIMELIM was programmed in milliseconds. From this version on, this parameter is set in percentage.

Version V5.31.01.

Apply or not tool radius compensation in G00.

In this version, movements in G00 do not apply tool radius compensation. This has been corrected in later versions with parameter G00COMP.

Version V4.40.

New software option "third-party CANopen".

From this version one, if a machine is using some non-Fagor CANopen module, the CNC will not recognize it and it will issue the corresponding error message when initializing the CAN bus.

Extended scope of the software option "Probing canned cycles".

The software option "Probing canned cycles" is now called "Probe". From this version on, functions G100, G101, G102, G103 and 104 need this software option.

1.

SOFTWARE INSTALLATION.
Updating the software version.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

1.4 Requirements before and after CNC setup.

1.

SOFTWARE INSTALLATION.

Requirements before and after CNC setup.

System with a write-protected (read-only) disk.

The CNC setup is carried out in setup mode. Fagor delivers the unit ready to start up in this mode. When powering the unit up, the desktop will show an image showing this circumstance.

After starting the CNC application, proceed with setup. When starting the application, it will display a warning indicating that it is operating in an unsafe mode.

When closing the application with the key sequence [ALT]+[F4] and choosing the options to "close" or to "close and make a backup copy", the CNC shows a second dialog box asking whether setup has been completed or not. If the "NO" is chosen, the next time the unit is turned on, it will start in setup mode again. If the "Yes" option is chosen, the CNC makes a backup copy of the MTB folder and, the next time it is turned on, the unit will start in user mode.



The unit must be delivered to the user ready to start up in user mode.

The unit will remain in setup mode until the manufacturer changes it to user mode in response to the dialog displayed when shutting the application down. If while in user mode, it is necessary to change something that has to do with setup, the setup mode must be accessed manually. See "[1.1.1 Work modes and software protection at the CNC.](#)" on page 45.

When starting the application up, either in setup or user mode, the CNC display can show warnings whose cause must be solved by the manufacturer during setup.

System without a write-protected (read-only) disk.

The configuration changes to a CNC without a write-protected (read only) disk are permanent. When closing the application by pressing the key combination [ALT]+[F4], the CNC shows the option to save a backup copy of the machine configuration, directory MTB, out o a compressed file.

If normal shutdown is selected and the CNC detects that the machine configuration has been changed, it will warn about it and will ask whether to make a backup copy of the machine configuration or not.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

1.5 Installing third-party software.

The CNC you have purchased is an industrial PC equipped with Windows XP Embedded or Windows 7 operating system that permits the installation of third-party applications. The installation of this software must meet the following requirements.

Any malfunction of the equipment due to the installation of third-party software frees Fagor Automation from any responsibility.

Installation in the hard disk.

The CNC that you purchased may have either just a compact flash or a compact flash and a hard disk. To install any software, the CNC must have enough free memory in the compact flash. To avoid ending up with little free memory in the compact flash, it may be necessary to install third-party software in a hard disk.

Also, in order to install third-party software, you must have an open system; i.e. with access to the administrator. The access to the administrator mode is enabled with the validation code. If you don't have this software option, you will not be able to access the administrator mode and, therefore, you will not be able to install third-party software. Software installation done in setup mode or user mode will disappear when turning the CNC off.

- In a system without write-protected disk, access to the administrator mode is protected with the password "machine parameters", defined in the utilities mode.
- In a system with write-protected disk, access to the administrator mode is protected with the password "administrator mode", defined in the utilities mode.

To obtain the relevant password, contact the supplier of your machine.

When installing third-party software in a system with write-protected disk, remember that the results of the software, generated programs, etc. must be saved in an unprotected folder, for example CNC8070\USERS. Never use a protected folder because it will be temporary and it will disappear when turning the CNC off.

System requirements.

Before installing the software, make sure that the CNC meets all the requirements of the software, both the CPU and the memory. You can consult this information in the CNC's diagnosis mode.

Also check that there is enough free memory space in the disk for your application.

Remember that the equipment, the operating system and the CNC (depending on the configuration of the machine) may consume between 50% and 60% of the available resources. Once the software has been installed, the CNC being up and running, check the status of the system resources and check that the CNC is running properly, screen refresh, etc.

1.

SOFTWARE INSTALLATION.
Installing third-party software.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

1.6 Software configuration.

The necessary files for the CNC are located in the folder C:\CNC8070 and its relevant subfolders.

Folder.	Contents.
BACKUPSDIR	Backup of previous installations. Every time the user installed a full new version, the installer saves a copy of the previous installation in this folder.
CONFIGURATION	MTB folders of the system. This folder contains the various MTB folders of the system; MTB_T for lathe and MTB_M for mill. When booting system and depending on the validation code, the system will move the corresponding folder from CONFIGURATION to MTB.
DIAGNOSIS	Information for the diagnosis. This folder contains relevant information for proper error diagnosis, including the reportfagor.zip file.
DRIVERS	CNC drivers. This folder contains some necessary drives for proper operation of the CNC.
FAGOR	Version folder. This folder contains the software corresponding to the CNC version installed. Software updates are carried out in this directory and they do not affect the contents of directories MTB and USERS. Do not change the contents of this directory. Only authorized personnel from Fagor Automation may modify the contents of this directory. Fagor Automation shall not be held responsible of the performance of this CNC if the contents of this directory have been changed.
MTB	OEM folder. This folder is especially directed at machine manufacturers. This folder contains the modifications made by the OEM at the CNC like, for example, the PLC program, machine parameters, new screens, integrating external applications, etc.
TMP	Temporary files. The CNC uses this folder to save the temporary files it generates while operating. The CNC deletes the contents of this folder on power-up.
USERS	User folder. This folder is especially directed at users. The purpose of this folder is to provide the user with a memory space for storing part-programs, profiles, etc. as they are generated.



The CNC deletes the contents of the TMP folder on power-up. Do not use this folder to save files that you'd like to keep.



1.6.1 MTB (Machine Tool Builder) folder.

This folder is especially directed at machine manufacturers. This folder contains the modifications made by the OEM at the CNC like, for example, the PLC program, machine parameters, new screens, integrating external applications, etc.

Folder.	Contents.
DATA	<p>This folder contains the following files or subfolders.</p> <ul style="list-style-type: none"> • The databases for machine parameters, tables, etc. and the safety backup (in ASCII) of those tables. • The files related to the data of the editor cycles (dat files). • The data the CNC must maintain after it has been turned off (coordinates, zero offsets, etc.). • Each language of the ..\LANG\ subfolder 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 ..\LANG\ENGLISH\cncError.txt; if it is not there, the CNC will issue the relevant error message.
DRIVE	This folder contains the information regarding the DDSSETUP mode.
GRAFDATA	This folder contains the information regarding graphics.
KINEMATIC	This folder contains the information regarding the OEM kinematics.
MMC	<p>This folder contains the CNC custom setting made by the machine builder.</p> <ul style="list-style-type: none"> • The "...\CONFIG" subfolder contains the configuration files (ini files) and the files that may be modified using the screen customizing tool (Fguim). • The ..\IMAGES subfolder contains the images, icons, videos, etc. that the OEM uses to customize the CNC. • Each language of the ..\LANG subfolder contains the texts used by the scripts. Each language of the ..\LANG subfolder contains the file with the texts for the language active at the CNC. When changing the CNC language, the CNC replaces this file with the one corresponding language.
PLC	<p>This folder contains the information regarding the PLC.</p> <ul style="list-style-type: none"> • Each language of the ..\LANG\ subfolder contains the messages and errors of the PLC in different languages. • The ..\PROJECT subfolder contains the files that make up the PLC project and the object file. • The ..\WATCH contains the data saved from the monitoring and logic analyzer services.
RELEASE	This folder contains the components (ocx files) used by the OEM to create his own application.
SUB	<p>This folder contains the following files or subfolders.</p> <ul style="list-style-type: none"> • The OEM subroutines (tool change, home search, etc.). • Each language of the ..\HELP folder contains the help files associated with OEM subroutines and the pcall.txt file that contains the list of OEM subroutines. If these files are not in the folder of the language active at the CNC, the editor will not provide them as help.
TUNING	This folder contains the information regarding the setup assistance.



SOFTWARE INSTALLATION.
Software configuration.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

1.6.2 USERS folder.

This folder is especially directed at users. The purpose of this folder is to provide the user with a memory space for storing part-programs, profiles, etc. as they are generated. We recommend to save these programs in the folders created for this purpose in order to make it easier and faster to find them and make safety backup copies.

In a system with a write-protected disk, the programs created by the user must be saved in this folder; the only one that is not protected. Any program saved in a protected folder will be temporary and will be deleted when the CNC is turned off.

Folder.	Contents.
HELP	Each language of the ..\HELP folder contains the help files associated with user-defined global subroutines and the pcall.txt file that contains the list of user subroutines.
POCKET	This folder contains the profiles that have been created using the profile editor and are related to the editor cycles.
PRG	This folder contains the part-programs created by the user who may create new subfolders and store the programs in a more orderly fashion. The subfolder "...PRG_8055_TO_8070" contains the programs translated from the 8055 format into the 8070 language. The CNC saves the converted (translated) program with the same name but with the extension m55 (milling program) or t55 (lathe program).
PROFILE	This folder contains the profiles that have been created using the profile editor.
REPORTS	This folder contains the bmp files that the CNC generates when printing a graphic to a file and the pm reports generated from the diagnosis mode.
SUB	This folder contains the subroutines created by the user, whose location must be fixed for the CNC (for example, the generic user subroutines G500-G599) and the subroutine program_start)

1.

SOFTWARE INSTALLATION.
Software configuration.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

MACHINE PARAMETERS.

2

In order for the machine tool to be able to properly execute the programmed instructions and interpret the elements connected to it, the CNC must know the specific machine data. Some of this data may be the number of axes, feedrates, accelerations, feedback, type of tool magazine, tool changer, etc. This data is set by the machine manufacturer and must be communicated to the CNC through the machine parameters. The machine parameters of the CNC are shown grouped as follows.

Machine parameters.	Meaning.
General parameters.	These parameters set the name of the axes and spindles, power-up conditions, subroutines associated with specific functions, etc. Some of these parameters must be defined first because they configure the axis parameter tables. For example, the number and name of the axes and spindles, etc.
Axis and spindle parameters.	These parameters indicate the properties of the axes and spindles, travel limits, moving conditions, related handwheels, probing, compensations, etc. Each axis or spindle may have up to 4 work sets. The following must be set for each one: feedrates and gains, home searches, accelerations, etc.
Jog parameters.	These parameters set the handwheels and the JOG keys.
Parameters of the M function table.	These parameters set the new M functions, the type of synchronization and the subroutine associated with each M function of the system.
Parameters for the Kinematics table.	These parameters indicate the type and characteristics of each kinematics.
Parameters for the magazines.	These parameters indicate the number of tool magazines and number of tool pockets (positions), etc.
HMI parameters.	These parameters are used to define the communication environment (interface) between the operator and the CNC.
OEM parameters.	These parameters set the reading/writing of drive variables, editing cams, defining a group of generic parameters so the manufacturer can use them like machine parameters, etc.

Icons associated with machine parameters.

Parameter validating method.



This icon may appear next to the parameter name and indicates that the CNC must be restarted for the parameter to assume its new value. The parameters that do not have this icon next to them can assume the new value by just pressing the "Validate" softkey.

Parameter matching.



This icon may appear next to the parameter name and indicates that it involved in the parameter matching between the CNC and the drive.







FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Selection of parameter values.

The icons appearing next to the parameter value permit accessing the list of pre-established values, a data table, a set of parameters or to refer to a file. For those parameters that do not have an icon next to them, a value must be edited within the indicated range.

Icon.	Meaning.
	The parameter has a list of options.
	To access a data table.
	To access a group of parameters.
	The parameter refers to a file.

Default value of the parameters at a simulator.

When the CNC is installed as a simulator at a PC, the default value of certain machine parameters may be different from those mentioned here. The changes basically have to do with the maximum axis feedrates, accelerations and jerk. The probe data has also been changed in order to have, in jog mode, the part centering and tool calibration cycles.

2.

MACHINE PARAMETERS.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.1 Parameter matching between the CNC and the Sercos drive.

Some parameters of the CNC and of the drive must be set equivalently to ensure proper system operation. While initializing the Sercos ring, on CNC power-up and when validating the machine parameters of the axes and spindles, the CNC updates at the drives the value of the parameters necessary to ensure that they are equivalent.



The CNC does not match the parameters when the drive is an ACSD.

Parameter matching and speed ranges (gears).

The number of parameter sets to define the speed ranges (gears) may be different at the CNC and at the drive. If there is a parameter set at the CNC, the parameter matching updates parameters NP121, NP122, NP131 and NP133 of each set of the drive with the values defined at the CNC. The parameter matching will save the CNC's default set values in the rest of the sets of the drive.

Understanding the parameter matching table.

The table indicates which CNC parameters are involved in parameter matching as well as the drive parameters that are affected. The different columns of the table have the following meaning.

- CNC List of CNC machine parameters.
- DRIVE List of drive parameters that are equivalent to each CNC parameter.
- Sercos It indicates writing the parameter at the drive depends or not on the type Sercos configuration, position or velocity).
- Feedback It indicates whether or not writing the parameter at the drive depends on the type of axis feedback, internal or external.

CNC	DRIVE	Sercos	Feedback	Remarks
AXISTYPE AXISMODE	PP76			PP76=65; Linear axis. PP76=66; Rotary axis without module. PP76=194; Rotary axis with module.
PITCH	NP123			
INPUTREV	NP121.x			It affects all the gears.
OUTPUTREV	NP122.x			It affects all the gears.
NPULSES2	PP115 (bit 0) NP117		External. External.	B0=0; If rotary encoder (NPULSES2<>0). B0=1; If linear encoder (NPULSES2==0). NP117; If rotary encoder (NPULSES2<>0).
PITCH2	NP133 NP117 NP118		External. External. External.	NP133; If rotary encoder (NPULSES2<>0). NP117; If linear encoder (NPULSES2==0). NP118; If linear encoder (NPULSES2==0).
INPUTREV2	NP131.x		External.	It affects all the gears. Only if rotary encoder (NPULSES2<>0).
OUTPUTREV2	NP132.x		External.	It affects all the gears. Only if rotary encoder (NPULSES2<>0).
FBACKSRC SINMAGNI	GP10			GP10=0; External feedback is not being used. GP10=1; TTL signal (SINMAGNI==0). GP10=2; Vpp signal (SINMAGNI<>0).
FBACKDIFF	PP5	Position.	External.	
FBMIXTIME	PP16	Position.	External.	
AXISCH LOOPCH	PP115 (bit 3)	Position.	External.	B3=0; Feedback sign AXISCH==LOOPCH. B3=1; Feedback sign AXISCH<>LOOPCH.
AXISCH	PP55 (bit 0,2,3)	Position.		B1=0 B2=0 B3=0; It does not change the sign of the feedback reading (AXISCH==NO) B1=1 B2=1 B3=1; It does not change the sign of the feedback reading (AXISCH==YES)

2.

MACHINE PARAMETERS.

Parameter matching between the CNC and the Sercos drive.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

MACHINE PARAMETERS.

Parameter matching between the CNC and the Sercos drive.

CNC	DRIVE	Sercos	Feedback	Remarks
I0TYPE	PP115 (bit 1,5)		External.	B1=0 B5=0; If distance-coded I0. B1=1 B5=0; if increasing distance-coded I0. B1=1 B5=1; if decreasing distance-coded I0.
I0CODD11 I0CODD12	NP166 NP165			Only when using distance-coded I0's. Only when using distance-coded I0's.
REFDIREC DECINPUT FBACKSRC	PP147 (bit 0) PP147 (bit 5) PP147 (bit 3) PP147 (bit 1)			B0=0; Positive home search direction. B0=1; Negative home search direction. B5=0; A home switch is being used. B5=1; A home switch is not being used. B3=0; Internal feedback. B3=1; External feedback. B1=0; The DECEL signal of the CNC is always positive.
REFFEED1	PP41			
REFFEED2	PP1			
REFVALUE	PP52 PP54	Position. Position.	Internal. External.	
REFSHIFT	PP150 PP151	Position. Position.	Internal. External.	
ABSOFF	PP177 PP178		Internal. External.	Only when using distance-coded I0's or absolute feedback.
PROGAIN	PP104			
BACKLASH	PP58	Position.		
BACKANOUT	PP2	Position.		
BACKTIME	PP3	Position.		Only if BACKANOUT<>0
REVEHYST	PP15			
PEAKDISP	PP14			
NPARSETS	GP6			
Activate software limits.	PP55 (bit 4)			B4=1; Check the limits. B4=0; Ignore the limits (for spindles, rotary axes with module and when both parameters LIMIT+ and LIMIT- are set to 0).
MODLIMUP	PP103			Only if it is a spindle or a rotary axis with module.
SZERO	SP42			Only if it is a spindle.
INPOSW	PP57			
MAXFLWE	PP159			Only if following error monitoring is active.
Calculation of the coordinate module.	PP76 (bit 7)			B7=0; On rotary axes or spindles working in velocity-Sercos, the CNC calculates the module of the coordinate.

2.2 Parameters to verify before the startup.

To ensure the proper performance of the CNC and prevent any damage to the machine, the machine parameters must be properly set, especially the ones related to alarms, travel limits, following errors, feedrate and speed. After the setup, make sure that the parameters related to these aspects are set with operative values.

For safety reasons, verify that these parameters are properly set; wrong values, even if they are within the permitted range could cause the machine to perform poorly.

Here are the parameters that you should at least check.

General machine parameters.

Parameter	Meaning
WARNCOUPE	Gantry axis. Maximum difference allowed between the following errors of both axes before issuing a warning.
MAXCOUPE	Gantry axis. Maximum difference allowed between the following errors of both axes.
MINCORFEED	Minimum feedrate for the HSC at the corners.
MAXFEED	Maximum machining feedrate. This parameter should be set to a value other than -0-.

Machine parameters for the axes and spindles.

Parameter	Meaning
FBACKDIFF	Maximum difference between feedbacks. The CNC takes this parameter into consideration the axes set as position Sercos and with external feedback or external+internal feedback. This parameter should be set to a value other than -0-.
DSYNVELW	Synchronization of axes and spindles. This parameter is defined for the slave element of the synchronization and indicates the velocity margin admitted for the synchronization to be OK.
DSYNPOSW	Synchronization of axes and spindles. This parameter is defined for the slave element of the synchronization and indicates the position margin admitted for the synchronization to be OK.
LIMIT+ LIMIT-	Axis travel limits. Both parameter should be set to a value other than -0-.
TENDENCY	Activation of tendency test. It detects axis runaway due to positive feedback. It should be activated during machine setup.
PROBERANGE	Maximum braking distance. This parameter sets the maximum braking distance for the probe after probing to avoid breaking it (ceramic, etc).
PROBEFEED	Maximum probing feedrate.
PROBEDELAY PROBEDELAY2	Delay for the probe 1 and 2 signal. This parameter is only set when there is a short delay from the probing instant to when the CNC actually receives the signals (infrared communication, etc.).
REPOSFEED	Repositioning feedrate after a tool inspection.
POSFEED	Positioning feedrate of the independent axis.
JOGFEED	JOG mode. Feedrate in continuous jog mode.
JOGRAPFEED	JOG mode. Rapid feedrate in continuous jog mode.
MAXMANFEED	JOG mode. Maximum feedrate in continuous jog mode.
MAXMANACC	JOG mode. Maximum acceleration in JOG mode.
INCJOGFEED	JOG mode. Feedrate in incremental jog mode.
FBACKAL	Feedback alarm activation. The feedback alarm must be activated.
G00FEED	Feedrate in G00.

2.

MACHINE PARAMETERS.

Parameters to verify before the startup.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

MACHINE PARAMETERS.

Parameters to verify before the startup.

Parameter	Meaning
MAXFEED	Maximum machining feedrate. This parameter should be set to a value other than -0-.
FLWEMONITOR	Type of monitoring of the following error (axis lag). Monitoring must be activated.
MINFLWE	This parameter indicates the maximum amount of following error allowed when the axis is stopped.
MAXFLWE	For standard monitoring of the following error, this parameter indicates the maximum amount of following error allowed when the axis is moving. For "linear" monitoring of the following error, this parameter indicates the value starting at which the following error will be dynamically monitored.
FEDYNFAC	For "linear" monitoring of the following error, this parameter indicates the percentage error, deviation of the real following error value with respect to the theoretical value.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3 General machine parameters.

2.3.1 Channel configuration.

NCHANNEL

Number of CNC channels.

Possible values: From 1 to 4.

Default value: 1.

Associated variable: (V.)MPG.NCHANNEL

The use of channels is oriented to machines like dual-spindle lathes where each channel has one of the spindles and two axes; machines with feeders, where the machine and the feeder will be different channels; tool magazine loading and unloading system controlled as an axis.

The CNC can have a single execution channel (single channel system) or several (multi-channel system). Each channel is a different work environment that can act upon a part of or the whole CNC system. The difference between a multi-channel system and several independent CNC's is that the channels can not only act independently, but also together; in other words, they can communicate, synchronize with each other and carry out coordinated actions.

The axes and spindles of a channel.

A channel can have a group of axes and spindles that act independently or in parallel with the rest of the channels. It is also possible to configure a channel without assigning axes or spindles to it at first; then, later on, axes and spindles may be added to it or removed from it via part-program or in the MDI/MDA.mode.

Operation of a channel.

In order to be able to move an axis or spindle, it must be assigned to a channel. Each channel can only control its axes and spindles, although via part-program or MDI/MDA it can command movements to axes or spindles of other channels.

Each channel can execute a different program, be in an different work mode and have its own data. The channels can share information through variables and arithmetic parameters and, if necessary, they may be synchronized via part-program.

2.

MACHINE PARAMETERS.

General machine parameters.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.2 Configuring the axes of the system.

NAXIS

Number of axes governed by the CNC.

Possible values: From 1 to 32.

Default value: 3.

Associated variable: (V.)MPG.NAXIS

This parameter sets the number of axes of the system, whether they are servo-controlled or not. The maximum number of axes depends on the number of spindles, the sum of axes plus spindles being 32 at the most. For instance, a system without spindles can have up to 32 axes (NSPDL=0 NAXIS=32) and a system with 6 spindles can have up to 26 axes (NSPDL=6 NAXIS=26).

Bear in mind that the number of axes does not depend on the number of channels. A channel may have one, several or no axes associated with it.

AXISNAME

List of system axes.

This parameter shows the table to define the names of the axes. Parameter NAXIS sets the number of axes of the system.

AXISNAME	
AXISNAME n	Name of the axes.

AXISNAME n

Name of the axes.

Possible values: X, X1..X9, .. , C, C1..C9.

Default value: Starting from AXISNAME1; X, Y, Z...

Associated variable: (V.)MPG.AXISNAME n

Parameter included in the AXISNAME table.

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 axes may be within the range X, X1...X9,...C, C1...C9. For example X, X1, Y3, Z9, W, W7, C...

When defining the axes, bear in mind that the order in which they are defined determines their logic number. The first axis of the table will be logic axis -1- and so on. As with the axis name, the logic number permits identifying the axis in PLC variables, marks, etc.

AXISNAME n	Logic order.
AXISNAME 1	Logic number ·1·.
AXISNAME 2	Logic number ·2·.
AXISNAME 3	Logic number ·3·.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.3 Configuration of a tandem system.

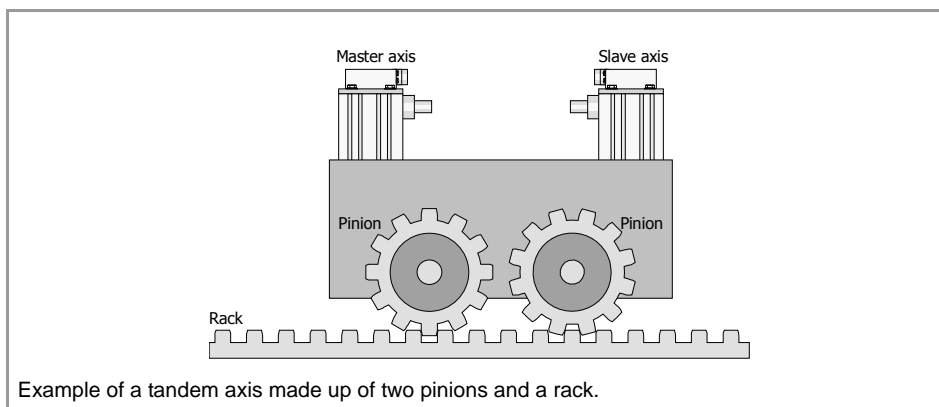
TANDEM

Table of tandem axes or spindles.

This parameter shows the table to define the tandem pairs of the system. A tandem axis consists of two electric motors mechanically coupled to each other forming a single transmission system (axis or spindle).



Managing tandem spindles and axes requires a drive version V6.14 or newer.



The following aspects may be pointed out in this configuration:

- A tandem axis helps provide the necessary torque to move a servo drive system when a single motor is not capable of supplying enough torque to do it.
- Applying a pre-load torque between the main motor and the slave motor reduces the backlash.

Requirements of the tandem systems

Each tandem pair (master and slave) must meet the following requirements:

- Each master tandem axis (spindle) admits one single slave tandem axis (spindle).
- A preload may be applied between the two motors.
- Each motor may have a different rated torque.
- The turning direction of each motor may be different from the other's.
- The torque distribution between both motors may be different from 1:1 ratio. For example, on motors whose rated torque is different.
- Both axes and drives must be of the same type (same AXISTYPE and DRIVETYPE parameters for both axes).
- The channel changing permission (parameter AXISEXCH) must be the same in both axes.
- Both axes and drives must have the same software limits (same LIMIT+ and LIMIT- parameters for both axes).
- When the axes are rotary, both axes must be of the same type (same AXISMODE and SHORTESTWAY parameters for both axes).
- The fast and slow home search speeds (parameters REFFEED1 and REFFEED2) must be the same for both axes.

TANDEM n

Configuration of each tandem pair.

This parameter shows the parameter table to define the tandem pairs of the system. Each tandem pair has the following machine parameters to configure it.

TANDEM	
MASTERAXIS	Tandem pair. Master or main axis/spindle.
SLAVEAXIS	Tandem pair. Slave axis/spindle.
TORQDIST	Tandem pair. Torque distribution.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

TANDEM	
PRELOAD	Tandem pair. Preload between both motors.
PRELFITI	Tandem pair. Filter time to apply the preload.
TPROGAIN	Tandem pair. Proportional gain (Kp) for the tandem.
TINTIME	Tandem pair. Integral gain (Ki) for the tandem.
TCOMPLIM	Tandem pair. Compensation limit.
TORQMODE	Tandem pair. Torque control mode.

MASTERAXIS**Tandem pair. Master or main axis/spindle.**

Possible values: Any axis defined in the parameter AXISNAME or spindle defined in SPDLNAME.

Associated variable: (V.)MPG.TMASTERAXIS[nb]

Parameter included in the TANDEM table.

This parameter sets which is the master axis or spindle of the tandem pair. The master motor of the tandem, besides generating torque, it is in charge of positioning.

The CNC closes the position loop only with the position of the master axis of the tandem. The velocity command of the master axis/spindle of the tandem is also sent to the slave axis/spindle of the tandem closing the velocity loop. The tandem control changes the velocity command of the master axis/spindle and that of the slave axis/spindle according to the torque distribution and the selected preload.

SLAVEAXIS**Tandem pair. Slave axis/spindle.**

Possible values: Any axis defined in the parameter AXISNAME or spindle defined in SPDLNAME.

Associated variable: (V.)MPG.TSLAVEAXIS[nb]

Parameter included in the TANDEM table.

This parameter sets which is the slave axis or spindle of the tandem pair. The slave motor only provides torque. The CNC closes the position loop only with the position of the master axis of the tandem.

TORQDIST**Tandem pair. Torque distribution.**

Possible values: from 1 to 99%.

Default value: 50 %.

Associated variable: (V.)MPG.TORQDIST[nb]

Parameter included in the TANDEM table.

This parameter sets the torque percentage supplied by each motor to obtain the total necessary torque in the tandem. The parameter indicates the percentage to be provided by the master motor. The difference between the value of this parameter and 100% is the percentage applied to the slave motor.

If the motors are identical and they're both supposed to output the same torque, this parameter should be set to 50%.

PRELOAD**Tandem pair. Preload between both motors.**

Possible values: Within $\pm 100\%$.

Default value: 0 (it disables the preload).

Associated variable: (V.)MPG.PRELOAD[nb]

Parameter included in the TANDEM table.

The preload is the amount of torque applied before, in both directions, to both motors of the tandem to set a traction between them in order to eliminate the backlash when the tandem is in rest position. The parameter indicates the percentage of the rated torque of the master motor that will be applied as preload.

In order for the two motors to supply opposite torques, the preload value must be greater than the maximum torque needed at all times, including accelerations.



Applying the preload necessarily implies mechanically joining the master and slave axes that make up the tandem; otherwise, the motors will move even without control command.

PRELFITI

Tandem pair. Filter time to apply the preload.

Possible values: From 0 to 65535 milliseconds (setting it to 0 disables the filter).

Default value: 0 ms.

Associated variable: (V.)MPG.PRELFITI[nb]

Parameter included in the TANDEM table.

First order filter that sets the time it takes the CNC to apply the preload gradually. This filter eliminates the torque steps at the input of the tandem compensator when setting a preload parameter, hence avoiding a step in the velocity command of the master motor and slave motor of the tandem.

TPROGAIN

Tandem pair. Proportional gain (Kp) for the tandem.

Possible values: from 0 to 100%.

Default value: 0 (no proportional gain is applied).

Associated variable: (V.)MPG.TPROGAIN[nb]

Parameter included in the TANDEM table.

The proportional controller generates an output proportional to the torque error between the two motors.

S	Speed.	
S.max	Maximum speed.	$S = Kp \times T.err$
Kp	Proportional gain.	
T.nom	Rated torque.	$Kp = \left(\frac{S.max}{T.nom} \right) \times TPROGAIN$
T.err	Torque error between motors.	
T.mst	Torque of the master motor.	$T.err = (-T.mst + T.slv + PRELOAD)$
T.slv	Torque of the slave motor.	

Example:

A tandem axis has a maximum speed of 2000 rpm and a rated torque of 20 Nm. TPROGAIN has been set to 10%.

$$Kp = (2000 \text{ rpm} / 20 \text{ Nm}) \cdot 0.1 = 10 \text{ rpm/Nm.}$$

TINTTIME

Tandem pair. Integral gain (Ki) for the tandem.

Possible values: From 0 to 65535 milliseconds.

Default value: 0 (no integral gain is applied).

Associated variable: (V.)MPG.TINTTIME[nb]

Parameter included in the TANDEM table.

The integral controller generates an output proportional to the integral of the torque error between the two motors.

S	Speed.	
S.max	Maximum speed.	$S = Ki \times \sum T.err$
Kp	Proportional gain.	
Ki	Integral gain.	$Ki = \left(\frac{ControlTime}{IntegralTime} \right) \times Kp$
T.nom	Rated torque.	
T.err	Torque error between motors.	

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

T.mst	Torque of the master motor.	$T.err = (-T.mst + T.slv + PRELOAD)$
T.slv	Torque of the slave motor.	

TCOMPLIM**Tandem pair. Compensation limit.**

Possible values: from 0 to 100%.

Default value: 0 (disables the tandem axis).

Associated variable: (V.)MPG.TCOMPLIM[nb]

Parameter included in the TANDEM table.

This parameter limits the maximum compensation applied by the tandem. This limit is also applied to the integral.

This parameter refers to the master motor. It is defined as percentage of the maximum speed of the master motor. If programmed with a "0" value, the output of the tandem control will be zero, thus disabling the tandem.

TORQMODE**Tandem pair. Torque control mode.**

Possible values: With torque control / Without torque control.

Default value: With torque control.

Associated variable: (V.)MPG.TORQMODE[nb]

Parameter included in the TANDEM table.

This parameter indicates whether the CNC takes the torque into account when managing the tandem. This parameter should be set to its default value.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

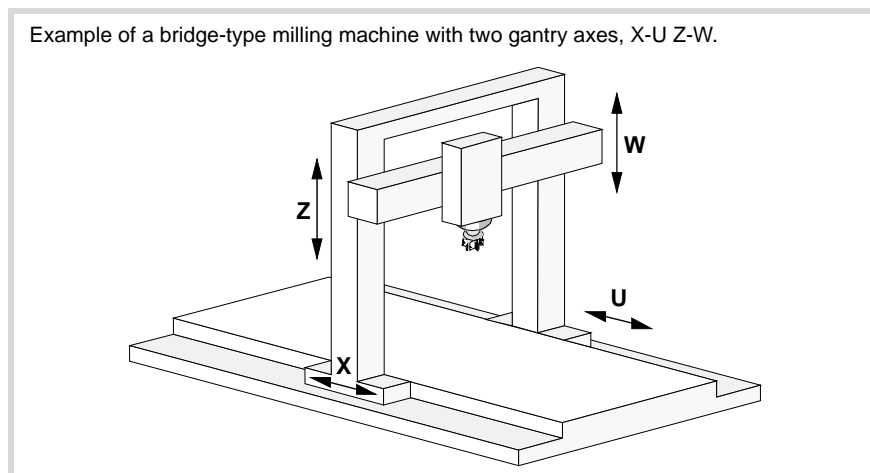
CNC 8070

(REF: 1709)

2.3.4 Configuration of a gantry axis.

GANTRY Gantry-axes table.

This parameter shows the table to define the gantry axes of the system. A gantry axis is a pair of axes that, due to the way the machine is built, must move at the same time and in synchronism. At the CNC, only the movements of one of the axes must be programmed (the master or main gantry axis). The other axis (slave gantry axis) is not programmable; it is controlled by the CNC.



Requirements of the gantry axes.

Each pair of axes (master and slave) must meet the following requirements:

- The master axis must be defined in the AXISNAME table before the slave axis. Both axes must belong to the same channel. The first three axes of the channel cannot be slaves.
- Both axes and drives must be of the same type (same AXISTYPE and DRIVETYPE parameters for both axes).
- The channel changing permission (parameter AXISEXCH) must be the same in both axes.
- Both axes and drives must have the same software limits (same LIMIT+ and LIMIT- parameters for both axes).
- When the axes are rotary, both axes must be of the same type (same AXISMODE and SHORTESTWAY parameters for both axes). Rotary axes that only turn in one direction (parameter UNIDIR = YES) cannot be gantry.
- Hirth axes (HIRTH = YES) cannot be gantry axes.
- The I0 type (parameter I0TYPE) must be the same for both axes either non-distance-coded or distance-coded (increasing or decreasing).
- When the gantry axis does not have distance-coded reference marks (I0), either both axes or just the master axis may have a home switch (parameter DECINPUT). The fast and slow home search speeds (parameters REFFEED1 and REFFEED2) must be the same for both axes.

GANTRY n Configuration of the gantry axes.

This parameter shows the parameter table to define the gantry axes of the system. Each gantry axis has the following machine parameters to configure it.

GANTRY	
MASTERAXIS	Master or main axis.
SLAVEAXIS	Gantry axis. Slave axis.
WARNCOUPE	Difference between the following errors of both axes for issuing a warning.

GANTRY	
MAXCOUPE	Difference between the following errors of both axes for issuing a warning.
DIFFCOMP	Compensate for the coordinate (position) difference between the two axes after G74.
MAXDIFF	Maximum position value (coordinate) difference between both axes to compensate for.

2.

MACHINE PARAMETERS.
General machine parameters.

MASTERAXIS

Gantry axis. Master or main axis.

Possible values: Any axes defined in AXISNAME.

Associated variable: (V.)MPG.MASTERAXIS[nb]

Parameter included in the GANTRY table.

This parameter sets which is the master axis of the gantry pair.

SLAVEAXIS

Gantry axis. Slave axis.

Possible values: Any axes defined in AXISNAME.

Associated variable: (V.)MPG.SLAVEAXIS[nb]

Parameter included in the GANTRY table.

This parameter sets which is the slave axis of the gantry pair.

WARNCOUPE

Gantry axis. Difference between the following errors of both axes for issuing a warning.

Possible values: From 0.0001 to 99999.9999 mm.

Possible values: From 0 to 3937.00787 inch.

Possible values: From 0.0001 to 99999.9999 degrees.

Default value: 1.0000 mm / 0.03937 inch / 1.0000 degree.

Associated variable: (V.)MPG.WARNCOUPE[nb]

Parameter included in the GANTRY table.

This parameter sets the maximum difference allowed between the following errors of both axes before issuing a warning. This warning lets the user act upon the machine before the CNC issues the error.

The value of parameter WARNCOUPE must be lower than that of parameter MAXCOUPE.

MAXCOUPE

Gantry axis. Maximum difference allowed between the following errors of both axes.

Possible values: From 0.0001 to 99999.9999 mm.

Possible values: From 0 to 3937.00787 inch.

Possible values: From 0.0001 to 99999.9999 degrees.

Default value: 1.0000 mm / 0.03937 inch / 1.0000 degree.

Associated variable: (V.)MPG.MAXCOUPE[nb]

Parameter included in the GANTRY table.

This parameter indicates the maximum difference allowed between the following errors of both axes. If the difference exceeds this value, the CNC will display an error message.

DIFFCOMP

Gantry axis. Compensate for the coordinate (position) difference between the two axes after G74.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)MPG.DIFFCOMP[nb]

Parameter included in the GANTRY table.

This parameter enables the correction of the position difference between the master and the slave axes after homing. The correction of the position difference is applied using the mark DIFFCOMP(axis).



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

To compensate the position value, the slave axis will move until reaching the position of the master axis at the feedrate set by parameter REFFEED2. This process can only be interrupted with RESET.

MAXDIFF

Gantry axis. Maximum position value (coordinate) difference between both axes to compensate for.

Possible values: From 0 to 99999.9999 mm.

Possible values: From 0 to 3937.00787 inch.

Possible values: From 0 to 99999.9999 degrees.

Default value: 0.

Associated variable: (V.)MPG.MAXDIFF[nb]

Parameter included in the GANTRY table.

To avoid correcting coordinate differences that are too big between the master and the slave axes, this parameter sets the maximum coordinate difference allowed to be corrected between both axes. The CNC will only correct the coordinate difference when it is smaller than the value set in this parameter.

2.

MACHINE PARAMETERS.
General machine parameters.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.5 Configuration of a multi-axis group.

MULTIAXIS

Table of multi-axis groups.

A multi-axis group may be used to control several axes with a single drive, controlling one or more motors. This parameter shows the multi-axis group definition table for defining the number of groups of the system and the composition of each group.

MULTIAXIS	
MULNGROUPS	Number of multi-axis groups in the system.
GROUP n	Multi-axis groups.

Requirements and limitations of a multi-axis group.

The axes associated with the same drive must meet the following requirements.

- Any Sercos axis of the system may be part of a multi-axis group, except those that belong to gantry or tandem axes.
- Since the axes are controlled by the same drive, all the axes of a group must have the same Sercos address (parameter DRIVEID).
- All the axes of a group must work in the same Sercos operating mode, position or speed, (parameter OPMODEP). If the external feedback (direct feedback) of the axis is connected to a drive, the axis may be operated in either position-Sercos mode or velocity-Sercos mode (velocity-Sercos mode is recommended). If the external feedback (direct feedback) of the axis is connected to local or remote feedback input, the axis can only work in velocity-Sercos mode.
- If the external feedback is local or remote, the CNC homes the axis as if it were analog; the drive does not know that the axis has been homed and does not update its position value, it only receives the velocity command. If the external feedback is connected to another drive, it will not be possible to home the axis.
- The axis being governed by the drive is selected via PLC (mark SWITCH(axis)). The PLC must either park the axes not governed by the drive or set them as DRO axes.
- The CNC keeps monitoring (following error, tendency test, etc.) all the axes sharing the drive even if the axis is not selected.

MULNGROUP

Number of multi-axis groups in the system.

Possible values: From 0 to 16.

Default value: 0.

Associated variable: (V.)MPG.MULNGROUP

Parameter included in the MULTIAXIS table.

This parameter sets the number of multi-axis groups of the system.

GROUP n

Multi-axis groups.

This parameter shows the parameter table to define each multi-axis group. The table shows a GROUP_n parameter for each multi-axis group with the following machine parameters for configuring the groups.

GROUP n	
MULNAXIS	Number of axes and/or spindles making up the multi-axis group.
MULNAXISNAME n	Name of the axes and/or spindles of the group.
KEEPPOS n	The deactivated axis maintains its coordinate regardless of the active axis.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

MULNAXIS

Number of axes and/or spindles making up the multi-axis group.

Possible values: From 1 to 8.

Default value: 2.

Associated variable: (V.)MPG.MULNAXIS[nb]

Parameter included in the MULTIAXIS // GROUP n table.

This parameter sets the number of axes and/or spindles making up the multi-axis group. Since each axis or spindle of the CNC can have several parameter sets, the number of axes of the group is limited by the 8 parameter sets available at the drive. The parameter sets of the drive are distributed among all the axes and/or spindles that make up the multi-axis group, so a group may be formed by 8 axes with 1 parameter set for each, 1 spindle with 4 sets plus 2 axes with 2 sets each, etc.

MULAXISNAME n

Name of the axes and/or spindles of the group.

Possible values: Any Sercos axis of the system, except tandem or gantry axes.

Associated variable: (V.)MPG.MULAXISNAMExn[nb]

Parameter included in the MULTIAXIS // GROUP n table.

Any Sercos axis of the system may be part of a group, except those that belong to gantry or tandem axes. The first axis of the group will be the master axis, the one the drive will use by default; the order of the rest of the axes is irrelevant. On system power-up, the CNC will use the parameters of the master axis to run the parameter matching with the drive.

KEEPPOS n

The deactivated axis maintains its coordinate regardless of the active axis.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)MPG.KEEPPOSxn[nb]

Parameter included in the MULTIAXIS // GROUP n table.

For a multi-axis group, this parameter defines whether the deactivated axis maintains the coordinate or not when the active axis moves.

KEEPPOS	Meaning.
Yes.	When the axis is deactivated, it maintains the coordinate; it ignores the movements of the active axis of the multi-axis group.
No.	When the axis is deactivated, it does not maintain the coordinate; it is affected by the movements of the active axis of the multi-axis group.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.6 Configuring the spindles of the system.

NSPDL

Number of spindles governed by the CNC.

Possible values: From 0 to 6.

Default value: 1.

Associated variable: (V.)MPG.NSPDL

This parameter sets the number of spindles of the system, whether they are servo-controlled or not.

Bear in mind that the number of spindles does not depend on the number of channels. A channel may have one, several or no spindles associated with it.

SPDLNAME

List of spindles of the system.

This parameter shows the table to define the names of the spindles. Parameter NSPDL sets the number of spindles of the system.

SPDLNAME	
SPDLNAME n	Name of the spindles.

SPDLNAME n

Name of the spindles.

Possible values: S, S1..S9.

Default value: Starting from SPDLNAME1; S, S1...

Associated variable: (V.)MPG.SPDLNAME n

Parameter included in the SPDLNAME table.

The spindle name is defined by 1 or 2 characters. The first character must be the letter -S-. 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 S, S1 ... S9.

When defining the spindles, bear in mind that the order in which they are defined determines their logic number. The logic numbering of the spindles continues from the last logic axis; hence, in a 3-axis system, the first spindle of the table will be logic spindle -4- and so on.

AXISNAME	SPDLNAME	Logic order.
AXISNAME 1		Logic number -1-.
AXISNAME 2		Logic number -2-.
AXISNAME 3		Logic number -3-.
	SPDLNAME 1	Logic number -4-.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.7 Time setting (system).

LOOPTIME

CNC cycle (loop) time.

Possible values: From 0 to 20 ms.

Default value: 4 ms.

Associated variable: (V.)MPG.LOOPTIME

This parameter sets the CNC's loop time. Loop time greatly depends on the number of inputs, outputs and analog axes of the bus. The shorter the loop time, the less time will the CPU have to process the data.

Influence of the analog axes on parameter LOOPTIME.

Use the following orientative values:

LOOPTIME	Number of analog axes.
4 ms.	Up to 8 analog axes.
5 ms.	Up to 12 analog axes.
6 ms.	Up to 16 analog axes.
8 ms.	Up to 20 analog axes.
10 ms.	Up to 24 analog axes.

Influence of the CAN bus on parameter LOOPTIME.

From the CAN bus point of view, the minimum loop time for a particular configuration mainly depends on the traffic requirements of the system; in other words, by the time consumed by the transmission of messages per cycle. The factors affect it are the following:

- Total number of remote feedbacks enabled.
- Total number of analog inputs, analog outputs and keyboard handwheels.
- Total number and changing frequency of digital inputs and outputs.
- Frequency of the PLC's PRG module (PRGFREQ).
- CAN transmission speed.

For an exact measurement of the minimum loop time required for a particular configuration, the most reliable thing to do is to use an off-the-shelf tool for a logic analysis of the CAN bus. This tool may be used to set parameter LOOPTIME with a value that covers the communicating requirements reliably, but without saturating the bandwidth of the system (we recommend not to take more than 60% of the bus) to prevent possible transient peaks of traffic or asynchronous eventualities.

If the value of parameter LOOPTIME is too low, the bus will behave as follows.

- CANfagor If the CNC's loop time is too short to manage the desired configuration, it will issue several errors indicating cycle overlapping and timeout in cyclic CAN readings.
- CANopen On CNC startup, it runs an estimate of the amount of time required and it will issue a warning if it is too short.

In either case, it would be necessary to increase the value of parameter LOOPTIME or decrease the communication time of the system (increasing the CAN speed, increasing the value of parameter PRGFREQ or removing physical elements).

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

PRGFREQ**Frequency of the PLC's PRG module.**

Possible values: from 1 to 100 cycles.

Default value: 2 cycles.

Associated variable: (V.)MPG.PRGFREQ

This parameter indicates how often (every how many CNC cycles) a full cycle of the PLC program is executed. This parameter also sets the refreshing frequency of the digital inputs and outputs as well as analog inputs.

Thus, with a sampling period LOOPTIME = 4 ms of and a frequency of PRGFREQ = 2, the PLC program will be executed every $4 \times 2 = 8$ ms.

2.**MACHINE PARAMETERS.**

General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.8 Sercos bus configuration.

SERBRATE

Sercos transmission rate.

Possible values: 2/4/8/16 Mbps (megabits per second).

Default value: 4 Mbps.

Associated variable: (V.)MPG.SERBRATE

This parameter indicates the Sercos transmission speed used when communicating with the drives and RCS-S modules (Sercos counter). All elements of the Sercos ring must have the same value.

Speeds of 8 Mbps and 16 Mbps require a Sercos board that can work at these speeds; otherwise the speed will be limited to 2 Mbps and 4 Mbps.



Sercos communication at 8 and 16 MHz requires a drive version V6.05 or higher.

SERPOWSE

Sercos optical power.

Possible values: From 1 to 6 ("Sercos I" board) / from 1 to 8 ("Sercos II" board).

Default value: 4 ("Sercos I" board) / 2 ("Sercos II" board).

Associated variable: (V.)MPG.SERPOWSE

This parameter sets the Sercos power or the intensity of the light going through the optic fiber. The optical power value depends on the total length of the cable being used. Set this parameter with the same value used by the drives.

Use the following approximate values. Assigning other values, e.g. a value of 6 for a length of 3 m, can cause communication errors due to fiber optic signal distortion.

Recommended values ("Sercos I" board).

SERPOWSE	Cable length.
2	Less than 7 meters.
4	From 7 to 15 meters.
6	More than 15 meters.

Recommended values ("Sercos II" board).

SERPOWSE	Cable length.	Recommended cable type.
1 to 4	Less than 15 meters.	SFO / SFO-FLEX
5 to 6	From 15 to 30 meters.	SFO-FLEX
7	From 30 to 40 meters.	SFO-FLEX
8	More than 40 meters.	SFO-V-FLEX

2.

MACHINE PARAMETERS.
General machine parameters.

2.3.9 Mechatrolink bus configuration.

MLINK

Mechatrolink mode.

Possible values: No / Mlink-I / Mlink-II.

Default value: No.

Associated variable: (V.)MPG.MLINK

This parameter may be used to set the Mechatrolink bus in either Mlink-I mode or Mlink-II mode. All the elements of the bus must be operating in the same mode (Mlink-I or Mlink-II); therefore, the selection will be conditioned by the mode available at the slaves connected to the bus.

Limitations of a system with a Mechatrolink bus.

- The CNC does not run any parameter matching process with Mechatrolink drives. The parameters of the drive must be set with the assistance of external tools provided by the drive manufacturer, through the interface of the device itself or writing the parameters from the DDSSetup and Tuning modes of the CNC.
- Second feedback cannot be used; therefore, the feedbacks cannot be combined (parameter FB MIXTIME) or swapped (parameter FB ACKSRC). No configuration similar to semi-Sercos is possible either.
- Tandem axes or spindles are not possible.
- Axes or spindles cannot be parked.
- Bode is not available.
- Set changing is not possible.
- The coordinate cannot be initialized.
- The dynamic changing of feed forward, AC forward and Kv will depend on the devices connected to the bus.

DATASIZE

Telegram size in Mlink-II mode.

Possible values: 17 / 32 bytes.

Default value: 32 bytes.

Associated variable: (V.)MPG.DATASIZE

This parameter sets the telegram size for the Mlink-II mode. All the elements of the bus must be operating with the same telegram size; therefore, the selection will be conditioned by the properties of the slaves connected to the bus.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.10 CAN bus configuration.

CANMODE

Type of CAN bus.

Possible values: CANfagor / CANopen.

Default value: CANopen.

Associated variable: (V.)MPG.CANMODE

CANfagor bus.

Using the CANfagor type bus requires the definition of the maximum length of the bus with parameter CANLENGTH.

CANopen bus.

When using the CANopen type bus, the work speed is defined in each node. All the nodes must work at the same speed. The work speeds at the bus depend on the total length of the bus.

Length (m)	20	40	100	500
Speed (KHz)	1000	800	500	250

CANLENGTH

CANfagor bus cable length.

Possible values: Up to 20, 30, 40, 50, 60, 70, 80, 90, 100, 110, 120, 130 and more than 130 meters.

Default value: Up to 20 meters.

Associated variable: (V.)MPG.CANLENGTH

The work speeds at the bus depend on the total length of the bus.

Length (m)												
20	30	40	50	60	70	80	90	100	110	120	130	>130
Speed (KHz)												
1000	888	800	727	666	615	571	533	500	480	430	400	250

CANOPENFREQ

CANopen bus communication frequency.

Possible values: Autoscan / 1 Mbps / 800 kbps / 500 kbps / 250 kbps.

Default value: Autoscan.

Associated variable: (V.)MPG.CANOPENFREQ

When using the CANopen protocol, the transmission speed at the bus is defined in each node and they all must run at the same speed. The transmission speed depends on the total length of the bus. Using the following illustrative values; assigning other values may cause communication errors due to signal distortion.

Speed	Length of the CAN bus.
Autoscan	The CNC adjusts the bus frequency at every startup depending on the speed of the rest of the modules. Depending on the configuration of the bus, this option can make the CNC startup slower than if there is a set frequency.
1000 kHz	Up to 20 meters.
800 kHz	From 20 to 40 meters.
500 kHz	From 40 to 100 meters.
250 kHz	From 100 to 500 meters.



The speed of 250 kHz is only available to communicate with the keyboards and RIOW and RIOR series remote modules; this speed is not available for the RIO5 series remote modules.

Peculiarities of the laser model.

For the laser model with RIOR modules, it is recommended to use a communications frequency of 1 MHz.

2.

MACHINE PARAMETERS.

General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.11 Serial line configuration.

RSTYPE

Type of serial line.

Possible values: RS232 / RS485 / RS422.

Default value: RS232.

Associated variable: (V.)MPG.RSTYPE

Type of serial line. Standard configuration for RS232 and full-duplex configuration for RS422. For the HBL5 (portable operator panel) select RS422.



In the RS485, the CNC uses the same signal to control "send data" and "receive data", therefore sending is disabled while receiving. In order to ensure proper communication, an 8-ms delay is required from when the CNC stops sending till it is ready to receive data. At the slaves connected to the CNC, this delay time must be set from when data is received till it is sent. An external RS232/RS485 adapter should be used if this delay cannot be set at the slave.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.12 MODBUS.

MODBUS Configure the ModBUS server.

This parameter shows the table to configure the ModBUS server.

MODBUS	
MODBUSSVRTCP	Activate the ModBUS sever on TCP.
MODBUSSVRRS	Activate the ModBUS sever on RS485.
MODSVRID	Identifier of the ModBUS sever on RS485.
MODBRATE	Communication speed of the ModBUS sever on RS485.

MODBUSSVRTCP Activate the ModBUS sever on TCP.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)MPG.MODBUSSVRTCP

Parameter included in the MODBUS table.

Activate the ModBUS sever on TCP.

MODBUSSVRRS Activate the ModBUS sever on RS485.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)MPG.MODBUSSVRRS

Parameter included in the MODBUS table.

Activate the ModBUS sever on RS485.

MODSVRID Identifier of the ModBUS sever on RS485.

Possible values: From 1 to 255.

Default value: 1.

Associated variable: (V.)MPG.MODSVRID

Parameter included in the MODBUS table.

Identifier of the ModBUS sever on RS485.

MODBRATE Communication speed of the ModBUS sever on RS485.

Possible values: 19200 Bd / 38400 Bd / 57600 Bd / 115200 Bd.

Default value: 19200 Bd.

Associated variable: (V.)MPG.MODBRATE

Parameter included in the MODBUS table.

Communication speed of the ModBUS sever on RS485.

2.

MACHINE PARAMETERS.
General machine parameters.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.13 Default conditions (system).

INCHES

Default work units (millimeters, inch).

Possible values: Millimeters / inches.

Default value: Millimeters.

Associated variable: (V.)MPG.INCHES

This parameter indicates the work units assumed by the CNC by default; i.e. on power-up, after executing an M02 or M30 or after a reset. To change the units from the part-program, use function G70 or G71.

PRESSURE

Default pressure units.

Possible values: Bar / Psi.

Default value: Bar.

Associated variable: (V.)MPG.PRESSURE

This parameter indicates the work units assumed by the CNC by default; i.e. on power-up, after executing an M02 or M30 or after a reset.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.14 Arithmetic parameters.

MAXLOCP

Maximum local arithmetic parameter.

Possible values: From 0 to 99.

Default value: 25.

Associated variable: (V.)MPG.MAXLOCP

See general machine parameter MINLOCP.

MINLOCP

Minimum local arithmetic parameter.

Possible values: From 0 to 99.

Default value: 0.

Associated variable: (V.)MPG.MINLOCP

Parameters MINLOCP and MAXLOCP define the group of local arithmetic parameters to be used. 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.

MAXGLBP

Maximum global arithmetic parameter.

Possible values: From 100 to 9999.

Default value: 299.

Associated variable: (V.)MPG.MAXGLBP

Check general machine parameter MINGLBP.

MINGLBP

Minimum global arithmetic parameter.

Possible values: From 100 to 9999.

Default value: 100.

Associated variable: (V.)MPG.MINGLBP

Parameters MAXGLBP and MINGLBP define the group of global arithmetic parameters to be used. Global parameters may be accessed from any program or subroutine called upon from the channel. There is a group of global parameters in each channel. The value of these parameters is shared by the program and the subroutines.

ROPARMAX

Maximum global read-only arithmetic parameter.

Possible values: From 100 to 9999.

Default value: 0.

Associated variable: (V.)MPG.ROPARMAX

See general machine parameter ROPARMIN.

ROPARMIN

Minimum global read-only arithmetic parameter.

Possible values: From 100 to 9999.

Default value: 0.

Associated variable: (V.)MPG.ROPARMIN

Parameters ROPARMAX and ROPARMIN may be used to protect a group of global arithmetic parameters so they cannot be modified. If both parameters are set to "0", there will be no protected parameters.

MAXCOMP

Maximum arithmetic parameter common to all the channels.

Possible values: From 10000 to 19999.

Default value: 10025.

Associated variable: (V.)MPG.MAXCOMP

Check general machine parameter MINCOMP.

2.

MACHINE PARAMETERS.

General machine parameters.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

MINCOMP**Minimum arithmetic parameter common to all the channels.***Possible values: From 10000 to 19999.**Default value: 10000.**Associated variable: (V.)MPG.MINCOMP*

Parameters MAXCOMP and MINCOMP the group of local arithmetic parameters common to all the channels to be used. The common parameters may be accessed from any channel. The value of these parameters is shared by all the channels.

BKUPCUP**Number of common non-volatile arithmetic parameters.***Possible values: From 0 to 20.**Default value: 0.**Associated variable: (V.)MPG.BKUPCUP*

This parameter indicates the number of common arithmetic parameters that are saved in RAM memory with a battery. The CNC saves the first parameters, from the first one to the one set in this parameter.

Non-volatile data means that their values must be maintained from one session to another and after turning the CNC off. The CNC saves this data when the CNC is turned off, when power is removed, when a hardware error occurs, etc.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

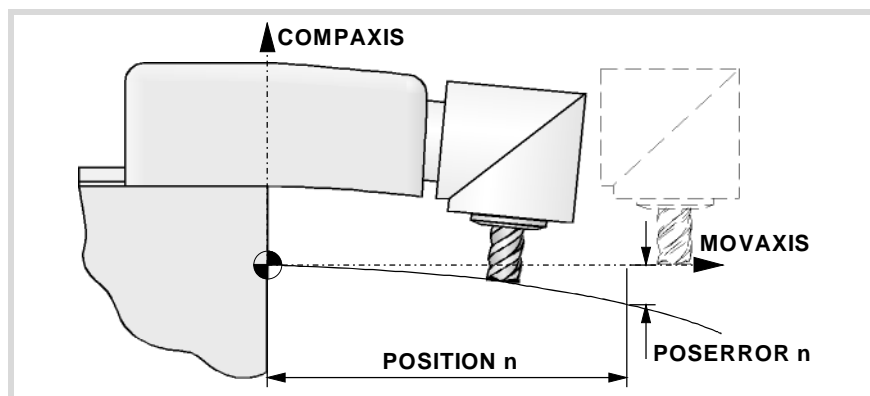
CNC 8070

(REF: 1709)

2.3.15 Cross compensation table.

CROSSCOMP Cross compensation tables.

This parameter shows the cross compensation table. Cross compensation is used when an axis suffers position variations due to the movement of another axis.



Define the tables in the order used when measuring; otherwise, the result will be different. The CNC calculates the compensation to be applied to each axis considering the order in which the tables are defined.

Although by default the CNC always activates the cross compensation tables defined in it, these tables may be disabled via PLC using the DISCROSS mark. The PLC has a mark for each table, the DISCROSS1 mark is for table 1, DISCROSS2 for table 2 and so on.

Import cross compensation tables.

In order to save time and eliminate transcription errors, instead of entering the data manually, it is possible to adapt the format of a text file that contains the result of the measurement and import it later on.



See the operating manual for further information on how to import cross compensation tables.

CROSSCOMP n Configuration of cross compensation tables.

The CNC shows the possible tables according to the number of axes defined. Each table has the following machine parameters to configure it.

CROSSCOMP	
MOVAXIS	Axis whose movement affects another axis (master).
COMPAXIS	Axis suffering the effects of the movement (compensated).
NPCROSS	Number of points in the table.
TYPCROSS	Type of compensation.
BIDIR	Bi-directional compensation.
REFNEED	Home search required for compensation.
DATA	Table to define the compensation values.

MOVAXIS Axis whose movement affects another axis (master).

Possible values: Any axes defined in AXISNAME.

Associated variable: (V.)MPG.MOVAXIS[tbl]

Parameter included in the CROSSCOMP table.

This parameter sets which is the master axis in the cross compensation.

COMPAXIS**Axis suffering the effects of the movement (compensated).**

Possible values: Any axes defined in AXISNAME.

Associated variable: (V.)MPG.COMPAXIS[tbl]

Parameter included in the CROSSCOMP table.

This parameter sets which is the compensated axis in the cross compensation.

NPCROSS**Number of points in the table.**

Possible values: From 0 to 1000.

Default value: 0 (there is no table).

Associated variable: (V.)MPG.NPCROSS[tbl]

Parameter included in the CROSSCOMP table.

Each cross compensation table can have up to 1000 points.

TYPCROSS**Type of compensation (type of coordinates).**

Possible values: Real / Theoretical.

Default value: Real.

Associated variable: (V.)MPG.TYPCROSS[tbl]

Parameter included in the CROSSCOMP table.

This parameter determines whether the cross compensation will be applied on to theoretical or real coordinates.

BIDIR**Bi-directional compensation.**

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)MPG.BIDIR[tbl]

Parameter included in the CROSSCOMP table.

This parameter indicates whether the compensation is bidirectional or not; i.e. if the compensation is different in each direction. If the compensation is not bidirectional, it applies the same compensation in both directions.

REFNEED**Home search required for compensation.**

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)MPG.REFNEED[tbl]

Parameter included in the CROSSCOMP table.

This parameter indicates whether both axes must be homed or not before applying the compensation.

DATA**Table to define the compensation values.**

This parameter shows the list of compensation points and values. The CNC only lets access this table when parameters are MOVAXIS, COMPAXIS and NPCROSS set.

The DATA table must indicate the amount of error to be compensated in specific positions of the moving axis. The number of points is set by parameter NPCROSS. Parameters POSITION, POSERROR and NEGERROR must be set for each point. Parameter NEGERROR only if the table has been defined with bidirectional compensation (BIDIR = YES).

CROSSCOMP DATA	
POSITION	Position of the master axis.
POSERROR	Amount of error to compensate when moving in the positive direction.
NEGERROR	Amount of error to compensate when moving in the negative direction.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

POSITION

Position of the master axis.

Possible values (1): Within ± 99999.9999 mm.

Possible values (2): Within ± 3937.00787 inch.

Possible values (3): Within ± 99999.9999 degrees.

Default value: 0.

Associated variable: (V.)MPG.POSITION[tbl][pt]

Parameter included in the CROSSCOMP // DATA table.

This parameter sets the position of the master axis in each point of the compensation. The points of the table must be ordered by their position on the axis and the table must begin by the most negative point (or least positive) to be compensated. For axis positioning outside this area, the CNC will apply the compensation that was defined for the nearest end.

POSERROR

Amount of error to compensate when moving in the positive direction.

Possible values (1): Within ± 99999.9999 mm.

Possible values (2): Within ± 3937.00787 inch.

Possible values (3): Within ± 99999.9999 degrees.

Default value: 0.

Associated variable: (V.)MPG.POSERROR[tbl][pt]

Parameter included in the CROSSCOMP // DATA table.

This parameter indicates the amount of error on the compensated axis when the master axis moves in the positive direction. If the table has not been defined with bi-directional compensation, this amount of error will also be applied on movements in the negative direction.

The machine reference point must have "0" error.

NEGERROR

Amount of error to compensate when moving in the negative direction.

Possible values (1): Within ± 99999.9999 mm.

Possible values (2): Within ± 3937.00787 inch.

Possible values (3): Within ± 99999.9999 degrees.

Default value: 0.

Associated variable: (V.)MPG.NEGERROR[tbl][pt]

Parameter included in the CROSSCOMP // DATA table.

This parameter indicates the amount of error on the compensated axis when the master axis moves in the negative direction. This compensation is only available if the table has been defined with bidirectional compensation.

The machine reference point must have "0" error.

2.

MACHINE PARAMETERS.
General machine parameters.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.16 Volumetric compensation tables.

VOLCOMP

Volumetric compensations tables.

This parameter shows the volumetric compensation table. Volumetric compensations are configured in the machine parameters and are activated from the PLC (marks VOLCOMP1 to VOLCOMP4).

VOLCOMP n

Configuration of volumetric compensation tables.

This parameter shows the parameter table to define the volumetric compensations of the system. Each table has the following machine parameters to configure it.

VOLCOMP	
VMOVAXIS1 VMOVAXIS2 VMOVAXIS3	In the basic volumetric compensation, the name of the axis that generates variations when moved. No function in the medium and large volumetric compensations.
NPOINTSAX1 NPOINTSAX2 NPOINTSAX3	Number of points of basic volumetric compensation on each axis. No function in the medium and large volumetric compensations.
INIPOSAX1 INIPOSAX2 INIPOSAX3	Initial position of the basic volumetric compensation on each axis. No function in the medium and large volumetric compensations.
INCREAX1 INCREAX2 INCREAX3	Interval between basic volumetric compensation points on each axis. No function in the medium and large volumetric compensations.
VCOMPAXIS1 VCOMPAXIS2 VCOMPAXIS3	Name of the first, second and third axis to be compensated.
VCOMPFILE	File containing volumetric compensation data.

VMOVAXIS1

VMOVAXIS2

VMOVAXIS3

Name of the axis that generates variations when moved.

Possible values: Any axes defined in AXISNAME.

Associated variable: (V.)MPG.VMOVAXIS1[tl]

Associated variable: (V.)MPG.VMOVAXIS2[tl]

Associated variable: (V.)MPG.VMOVAXIS3[tl]

Parameter included in the VOLCOMP table.

Axes that define the volume to be compensated. First, second and third axis that, when moved, generates variations in the axes to be compensated. If it is a gantry axis, it is only necessary to define the master axis; the CNC also applies the compensation to the slave axis.

These parameters are only available for basic volumetric compensation; they have no function in medium and large volumetric compensations. In the medium and large volumetric compensations, the order of the axes is implicit in the file.

2.

MACHINE PARAMETERS.
General machine parameters.

NPOINTSAX1
NPOINTSAX2
NPOINTSAX3

Number of points of the basic volumetric compensation for the MOVAXIS axis.

Possible values: From 2 to 25 points.

Default value: 10 points.

Associated variable: (V.)MPG.NPOINTSAX1[tl]

Associated variable: (V.)MPG.NPOINTSAX2[tl]

Associated variable: (V.)MPG.NPOINTSAX3[tl]

Parameter included in the VOLCOMP table.

Number of points of basic volumetric compensation on each axis. Each volumetric compensation axis can have 25 points.

These parameters are only available for basic volumetric compensation; they have no function in medium and large volumetric compensations.

INIPOSAX1
INIPOSAX2
INIPOSAX3

Initial position of the basic volumetric compensation for the MOVAXIS axis.

Possible values: Within ±99999.9999 mm / Within ±3937.00787 inch.

Default value: 0.

Associated variable: (V.)MPG.INIPOSAX1[tl]

Associated variable: (V.)MPG.INIPOSAX2[tl]

Associated variable: (V.)MPG.INIPOSAX3[tl]

Parameter included in the VOLCOMP table.

Initial position of the basic volumetric compensation on each axis.

These parameters are only available for basic volumetric compensation; they have no function in medium and large volumetric compensations.

INCREAX1
INCREAX2
INCREAX3

Interval between points for the MOVAXIS axis in basic volumetric compensation.

Possible values: Within ±99999.9999 mm / Within ±3937.00787 inch.

Default value: 10 mm / 0.3937 inch.

Associated variable: (V.)MPG.INCREAX1[tl]

Associated variable: (V.)MPG.INCREAX2[tl]

Associated variable: (V.)MPG.INCREAX3[tl]

Parameter included in the VOLCOMP table.

Interval between points on each axis.

These parameters are only available for basic volumetric compensation; they have no function in medium and large volumetric compensations.

VCOMPAXIS1
VCOMPAXIS2
VCOMPAXIS3

Name of the axis to be compensated.

Possible values: Any axes defined in AXISNAME.

Associated variable: (V.)MPG.VCOMPAXIS1[tl]

Associated variable: (V.)MPG.VCOMPAXIS2[tl]

Associated variable: (V.)MPG.VCOMPAXIS3[tl]

Parameter included in the VOLCOMP table.

These parameters set the axes to be compensated with volumetric compensation. The axes associated with the same compensation may belong to different channels and they may be interchanged from one channel to another while the compensation is active. An axis can be included in several different compensations, but compensations that share axes cannot be active at the same time.

2.

MACHINE PARAMETERS.
 General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

VCOMPFILE

File containing volumetric compensation data.

Associated variable: (V.)MPG.VCOMPFILE[tbl]

Parameter included in the VOLCOMP table.

File containing volumetric compensation data. The compensation tables are generated by the calibration application; they are not editable from the CNC. The units in the file data (millimeters or inches) must be those defined by the CNC (parameter INCHES).

2.

MACHINE PARAMETERS.

General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.17 Execution times.

MINAENDW

Minimum duration of the AUXEND signal.

Possible values: From 0 to 65535 ms.

Default value: 10 ms.

Associated variable: (V.)MPG.MINAENDW

This parameter has the following meanings.

- This parameter sets the time that the AUXEND signal must stay active for the CNC to consider it a valid signal. AUXEND is the synchronization signal that the PLC sends to the CNC to indicate that the –M–, –S–, –T– functions have been executed.
- For –M– functions (which do not need synchronization), this parameter indicates the duration of the MSTROBE signal.
- For –H– functions (which do not need synchronization), this parameter indicates the duration of the HSTROBE signal.

The value assigned to this parameter must be equal to or greater than the PLC's input frequency (LOOPTIME x PRGFREQ).

REFTIME

Estimated home searching time.

Possible values: From 0 to 1000000 ms.

Default value: 0. ms.

Associated variable: (V.)MPG.REFTIME

See general machine parameter TTIME.

HTIME

Estimated time for an H function.

Possible values: From 0 to 1000000 ms.

Default value: 0. ms.

Associated variable: (V.)MPG.HTIME

See general machine parameter TTIME.

DTIME

Estimated time for a D function.

Possible values: From 0 to 1000000 ms.

Default value: 0. ms.

Associated variable: (V.)MPG.DTIME

See general machine parameter TTIME.

TTIME

Estimated time for a T function.

Possible values: From 0 to 1000000 ms.

Default value: 0. ms.

Associated variable: (V.)MPG.TTIME

In Editing - Simulation mode, there is an option that allows calculating the time required to execute a part with the machining conditions established in the program. To fine tune that calculation, one may define these parameters that indicate the estimated time for processing particular functions.

The values are generic, for any H, D, T function or for homing one or several axes at a time. Spindle machine parameter SPDLTIME indicates the estimated time for executing an S function and the machine parameter of the M function table –M– MTIME indicates the time estimated to execute an M function.

2.

MACHINE PARAMETERS.
General machine parameters.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.18 Numbering of the digital inputs (CANfagor bus).

NDIMOD

Total number of digital input modules.

Possible values: From 0 to 64.

Default value: 0 (no numbering is defined).

Associated variable: (V.)MPG.NDIMOD

This parameter shows the number of digital input modules connected to the same bus CAN. After defining this parameter, it is possible to set the numbering of the digital inputs of each module by assigning a base index to the first of them. If this parameter is not set, the CNC numbers the digital inputs sequentially according to the order of the modules in the bus.

DIMODADDR

Table of digital input modules.

This parameter shows the list of digital input modules connected to the same bus CAN. When inserting a new module, the CNC will assign the numbering of the table to the first modules and the next valid base index after the highest one assigned until then to the last module.

DIMOD n

Base index of the digital input modules.

Possible values: From 0 to 1009.

Default value: The first valid value.

Associated variable: (V.)MPG.DIMODADDR[nb]

Parameter included in the DIMODADDR table.

This parameter sets the base index from which the digital inputs of the module are numbered. The values of the base index must comply with the formula " $16n + 1$ " (i.e. 1, 17, 33, etc.). If an invalid base index is entered, it assumes the nearest previous valid one. The base indexes may follow any order, they do not have to be sequential.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.19 Numbering of the digital outputs (CANfagor bus).

NDOMOD

Total number of digital output modules.

Possible values: From 0 to 64.

Default value: 0 (no numbering is defined).

Associated variable: (V.)MPG.NDOMOD

This parameter shows the number of digital outputs modules connected to the same bus CAN. After defining this parameter, it is possible to set the numbering of the digital outputs of each module by assigning a base index to the first of them. If this parameter is not set, the CNC numbers the digital outputs sequentially according to the order of the modules in the bus.

DOMODADDR

Table of digital output modules.

This parameter shows the list of digital output modules connected to the same bus CAN. When inserting a new module, the CNC will assign the numbering of the table to the first modules and the next valid base index after the highest one assigned until then to the last module.

DOMOD n

Base index of the digital output modules.

Possible values: From 0 to 1009.

Default value: The first valid value.

Associated variable: (V.)MPG.DOMODADDR[nb]

Parameter included in the DOMODADDR table.

This parameter sets the base index from which the digital outputs of the module are numbered. The values of the base index must be comply with the formula " $16n + 1$ " (i.e. 1, 17, 33, etc.). If an invalid base index is entered, it assumes the nearest previous valid one. The base indexes may follow any order, they do not have to be sequential.

2.

MACHINE PARAMETERS.

General machine parameters.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.20 Numbering of the digital inputs (CANopen bus).

NDIMOD

Total number of digital input logic blocks.

Possible values: From 0 to 42.

Default value: 0 (no numbering is defined).

Associated variable: (V.)MPG.NDIMOD

This parameter shows the number of logic blocks that the digital input modules connected to the CANopen bus are divided into. After defining this parameter, it is possible to set the numbering of the digital inputs of each logic block by assigning a base index to the first of them.

- In RIO5 and RIOR nodes, each module is a logic block, several modules cannot be grouped in a block or divide a module into blocks. The logic blocks must have the same number and size as the modules physically detected in the bus.
- In the RIO5 nodes, each double module (module with two boards) it counts as two, in other words, two logic blocks.
- In RIOW nodes and third-party nodes, it is possible to display as many logic blocks as desired, both in number and in size.

If this parameter is not set, the CNC numbers the digital inputs sequentially according to the order of the modules in the bus.

DIMODADDR

Base addresses of the digital input logic blocks.

This parameter shows the table to set the digital input logic blocks. The following fields must be defined for each block.

Column.	Meaning.
MNEMONIC	Mnemonic of the logic block.
NODE	Node number of the header to which the I/O logic block belongs.
BLOCK	Number of the logic block.
ADDRESS	Base address of the logic block.
NDI	Number of digital inputs of the module.

The configuration must meet the following requirements. If the parameters are not set according to these requirements, the CNC will ignore it and it will number the digital inputs sequentially according to the order of the modules in the bus.

- The parameters of all I/O nodes of the bus must be set; no node can be left undefined.
- The parameters of all the resources of a node must be set; no I/O can be left without a number.
- The maximum number of logic blocks in the system will be 64, both for digital inputs and digital outputs.
- The number of I/Os in a logic block must be other than zero and a multiple of 8 (8, 16, 24, 32, ...1024).
- The values of the base index must be comply with the formula " $8n + 1$ " (i.e. 1, 9, 17, 25, etc.).
- The blocks may be distributed at will within the range of resources managed by PLC (1...1024). The PLC admits blanks, but the blocks cannot overlap; in other words, a PLC digital input or output can only belong to a single node-module of the bus.

NODE

Node number of the header to which the I/O logic block belongs.

Possible values: From 1 to 126.

Default value: 0.

Associated variable: (V.)MPG.DIMODNODE[nb]

Node number of the header to which the I/O logic block belongs. The parameters of all I/O nodes of the bus must be set; no node can be left undefined.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

BLOCK

Number of the logic block.

Possible values: From 1 to 64.

Default value: 0.

Associated variable: (V.)MPG.DIMODBLOCK[nb]

Number of the logic block. The maximum number of logic blocks in the system will be 64, both for digital inputs and digital outputs.

ADDRESS

Base address of the logic block.

Possible values: From 1 to 1017.

Default value: The first valid value.

Associated variable: (V.)MPG.DIMODADDR[nb]

This parameter sets the base index from which the digital inputs of the logic block are numbered. The values of the base index must comply with the formula " $8n + 1$ " (i.e. 1, 9, 17, 25, etc.). If an invalid base index is entered, it assumes the nearest previous valid one. The base indexes may follow any order, they do not have to be sequential.

In the RIO5 nodes, each double module (module with two boards) it counts as two, in other words, two logic blocks. These modules must be assigned two base indexes for the inputs and two base indexes for the outputs; one for each board.

NDI

Number of digital inputs of the logic block.

Possible values: From 8 to 1024.

Default value: 0.

Associated variable: (V.)MPG.DIMODNDI[nb]

Number of digital inputs of the logic block; it must be other than zero and a multiple of 8 (8, 16, 24, 32, ...1024). The parameters of all the resources of a node must be set; no I/O can be left without a number.

2.

MACHINE PARAMETERS.
General machine parameters.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.21 Numbering of the digital outputs (CANopen bus).

NDOMOD

Total number of digital output logic blocks.

Possible values: From 0 to 42.

Default value: 0 (no numbering is defined).

Associated variable: (V.)MPG.NDOMOD

This parameter shows the number of logic blocks that the digital output modules connected to the CANopen bus are divided into. After defining this parameter, it is possible to set the numbering of the digital outputs of each logic block by assigning a base index to the first of them.

- In RIO5 and RIOR nodes, each module is a logic block, several modules cannot be grouped in a block or divide a module into blocks. The logic blocks must have the same number and size as the modules physically detected in the bus.
- In the RIO5 nodes, each double module (module with two boards) it counts as two, in other words, two logic blocks.
- In RIOW nodes and third-party nodes, it is possible to display as many logic blocks as desired, both in number and in size.

If this parameter is not set, the CNC numbers the digital outputs sequentially according to the order of the modules in the bus.

DOMODADDR

Base addresses of the digital output logic blocks.

This parameter shows the table to set the digital output logic blocks. The following fields must be defined for each block.

Column.	Meaning.
MNEMONIC	Mnemonic of the logic block.
NODE	Node number of the header to which the I/O logic block belongs.
BLOCK	Number of the logic block.
ADDRESS	Base address of the logic block.
NDO	Number of digital outputs of the module.

The configuration must meet the following requirements. If the parameters are not set according to these requirements, the CNC will ignore it and it will number the digital outputs sequentially according to the order of the modules in the bus.

- The parameters of all I/O nodes of the bus must be set; no node can be left undefined.
- The parameters of all the resources of a node must be set; no I/O can be left without a number.
- The maximum number of logic blocks in the system will be 64, both for digital inputs and digital outputs.
- The number of I/Os in a logic block must be other than zero and a multiple of 8 (8, 16, 24, 32, ...1024).
- The values of the base index must be comply with the formula " $8n + 1$ " (i.e. 1, 9, 17, 25, etc.).
- The blocks may be distributed at will within the range of resources managed by PLC (1...1024). The PLC admits blanks, but the blocks cannot overlap; in other words, a PLC digital input or output can only belong to a single node-module of the bus.

NODE

Node number of the header to which the I/O logic block belongs.

Possible values: From 1 to 125.

Default value: 0.

Associated variable: (V.)MPG.DOMODNODE[nb]

Node number of the header to which the I/O logic block belongs. The parameters of all I/O nodes of the bus must be set; no node can be left undefined.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

BLOCK

Number of the logic block.

Possible values: From 1 to 64.

Default value: 0.

Associated variable: (V.)MPG.DOMODBLOCK[nb]

Number of the logic block. The maximum number of logic blocks in the system will be 64, both for digital inputs and digital outputs.

ADDRESS

Base address of the logic block.

Possible values: From 1 to 1017.

Default value: The first valid value.

Associated variable: (V.)MPG.DOMODADDR[nb]

This parameter sets the base index from which the digital outputs of the logic block are numbered. The values of the base index must be comply with the formula " $8n + 1$ " (i.e. 1, 9, 17, 25, etc.). If an invalid base index is entered, it assumes the nearest previous valid one. The base indexes may follow any order, they do not have to be sequential.

In the RIO5 nodes, each double module (module with two boards) it counts as two, in other words, two logic blocks. These modules must be assigned two base indexes for the inputs and two base indexes for the outputs; one for each board.

NDO

Number of digital outputs of the logic block.

Possible values: From 8 to 1024.

Default value: 0.

Associated variable: (V.)MPG.DOMODNDO[nb]

Number of digital outputs of the logic block; it must be other than zero and a multiple of 8 (8, 16, 24, 32, ...1024). The parameters of all the resources of a node must be set; no I/O can be left without a number.

2.

MACHINE PARAMETERS.
General machine parameters.

2.3.22 Numbering of analog inputs for temperature sensors PT100.

NPT100

Total number of active PT100 inputs.

Possible values: From 0 to 10.

Default value: 0 (there are no active PT100 inputs).

Associated variable: (V.)MPG.NPT100

This parameter indicates the number of PT100 inputs active in the CAN bus. A PT100 input must be active if it has one of these temperature sensors connected to it. If the input is active in the machine parameter and there is no sensor connected, the CNC returns the corresponding error.

PT100

Table of active PT100 inputs.

This parameter shows the list of PT100 inputs active in the CAN bus.

PT100 n

Analog input associated with the PT100 input.

Possible values: From 0 to 32.

Default value: 0 (Do not activate the PT100 input).

Associated variable: (V.)MPG.PT100[nb]

This parameter indicates the number of the analog input associated with each active PT100 input. Each parameter can take any valid value; there is no need to follow a particular order.

When numbering the analog inputs, the CNC considers the PT100 inputs as analog inputs. This way, for numbering, the CNC considers the following:

- The RIO5 modules have 6 analog inputs; the four analog inputs (AI1..AI4) and the two PT100 inputs (AI5..AI6).
- The RIOR modules have 4 analog inputs; the two analog inputs (AI1..AI2) and the two PT100 inputs (AI3..AI4).

Example.

In a system with an RIO5 module (node 1) and an RIOR (node 2), the CNC identifies the analog inputs as follows.

- (address = 0) CNC
- (address = 1) RIO5
 - Analog inputs. - 1..4
 - PT100 inputs. - 5..6
- (address = 2) RIOR
 - Analog inputs. - 7..8
 - PT100 inputs. - 9..10

To have 3 PT100 inputs active (the two of the first module and the first one of the second module), the PT100 parameters may be set as follows.

```
NPT100 = 3
PT100 1 = 5
PT100 2 = 6
PT100 3 = 9
```

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.23 Probe setting.

PROBE

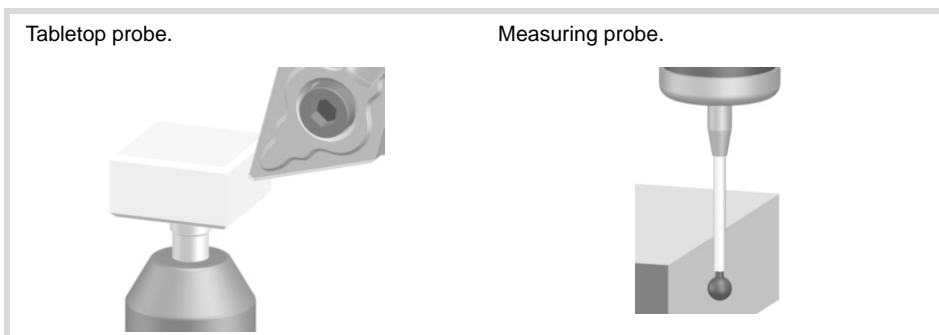
There is a probe.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)MPG.PROBE

This parameter indicates whether there is a probe on the machine or not. The CNC may have configured two probes, it will usually be a tabletop probe to calibrate tools and a touch probe to measure the part.



PROBEDATA

Probe parameters.

This parameter shows the table of parameters needed to set up the probe. If the CNC uses a tabletop probe, besides these parameters it is necessary to define the probe position.

PROBEDATA	
PROBETYPE1	Probe type 1.
PROBETYPE2	Probe type 2.
PRBDI1	Number of the input associated with probe 1.
PRBDI2	Number of the input associated with probe 2.
PRBPULSE1	Logic level to activate probe 1.
PRBPULSE2	Logic level to activate probe 2.

PROBETYPE1

Probe type 1.

Possible values: Remote / Local.

Default value: Remote.

Associated variable: (V.)MPG.PROBETYPE1

Parameter included in the PROBEDATA table.

This parameter indicates the type of probe depending on where it is connected; a remote probe if it is connected to a digital input at the remote modules or to a local probe if it is connected to a probe input at the central unit.

PROBETYPE2

Probe type 2.

Possible values: Remote / Local.

Default value: Remote.

Associated variable: (V.)MPG.PROBETYPE2

Parameter included in the PROBEDATA table.

This parameter indicates the type of probe depending on where it is connected; a remote probe if it is connected to a digital input at the remote modules or to a local probe if it is connected to a probe input at the central unit.

2.

PRBDI 1**Number of the input associated with probe 1.**

Possible values: From 1 to 1024 for the remote probe / 1 or 2 for the local probe.

Default value: 0 (no probe is connected).

Associated variable: (V.)MPG.PRBDI1

Parameter included in the PROBADATA table.

This parameter indicates the number of the input that the probe is connected to. For remote probes, this parameter indicates the digital input number; for local probes, it indicates the local probe input number.

The digital input will be a physical input of the module or a logic input of the PLC. The PLC considers logic inputs the inputs that do not exist physically; for example, if inputs 1 through 256 are numbered at the remote modules, the PLC considers logic inputs the ones from 257 to 1024.

Two remote probes cannot be connected to the same module if they are going to be involved in simultaneous probing operations; for example, a probe in each channel and both making a probing movement.

PRBDI 2**Number of the input associated with probe 2.**

Possible values: From 1 to 1024 for the remote probe / 1 or 2 for the local probe.

Default value: 0 (no probe is connected).

Associated variable: (V.)MPG.PRBDI2

Parameter included in the PROBADATA table.

This parameter indicates the number of the input that the probe is connected to. For remote probes, this parameter indicates the digital input number; for local probes, it indicates the local probe input number.

The digital input will be a physical input of the module or a logic input of the PLC. The PLC considers logic inputs the inputs that do not exist physically; for example, if inputs 1 through 256 are numbered at the remote modules, the PLC considers logic inputs the ones from 257 to 1024.

Two remote probes cannot be connected to the same module if they are going to be involved in simultaneous probing operations; for example, a probe in each channel and both making a probing movement.

PRBPULSE1**Logic level to activate probe 1.**

Possible values: Positive / Negative.

Default value: Positive.

Associated variable: (V.)MPG.PRBPULSE1

Parameter included in the PROBADATA table.

This parameter indicates whether the probe functions are active high (24 V or 5 V) or active low (0V) of the signal provided by the probe. In any case, the probe signal must be at least 20 ms long for the CNC to consider it valid.

PRBPULSE2**Logic level to activate probe 2.**

Possible values: Positive / Negative.

Default value: Positive.

Associated variable: (V.)MPG.PRBPULSE2

Parameter included in the PROBADATA table.

This parameter indicates whether the probe functions are active high (24 V or 5 V) or active low (0V) of the signal provided by the probe. In any case, the probe signal must be at least 20 ms long for the CNC to consider it valid.

2.3.24 Shared PLC memory.

PLCDATASIZE

Size of the PLC's shared data area.

Possible values: From 0 to 500.000 bytes.

Default value: 0.

Associated variable: (V.)MPG.PLCDATASIZE

This parameter may be used to define a memory area to exchange data between a PLC program written in C language and an external application.

2.3.25 Management of local I/O.

NLOCOUT

Number of local digital outputs.

Possible values: From 0 to 8.

Default value: 0.

Associated variable: (V.)MPG.NLOCOUT

The central unit has set of 8 local digital signals, identified as LI/O1 through LI/O8, that may be configured both as input and output. This parameter indicates how many of these signals, starting with LI/O1, are configured as digital outputs; the rest of the signals will act as digital inputs. The following table shows the logict numbers of pins LI/O1-LI/O8 according to parameter NLOCOUT.

Pin.	NLOCOUT								
	8	7	6	5	4	3	2	1	0
LI/O8	O8	O7	O6	O5	O4	O3	O2	O1	I16
LI/O7	O7	O6	O5	O4	O3	O2	O1	I15	I15
LI/O6	O6	O5	O4	O3	O2	O1	I14	I14	I14
LI/O5	O5	O4	O3	O2	O1	I13	I13	I13	I13
LI/O4	O4	O3	O2	O1	I12	I12	I12	I12	I12
LI/O3	O3	O2	O1	I11	I11	I11	I11	I11	I11
LI/O2	O2	O1	I10	I10	I10	I10	I10	I10	I10
LI/O1	O1	I9	I9	I9	I9	I9	I9	I9	I9

For the PWM and synchronized switching, only local outputs associated with pins LI/O1 and LI/O2 may be used because they have been set for Laser applications. That requires setting parameter NLOCOUT = 8.

EXPSCHK

Activate the 24 V monitoring at the local digital outputs.

Possible values: ON / OFF.

Default value: ON.

Associated variable: (V.)MPG.EXPSCHK

When a local digital output is active, the connector must be supplied with 24 V DC. This parameter enables the detection of these 24 V at the connector.

The 24 V monitoring must be active when a local output has been set; if no local digital output is active, the 24 V monitoring must be deactivated.

2.3.26 Synchronized switching.

SWTOUTPUT

Local digital output associated with synchronized switching.

Possible values: From 0 to NLOCOUT (Number of local digital outputs).

Default value: 0.

Associated variable: (V.)MPG.SWTOUTPUT

Local digital output associated with synchronized switching. For synchronized switching, only local outputs associated with pins LI/O1 (local output 1) and LI/O2 (local output 2) may be used because they have been set for Laser applications. That requires setting parameter NLOCOUT = 8.

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. 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.

In order for this feature to work better, the parameters of the axes involved in the movements must be set with the same values (gain, filters, acceleration and deceleration, feed forward, etc.) and must also work with the least amount of following error (lag) as possible.

SWTDELAY

Delay of the device associated with synchronized switching.

Possible values: From 0 to 100 ms.

Default value: 0 ms.

Associated variable: (V.)MPG.SWTDELAY

This parameter sets the delay between digital output on/off and the device that is connected to it.

SWTG0FEED

Observe the G00 feedrate during the synchronized switching.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)MPG.SWTG0FEED

The SWTG0FEED parameter determines the feedrate for the sections in G0.

SWTG0FEED	Meaning.
Yes	Movements in G0 are carried out at a rapid feedrate (parameter G00FEED).
No	The cut is made at a constant feedrate (with the feedrate of the previous G1/G2/G3), without accelerating in the sections in G0. This option makes it possible to prevent possible problems with machine frequency excitation, thus producing better quality precision and surface cuts.

The behavior can be modified from the part program with the instruction #SWTOUT. If according to the parameter or the command (#SWTOUT) the G0s are executed as G1 (constant feedrate), G0 is displayed in the history.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.27 PWM (Pulse-Width Modulation).

PWMOUTPUT

Local digital output associated with the PWM.

Possible values: From 0 to 2.

Default value: 0.

Associated variable: (V.)MPG.PWMOUTPUT

Local digital output associated with the PWM. For the PWM, only local outputs associated with pins LI/O1 (local output 1) and LI/O2 (local output 2) may be used because they have been set for Laser applications. That requires setting parameter NLOCOUT = 8.

PWMOUTPUT	Meaning.
0	PWM missing.
1	Local output 1 (pin LI/O1).
2	Local output 2 (pin LI/O2).

The output for the PWM and for synchronized switching may be the same one and they may be used at the same time (same value at parameters SWTOUTPUT and PWMOUTPUT). However, this configuration is not recommended for certain types of Laser generators.

PWMCANCEL

Cancel the PWM after an M30 or a reset.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)MPG.PWMCANCEL

This parameter indicates whether the CNC cancels the PWM after executing M02, M30 or after a reset.

When there is a CNC error in the channel that has the PWM active (or in any channel if it is managed via PLC), if the parameter is set to "Yes", the CNC deactivates the PWM. If the parameter is set to "No", the CNC keeps the status of the output; i.e. it does not cancel the output if it is active. Once the PWM is deactivated for some error, that PWM will not be reactivated when the error disappears; therefore, they have to be reactivated either by program or via PLC.

2.

MACHINE PARAMETERS.
General machine parameters.

2.3.28 Power control.

PWRCTRLACT

Manage power based on the actual feed.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)MPG.PWRCTRLACT

The CNC allows control of the power, laser or PWM duty, based on the feed. This parameter defines the feed to be used by the CNC to manage the power.

Value.	Meaning.
Yes.	The CNC manages the power based on the actual feed (variable (V.)A.FEED.xn).
No	The CNC manages the power based on the theoretical feed (variable (V.)A.TFEED.xn).

PWRCTRLCANCEL

Canceling power control with reset/M2/M30.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)MPG.PWRCTRLCANCEL

This parameter indicates whether the CNC cancels the power control after executing M02, M30 or after a reset.

Value.	Meaning.
Yes.	The CNC cancels the power control. A channel error cancels the power control. If an error disables the power control, it will not be re-enabled when the error disappears. It should be programmed again via the program or PLC.
No	The CNC does not cancel the power control. A channel error does not cancel the power control; this will maintain its previous status. At the beginning of the execution, after a reset and upon accessing the tool inspection, the CNC synchronizes the preparation of blocks with the execution.

2.

MACHINE PARAMETERS.
General machine parameters.

2.3.29 Gap control.

GAPCONTROL

Table of gap control parameters.

This parameter shows the table to define the gap control/

GAPCONTROL	
GAPANAINTYPE	Type of analog input connected to the sensor.
GAPANAINID	Number of the analog input connected to the sensor.
GAPDISTLIMIT	Distance corresponding to the limit analog signal of the sensor.
GAPVOLTLIMIT	Voltage corresponding to the limit path of the sensor .
GAPSENSOROFFSET	Offset applied to the sensor.
GAPSENSORCH	Change the sensor signal sign.
GAPGAIN	Proportional gain when applying in GAPCTRL.
GAPAPPROACHDYN	Dynamic response during the final part of the movement approaching the plate.
GAPSENSORFILTER	Table for configuring the sensor filter.
GAPERRORCANCEL	Canceling the gap error outside the range defined by GAPMIN/GAPMAX.
GAPMIN	Minimum sensor limit.
GAPMAX	Maximum sensor limit.
GAPTOLCANCEL	Canceling the gap error outside the range defined by GAPTOL.
GAPTOL	Error tolerance margin.
GAPCOLLISIONMODE	CNC behavior in the event of a sensor collision.
GAPTONEG	Movement block in #GAPCTRL towards descending coordinate.

GAPANAINTYPE

Type of analog input connected to the sensor.

Possible values: CAN remote / Drive.

Default value: Drive.

Associated variable: (V.)MPG.GAPANAINTIME

This parameter indicates the location of the analog output connected to the sensor; either in the CAN remote modules or in a Sercos drive. It is advisable to use a drive input.

GAPANAINID

Number of the analog input connected to the sensor.

Possible values: 1-60 for remote CAN / 101-132 or 201-232 for drive.

Default value: None (no gap control).

Associated variable: (V.)MPG.GAPANAINID

The sensor can be connected to an analog output of the CAN remote modules or a Sercos drive. In either case, this parameter indicates the analog output being used. It is advisable to use a drive input.

Analog input of the remote modules.

In this case, this parameter indicates the number of the analog output used for the command. The analog output modules are numbered following the logic order of the remote groups. If there are several analog-output modules in each group, the order is from left to right.

Managing the analog command from Sercos drives.

In this case, this parameter indicates the number of the analog output and the drive to be used. The parameter is set with a three-digit number; the first digit indicates the number of the analog output to be used (1 or 2) and the other two digits indicate the logic address of the drive (1 through 32). The "address" rotary selector switch sets the logic address.

2.

MACHINE PARAMETERS.

General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

For example, if the parameter is with a value of $\cdot 107\cdot$, it indicates that the CNC uses analog input $\cdot 1\cdot$ of the drive whose logic address is $\cdot 7\cdot$.

GAPDISTLIMIT

Distance corresponding to the limit analog signal of the sensor.

Possible values: From 0 to 99999.9999 mm / From 0 to 3937.00787 inches.

Default value: 10.0000 mm / 0.39370 inch.

Associated variable: (V.)MPG.GAPDISTLIMIT

Distance corresponding to the limit analog signal of the sensor (parameter GAPVOLTLIMIT).

GAPVOLTLIMIT

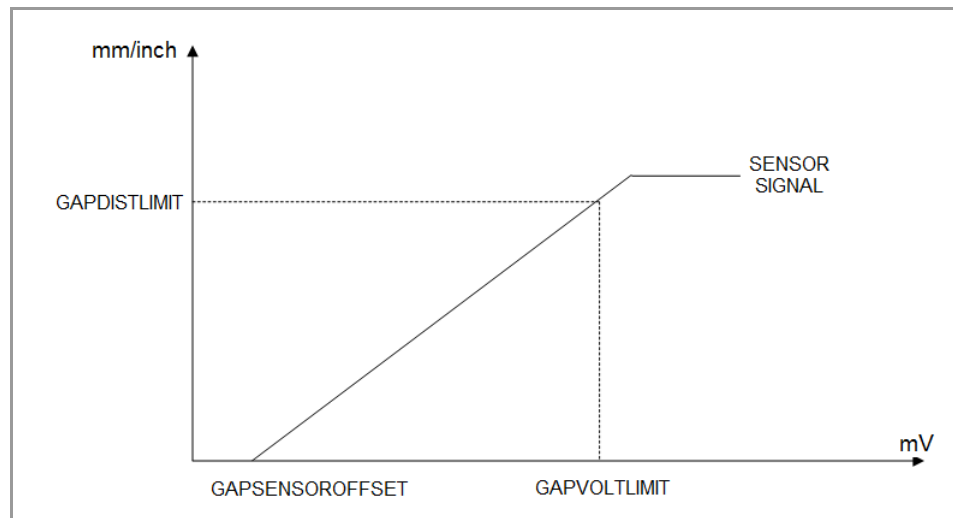
Voltage corresponding to the travel limit of the sensor.

Possible values: From 0 to 10000 mV

Default value: 10000 mV.

Associated variable: (V.)MPG.GAPVOLTLIMIT

Voltage (in millivolts) corresponding to the limit path of the sensor (parameter GAPDISTLIMIT).



GAPSENSOROFFSET

Offset to apply to the sensor.

Possible values: Between ± 10000 mV.

Default value: 0 mV.

Associated variable: (V.)MPG.GAPSENSOROFFSET

Offset (in millivolts) to apply to the sensor from the CNC. If a Fagor analog input is used, this parameter is not required, as each has its own offset.

GAPSENSORCH

Change the sensor signal sign.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)MPG.GAPSENSORCH

Change the sensor signal sign.

GAPGAIN

Proportional gain when applying in GAPCTRL.

Possible values: From 0.0000 to 100.0000.

Default value: 0.2.

Associated variable: (V.)MPG.GAPGAIN

Proportional gain applied to the signal sensor, in position. To use values greater than 0.2 (close to 1), it is advisable to use the parameter GAPSENSORFILTER.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

GAPAPPROACHDYN

Dynamic response during the final part of the movement approaching the plate.

Possible values: From 10 to 120 %.

Default value: 80 %.

Associated variable: (V.)MPG.GAPAPPROACHDYN

This parameter reduces, by a set percentage, the GAPGAIN value of the sensor axis during the approach to the sheet. The CNC applies this parameter in the approach associated with the gap control (instruction #GAPCTRL) as well as in that associated with leapfrog with gap control (instruction #LEAP). This parameter does not have any effect on the rest of the movements when the gap control is active.

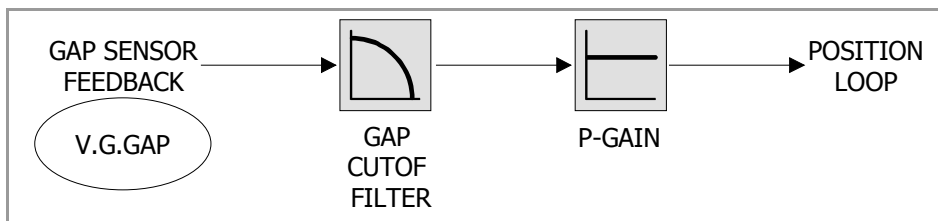
Values below 100% permit a smoother approach, eliminating any possible overshooting of the gap during these approaches. Increasing the value of this parameter can improve the lowering speed, but it can also cause a small overshooting of the position. Reducing this parameter too much can cause a small jolt at the end of the approach when the movement begins on the XY axis with gap control, as there is a sudden change in GAPGAIN.

GAPSENSORFILTER

Table for configuring the sensor filter.

This parameter shows the table to configure the sensor filter.

GAPSENSORFILTER	
ORDER	Filter order.
TYPE	Filter type.
FREQUENCY	Break frequency.



ORDER

Filter order.

Possible values: From 0 to 50.

Default value: 0 (the filter is not applied).

Associated variable: (V.)MPG.ORDER

Parameter included in the GAPSENSORFILTER table.

The higher the order number, the greater the slope of attenuation. An order 2 is recommended.

TYPE

Filter type.

Possible values: Low Passing.

Default value: Low Passing.

Associated variable: (V.)MPG.TYPE

Parameter included in the GAPSENSORFILTER table.

A low-pass filter allows the passage of lower frequencies and attenuates higher frequencies.

FREQUENCY

Break frequency.

Possible values: From 0 to 500.0000 Hz

Default value: 30.0000 Hz.

Associated variable: (V.)MPG.FREQUENCY

Parameter included in the GAPSENSORFILTER table.

Filter cutoff frequency. A value of 30 Hz is recommended.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

GAPERRORCANCEL**Canceling the gap error outside the range defined by GAPMIN/GAPMAX.***Possible values: Yes / No.**Default value: No.**Associated variable: (V.)MPG.GAPERRORCANCEL*

The GAPERRORCANCEL parameter establishes the CNC behavior when the gap exceeds the GAPMIN - GAPMAX range. Regardless of the value of this parameter, the CNC keeps monitoring the INPOSGAP mark.

Value.	Meaning.
Yes.	The CNC cancels the out-of-range gap error and does not stop the movement of the axes.
No	The CNC returns an out-of-range gap error and stops the movement of the axes, according to the braking ramp and controlling the gap during the ramp.

GAPMIN**Minimum sensor limit.***Possible values: From 0 to 99999.9999 mm / From 0 to 3937.00787 inches.**Default value: 0.**Associated variable: (V.)MPG.GAPMIN*

Minimum value permitted for the gap. If the gap exceeds the range defined in the parameters GAPMIN and GAPMAX, the PLC deactivates the INPOSGAP mark. The behavior of the CNC depends on the GAPTOLCANCEL parameter.

GAPMAX**Maximum sensor limit.***Possible values: From 0 to 99999.9999 mm / From 0 to 3937.00787 inches.**Default value: 5.0000 mm / 0.19685 inch.**Associated variable: (V.)MPG.GAPMAX*

Maximum value permitted for the gap. This parameter should be lower than GAPDISTLIMIT. If the gap exceeds the range defined in the parameters GAPMIN and GAPMAX, the PLC deactivates the INPOSGAP mark. The behavior of the CNC depends on the GAPTOLCANCEL parameter.

GAPTOLCANCEL**Canceling the gap error outside the range defined by GAPTOL.***Possible values: Yes / No.**Default value: No.**Associated variable: (V.)MPG.GAPTOLCANCEL*

The GAPTOLCANCEL parameter establishes the behavior of the CNC when the gap exceeds the margin set in the GAPTOL parameter.

Value.	Meaning.
Yes.	The CNC cancels the out-of-tolerance gap error and does not stop the movement of the axes.
No	The CNC returns an out-of-tolerance gap error and stops the movement of the axes, according to the braking ramp and controlling the gap during the ramp.

GAPTOL**Error tolerance margin.***Possible values: From 0 to 99999.9999 mm / From 0 to 3937.00787 inches.**Default value: 0.1000 mm / 0.00394 inch.**Associated variable: (V.)MPG.GAPTOL*

This parameter defines a positioning window on the programmed gap, to finish the approach block. If the axis exceeds the tolerance set in the GAPTOL parameter, the PLC deactivates the INTOL mark. The behavior of the CNC depends on the GAPTOLCANCEL parameter.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

GAPCOLLISIONMODE

CNC behavior in the event of a sensor collision.

Possible values: Do not display error / Display error.

Default value: Issue an error.

Associated variable: (V.)MPG.GAPCOLLISIONMODE

The gap control sensor may have a collision signal connected to the CNC, which should be managed by the PLC via the GAPCOLLISION mark. If the PLC activates the GAPCOLLISION mark, the CNC will display an error or not, depending on how this parameter has been set. The error does not activate the emergency.

GAPTONEG

Movement block in #GAPCTRL to the descending coordinate.

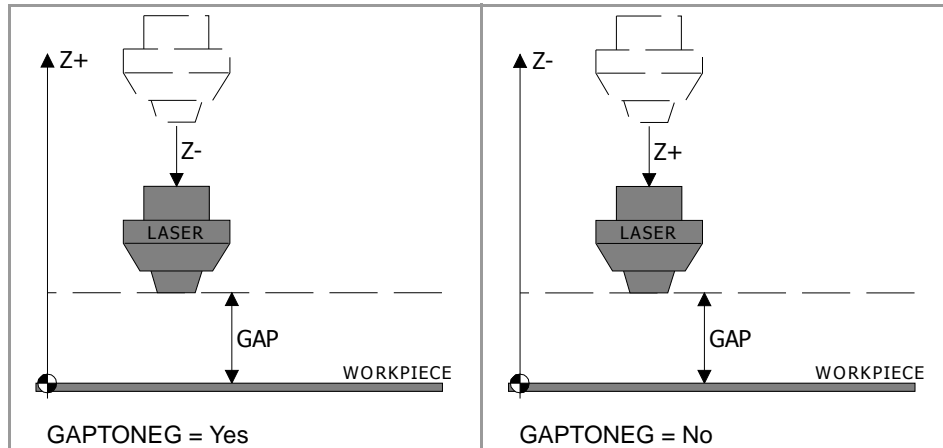
Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)MPG.GAPTONEG

This parameter indicates whether the movement down from the Z axis to reach the gap is in a negative direction (normal conditions) or a positive direction (inverted Z axis).

Value.	Meaning.
Yes.	The positive direction of the Z axis is up and the negative is down (under normal conditions). The axis moves in the negative direction (down) to reach the gap.
No	The positive direction of the Z axis is down and the negative direction is up (inverted Z axis). The axis moves in a positive direction (down) to reach the gap.



2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.30 Leapfrog.

LEAPDYNOVR

Dynamic response of the leap during the leapfrog.

Possible values: From 10 to 100 %.

Default value: 100 %.

Associated variable: (V.)MPGLEAPDYNOVR

This parameter enables the dynamic response of the Z axis (the longitudinal axis) to be reduced during the leapfrog for a smoother leap, in both the up and down flanks. Making the movement smoother will result in a longer machining time when there are leapfrogs.

Height of the leapfrog.

The height level that is reached in a leapfrog (#LEAP) depends on the ACCJERK, DECJERK, ACEL and DECEL values. The leapfrog requires high ACEL and DECEL values (of around 15,000 mm/s² or greater), as well as ACCJERK and DECJERK values (of around 20 times the ACEL and DECEL).

Parameter.	Meaning.
SLOPETYPE	Type of acceleration by default; linear, trapezoidal and square sine.
ACCEL DECEL	Acceleration and deceleration. Parameters ACCEL and DECEL set the acceleration values when it is trapezoidal or square-sine (bell shaped) (parameter SLOPETYPE).
ACCJERK DECJERK	Acceleration and deceleration jerk. Parameters ACCJERK and DECJERK set the slope of the acceleration and deceleration. Both parameters help limit the acceleration changes so the machine runs more smoothly in small speed increments or decrements and with FFGAIN values close to 100%. The lower the value assigned to these parameters, the smoother the machine response, but the acc/dec time will increase.

Plate approach.

The CNC executes the approach to the plate with linear acceleration LACC1 and LACC2, and thus this must not be very high (around a third of the ACEL and DECEL).

Parameter.	Meaning.
LACC1 LACC2	Acceleration of the first and second section. Parameters LACC1 and LACC2 set the acceleration values when it is linear (SLOPETYPE parameter).

Example.

For an ACEL and DECEL = 18,000 mm/s².

- LACC1 and LACC2 = 6000 mm/s².
- ACCJERK and DECJERK = 360,000 mm/s³.

2.

MACHINE PARAMETERS.
 General machine parameters.



FAGOR AUTOMATION

CNC 8070

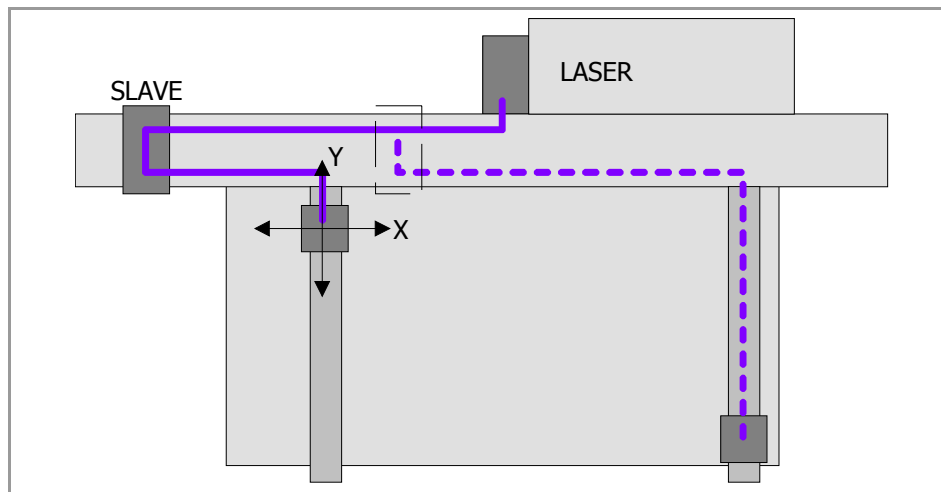
(REF: 1709)

2.3.31 Compensating the dispersion by the CO2 laser path.

LASERFOLLOW

Compensate the dispersion by the CO2 laser path.

This parameter shows the table to define the compensation. The goal of the compensation is to keep the total laser path constant in order to prevent dispersion variations in the cutoff point on the sheet.



For the compensation, a slave axis, which follows the main axes, is used. The dispersion compensation is activated from the program using the instruction #FOLLOW or from the PLC using the instruction FOLLOW.

LASERFOLLOW	
LASERFOLLOWAXIS	Slave axis applied to which FOLLOW is applied.
LASERFOLLOWOFFSET	Offset of the X Y Z axes with respect to the slave axis.
LASERFOLLOW1	Indicates how the first of the X Y Z axes affect the slave axis.
LASERFOLLOW2	Indicates how the second of the X Y Z axes affect the slave axis.
LASERFOLLOW3	Indicates how the third of the X Y Z axes affect the slave axis.

LASERFOLLOWAXEIS

Indicates the slave axis to which FOLLOW is applied.

Possible values: X, X1..X9, .., C, C1..C9.

Default value: 0 (no compensation).

Associated variable: (V.)MPG.LASERFOLLOW

Parameter included in the LASERFOLLOW table.

Name of the slave axis for compensation (axis on which the instruction #FOLLOW or the PLC command FOLLOW acts). The variable V.A.FLWMASTER.xn of this axis will be the main element of the synchronization.

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. For example X, X1, Y3, Z9, W, W7, C...

LASERFOLLOWOFFSET

Source offset of the X Y Z axes regarding the slave axis.

Possible values: Within ±99999.9999 mm / Within ±3937.00787 inch.

Default value: 0.

Associated variable: (V.)MPG.LASERFOLLOWOFFSET

Parameter included in the LASERFOLLOW table.

This offset allows the minimization of the total laser path.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

LASERFOLLOW1**Effect of the first of the X Y Z axes on the slave axis.***Possible values: No effect / Positive / Negative.**Default value: Not affected.**Associated variable: (V.)MPG.LASERFOLLOW1**Parameter included in the LASERFOLLOW table.*

This parameter indicates if the first axis (according to the machine parameters) of the channel that manages the compensation affects the slave axis, as well as its direction.

LASERFOLLOW2**Effect of the second of the X Y Z axes on the slave axis.***Possible values: No effect / Positive / Negative.**Default value: Not affected.**Associated variable: (V.)MPG.LASERFOLLOW2**Parameter included in the LASERFOLLOW table.*

This parameter indicates if the second axis (according to the machine parameters) of the channel that manages the compensation affects the slave axis, as well as its direction.

LASERFOLLOW3**Effect of the second of the X Y Z axes on the slave axis.***Possible values: No effect / Positive / Negative.**Default value: Not affected.**Associated variable: (V.)MPG.LASERFOLLOW3**Parameter included in the LASERFOLLOW table.*

This parameter indicates if the third axis (according to the machine parameters) of the channel that manages the compensation affects the slave axis, as well as its direction.

- If any of the parameters LASERFOLLOW1, LASERFOLLOW2 or LASERFOLLOW3 is different from 0, the CNC will automatically update the variable V.A.FLWMASTER.xn of the slave axis (LASERFOLLOWAXIS parameter) as follows.

$$\begin{aligned} V.A.FLWMASTER.xn = & V.MPG.LASERFOLLOWOFFSET + \\ & V.MPG.LASERFOLLOW1 * V.A.TPOS.X + \\ & V.MPG.LASERFOLLOW2 * V.A.TPOS.Y + \\ & V.MPG.LASERFOLLOW3 * V.A.TPOS.Z \end{aligned}$$

- If parameters LASERFOLLOW1, LASERFOLLOW2 and LASERFOLLOW3 are 0, the OEM should enter the value of the variable V.A.FLWMASTER.xn (e.g. from the PLC).

2.3.32 Backup of non-volatile data.

BKUPREG

Number of non-volatile PLC registers.

Possible values: From 0 to 20.

Default value: 0.

Associated variable: (V.)MPG.BKUPREG

This parameter indicates the number of PLC registers that are saved in RAM memory with a battery. The CNC saves the first registers, from the first one to the one set in this parameter.

Non-volatile data means that their values must be maintained from one session to another and after turning the CNC off. The CNC saves this data when the CNC is turned off, when power is removed, when a hardware error occurs, etc.

BKUPCOUN

Number of non-volatile PLC counters.

Possible values: From 0 to 20.

Default value: 0.

Associated variable: (V.)MPG.BKUPCOUN

This parameter indicates the number of PLC counters that are saved in RAM memory with a battery. The CNC saves the first counters, from the first one to the one set in this parameter.

Non-volatile data means that their values must be maintained from one session to another and after turning the CNC off. The CNC saves this data when the CNC is turned off, when power is removed, when a hardware error occurs, etc.

2.3.33 Tool offset and wear.

TOOLOFSG

Sign criteria for tool offsets and tool wear.

Possible values: Positive / Negative.

Default value: Negative.

Associated variable: (V.)MPG.TOOLOFSG

This parameter sets the sign criteria for the offsets and their 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.

TOOLOFSG	TOOLOFSG	Meaning.
Negative.		Tool calibration returns a negative offset. The offset wear must be entered with a positive value.
Positive.		Tool calibration returns a positive offset. The offset wear must be entered with a negative value.

In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.34 Spindle synchronization.

SYNCCANCEL

Cancels the spindle synchronization.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)MPG.SYNCCANCEL

This parameter indicates whether the CNC cancels spindle synchronization or not after executing M02, M30 or after an error or reset.

2.3.35 Define the number of jog panels and their relationship with the channels.

NKEYBD

Number of jog panels.

Possible values: From 1 to 8.

Default value: 1.

Associated variable: (V.)MPG.NKEYBD

Number of jog panels.

KEYBDCH

List of jog panels of the system.

This parameter shows the table to assign the jog panels to the channels.

KEYBDCH	
KEYBDnCH	Channel to which the jog panel is assigned.

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.

KEYBDnCH

Channel to which the jog panel is assigned.

Possible values: Active channel / CH1 / CH2 / CH3 / CH4.

Default value: Active channel.

Associated variable: (V.)MPG.KEYBDCH[jog]

Parameter included in the KEYBDCH table.

Each jog panel may be assigned to a particular channel or always to the active channel. When a jog panel is assigned to a channel, it is always operative even if the channel is not the active one. When there are several jog panels assigned to the same channel, operations may be carried out from any of them indistinctively.

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.36 PLC type.

PLCTYPE
PLC type.

Possible values: IEC / IEC+Fagor / Fagor.

Default value: IEC.

Associated variable: (V.)MPG.PLCTYPE

This parameter selects the type of PLC to be used; the Fagor one, the standard IEC-61131 or both. The following table shows who is affected by the startup (RUN) and stop (STOP) of the PLC depending on this parameter as well as the status of the mark PLCREADY.

PLCTYPE	RUN		STOP		PLCREADY	
	IEC	Fagor	IEC	Fagor	IEC	Fagor
IEC	Yes	No	Yes	No	ON	- - -
IEC+Fagor	Yes	Yes	Yes	Yes	ON	ON
Fagor	No	Yes	No	Yes	OFF	ON

2.3.37 Rename the axes and the spindles.

RENAMECANCEL

Cancel the name change for the axes and the spindles.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)MPG.RENAMECANCEL

The #RENAME instruction may be used to rename the axes and spindles via part-program or MDI/MDA mode. This parameter indicates whether the CNC keeps or cancel the names of the axes and spindles (#RENAME) 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.

2.

MACHINE PARAMETERS.
 General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.3.38 Zero offsets.

FINORG

Fine definition of zero offsets.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)MPG.FINEORG

This parameter enables the possibility to set each zero offset in the zero offset table with a coarse (or absolute) value and a fine (incremental) one. When executing function G159, the CNC assumes as new zero offset the sum of both parts.

- Absolute zero offset table (without fine setting of the zero offset).

Channel 1 : Zero offsets			
Origin	X (mm)	Y (mm)	Z (mm)
PLCOF	00000.0000	00000.0000	00000.0000
G158	00054.5000	00010.0000	00000.0000
G54 (G159=1)	00000.0000	00000.0000	00000.0000
G55 (G159=2)	00000.0000	00000.0000	00000.0000
G56 (G159=3)	00000.0000	00000.0000	00000.0000

- Absolute zero offset table (with fine setting of the zero offset).

Channel 1 : Zero offsets			
Origin	X (mm)	Y (mm)	Z (mm)
PLCOF	00000.0000	00000.0000	00000.0000
G158	00054.5000	00010.0000	00000.0000
G54 (G159=1)	00050.0000	00000.0000	00000.0000
△	00003.0000	00000.0000	00000.0000
G55 (G159=2)	00000.0000	00000.0000	00000.0000
△	00000.0000	00000.0000	00000.0000
G56 (G159=3)	00010.0000	00000.0000	00000.0000
△	00000.0000	00000.0000	00000.0000

2.3.39 Remote module RCS-S (Sercos Counter).

NSERCOUNT

Number of Sercos counter nodes in the bus.

Possible values: From 0 to 8.

Default value: 0.

Number of RCS-S modules in the bus.

SERCOUNTID n

Table of thumbwheel identifiers associated to the Sercos counters.

Possible values: From 1 to 15.

These parameters may be used to assign a logic number to the RCS-S modules of the bus based on their physical address, set by its rotary switch (node number). Define in each parameter the address (node number) of the RCS-S module.

Although the Sercos ring may have up to 32 nodes, the RCS-S modules always occupy positions from 1 to 15.



If drives and RCS-S modules are going to coexist on the Sercos bus, in order to achieve better system performance, it is recommended to assign the drives the lower node numbers.

(REF: 1709)

2.

MACHINE PARAMETERS.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

2.3.40 Access to the parameter table of the channel.

CHANNEL n
Parameter table of channel.

This parameter shows the machine parameter table of the channel.

2.

MACHINE PARAMETERS.
General machine parameters.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4 General machine parameters. Execution channels.

2.4.1 Channel configuration.

GROUPID

Group the channel belongs to.

Possible values: From 0 to 2.

Default value: 0 (it does not belong to any group).

Associated variable: (V.)[ch].MPG.GROUPID

Two or more channels may be configured to form a group. The channels of the same group behave as follows.

- Each channel may be in a different work mode, except in jog and automatic modes. Toggling between the jog mode and automatic mode of a channel will affect all the channels of the group that are in any of these modes; the channels that are in a different mode will not be affected.
- A reset in any of the channels of the group affects all of them.
- Any error in any of the channels of the group interrupts the execution in all of them.

CHTYPE

Type of channel.

Possible values: CNC / PLC / CNC+PLC.

Default value: CNC.

Associated variable: (V.)[ch].MPG.CHTYPE

A channel may be governed from the CNC, from the PLC or from both.

Channels governed by the PLC have neither jog mode nor MDI/MDA mode. The automatic and EDISIMU modes are available, but programs cannot be executed or simulated.

If during setup, it is necessary to display these work modes or execute or simulate programs, set the channel as being governed from the CNC+PLC and once setup is completed, set it back as a PLC channel.

HIDDENCH

Hidden channel.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPG.HIDDENCH

Hidden channels are not displayed and cannot be selected.

A hidden channel cannot be reset from the operator panel; to reset it, either group it with another channel or reset it from the PLC mark RESETIN.

2.

2.4.2 Configuring the axes of the channel.

CHNAXIS

Number of axes of the channel.

Possible values: From 0 to 28.

Default value: 3.

Associated variable: (V.)[ch].MPG.CHNAXIS

This parameter sets the number of axes of the channel, whether they are servo-controlled or not.

A channel may have initially associated with it one, several or no of the axes defined in the system. In any case, the number of axes assigned to the channel cannot be higher than the number of axes of the system, defined by parameter NAXIS. The sum of the axes assigned to the channels cannot exceed the number of axes of the system either.

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.

CHAXISNAME

List of the axes of the channel.

This parameter shows the table to define the names of the axes. Parameter CHNAXIS sets the number of axes of the channel.

CHAXISNAME	
CHAXISNAME n	Name of the axes.

CHAXISNAME n

Name of the axes.

Possible values: Any axes defined in AXISNAME.

Default value: Starting from CHAXISNAME1; X, Y, Z...

Associated variable: (V.)[ch].MPG.CHAXISNAME n

Parameter included in the CHAXISNAME table.

Any axis defined by parameter AXISNAME may belong to the channel.

When defining the axes, bear in mind that the order in which they are defined determines their index in the channel. The index of the first axis of the table will be ·1· and so on. As with the axis name, the index in the channel permits identifying the axis in PLC variables, marks, etc.

CHAXISNAME n	Index in the channel.
CHAXISNAME 1	Index ·1·.
CHAXISNAME 2	Index ·2·.
CHAXISNAME 3	Index ·3·.

The order of the axes and work planes (mill model).

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 axes of the channel.

Plane.	Abscissa axis.	Ordinate axis.	Longitudinal axis.
G17	CHAXISNAME 1	CHAXISNAME 2	CHAXISNAME 3
G18	CHAXISNAME 3	CHAXISNAME 1	CHAXISNAME 2
G19	CHAXISNAME 2	CHAXISNAME 3	CHAXISNAME 1

The order of the axes and work planes (lathe model).

The order of the axes of the channel and parameter GEOCONFIG set the main work planes. Refer to the parameter for further detail.

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

GEOCONFIG

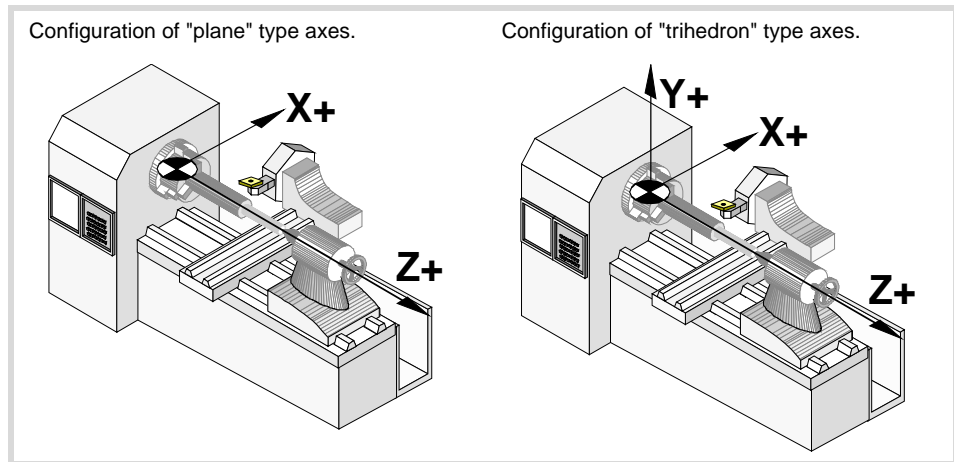
Geometrical configuration of the axes of the channel.

Possible values: Plane / Trihedron.

Default value: Trihedron.

Associated variable: (V.)[ch].MPG.GEOCONFIG

Not being used at the mill model. On the lathe model, this parameter indicates the axis configuration of the machine, trihedron or plane.



Configuration of "Trihedron" type axes.

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

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 in parameter IPLANE). 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 axes of the channel.

Plane.	Abscissa axis.	Ordinate axis.	Longitudinal axis.
G17	CHAXISNAME 1	CHAXISNAME 2	CHAXISNAME 3
G18	CHAXISNAME 3	CHAXISNAME 1	CHAXISNAME 2
G19	CHAXISNAME 2	CHAXISNAME 3	CHAXISNAME 1

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

Configuration of "plane" type axes.

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; they 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. If the X (firs axis of the channel) and Z (second axis of the channel) have been defined, the work plane will be the ZX (Z as abscissa and X as ordinate).

The work plane is always G18; machine parameter IPLANE is not applied and it is not possible to change planes via part-program. 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.

2.

MACHINE PARAMETERS. Execution channels. General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Configuration of "plane" type axes. Arc programming.

Programming the arc center I K depends on the active work plane.

- With function G18, in circular interpolations, the arc center I is associated with the first axis of the channel (usually X) and K with the second axis of the channel (usually Z).
- With function G20, in circular interpolations, the arc center I is associated with the abscissa axis (usually Z) and K with the ordinate axis of the channel (usually X).

Configuration of "plane" type axes. The longitudinal axis.

In this configuration, the second axis of the channel is considered as longitudinal axis. 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. 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.

When using milling tools on a lathe, the longitudinal compensation axis may be changed by means of the #TOOLAX instruction or the G20 function.

Configuration of "plane" type axes. Axis swapping.

The axes may be swapped, but it must be borne in mind that the previous behavior stays the same for the first and second axes of the channel resulting from the swap.

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4.3 Configuring the spindles of the channel.

CHNSPDL

Number of spindles of the channel.

Possible values: From 0 to 4.

Default value: 1.

Associated variable: (V.)[ch].MPG.CHNSPDL

This parameter sets the number of spindles of the channel, whether they are servo-controlled or not.

A channel may have initially one, several or no spindles associated with it. In any case, the number of spindles assigned to the channel cannot be higher than the number of spindles of the system, defined by parameter NSPDL. The sum of the spindles assigned to the channels cannot exceed the number of spindles of the system either.

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.

CHSPDLNAME

List of the spindles of the channel.

This parameter shows the table to define the names of the spindles. Parameter CHNSPDL sets the number of spindles of the channel.

CHSPDLNAME	
CHSPDLNAME n	Name of the spindles.

CHSPDLNAME n

Name of the spindles.

Possible values: Any spindle defined in SPDLNAME.

Default value: Starting from CHSPDLNAME1; S, S1...

Associated variable: (V.)[ch].MPG.SPDLNAME n

Parameter included in the CHSPDLNAME table.

Any spindle defined by parameter SPDLNAME may belong to the channel.

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.

When defining the spindles, bear in mind that the order in which they are defined determines their index in the channel. The index of the first spindle of the table will be ·1· and so on. As with the spindle name, the index in the channel permits identifying the spindle in PLC variables, marks, etc.

CHSPDLNAME	Index in the channel.
CHSPDLNAME 1	Index ·1·.
CHSPDLNAME 2	Index ·2·.
CHSPDLNAME 3	Index ·3·.

2.

MACHINE PARAMETERS. Execution channels.
General machine parameters. Execution channels.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4.4 Configuration of the C axis.

CAXNAME

Default name of the C axis.

Possible values: Any valid axis name; X, X1..X9, .. , C, C1..C9.

Default value: C.

Associated variable: (V.)[ch].MPG.CAXNAME

This parameter must be defined whenever there is an axis or spindle set as C axis.

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.

ALIGNC

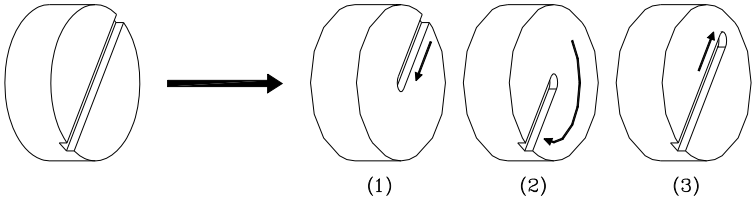
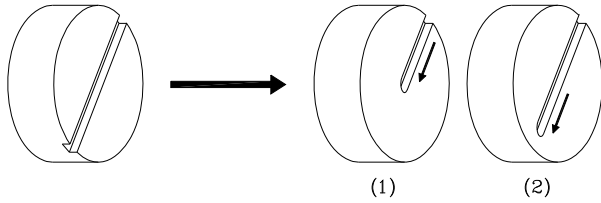
"C" axis alignment for diametrical machining.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)[ch].MPG.ALIGNC

This parameter indicates whether the tool can machine the whole surface diametrically in a single run (ALIGNC = No) or the "C" axis must be aligned (ALIGNC = Yes).

ALIGNC	Meaning.
Yes.	 <p>(1) Machining up to the center. (2) Rotate the "C" axis 180°. (3) Resume machining by moving the tool back.</p>
No.	 <p>(1) Machining up to the center. (2) Resume machining by moving the tool back.</p>

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4.5 Time setting (channel).

PREPFREQ

Maximum number of blocks to prepare per cycle.

Possible values: From 1 to 8.

Default value: 1.

Associated variable: (V.)[ch].MPG.PREPFREQ

While executing a program, the CNC reads in advance the blocks to be executed in order to calculate the path to follow; this is known as block preparations. The number of blocks to be prepared depends on the type of machining operation; it is not the same to process blocks generated with a CAD-CAM and the blocks coming from a program edited manually or with canned cycles.

This parameter sets the maximum number of blocks the CNC must process in each cycle (parameter LOOPTIME). The CNC will try processing the number of blocks set in this parameter and, if not possible, it will only process the number of blocks that it can.

Besides preparing the blocks, the CNC carries out several tasks in each cycle. Increasing the value of parameter PREPFREQ unnecessarily can jeopardize the rest of the task. Before modifying the value of this parameter, check with the service department.

ANTIME

Anticipation time.

Possible values: From 0 to 10000000 ms.

Default value: 0.

Associated variable: (V.)[ch].MPG.ANTIME

This parameter is used on punch presses that have an eccentric cam as a punching system. This parameter indicates how long before the axes reach position, the anticipation logic signal ADVINPOS of the channel is activated .

This signal may be used to start the movement of the punch before the axes reach the position. This reduces idle time, thus increasing the number of punches per minute.

If the total duration of the movement is lower than the value in the parameter, the anticipation signal ADVINPOS will be activated immediately.

If set to zero, the anticipation signal ADVINPOS is always active.

2.

MACHINE PARAMETERS. Execution channels.
General machine parameters. Execution channels.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4.6 Configuration of the HSC mode (channel).

HSC

Parameter table of the HSC mode.

This parameter shows the table to define the HSC work mode.

HSC	
HSCDEFAULTMODE	Modo por defecto al programar #HSC ON.
FEEDAVRG	Calculating feedrate average.
MINCORFEED	Minimum feedrate at the corners.
CORNER	Maximum angle of the corner to machine it in square corner mode.
SMOOTHFREQ	Smoothing frequency in the interpolation.
HSCFILTFREQ	Filter frequency (CONTERROR mode).
FASTFACTOR	Default feedrate percentage (FAST mode).
FTIMELIM	Time difference permitted in feedrate interpolation (FAST mode).
FSMOOTHFREQ	Smoothing frequency in the interpolation (FAST mode).
FASTFILTFREQ	Filter frequency (FAST mode).
SURFFILTFREQ	Axis filter frequency (SURFACE mode).
FREQRES	First resonance frequency of the machine.
SOFTFREQ	Path filter frequency for profiles of linear acceleration.
HSCROUND	Default value of the maximum path error in HSC.
ORISMOOTH	Orientation smoothing of the rotary axes working with RTCP.

HSCDEFAULTMODE

Default mode when programming #HSC ON.

Possible values: SURFACE / CONTERROR / FAST.

Default value: SURFACE.

Associated variable: (V.)[ch].MPG.HSCDEFAULTMODE

Default mode to activate HSC machining.

FEEDAVRG

Calculating feedrate average.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)[ch].MPG.FEEDAVRG

This parameter enables the adjustment of the feedrate according to the block reading speed and their size. This adjustment eliminates the need to slow down due to a poor supply of small blocks; although as a result, the feedrate reached is lower, the overall machining time will improve. The block reading speed depends on machine parameter PREPFREQ.

This parameter is only valid when the acceleration profile is trapezoidal or square sine (parameter SLOPETYPE), that is the default acceleration profile for the HSC CONTERROR mode.

MINCORFEED

Minimum feedrate at the corners.

Possible values: From 0 to 200000.0000 mm/min / from 0 to 7874.01575 inch/min.

Default value: 0.

Associated variable: (V.)[ch].MPG.MINCORFEED

This value should not be modified because it is possible to exceed the dynamics of the axes.

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

CORNER**Maximum angle of the corner to machine it in square corner mode.***Possible values: From 0 to 180.0000°.**Default value: 0.**Associated variable: (V.)[ch].MPG.CORNER*

This parameter indicates the maximum angle between two paths, under which the machining operation is carried out in square corner mode.

SMOOTHFREQ**Smoothing frequency in the interpolation.***Possible values: From 0 to 500.0000.**Default value: 0 (not active).**Associated variable: (V.)[ch].MPG.SMOOTHFREQ*

This parameter sets the smoothing frequency in path interpolation. This parameter avoids accelerating and decelerating throughout a path, beyond a particular frequency when generating an averaged speed.

HSCFILTFREQ**Filter frequency (CONTERROR mode).***Possible values: From 0 to 500.0000.**Default value: 40.**Associated variable: (V.)[ch].MPG.HSCFILTFREQ*

This parameter activates an automatic FIR filter for all the axes of the channel during the execution in CONTERROR mode. This filter may be used to smooth the response of the axes generating a smoother tool path and, if necessary, reduce the feedrate so the path error is adjusted to the programmed path.

This parameter inserts a variable (non constant) phase shift depending on the frequencies. This phase shift can cause a change on the path if it is not executed at the same feedrate as, for example, when changing the feedrate override percentage or when moving back and forth on the same path.

FASTFACTOR**Default feedrate percentage (FAST mode).***Possible values: From 0 to 100.**Default value: 100.**Associated variable: (V.)[ch].MPG.FASTFACTOR*

This parameter indicates the feedrate at the corners and indicates the acceleration percentage to be reached, of the maximum that the CNC can reach in FAST mode. This parameter indicates the default value and it may be changed from the part program.

FTIMELIM**Time difference permitted in feedrate interpolation (FAST and SURFACE modes).***Possible values: from 0 to 100000.0000 %.**Default value: 200 %.**Associated variable: (V.)[ch].MPG.FTIMELIM*

The feedrate interpolation in HSC FAST and SURFACE modes improves the machining quality. Parameter FTIMELIM allows limiting the time it takes the CNC to interpolate the feedrate in each segment. When programmed with a 0 value, the CNC does not interpolate the feedrate and, therefore, the execution is faster. When increasing the value of this parameter, it increases the maximum total execution time allowed; e.g. a value of 200% or 300% will allow an execution time twice or three times as long in the interpolated segment.

In rather large blocks where the feedrate adaptation represents a longer additional time than the one indicated in FTIMELIM, the CNC will adapt the feedrate using the maximum dynamics to avoid unnecessary waste of time.

(REF: 1709)

FSMOOTHFREQ**Smoothing frequency in the interpolation (FAST and SURFACE modes).**

Possible values: From 0 to 500.0000.

Default value: 20.

Associated variable: (V.){ch}.MPG.FSMOOTHFREQ

This parameter sets the smoothing frequency in path interpolation for the HSC FAST and SURFACE modes. This parameter avoids accelerating and decelerating throughout a path, beyond a particular frequency when generating an averaged speed. It is only applied when the HSC FAST or SURFACE mode is active.

FASTFILTFREQ**Filter frequency (FAST mode).**

Possible values: From 0 to 500.0000.

Default value: 17.

Associated variable: (V.){ch}.MPG.FASTFILTFREQ

This parameter activates an automatic "low pass" filter for all the axes of the channel while executing in HSC FAST mode that permits smoothing the response of the axes by generating a smoother path. This filter has the drawback of rounding the corners slightly.

This parameter inserts a constant phase shift regardless of the frequencies.

SURFFILTFREQ**Axis filter frequency (SURFACE mode).**

Possible values: From 0 to 500.0000 Hz.

Default value: 25 Hz.

Associated variable: (V.){ch}.MPG.SURFFILTFREQ

This parameter activates an automatic filter for all the axes of the channel while executing in HSC SURFACE mode that permits smoothing the response of the axes by generating a smoother path.

FREQRES**First resonance frequency of the machine.**

Possible values: From 0 to 500.0000.

Default value: 0.

Associated variable: (V.){ch}.MPG.FREQRES

Resonance frequency that the CNC must eliminate when generating the velocity command.

This parameter is only valid when the acceleration profile is trapezoidal or square sine (parameter SLOPETYPE), that is the default acceleration profile for the HSC CONTERROR mode.

SOFTFREQ**Path filter frequency for profiles of linear acceleration.**

Possible values: From 0 to 500.0000 Hz.

Default value: 25 Hz.

Associated variable: (V.){ch}.MPG.SOFTFREQ

This parameter may be used to smooth the velocity profile in HSC SURFACE mode, which improves machining time and surface quality. This parameter must be set using the FineTune application.

HSCROUND**Default value of the maximum path error in HSC.**

Possible values: From 0.0001 to 99999.9999 mm / from 0 to 3937.00787 inches.

Default value: 0.06 mm / 0.00236 inch.

Associated variable: (V.){ch}.MPG.HSCROUND

Maximum path error in HSC.

2.**MACHINE PARAMETERS.**

General machine parameters. Execution channels.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

ORISMOOTH

Orientation smoothing of the rotary axes working with RTCP.

Possible values: From 0 to 200 ms.

Default value: 60 ms.

Associated variable: (V.)[ch].MPG.ORISMOOTH

This parameter smooths the orientation of the rotary axes, without tool tip error, when working with RTCP in HSC SURFACE mode.

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4.7 Virtual tool axis.

A virtual axis of the tool is a fictitious axis that always moves in the direction in which the tool is oriented. The purpose of the virtual tool is to facilitate the movement in the direction of the tool when it is not aligned with the axes of the machine. 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.

VIRTAXISNAME

Virtual tool axis name.

Possible values: Any valid axis name; X, X1..X9, .. , C, C1..C9.

Default value: none.

Associated variable: (V.)[ch].MPG.VIRTAXISNAME

Name identifying the virtual axis of the tool.

The virtual tool axis must be a linear axis, must belong to the channel and cannot be part of the main trihedron when it is active. Since it is an axis of the channel, it 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.

VIRTAXCANCEL

Automatic cancellation of the virtual tool axis after an M30 and Reset.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPG.VIRTAXCANCEL

This parameter indicates whether the CNC cancels or not the virtual tool axis after executing an M02/M30 and after a reset.

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.

2.4.8 Default conditions (channel).

The following parameters indicate the conditions assumed by the channel on power-up, after executing an M02 or M30 or after a reset.

KINID

Default kinematics number.

Possible values: From 0 to 6 or nothing.

Default value: Nothing (the CNC does not assume any kinematics on power-up).

Associated variable: (V.)[ch].MPG.KINID

This parameter indicates the kinetics number (not type) active by default. If set to ·0·, on power-up the CNC restores the kinematics that was active when it was turned off. If no value is set, the CNC does not activate any kinematics by default.

There can be up to 6 different kinematics defined at the CNC. To select another kinematics from the part-program, use the #KIN ID instruction.

CSCANCEL

Cancel the inclined plane on start-up.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)[ch].MPG.CSCANCEL

This parameter indicates whether on start-up the CNC cancels the inclined plane (#CS/#ACS) that was active when the CNC was turned off.

LINKCANCEL

Cancel axis coupling by default.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)[ch].MPG.LINKCANCEL

This parameter indicates whether the active axis couplings (#LINK) are canceled or not at the end of the part-program, after an emergency or after a reset.

MIRRORCANCEL

Cancel mirror image (G11/G12/G13/G14) after M30 and reset.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)[ch].MPG.MIRRORCANCEL

This parameter indicates whether mirror image is cancelled or not at the end of the part program or after a reset.

SLOPETYPE

Default acceleration type.

Possible values: Linear / Trapezoidal / Square sine (bell shaped).

Default value: Square sine (bell shaped).

Associated variable: (V.)[ch].MPG.SLOPETYPE

This parameter indicates the type of acceleration applied by default in automatic movements. There are three types of acceleration, namely: linear, trapezoidal and square sine (bell shaped). It is recommended to use square-sine type acceleration. When working in manual (JOG) mode, the CNC always applies linear acceleration.

The dynamics of trapezoidal and square sine (bell shaped) accelerations are similar. Trapezoidal acceleration may be used to program ramps in order to smooth out the acc/dec changes. The square sine acceleration is an improvement to the trapezoidal acceleration; it smoothes out the jerk so the movements are softer and the axis mechanics suffers less.

Depending on the type of acceleration selected, the machine parameters will show the ones needed to configure the acceleration. To select a different acceleration via part-program, for the automatic mode, use the instruction #SLOPE.

2.

MACHINE PARAMETERS. Execution channels.
General machine parameters. Execution channels.



FAGOR AUTOMATION

CNC 8070

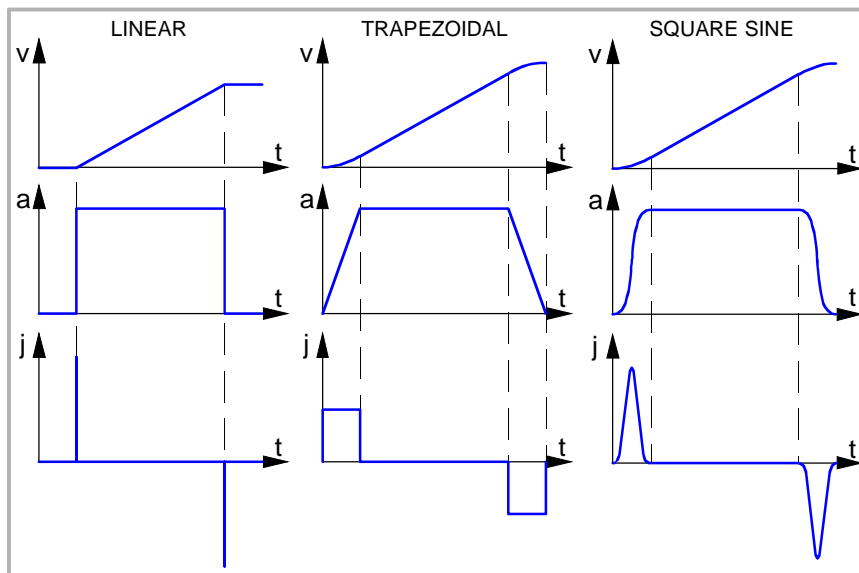
(REF: 1709)

Description of the types of acceleration.

Square-sine acceleration provides the system's best response. The movements are smoother and the axis mechanics does not suffer as much. Linear acceleration provides the poorest response.

However, the smoother the system's response, the slower the movements. Linear acceleration provides the fastest movements and the square sine the slowest.

The figure below shows the graphs for velocity (v), acceleration graph (a) and jerk (j) for each case. As acceleration represents the velocity change per time unit, the jerk represents the acceleration change per time unit.



Parameter SLOPETYPE for the spindle.

The parameter SLOPETYPE does not affect the spindle (M3/M4), which always uses jerk acceleration (parameters ACCEL/DECEL and ACCJERK/DECJERK). When the spindle works as a C axis, it will use linear acceleration or jerk acceleration depending on the type of machining.

Acceleration type.	Spindle as C axis. Type of machining.
Linear.	EI CNC uses the parameters LACC1/LACC2/LFEED. <ul style="list-style-type: none"> • Non-HSC Mode (G7/G50/G5) when SLOPETYPE=Linear. • HSC CONTERROR mode when SLOPETYPE=Linear. • HSC SURFACE/FAST mode.
Trapezoidal.	EI CNC uses the parameters ACCEL/DECEL/ACCJERK/DECJERK. <ul style="list-style-type: none"> • Non-HSC Mode (G7/G50/G5) when SLOPETYPE=Trapezoidal. • HSC CONTERROR mode when SLOPETYPE=Trapezoidal.
Square sine (bell shaped).	EI CNC uses the parameters ACCEL/DECEL/ACCJERK/DECJERK. <ul style="list-style-type: none"> • Non-HSC Mode (G7/G50/G5) when SLOPETYPE=Square sine. • HSC CONTERROR mode when SLOPETYPE=Square sine.

IPLANE

Main plane (G17/G18) by default.

Possible values: G17 / G18.

Default value: G17.

Associated variable: (V.)[ch].MPG.IPLANE

This parameter indicates the main work plane assumed by the CNC by default. The axes that form the work plane depend on machine parameter CHAXISNAME.

Plane.	Abcissa axis.	Ordinate axis.	Longitudinal axis.
G17	CHAXISNAME 1	CHAXISNAME 2	CHAXISNAME 3
G18	CHAXISNAME 3	CHAXISNAME 1	CHAXISNAME 2

M models (mill) and L models (laser).

It is recommended to define the IPLANE parameter as G17. To change the work plane via part-program, use function G17, G18, G19 or G20.

T model (lathe). Configuration of "Trihedron" type axes.

It is recommended to define the IPLANE parameter as G18. To change the work plane via part-program, use function G17, G18, G19 or G20.

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

The work plane is always G18; machine parameter IPLANE is not applied and it is not possible to change planes via the part-program, only the longitudinal axis (G20).

ISYSTEM**Type of coordinates (G90/G91) by default.**

Possible values: G90 / G91.

Default value: G90.

Associated variable: (V.)[ch].MPG.ISYSTEM

This parameter indicates the type of coordinates assumed by the CNC by default. 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.

To change the type of coordinates via part-program, use function G90 or G91.

IMOVE**Type of movement (G0/G1) by default.**

Possible values: G0 / G1.

Default value: G01.

Associated variable: (V.)[ch].MPG.IMOVE

This parameter indicates the type of movement, G0 or G1, assumed by the CNC by default (after a start-up, following a reset or M30). To change them from the part-program, use function G0 or G1.

IMOVE	Meaning.
G0.	<ul style="list-style-type: none"> The CNC assumes the G0 function. Movements in G0 are carried out in rapid as set by parameter G00FEED. The CNC will display G0 in the G function history. When opening the MDI/MDA mode and programming a movement without G, the CNC will execute it as G0. When programming a movement via the manual mode, the axis moves as G0.
G1.	<ul style="list-style-type: none"> The CNC assumes function G1. Movements in G1 are carried out at the feedrate active at the CNC. The CNC does not display G1 in the G function history. When opening the MDI/MDA mode and programming a movement without G, the CNC will execute it as G1. When programming a movement via the manual mode, the axis moves as G1.

Parameter G0MODAL; maintaining function G0 as modal or non-modal.

The value IMOVE=G0 has priority over G0MODAL=No. The G0 function is defined as non-modal, however, with IMOVE=G0, the CNC assumes the G0 function at start-up after a reset or M30. After programming G1, G2 or G3, the CNC recovers G1 as a modal function until the following reset or M30. See "[G0MODAL Programming for G0 remains as modal.](#)" on page 135.

2.

MACHINE PARAMETERS. Execution channels.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

GOMODAL

Programming for G0 remains as modal.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)[ch].MPG.GOMODAL.

This parameter defines whether the G00 function is modal or not.

GOMODAL	Meaning.
Yes.	The G00 function is modal. After it has been programmed, it remains active until G01, G02, G03, G33 or G63 are programmed.
No.	The G00 function is non-modal, it must be programmed for each rapid traverse block. For the following program block, if there is no programmed movement function (G0, G1, G2, G3, G33 or G63), the CNC uses G1, meaning that G1 remains as modal. Within the range of influence of a canned cycle, the last programmed G will be maintain active, while G0 remains being modal.

IMOVE parameter; type of movement (G0/G1) at startup, after a reset or M30.

The value IMOVE=G0 has priority over GOMODAL=No. The G0 function is defined as non-modal, however, with IMOVE=G0, the CNC assumes the G0 function at start-up after a reset or M30. After programming G1, G2 or G3, the CNC recovers G1 as a modal function until the following reset or M30. See "*IMOVE Type of movement (G0/G1) by default.*" on page 134.

G2G3MODAL

Programming in G2/G3 remains as modal.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)[ch].MPG.G2G3MODAL.

This parameter defines whether the G2/G3 functions are modal or not.

G2G3MODAL	Meaning.
Yes.	The G02 and G03 functions are modal. Once they are programmed, they remain active until G00, G01, G33 or G63 are programmed.
No.	The G02 and G03 functions are non-modal. These must be programmed for each circular interpolation block. For the following program block, if there is no programmed movement function (G0, G1, G2, G3, G33 or G63), the CNC uses G1.

IFEED

Type of feedrate (G94/G95) by default.

Possible values: G94 / G95.

Default value: G94.

Associated variable: (V.)[ch].MPG.IFEED

This parameter indicates the type of feedrate assumed by the CNC by default.

- With G94, the feedrate is assumed in mm/min or degrees/min or inches/min.
- With G95, the feedrate is assumed in mm/rev or degrees/rev or inches/rev.

The typical configuration for the mill model will be G94. The typical configuration for the lathe model will be G95.

To change the type of feedrate via part-program, use function G93 or G94.

FPRMAN

Function G95 admitted in jog mode.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPG.FPRMAN

Parameter valid on models -T-, -BL- and -OL-; not being used on the -M- model.

This parameter indicates whether function G95 (feedrate in mm/turn or inch/turn) is allowed in jog mode, for jog movements; it does not affect handwheel movements or in MDI/MDA mode.

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

The CNC will apply function G95 to the main axes of the channel (parameter GEOCONFIG); G95 does not affect the rest of the axes (auxiliary axes, steady rest, tailstock, turrets, etc.) even if FPRMAN = YES.

GEOCONFIG.	Meaning.
Plane	The CNC will apply function G95 to the first two axes of the channel.
Trihedron	The CNC will apply function G95 to the first three axes of the channel.

When G95 is active and FPRMAN = YES, the real feedrate (Freal) displayed in jog and conversational modes always is in mm/turn or inch/turn, whether the moving axis is a main axis or not.

IRCOMP

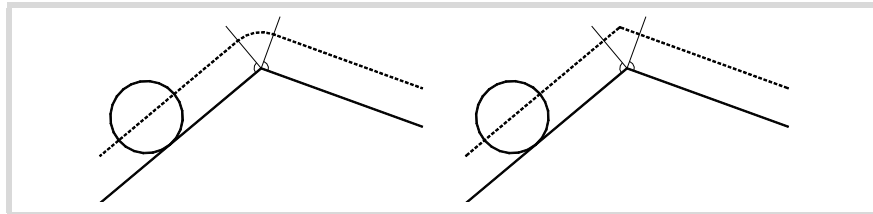
Tool radius compensation mode (G136/G137) by default.

Possible values: G136 / G137.

Default value: G136.

Associated variable: (V.)[ch].MPG.IRCOMP

This parameter indicates the compensation mode assumed by the CNC by default. Being the radius compensation active, the compensated paths may be blended together using circular paths (G136) or linear paths (G137).



To change the type of compensation via part-program, use function G136 or G137.

COMPCANCEL

How to cancel tool radius compensation.

Possible values: With motion / Without motion.

Default value: Without motion.

Associated variable: (V.)[ch].MPG.COMPCANCEL

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.

GOOCOMP

Apply tool radius compensation in G00.

Possible values: Yes / No.

Default value: Yes (milling and lathe) / No (laser).

Associated variable: (V.)[ch].MPG.GOOCOMP

This parameter indicates whether the CNC applies tool radius compensation or not in G00 movements.

2.

MACHINE PARAMETERS. Execution channels.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

LCOMPTYP

Keep the longitudinal axis when changing planes (G17/G18/G19).

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPG.LCOMPTYPE

This parameter indicates which will be the longitudinal axis when changing planes (G17/G18/G19). In either case, the longitudinal axis may be changed via part-program using G20 functions or #TOOL AX.

LCOMPTYP	Meaning.
Yes.	When changing work planes (G17/G18/G19), the CNC el CNC keeps the longitudinal axis that was active before the plane change.
No.	When changing the work planes (G17/G18/G19), the CNC assumes the axis perpendicular to the plane as the new longitudinal axis.

PLANECANCEL

Assume IPLANE as active plane after M30/RESET or keep the active one.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)[ch].MPG.PLANECANCEL

This parameter defines which is the active plane after executing M02, M30 or reset.

PLANECANCEL.	Meaning.
Yes	On power-up, after executing an M02 or M30, and after a reset, the CNC assumes the plane defined in parameter IPLANE.
No	On power-up, the CNC assumes the plane defined in parameter IPLANE; after executing an M02 or M30, and after a reset, the CNC maintains the main active plane.

ICORNER

Type of corner (G5/G7/G50) by default.

Possible values: G50 / G5 / G7.

Default value: G50.

Associated variable: (V.)[ch].MPG.ICORNER

This parameter indicates the type of corner assumed by the CNC by default. There are three types of corners, namely: square (G7), rounded (G5) and semi-rounded (G50). To change the type of corner via part-program, use function G5, G7 or G50.

- When working in square corner, the CNC starts executing the next movement when the axis gets into the in-position zone defined by parameter INPOSW.
- When working in round corner, it is possible to control the corner of the programmed profile.
- When working in semi-rounded corner, the CNC starts executing the next movement once the theoretical interpolation of the current move is completed.

If G5is selected, parameter ROUNTYPE must be set.

ROUNDTYPE

Rounding type in G5 (by default).

Possible values: Chordal error / % Feedrate.

Default value: Chordal error.

Associated variable: (V.)[ch].MPG.ROUNDTYPE

This parameter indicates the type of rounding applied by default when working in round corner. To change the type of rounding via program, use the #ROUNDPAR instruction.

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. Depending on the option selected, either parameter MAXROUND or ROUNDFEED will have to be set.

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.

MAXROUND**Maximum rounding error in G5.***Possible values: From 0.02 to 99999.9999 mm or degrees / from 0.00079 to 3937.00787 inch.**Default value: 0.1000 mm or degrees / 0.00394 inch.**Associated variable: (V.)[ch].MPG.MAXROUND*

This parameter sets the maximum deviation allowed between the programmed point and the profile resulting from rounding the corner. The CNC takes it into account if ROUNDTYPE = Chordal error.

This parameter also sets the maximum contour allowed for the HSC mode when not programming CONTERROR (CONTERROR mode) or E (FAST mode).

ROUNDFEED**Percentage of feedrate in G5.***Possible values: From 0 to 100.**Default value: 100.**Associated variable: (V.)[ch].MPG.ROUNDFEED*

This parameter sets the percentage of the active feedrate to be used for machining. The CNC takes it into account if ROUNDTYPE = % Feedrate.

2.4.9 Arc center correction.

CIRINERR

Maximum absolute radius error permitted.

Possible values: From 0 to 99999.9999 mm or degrees / from 0 to 3937.00787 inch.

Default value: 0.0100 mm or degrees / 0.00039 inch.

Associated variable: (V.)[ch].MPG.CIRINERR

See general machine parameter CIRINFACT.

CIRINFACT

Maximum relative radius error permitted.

Possible values: from 0 to 100.0 %.

Default value: 0.1 %.

Associated variable: (V.)[ch].MPG.CIRINFACT

These parameters set the conditions for correcting the center position in circular interpolations. On circular interpolations, the CNC calculates the radius of the starting point and end point of the tool path. Theoretically, they should be the same; but these parameters may be used to set the maximum difference allowed between both radius.

Parameter CIRINERR indicates the maximum absolute error allowed. Parameter CIRINFACT indicates the maximum relative error allowed (% of the radius). Both parameters are taken into account. The CNC will show the relevant error message when this difference between them is greater than CIRINERR and greater than $\text{—CIRINFACT} \times \text{Radius—}$.

This function is controlled via program using functions G264 and G265.

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.

2.4.10 Behavior of the feedrate and the feedrate override.

MAXOVR

Maximum override (%) permitted.

Possible values: From 0 to 255.

Default value: 200.

Associated variable: (V.)[ch].MPG.MAXOVR

This parameter indicates the maximum percentage to be applied to the programmed axis feedrate (feedrate override).

The percentage applied to the programmed feedrate may be set by program, via PLC or by the switch of the panel. The one set by program has the highest priority and the one set by the switch has the lowest priority.

Different values may be set for each axis via PLC and by program. The one selected at the switch is common to all of them.

RAPIDOVR

Override acts in G00 (from 0 to 100%).

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)[ch].MPG.RAPIDOVR

This parameter indicates whether the feedrate % may be modified (between 0% and 100%) or not when working in G0. If not allowed, the percentage 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.

FEEDND

Apply the programmed feedrate to all the axes of the channel.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPG.FEEDND

This parameter indicates whether the programmed feedrate is applied to all the axes of the channel or only to the main axes.

FEEDND	Meaning.
Yes.	The programmed feedrate will be the result of composing the movements onto all the axes of the channel.
No.	If a movement has been programmed on any of the main axes, the programmed feedrate will be the result of composing the movement only onto these axes. The rest of the axes move at their corresponding feedrate to end the movement of them all at the same time. The programmed feedrate is limited only if an axis could exceed its MAXFEED. If none of the main axes are programmed, the programmed feedrate will be reached on the axis moving the farthest so they can all reach their destination at the same time.

2.

MACHINE PARAMETERS. Execution channels.
General machine parameters. Execution channels.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4.11 Override of the dynamics for HSC.

MINDYNOVR

Minimum override for the dynamics in HSC.

Possible values: From 10 to 100 %.

Default value: 30 %.

Associated variable: (V.)[ch].MPG.MINDYNOVR

The screen of the automatic mode offers a slider for changing the dynamics defined by program for an HSC machining operation. This parameter sets the minimum percentage to be applied using the slider.

MAXDYNOVR

Maximum override for the dynamics in HSC.

Possible values: From 100 to 500 %.

Default value: 200 %.

Associated variable: (V.)[ch].MPG.MAXDYNOVR

The screen of the automatic mode offers a slider for changing the dynamics defined by program for an HSC machining operation. This parameter sets the maximum percentage to be applied using the slider.

2.4.12 Movement of the independent axes.

IMOVEMACH

Movement of the independent axis referred to machine coordinates.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPG.IMOVEMACH

This parameter indicates whether the movements of the independent axes are referred to machine coordinates (IMOVEMACH = YES) or to part coordinates (IMOVEMACH = NO), before the coordinate transformations.

XFITOIND

The transfer inhibit of the channel affects the independent axes.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)[ch].MPG.XFITOIND

This parameter indicates whether the transfer inhibit of the channel (_XFERINH mark) affects the movements of the independent axis programmed from the PLC or not. The transfer inhibit of the channel always affects the movements of the independent axis programmed from the CNC.

XFITOIND	Meaning.
Yes	The _XFERINH mark affects the movement of the independent axis programmed from the PLC and from the CNC.
No	The _XFERINH mark does not affect the movements of the independent axis programmed from the PLC; but it does affect the ones programmed from the CNC.

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4.13 Definition of the subroutines.

SUBTABLE OEM-subroutines table.

The manufacturer's subroutines are those stored in folder ..\MTB\Sub. These subroutines are associated with functions of the CNC such as tool change, G74, G180 to G189, G380 to G399, etc.

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. To load a subroutine into RAM, it must have the extension .fst.

Being the CNC in USER mode, the OEM routines having a .fst extension are loaded into RAM memory when starting up the CNC application. 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.

TOOLSUB Subroutine associated with "T".

Possible values: any text with up to 64 characters.

Associated variable: (V.)[ch].MPG.TOOLSUB

This subroutine is executed automatically every time a T function (tool selection) is executed.

REFPSUB (G74) Subroutine associated with function G74.

Possible values: any text with up to 64 characters.

Associated variable: (V.)[ch].MPG.REFPSUB

Function G74 (home search) may be programmed in two ways. indicating the axes and the order they will be homed or by programming G74 alone (without axes).

When executing a block that only contains the G74 function (without axes), it calls the subroutine indicated in this parameter. This subroutine must contain the axes and the order (sequence) to search home.

This subroutine is also called when homing the axis in JOG mode without selecting the axes.

INT1SUB..INT4SUB Interruption subroutines.

Possible values: any text with up to 64 characters.

Associated variable: (V.)[ch].MPG.INT1SUB / (V.)[ch].MPG.INT4SUB

These parameters indicate the name of the interruption subroutines associated with marks INT1 to INT4 of the PLC. When the PLC activates one of these marks, the channel interrupts the execution of the program and executes the corresponding interruption subroutine. If the program is interrupted (STOP) or no program is in execution (channel in READY state), the execution of the subroutine depends on parameter SUBINTSTOP.

INITIALSUB Subroutine associated with the command #INITIALSUB.

Possible values: any text with up to 64 characters.

Associated variable: (V.)[ch].MPG.INITIALSUB

This parameter indicates the name of the subroutine associated with the instruction #INITIALSUB. Every time the CNC executes this instruction, its associated subroutine is called upon.

2.

MACHINE PARAMETERS. Execution channels.
General machine parameters. Execution channels.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

PIERCING**Subroutine associated with the command #PIERCING.**

Possible values: any text with up to 64 characters.

Associated variable: (V.)[ch].MPG.PIERCING

This parameter indicates the name of the subroutine associated with the instruction #PIERCING. Every time the CNC executes this instruction, its associated subroutine is invoked (by default, Piercing.fst).



The OEM is responsible for ensuring that the subroutine has all the safety aspects related to handling the laser.

At the start of the subroutine (or when the OEM deems it necessary), the OEM must activate the PIERCING mark (assigning it the value 1) to indicate to the PLC that it must start the piercing operation. At the end of the subroutine (or when the OEM deems it necessary), the OEM must deactivate this mark (assigning it the value 0) to indicate to the CNC that it has finished the piercing operation.

CUTTING ON**Subroutine associated with the command #CUTTING ON.**

Possible values: any text with up to 64 characters.

Associated variable: (V.)[ch].MPG.CUTTING ON

This parameter indicates the name of the subroutine associated with the instruction #CUTTING ON. Every time the CNC executes this instruction, its associated subroutine is invoked (by default, Cuttingon.fst).



The OEM is responsible for ensuring that the subroutine has all the safety aspects related to handling the laser.

At the start of the subroutine Cuttingon.fst (or when the OEM deems it necessary), the OEM must activate the CUTTING mark (assigning it a value of 1) to indicate to the PLC that it should start the cutting operation.

CUTTING OFF**Subroutine associated with the command #CUTTING OFF.**

Possible values: any text with up to 64 characters.

Associated variable: (V.)[ch].MPG.CUTTING OFF

This parameter indicates the name of the subroutine associated with the instruction #CUTTING OFF. Every time the CNC executes this instruction, its associated subroutine is invoked (by default, Cuttingoff.fst).



The OEM is responsible for ensuring that the subroutine has all the safety aspects related to handling the laser.

At the end of the Cuttingoff.fst subroutine (or when the OEM deems it necessary), the OEM must deactivate the CUTTING mark (assigning it a value of 0) to indicate to the CNC that it has finished the cutting operation.

FINALSUB**Subroutine associated with the command #FINALSUB.**

Possible values: any text with up to 64 characters.

Associated variable: (V.)[ch].MPG.FINALSUB

This parameter indicates the name of the subroutine associated with the instruction #FINALSUB. Every time the CNC executes this instruction, its associated subroutine is called upon.

OEMSUB (G18x)**Subroutines associated with functions G180 through G189.**

Possible values: any text with up to 64 characters.

Associated variable: (V.)[ch].MPG.OEMSUB1..10

These parameters indicate the number of the subroutines associated with functions G180 through G189. Every time one of these functions is executed, its associated subroutine is called upon.

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.

FAGOR

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

OEMSUB (G38x)**Subroutines associated with functions G380 through G399.**

Possible values: any text with up to 64 characters.

Associated variable: (V.)[ch].MPG.OEMSUB11..30

These parameters indicate the number of the subroutines associated with functions G380 through G399. Every time one of these functions is executed, its associated subroutine is called upon.

SUBPATH**Path of the program subroutines.**

Associated variable: (V.)[ch].MPG.SUBPATH

This parameter indicates the directory, by default, containing the user subroutines and the necessary subroutines to execute programs of the 8055 MC and 8055 TC.

User subroutines.

User subroutines are those associated with part-programs. These subroutines must be in an unprotected folder for the user mode. When calling any of these subroutines (instructions #PCALL, #CALL, etc.) without indicating the path, the subroutine looks for them in this order and in the following directories:

- 1 Folder selected with the #PATH instruction.
- 2 Folder of the program in execution.
- 3 Folder indicated in the machine parameter SUBPATH.

When the call indicates the whole path, it will only look for it in the indicated folder.

Subroutines 9998 to 9999.

In order to be able to execute programs of the 8055 MC/TC, the CNC must have two subroutines called 9998 and 9999 written in 8070 CNC language.

9998 Subroutine that the CNC will execute at the beginning of each program.

9999 Subroutine that the CNC will execute at the end of each program.

Each program in the language of the 8055 MC/TC has a call to the corresponding subroutine at the beginning and at the end. Both subroutines must be defined, even if no operation is to be carried out at the beginning or at the end of the part-program, in which case the subroutines will be empty (except for the end-of-subroutine block). If any of these subroutines is missing, the CNC will issue an error message every time when trying to execute a part-program.

2.

2.4.14 Tabletop probe position.

PROBEDATA

Channel related probe parameters.



This parameter shows the table to define the position of the tabletop probe on the first three axes of the channel according to parameter CHAXISNAME. The table shows the following parameters to configure it.

PROBEDATA	
PRB1MAX	Maximum probe coordinate (first axis of the channel).
PRB1MIN	Minimum probe coordinate (first axis of the channel).
PRB2MAX	Maximum probe coordinate (second axis of the channel).
PRB2MIN	Minimum probe coordinate (second axis of the channel).
PRB3MAX	Maximum probe coordinate (third axis of the channel).
PRB3MIN	Minimum probe coordinate (third axis of the channel).

PRB1MAX

Maximum probe coordinate (first axis of the channel).

Possible values: Within ±99999.9999 mm / within ±3937.00787 inch.

Default value: 0.

Associated variable: (V.)[ch].MPG.PRB1MAX

See general machine parameter PRB3MIN.

PRB1MIN

Minimum probe coordinate (first axis of the channel).

Possible values: Within ±99999.9999 mm / within ±3937.00787 inch.

Default value: 0.

Associated variable: (V.)[ch].MPG.PRB1MIN

See general machine parameter PRB3MIN.

PRB2MAX

Maximum probe coordinate (second axis of the channel).

Possible values: Within ±99999.9999 mm / within ±3937.00787 inch.

Default value: 0.

Associated variable: (V.)[ch].MPG.PRB2MAX

See general machine parameter PRB3MIN.

PRB2MIN

Minimum probe coordinate (second axis of the channel).

Possible values: Within ±99999.9999 mm / within ±3937.00787 inch.

Default value: 0.

Associated variable: (V.)[ch].MPG.PRB2MIN

See general machine parameter PRB3MIN.

PRB3MAX

Maximum probe coordinate (third axis of the channel).

Possible values: Within ±99999.9999 mm / within ±3937.00787 inch.

Default value: 0.

Associated variable: (V.)[ch].MPG.PRB3MAX

See general machine parameter PRB3MIN.

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

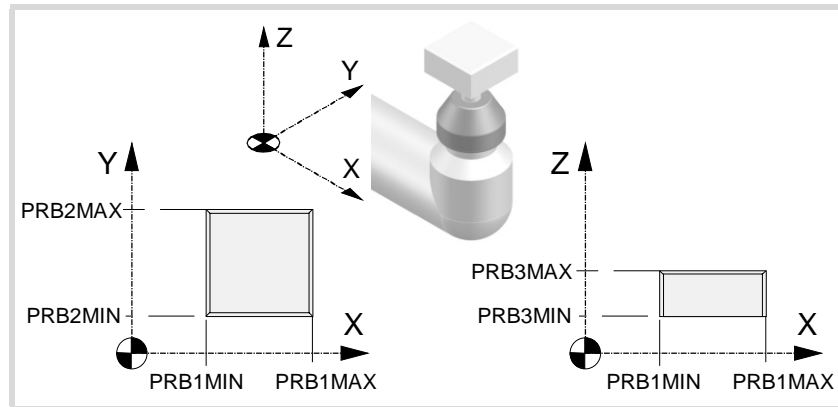
PRB3MIN**Minimum probe coordinate (third axis of the channel).**

Possible values: Within ± 99999.9999 mm / within ± 3937.00787 inch.

Default value: 0.

Associated variable: (V.)[ch].MPG.PR3MIN

These parameters define the position of the tabletop probe used for tool calibration. They must be defined in absolute coordinates referred to machine reference zero. For a LATHE model CNC, the coordinates must be given in radius.



2.

MACHINE PARAMETERS. Execution channels.
General machine parameters.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4.15 Block search.

FUNPLC

Send the M, H, S functions to the PLC during block search.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPG.FUNPLC

With block search, it is possible to recover the program history up to a particular block, that may be in a program or in a subroutine, in such way that if program execution is resumed at that block, it will do so with the same conditions as if it were executed from the beginning.

This parameter determines whether or not the CNC sends the M, H, S functions to the PLC during block search. The M function table has an MPLC field to define whether the CNC sends the function to the PLC or not. All M functions set in the table will be sent out to the PLC or not depending on this field regardless of the setting of machine parameter FUNPLC.

FUNPLC	Meaning.
Yes	In this case, the functions are sent out to the PLC during block search as they are being read. Once block search ends and after repositioning the axes, the CNC shows the "MHSF" softkey for executing any function of this type, thus modifying the machining conditions.
No	In this case, the functions are not sent out to the PLC during block search. After the search, the CNC screen shows the history of those functions so the user can enable them in the desired order.

2.4.16 Interruption subroutines.

SUBINTSTOP

Execute interruption subroutines while the program is stopped or while no program is in execution.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPG.SUBINTSTOP

This parameter determines whether the CNC must execute the interruption subroutine when the program is interrupted (STOP) or no program is in execution (channel in READY state). 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.

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4.17 Machining feedrate.

MAXFEED

Maximum machining feedrate on the tool path.

Possible values: From 0 to 500000.0000 mm/min / from 0 to 19685.03937 inch/min.

Default value: 0.

Associated variable: (V.)[ch].MPG.MAXFEED

This parameter sets the maximum machining feedrate (movements in G01/G02/G03) along the path; when set to ·0·, the feedrate is not limited. This parameter cannot be set with a value higher than that of parameter G00FEED.

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 in MAXFEED without showing any error message or warning.

If the machining feedrate is not limited, the CNC assumes for all the movements the one set in machine parameter G00FEED as the maximum feedrate.

How to limit temporarily the maximum feedrate via PLC

The PLC has the variable (V.)[ch].PLC.G00FEED that may be used to limit the feedrate in the channel for any type of movement (G00, G01, etc). This variable limits the feedrate of the path and affects all the axes whether they move interpolated or one axis at a time.

The CNC assumes the change immediately and the change stays active until the variable takes a value of ·0· restoring the limits set by machine parameters.

DEFAULTFEED

Assume MAXFEED for movements in G1/G2/G3 without active feedrate.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPG.DEFAULTFEED

This parameter sets whether or not the axes move at the feedrate set in MAXFEED when programming a movement in G01/G02/G03 without programming the feedrate. If this parameter is set to ·No·, the CNC will issue an error message when programming a movement in G01/G02/G03 without programming the feedrate.

The CNC takes this parameter into account when MAXFEED has been set with a value other than ·0·.

2.

MACHINE PARAMETERS. Execution channels.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4.18 Rapid traverse for the automatic mode.

RAPIDEN

It is used to enable rapid traverse for the automatic mode.

Possible values: Disabled / EXTRAPID or rapid key / EXTRAPID and rapid key.

Default value: Disabled.

Associated variable: (V.)[ch].MPG.RAPIDEN

This parameter indicates whether rapid traverse may be enabled or not during the execution of a program for the programmed movements. Depending on the selected option, activating rapid traverse will require activating the PLC mark EXTRAPID or pressing the "rapid" key or both.

RAPIDEN	Meaning.
Disabled.	Rapid traverse is not available for the automatic mode.
EXTRAPID or rapid key.	To activate the rapid feed, just activate the PLC mark EXTRAPID or press the "rapid" key of the jog panel.
EXTRAPID and rapid key.	To activate the rapid feed, just activate the PLC mark EXTRAPID and press the "rapid" key of the jog panel.

When activating rapid traverse while executing a program, each axis will assume the feedrate set by its parameter FRAPIDEN, as long as G0 or a thread is not active. Movements in G0 are carried out at the feedrate set in parameter G00FEED. The threads are executed at the programmed feedrate. The rapid traverse will also be limited by channel parameter FRAPIDEN.



If "EXTRAPID or rapid key" is the chosen option for RAPIDEN and the CNC has several channels, the EXTRAPID mark will only affect the corresponding channel. The rapid key, on the other hand, affects simultaneously all the channels that may be affected at the time. If the active channel is in jog mode and another channel is executing a program, when pressing the rapid key in the active channel (jog mode), the rapid traverse will also be applied in the channel that is executing the program.

In a system where the rapid key may affect several channels at the same time, we recommend to choose the "EXTRAPID and rapid key" option or manage by PLC the behavior of the rapid key according to the active channel and the selected work mode (jog or automatic).

FRAPIDEN

Rapid traverse in the channel for movements in automatic mode.

Possible values: From 0 to 500000.0000 mm/min / from 0 to 19685.03937 inch/min.

Default value: 0.

Associated variable: (V.)[ch].MPG.FRAPIDEN

Maximum feedrate in the channel for the program being executed, 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.

Rapid traverse cannot exceed the value set in axis parameters G00FEED and FRAPIDEN 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).

2.

MACHINE PARAMETERS.

General machine parameters. Execution channels.

FAGOR

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4.19 Maximum acceleration and jerk on the tool path.

MAXACCEL

Maximum acceleration on the machining path.

Possible values: From 0.0010 to 600000000.0000 mm/min / from 0.00004 to 23622047.24409 inch/min.

Default value: There is no maximum feedrate.

Associated variable: (V.)[ch].MPG.MAXACCEL

This parameter may be used to smooth the feedrate on the tool path by limiting the maximum acceleration on it. Regardless of the value of this parameter, the CNC always respects the dynamics of the axes involved in the path; i.e. it respects the acceleration set for each axis.

How to change temporarily the maximum acceleration defined.

The variable "(V.)[ch].G.MAXACCEL" may be used to temporarily change at a particular time and in real time the value set by the machine parameter. 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.

MAXJERK

Maximum jerk on the machining path.

Possible values: From 0.0010 to 6E11 mm/min / from 0.00004 to 2.362E10 inch/min.

Default value: There is no maximum feedrate.

Associated variable: (V.)[ch].MPG.MAXJERK

This parameter may be used to smooth the feedrate on the tool path by limiting the maximum jerk on it. Regardless of the value of this parameter, the CNC always respects the dynamics of the axes involved in the path; i.e. it respects the jerk set for each axis.

How to change temporarily the maximum jerk defined.

The variable "(V.)[ch].G.MAXJERK" may be used to temporarily change at a particular time and in real time the value set by the machine parameter. 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.

2.4.20 Maximum frequency on the tool path.

MAXFREQ

Maximum frequency generated on the machining path.

Possible values: From 0 to 500 Hz.

Default value: 0.

Associated variable: (V.)[ch].MPG.MAXFREQ

This parameter may be used to set the parameters the damping (smoothing) filter for machining in G5. This filter improves the machining of CAD parts without having to activate the HSC feature. This parameter must be set using the FineTune application.

2.

MACHINE PARAMETERS. Execution channels.
General machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.4.21 Resonance frequency of the machine.

CURVFREQRES

First resonance frequency of the machine in the arcs.

Possible values: From 0 to 500.0000 Hz

Default value: 0.

Associated variable: (V.)[ch].MPG.CURVFREQRES

Resonance frequency that the CNC must eliminate when generating the command during circular tool paths. The CNC applies this filter in HSC, G50, G5 and G7.

2.4.22 "Retrace" function.

RETRACAC

Permit enabling the "retrace" function.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPG.RETRACAC

This parameter indicates whether the "retrace" function may be used at the CNC or not. Once the "retrace" function is permitted, it may be activated from the PLC using the RETRACE mark.

NRETBK

Maximum number of blocks allowed for the "retrace" function.

Possible values: From 0 to 300.

Default value: 75.

Associated variable: (V.)[ch].MPG.NRETBK

This parameter sets the maximum number of blocks that may be retraced (executed backwards) with the "retrace" function. Once the CNC has retraced all the blocks, it goes on executing forward.

It is recommended not to increase the value of this parameter unnecessarily. In order to be able to execute the "retrace" function, the CNC stores the information of the last blocks executed. The higher the value assigned to this parameter, the greater the information the CNC must store.

RETMFUNC

Treatment of the M functions with the "retrace" function.

Possible values: Ignore / Cancel.

Default value: Ignore.

Associated variable: (V.)[ch].MPG.RETMFUNC

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.

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 only cancels the retrace function if it implies stopping the spindle; in other words, if the spindle is not running, an M05 function does not cancel the retrace function.

2.

2.4.23 Withdrawal the tool.

RETRACTENABLE Enable tool withdrawal.

This parameter shows the parameter table to enable tool withdrawal (retraction) in various situations.

RETRACTENABLE	
RETRACTTHREAD	Enable tool withdrawal (retraction) in threading.

RETRACTTHREAD Enable tool withdrawal (retraction) in threading.

Possible values: ON / OFF.

Default value: ON.

Associated variable: (V.)[ch].MPG.RETRACTTHREAD

This parameter defines the CNC behavior when interrupting a threading ([STOP] key or PLC mark _FEEDHOL).

RETRACTTHREAD	Meaning.
OFF	The CNC stops the axes at the end of the pass.
ON	<ul style="list-style-type: none"> In electronic threading (G33/G34), function G233 defines the distance that the axes withdraw from the part. If G233 is not active, the axes stop at the end of the pass. In threading canned cycles, (-T- model), both ISO and conversational, the CNC withdraws the axes from the part. The distance that the tool withdraws depends on how the cycle is programmed.

2.4.24 Master spindle.

MASTERSPDL Master spindle maintained.

Possible values: Temporary / maintained.

Default value: Temporary.

Associated variable: (V.)[ch].MPG.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.

2.

MACHINE PARAMETERS. Execution channels.
General machine parameters. Execution channels.

2.5 Machine parameters for the axes and spindles.

The CNC only shows the parameters for the selected type of axis and drive. That's why it displays some characters next to each parameter indicating the corresponding type of axis and drive.

In the laser model, the parameters related to spindle speed (defined in rpm) define the power (defined in watts).

2.5.1 Belonging to the channel.

AXISEXCH

Channel change permission.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: No / Temporary / Maintained.

Default value: No.

Associated variable: (V)[ch].MPA.AXISEXCH.xn

Initially, each channel has some axes and spindles assigned to it. The CNC can change channel axes and spindles or just change the configuration of a channel by modifying the position of its axes and spindles or eliminating some of them.

In order for the CNC to be able to change the channel axes and spindles, they must have permission. Parameter AXISEXCH sets whether the axis or the spindle has permission to change channels and if it does, whether the change is temporary or permanent; in other words, whether the change is maintained after an M02, M30, a reset or after restarting the CNC.

AXISEXCH.	Meaning.
No.	<ul style="list-style-type: none"> The CNC can modify the position of the axis or spindle in its channel or even remove it from the channel. The CNC cannot change the axis or the spindle of the channel. The CNC does not maintain the changes when restarting the part-program, after a reset or after restarting the CNC. The axis or spindle returns to its original position, set by machine parameters.
Temporary.	<ul style="list-style-type: none"> The CNC can modify the position of the axis or spindle in its channel or even remove it from the channel. The CNC can change the axis or the spindle of the channel. The CNC does not maintain the changes when restarting the part-program, after a reset or after restarting the CNC. The axis or spindle returns to its channel and original position, set by machine parameters.
Maintained.	<ul style="list-style-type: none"> The CNC can modify the position of the axis or spindle in its channel or even remove it from the channel. The CNC can change the axis or the spindle of the channel. The CNC maintains the changes when restarting the part-program, after a reset or after restarting the CNC. The axis or spindle stays in its new channel, but it can change the position to let the original axes of the channel return.

The original configuration (the one defined in the machine parameters) of a channel with axes and spindles of the type "AXISEXCH = Maintained" may be restored either by validating the machine parameters or by undoing the changes, for example using a part-program. It must be borne in mind that validating the machine parameters restores the configuration of all the channels.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.2 Type of axis and drive.

AXISTYPE

Type of axis.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Linear / Rotary / Spindle.

Default value: Linear.

Associated variable: (V.)[ch].MPA.AXISTYPE.xn

The axes defined here may be configured as gantry or tandem axes.

DRIVETYPE

Type of drive.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives, Sercos, Mlink (Mechatrolink) and simulated drives.

Possible values: Analog / Sercos / Mlink / Simulated.

Default value: Simulated.

Associated variable: (V.)[ch].MPA.DRIVETYPE.xn

This parameter sets the type of drive, analog, Sercos, Mlink (Mechatrolink) or simulated. The CNC does not allow having Sercos and Mechatrolink drives in the same system; but it does allow combining these drives with analog drives. The simulated axis option must be used when there is no physical axis or during setup. With the Simulated option, the CNC simulates all the movements, it assumes the theoretical coordinate as real and does not output velocity commands.

The simulated axes are not activated with the validation code. It is possible to use as many simulated axes as you wish as long as the sum of the simulated axes and the physical axes does not exceed the maximum number of axes possible (as set by parameter NAXIS).

Type of drive when the CNC has been installed as simulator at a PC.

When the CNC is installed as a simulator at a PC, the drive may be a simulated type, Sercos or Mechatrolink; it cannot be analog. In either case, the CNC simulates all the movements and assumes the theoretical coordinate as the real one.

POSUNITS

Feedback system units.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for Sercos, Mlink (Mechatrolink) and simulated drives.

Possible values: Measurement / Pulses.

Default value: Measurement.

Associated variable: (V.)[ch].MPA.POSUNITS.xn

This parameter sets the units system used by the feedback system.

POSUNITS	Meaning.
Measurement.	The communication with the drive is in millimeters or degrees.
Pulses.	The communication with the drive is in pulses.

Refer to the drive manual for further detail.

2.

MACHINE PARAMETERS.
Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.3 Configuring a Sercos drive.

SERCOSDATA

Sercos drive data.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Sercos drive.

This parameter shows the table to define the communication with the Sercos drive.

SERCOSDATA	
DRIVEID	Sercos drive address.
OPMODEP	Main operating mode of the Sercos drive.
FBACKSRC	Type of feedback.
FBACKDIFF	Maximum difference between feedbacks.
ULTRAIPO	Parameter not being used at this time.
FBACKTIME	Time constant for combined feedback.

DRIVEID

Sercos drive address.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Sercos drive.

Possible values: From 1 to 32.

Default value: 1.

Associated variable: (V.)[ch].MPA.DRIVEID.xn

Parameter included in the SERCOSDATA table.

This parameter indicates the position (node) occupied by the drive in the Sercos connection.

OPMODEP

Drive's main operating mode.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Sercos drive.

Possible values: Position / Speed.

Default value: Position.

Associated variable: (V.)[ch].MPA.OPMODEP.xn

Parameter included in the SERCOSDATA table.

This parameter indicates the operating mode of the Sercos drive; velocity command or position command. Refer to the drive manual for further detail.

We recommend to set the axes (except tandem axes) to operate in position-Sercos mode and the spindles in velocity-Sercos mode. However, tandem axes must necessarily work in velocity-Sercos mode. All the axes that will be interpolated with the tandem axis must also operate in velocity-Sercos mode.

Operating mode of the drive in velocity-Sercos.

- The velocity command is sent out to the drive in ten-thousandths rpm of the motor. When needing to use the SANALOG from the PLC, its value will also be given in ten-thousandths rpm of the motor.
- The CNC receives the drive's feedback as absolute coordinate in ten-thousandths of a millimeter or thousandths of a degree (same as in Sercos position mode).
- The CNC controls the home search. The home search is carried out without stopping the spindle if it was turning. If it is turning in M03 or M04 and M19 is programmed, the spindle decelerates down to REFFEED1 and starts searching home.
- The following error and the backlash compensation are calculated at the CNC.
- The CNC applies feed-forward and AC-forward.
- When the spindle is turning in open loop, the coordinate is simulated.

Operating mode of the drive in position-Sercos.

- The command is sent out to the drive in absolute coordinates, in ten-thousandths of a millimeter (linear axes) or thousandths of a degree (rotary axes). When needing to use

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

the SANALOG from the PLC, its value will also be given in ten-thousandths of a millimeter or thousandths of a degree. The drive smoothes the command using a cubic interpolation and closes the position loop every 256 microseconds.

- The feedback from the drive to the CNC arrives as absolute coordinate, in ten-thousandths of a millimeter or thousandths of a degree (same as in velocity-Sercos mode).
- The drive controls the home search. Before starting the home search, the spindle stops if it is turning.
- The drive calculates the following error (lag).
- The drive applies feed-forward and AC-forward.

FBACKSRC

Feedback type.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Sercos drive.

Possible values: Internal / External / Internal+External.

Default value: Internal.

Associated variable: (V.)[ch].MPA.FBACKSRC.xn

Parameter included in the SERCOSDATA table.

Type of feedback used to close the position loop. Refer to the drive manual for further detail.

When using internal feedback, the position value is taken from motor feedback whereas when using external feedback, it is taken from direct feedback. When using internal+external feedback, the feedback may be swapped via PLC using the FBACKSEL(axis) mark. When using external or internal+external feedback, we recommend setting parameter FBACKDIFF.

When the type of feedback is external or internal+external, the CNC can enable the combined feedback; parameter FBMIXTIME.

FBACKSRC	FBMIXTIME	Type of feedback.
Internal	(Not being used)	Internal feedback.
External	0	External feedback.
External	Value greater than ·0·.	The combined (mixed) feedback is active.
Internal+External	0	The feedback type may be swapped from the PLC using the mark FBACKSEL(axis). Internal or external feedback.
Internal+External	Value greater than ·0·.	The feedback type may be swapped from the PLC using the mark FBACKSEL(axis). Internal feedback or combined (mixed) feedback.

When the type of feedback is internal+external, the CNC uses the internal feedback (motor feedback) on power-up, when resetting the drive and when initializing the Sercos ring.

Tandem axes do not admit the internal+external type of feedback. In a tandem axis, both axes must have the same type of feedback, external or internal, but usually, the master axis uses external feedback and the slave axis the internal one.

FBACKDIFF

Maximum difference between feedbacks.

Parameter valid for linear and rotary axes.

Parameter valid for a position-Sercos drive.

Possible values: Within ± 99999.9999 mm or degrees / within ± 3937.00787 inch.

Default value: 0 (no monitoring).

Associated variable: (V.)[ch].MPA.FBACKDIFF.xn

Parameter included in the SERCOSDATA table.

The CNC takes this parameter into account only on axes with external or external+internal feedback.

When the system has dual feedback, this parameter may be used to monitor the difference between both feedbacks. If the difference exceeds the set value, the CNC will display the corresponding error message. When defined with a "0" value, there will be no monitoring.

2.

ULTRAIPO
(Not being used at this time)

Parameter not being used at this time.

FBMIXTIME
Time constant for combined (mixed) feedback.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Sercos drive.

Possible values: From 0 to 3200.0 ms.

Default value: 0.

Associated variable: (V)[ch].MPA.FBMIXTIME.xn

Parameter included in the SERCOSDATA table.

This parameter may be used to set the time constant to be used when combining feedbacks; i.e. it sets the delay between the coordinates entered in the position loop of the rotary encoder and that of the linear encoder. The CNC takes this parameter into account only when using external or external+internal feedback.

If the parameter is set to a value other than -0., the combined feedback is enabled; if set to -0., only the external feedback is enabled. When changing this parameter, the parameter validation will force resetting, initializing, the Sercos ring.



This parameter requires a drive version V6.13 or newer.

On machines that have a lot of backlash and use external feedback to obtain greater accuracy may suffer some instability. This kind of machines, with internal feedback, run smoothly, but they may lose accuracy; using external feedback on the other hand, the machine is more accurate but it may run more abruptly. The combination of the two feedbacks achieves the desired result of accuracy and smoothness.

The CNC uses the combined feedback to calculate the velocity command, but it uses the external (direct) feedback to calculate the compensations, circularity test, etc.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.

2.5.4 Configuring a Mechatrolink drive.

MLINKDATA

Data of the Mechatrolink drive.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Mechatrolink drive.

This parameter shows the table to define the communication with the Mechatrolink drive.

MLINKDATA	
DRIVEID	Drive select.
OPMODEP	Main operating mode of the drive.
OPTION	Activate the drive options.

DRIVEID

Drive address.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Mechatrolink drive.

Possible values: From 1 to 14 (Mlink-I) / From 1 to 30 (Mlink-II).

Default value: 1.

Associated variable: (V.)[ch].MPA.DRIVEID.xn

Parameter included in the MLINKDATA table.

This parameter indicates the position (node) occupied by the drive in the Mechatrolink bus. The drives must occupy consecutive positions starting at .1.. Although the relationship between the node number and the logic number of the axis or spindle is irrelevant, for clarity sake we recommend to configure the nodes in the same order the axes and spindles are defined at the CNC.

OPMODEP

Drive's main operating mode.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Mechatrolink drive.

Possible values: Position / Velocity / Inverter.

Default value: Position (axes) / Velocity (spindles).

Associated variable: (V.)[ch].MPA.OPMODEP.xn

Parameter included in the MLINKDATA table.

This parameter indicates the type of drive and the operating mode; servo with a velocity command, servo with a position command or inverter control; the latter is only available for spindles. We recommend to set the servos of the axes to operate in position mode and the spindles in velocity mode. The inverter option must be used when the spindle is connected to this type of device.

OPTION

Activate the drive options.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Mechatrolink drive.

Possible values: From \$00000000 to \$0000FFFF.

Default value: \$00000000.

Associated variable: (V.)[ch].MPA.OPTION.xn

Parameter included in the MLINKDATA table.

This parameter lets configure different options of the Mechatrolink drive, such as activating acc/dec filters, activate the first or second feedback, choose either a P or a PI control for the velocity loop, etc.

The drive options are coded in 16-bit format. Refer to the drive manual for the meaning of each bit.

2.

MACHINE PARAMETERS. Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.5 Hirth axis.

HIRTH

Axis with Hirth tothing.

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.HIRTH.xn

A Hirth axis is the one that can only be positioned at positions multiple of a given value. Even if the position of a Hirth axis does not coincide with its pitch, the axis may be moved to a valid position in both automatic and jog modes. If the position to move the axis does not match its pitch, the CNC will issue an error message.

HPITCH

Hirth axis pitch.

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 99999.9999 mm.

Possible values (2): From 0 to 3937.00787 inch.

Possible values (3): From 0 to 99999.9999 degrees.

Default value: 1 mm / 0.03937 inch / 1 degree.

Associated variable: (V.)[ch].MPA.HPITCH.xn

When the axis is set as Hirth axis, this parameter indicates its pitch. Parameter REFVALUE will be taken as initial coordinate for the next increments.

Any stop or movement in continuous jog will stop the axis in coordinates multiple of HPITCH. Movements in incremental jog will only be valid if they are greater than HPITCH and will be limited to multiples of this value. The positions of the switch for jog movements may be customized at the axis parameter table so they match the desired steps.



The way the decimals of a Hirth axis will be displayed on the screen will be configured using FGUM.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.6 Axis configuration for lathe type machines.

FACEAXIS

Face axis on a lathe.

Parameter valid for a linear axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.FACEAXIS.xn

See axis machine parameters LONGAXIS.

LONGAXIS

Longitudinal axis on a lathe.

Parameter valid for a linear axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.LONGAXIS.xn

On turning machines, you must indicate which one is the longitudinal axis and which one is the cross axis.

Typical lathe setting:		
X axis.	FACEAXIS = Yes	LONGAXIS = No
Z axis.	FACEAXIS = No	LONGAXIS = Yes
Rest of the axes.	FACEAXIS = No	LONGAXIS = No

Typical Mill setting:		
All the axes.	FACEAXIS = No	LONGAXIS = No

These parameters only affect the axes that can work in lathe mode. If it is a combined machine and any or several milling axes can work as lathe, it is necessary to set these parameters. In some cases, the same axis can work as a face axis (FACEAXIS=Yes) and as longitudinal axis (LONGAXIS=Yes) and consequently, both parameters must be set to Yes.

2.

MACHINE PARAMETERS.
Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.7 Synchronization of axes and spindles.

SYNCSET

Set of parameters for synchronization.

Parameter valid for rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 4.

Default value: 1.

Associated variable: (V.)[ch].MPA.SYNCSET.xn

Parameter set assumed by the axis or spindle when it is synchronized. If the parameter is set to ·0· (zero), the CNC will try to synchronize the spindle without forcing the set change.

- A slave axis or spindle always assumes the set defined in its parameter SYNCSET.
- If the master and slave spindles are in the same channel, master spindle assumes the set defined in its parameter SYNCSET. If both spindles are in different channels, the parameter set of the master spindle must be selected before synchronizing.

DSYNVELW

Speed synchronization window.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 200000.0000 mm/min.

Possible values (2): From 0 to 7874.01575 inch/min.

Possible values (3): From 0 to 36000000.0000 degrees/min.

Possible values (4): From 0 to 100000 rpm.

Default value: 100 mm/min / 3.937 inch/min / 3600 degrees/min / 10 rpm.

Associated variable: (V.)[ch].MPA.DSYNVELW.xn

This parameter is defined for the slave element of the synchronization and indicates the velocity margin admitted for the synchronization to be OK.

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 parameter is exceeded, the SYNSPEED signal goes low; the movement is not stopped and no error message is issued.

When synchronizing axes, the slave axis moves at the same feedrate as the master (considering the ratio). If the difference between the synchronism speed calculated for the slave axis and its real speed exceeds the value set in this parameter, it cancels the INSYNC mark of the PLC.

DSYNCPOSW

Position synchronization window.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 99999.9999 mm/min.

Possible values (2): From 0 to 3937.00787 inch/min.

Possible values (3): From 0 to 99999.9999 degrees/min.

Default value: 0.0100 mm/min / 0.00039 inch/min / 0.0100 degrees/min.

Associated variable: (V.)[ch].MPA.DSYNCPOSW.xn

This parameter is defined for the slave element of the synchronization and indicates the position margin admitted for the synchronization to be OK.

When the spindles are synchronized in position, the slave spindle follows the master keeping the programmed offset (bearing the ratio in mind) and it activates the SYNCHRONP mark of the PLC. If the value set in this parameter is exceeded, it cancels the SYNPOSI mark of the PLC; it does not stop the movement or issue any error message.

When synchronizing axes, the slave axis follows the master keeping the offset (considering the ratio). If the difference between the synchronism position calculated for the slave axis and its real position exceeds the value set in this parameter, it cancels the INSYNC mark of the PLC.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.8 Configuration of the rotary axes.

AXISMODE

Work mode of the rotary axis.

Parameter valid for a rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Linearlike / Module.

Default value: Module.

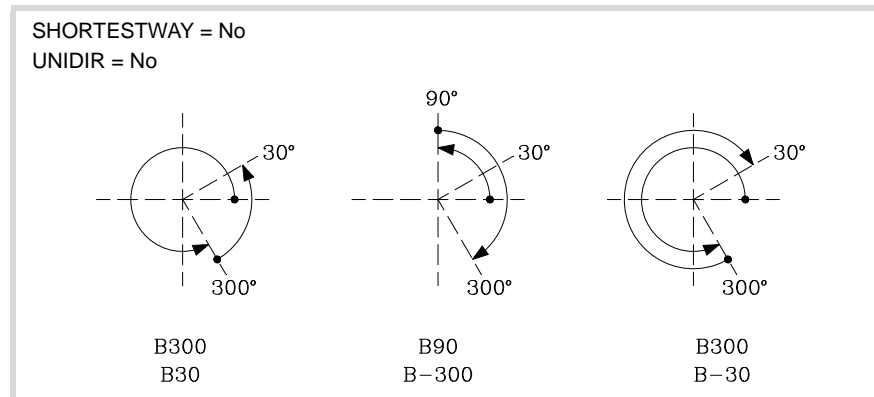
Associated variable: (V.)[ch].MPA.AXISMODE.xn

This parameter indicates how the rotary axis will behave in terms of number of turns or the display of coordinates.

Behavior when AXISMODE = Module.

The axis behaves like a rotary axis. Movements in G0/G1 and G90/G91 may be programmed.

- For movements in G90, more than a complete turn may be programmed or values outside the module, but the whole travel must always be less than a complete turn. If the axis is neither SHORTESTWAY nor UNIDIR, the programmed sign indicates the turning direction and the absolute value of the coordinate indicates the target position.



- In movements in G91, the sign indicates the turning direction and the absolute value of the coordinate indicates the distance to move.

The module limits (travel limits of the rotary axis) are set by set parameters MODUPLIM and MODLOWLIM. For analog and simulated axes, the lower limit MODLOWLIM must be lower than the upper limit MODUPLIM, for example 0° to 360°, 0° to 400° or -230 to 95; it is not possible, for example -100° to -230° or 360° to 0°. For Sercos axes, the module limits must be 0° and 360°.

The coordinates are always displayed within the module limits, by default 0 and 360°.

Parameters SHORTESTWAY and UNIDIR must be set. Parameters LIMIT+ and LIMIT- have not meaning.

Behavior when AXISMODE = Linearlike.

It behaves like a linear axis. Movements in G0/G1 and G90/G91 may be programmed.

The reading is free and in degrees (not affected by mm/inch). There are travel limits set by "LIMIT+" and "LIMIT-".

Parameters SHORTESTWAY, UNIDIR and those for set MODUPLIM and MODLOWLIM do not apply.

2.

MACHINE PARAMETERS.
Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

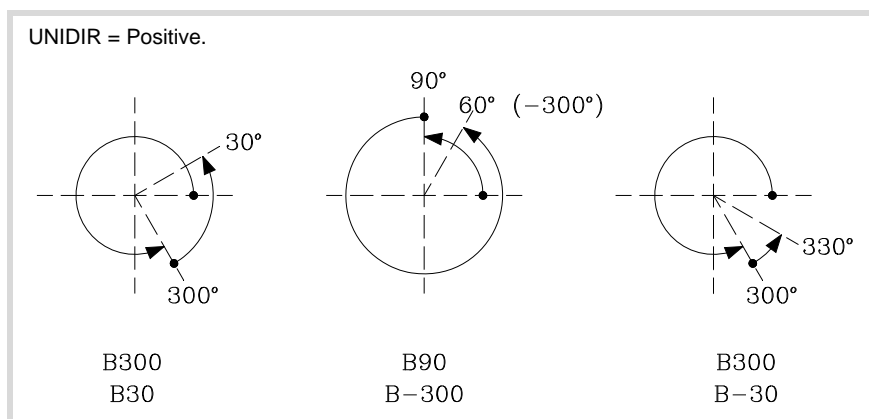
(REF: 1709)

UNIDIR

Single turning direction.

Parameter valid for a rotary axes.
 Parameter valid for analog drive, Sercos and simulated.
 Possible values: No (both directions) / Positive / Negative.
 Default value: No (both directions).
 Associated variable: (V)[ch].MPA.UNIDIR.xn

The CNC takes this parameter into consideration only when AXISMODE = Module and SHORTESTWAY = No. This parameter indicates whether G00/G01 movements of the rotary axes in G90 may be carried out in both directions or they must always turn in the same direction (either positive or negative). If the axis is not UNIDIR, the programmed sign indicates the turning direction and the absolute value of the coordinate indicates the target position.



The movements in G91 are carried out in the programmed direction. If it is a UNIDIR axis, the programmed direction must be the same as the one preset for the axis; otherwise an error message will be issued because the axis cannot turn in the opposite direction. Likewise, the error will also come up when programming a mirror image on these axes.

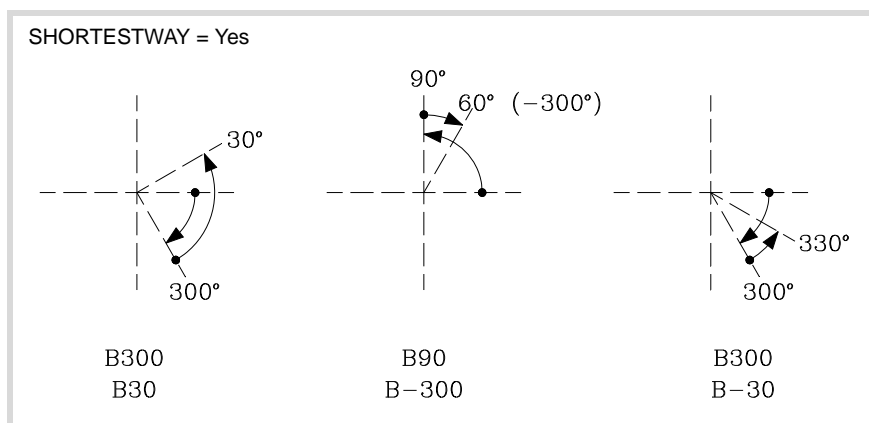
SHORTESTWAY

Via the shortest way.

Parameter valid for a rotary axes.
 Parameter valid for analog drive, Sercos and simulated.
 Possible values: Yes / No.
 Default value: No.
 Associated variable: (V)[ch].MPA.SHORTESTWAY.xn

The CNC takes this parameter into consideration only when AXISMODE = Module and UNIDIR = No.

This parameters indicates whether the rotary axis movements in G90 are carried out via the shortest way or not. Otherwise, the programmed sign will indicate the turning direction whereas the absolute value of the coordinate will indicate the target position.



The movements in G91 are carried out in the programmed direction.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.9 Module configuration (rotary axes and spindle).

MODCOMP

Module compensation.

Parameter valid for rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No (without compensation).

Associated variable: (V.)[ch].MPA.MODCOMP.xn

The CNC takes this parameter into consideration only when AXISMODE = Module.

The module compensation must be activated when the axis resolution is not exact. Range parameters MODNROT and MODERR set the compensation to be applied to obtain the exact reading. The CNC applies module compensation throughout the entire revolution.

2.**MACHINE PARAMETERS.**

Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.10 Configuration of the C axis.

CAXIS

Possibility to work as a C axis.

Parameter valid for rotary axes and spindles.

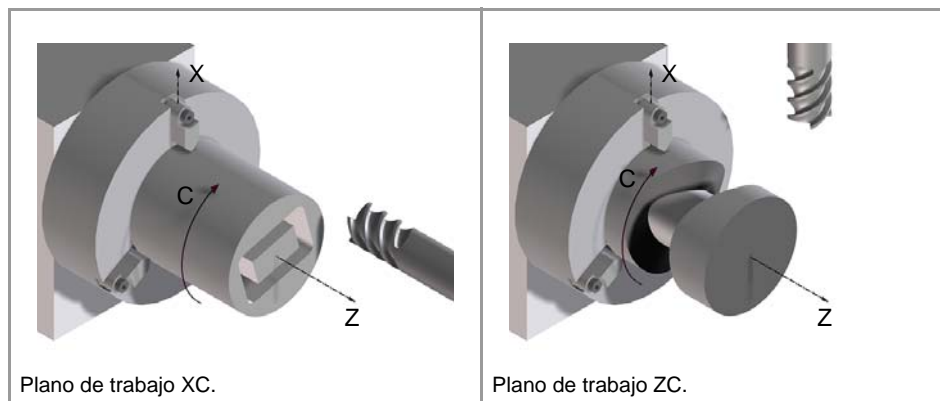
Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.CAXIS.xn

This parameter indicates whether the axis or spindle can work as C axis.



CAXSET

Set of parameters to work as C axis.

Parameter valid for rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 1 to 4.

Default value: 1.

Associated variable: (V.)[ch].MPA.CAXSET.xn

The CNC takes this parameter into consideration only when CAXIS = Yes. This parameter indicates which set of parameters (NPARSETS) the axis or spindle uses when it works as C axis.

PERCAX

"C" axis kept active.

Parameter valid for rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.PERCAX.xn

The CNC takes this parameter into consideration only when CAXIS = Yes. This parameter indicates whether the CNC keeps the C axis active or not after executing M02, M30 or after an emergency or reset. After turning the CNC off, the CNC deactivates the C axis.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.11 Configuration of the spindle.

AUTOGEAR

Automatic gear change.

Parameter valid for spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.AUTOGEAR.xn

This parameter indicates whether the gear change is automatically generated by activating (if necessary) the auxiliary functions M41, M42, M43 and M44 when programming the speed.

LOSPDLIM

Lower "rpm OK" percentage.

Parameter valid for spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 255.

Default value: 50.

Associated variable: (V.)[ch].MPA.LOSPDLM.xn

See axis machine parameters UPSPDLIM.

UPSPDLIM

Upper "rpm OK" percentage.

Parameter valid for spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 255.

Default value: 150.

Associated variable: (V.)[ch].MPA.UPSPDLIM.xn

When working with M3 and M4, the REVOK signal is set high when the actual spindle rpm are between these percentages.

The REVOK signal may be used to manage the Feedhold signal and avoid machining at lower or higher rpm than the ones programmed.

SPDLTIME

Estimated time for an S function.

Parameter valid for spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 1000000 ms.

Default value: 0 ms.

Associated variable: (V.)[ch].MPA.SPDLTIME.xn

In EDISIMU mode, there is an option that allows calculating the time required to execute a part with the machining conditions established in the program.

To fine tune that calculation, one may define this parameter that indicates the estimated time for processing the S function.

When assigning a value other than "0", the CNC interprets that the "S" value must be passed on to the PLC using signals SSTROBE + SFUN1.

SPDLSTOP

Functions M2 and M30, an error or a reset stop the spindle.

Parameter valid for spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)[ch].MPA.SPDLSTOP.xn

This parameter indicates whether the spindle is stopped or not when executing M02, M30, when doing a reset or when an error occurs. Otherwise, it will be necessary to program function M5.

The spindle errors and an emergency always stop the spindle.

2.

MACHINE PARAMETERS.
Machine parameters for the axes and spindles.

SREVM05

With G84, the spindle must be stopped to change the turning direction.

Parameter valid for spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.SREVM05.xn

This parameter indicates whether the spindle must be stopped (with M5) when reversing the spindle in tapping cycle.

M19SPDLEREV

The SPDLEREV mark (reverse turning direction) affects the spindle in M19.

Parameter valid for spindles.

Parameter valid for analog drives and velocity-Sercos drives.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.M19SPDLEREV.xn

This parameter defines whether the PLC mark SPDLEREV (reverse turning direction) also affects the spindles in M19. The SPDLEREV mark always affects the spindles in open loop (M3/M4), regardless of how this parameter has been set.

STEPOVR

Spindle override step.

Parameter valid for spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 255.

Default value: 5.

Associated variable: (V.)[ch].MPA.STEPOVR.xn

This parameter sets the incremental step used to change the programmed spindle speed with the spindle override keys of the operator panel. Parameter ignored when the operator panel has a spindle speed override switch.

MINOVR

Minimum override (%) permitted for the spindle.

Parameter valid for spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 255.

Default value: 50.

Associated variable: (V.)[ch].MPA.MINOVR.xn

This parameter sets the minimum percentage allowed to apply to the spindle speed when it is changed from the operator panel (keys or speed override switch).

MAXOVR

Maximum override (%) permitted for the spindle.

Parameter valid for spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 255.

Default value: 150.

Associated variable: (V.)[ch].MPA.MAXOVR.xn

This parameter sets the maximum percentage allowed to apply to the spindle speed when it is changed from the operator panel (keys or speed override switch).

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.12 Spindle override change while threading.

THREADOVR

Maximum variation allowed for the override while threading.

Parameter valid for spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: from 0 to 100%.

Default value: 0 (The override cannot be changed).

Associated variable: (V.)[ch].MPA.THREADOVR.xn

If this parameter is set with a value other than 0, it is possible to change the spindle override during electronic threading (G33) and in the threading canned cycles of the T model (G86, G87 and their equivalent of the cycle editor).

The parameter sets the maximum spindle override variation possible, both to increase it and to reduce it. For example, if set to 30, the override may be varied between 80% and 130%. The limits set by machine parameters MINOVR and MAXOVR can never be exceeded.

In order to avoid damaging the thread when varying the spindle override, you should use a feed-forward value close to 100% on the axes involved in threading so the following error is as small as possible. The CNC allows modifying the spindle override while threading if it detects that the feed forward (parameter FFWTYPE) is not active in a gear of the axes involved or if the active feed forward is lower than 90%. Feed forward is defined with parameter FFGAIN and may be modified from the PLC using variables.

Threading canned cycles of the T model. Considerations related to changing the override value.

- In threading cycles, it is possible to change the spindle speed during the threading passes, except in those done all the way to the final depth of the thread (one or two final passes) which are done at the override value active at the beginning of the pass.
- Even if the override is changed, the CNC respects the pitch and entry of the thread.
- It is recommended not to change the override on threads with flank infeed.

Electronic threading G33. Considerations related to changing the override value.

- 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.

OVRFILTER

Time to make the override change effective.

Parameter valid for spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 1000000 ms.

Default value: 0.

Associated variable: (V.)[ch].MPA.OVRFILTER.xn

The CNC takes this parameter into account only if THREADOVR other than 0 (zero).

This parameter acts as a filter to make the override change effective. The override change is applied progressively during the indicated period of time.

2.

MACHINE PARAMETERS. Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.13 Software axis limits.

LIMIT+

Positive software limit.

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): Within ± 99999.9999 mm.

Possible values (2): Within ± 3937.00787 inch.

Possible values (3): Within ± 99999.9999 degrees.

Default value: 99999.9999 mm / 3937.00787 inch / 99999.9999 degrees.

Associated variable: (V.)[ch].MPA.POSLIMIT.xn

See axis machine parameters LIMIT-.

LIMIT-

Negative software limit.

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): Within ± 99999.9999 mm.

Possible values (2): Within ± 3937.00787 inch.

Possible values (3): Within ± 99999.9999 degrees.

Default value: -99999.9999 mm / -3937.00787 inch / -99999.9999 degrees.

Associated variable: (V.)[ch].MPA.NEGLIMIT.xn

On rotary axes, the CNC takes this parameter into account when AXISMODE = Linearlike.

On linear and rotary axes, these parameters set the axis travel limits. If both are set to "0", there will be no software limits.



The software limits are always set in radius, regardless of the setting of parameter DIAMPROG.

SWLIMITTOL

Software limit tolerance.

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): Within ± 99999.9999 mm.

Possible values (2): Within ± 3937.00787 inch.

Possible values (3): Within ± 99999.9999 degrees.

Default value: 0.1000 mm / 0.00394 inch / 0.1000 degrees.

Associated variable: (V.)[ch].MPA.SWLIMITTOL.xn

This parameter indicates the maximum variation or oscillation allowed for an axis located at the limit.

This parameter indicates the maximum variation or oscillation allowed to a real coordinate of an axis referred to the software limits, before issuing an error indicating travel limit overrun. The programmed theoretical movement of the axis is only possible up to the exact limit, but the real axis coordinate is allowed this margin before the error is issued.

When they are DRO axes, the error is also issued when the real coordinate exceeds the limit over the tolerance.

When no theoretical movement has been programmed, the limit overrun error will only be issued when exceeding the tolerance in a sampling period (cycle time); for example, when hitting the axis causing it to overrun the limits abruptly. In any other case, if no theoretical movement has been programmed for the axis, the error will not be issued even if it overruns the limits.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.

FAGOR

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.14 Work zones.

ZONELIMITTOL

Safety distance of the limits of the work zones.

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 99999.9999 mm or degrees / from 0 to 3937.00787 inch.

Default value: 0.1000 mm or degrees / 0.00394 inch.

Associated variable: (V.)[ch].MPA.ZONELIMITTOL.xn

Safety distance applied by the CNC to the axis with respect to the limit of the work zone. When a work zone is active, the CNC stops the axes when one of them reaches the safety distance. The safety distance is defined per axis and each axis will have the same safety distance for all the zones..

The safety distance may be modified with the following variable which will assume the value of this parameter on startup.

(V.)[ch].A.ZONELIMITTOL.xn Safety distance of the limits of the work zones.

2.5.15 Runaway protection and tendency test.

TENDENCY

Activation of tendency test.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and simulated drives.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.TENDENCY.xn

This parameter enables two tests:

- Axis tendency watch.

This test watches that the real movement coincides with the theoretical movement, from the time period set in parameter ESTDELAY. If no theoretical movement has been programmed for the axis, the test detects whether the axis is moving in one direction longer than the time indicated in parameter ESTDELAY. If no theoretical movement has been programmed for the axis, the test detects whether it is moving in the opposite direction longer than the time indicated by ESTDELAY.

- Runaway protection.

This test watches the real movement of the axis from startup to detect if runs away considering the time defined in the parameter TENDTIME.

If this alarm is canceled, the CNC shows on power-up a message indicating that this safety is disabled. This situation can only be allowed during setup; once setup is completed, this alarm must be enabled.

TENDTIME

Time to detect axis runaway.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and simulated drives.

Possible values: From 0 to 65535 ms.

Default value: 0.

Associated variable: (V.)[ch].MPA.TENDTIME.xn

When the runaway test is active, this parameter defines the time the CNC must wait before issuing an error message. We recommend to set a short time (about 4 sampling periods) to prevent the axis from moving too far.

2.

MACHINE PARAMETERS.
Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.16 PLC Offset.

PLCOINC

PLC offset increment per cycle.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 99999.9999 mm/min.

Possible values (2): From 0 to 3937.00787 inch/min.

Possible values (3): From 0 to 99999.9999 degrees/min.

Default value: 0 (they are assumed instantaneously).

Associated variable: (V.)[ch].MPA.PLCOINC.xn

The PLC offset is an additional compensation over the real coordinate of the axis. The CNC applies this compensation in a transparent way for the user who will not see it reflected in the coordinates. A typical utility is to correct the axis dilatations due to temperature.

This parameter indicates whether the CNC assumes the PLC offsets instantaneously in a sampling period (cycle time) or in steps.

Example:

It is set PLCOINC = 0.001 mm (one micron per CNC cycle). If the PLC Offset had an initial value of 0.25 mm and the new value is 0.30 mm, the PLC offset applied per cycle will be:

0.250 0.251 0.252 0.253 . . . 0.297 0.298 0.299 0.300

The PLC offset to be applied is set in the variable (V.)A.PLCOF.xn. The PLC offset applied so far may be checked in the variable (V.)A.ACTPLCOF.xn.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.

2.5.17 Dwell for dead axes.

DWELL

Dwell for dead axes.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

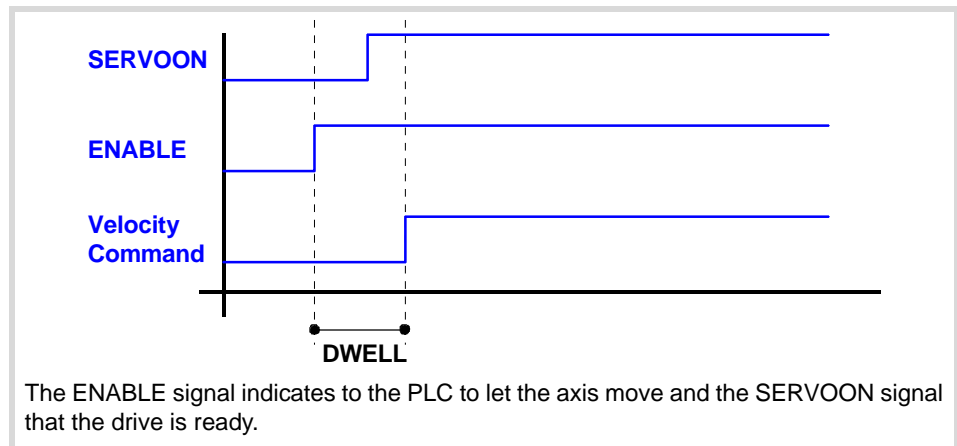
Possible values: From 0 to 1000000 ms.

Default value: 0 (no dwell).

Associated variable: (V.)[ch].MPA.DWELL.xn

When an axis has a brake, for example very heavy vertical axes, it is only governed while it is moving. When it is governed by the CNC (movements), it is referred to as being alive and, when not moving (brake on), it is referred to as being "dead".

To bring it to "life", release the brake and close the position loop. The time required for this operation must be defined by the DWELL parameter.



The CNC waits the amount of time set in parameter DWELL before generating a movement on the "dead" axis if it does not have the SERVOON signal active. Once the dwell has started, the CNC will wait the time set in parameter DWELL before starting the movement, even if the SERVOON signal has been activated before.

Special care must be taken with the delayed disconnection of the axis, when it becomes a "dead" axis. If the SERVOON signal has been turned off and a new movement of the axis begins, the CNC will not start timing but it could issue an error indicating that the axis is locked up if as a result of disconnecting logically, the SERVOON signal ends up dropping.

On a tandem axis, both the master and slave axes must be enabled so they can be moved. In this case, the CNC only applies the time set in DWELL to the SERVOON of the master axis; if the enabling of the slave axis is slower, the PLC maneuver must verify that both axes are enabled before generating the movement.

If an axis is going to work as a "dead" axis in a program with path joints, G5, G50 or HSC, it will be necessary to enable the axis via PLC as "dead" axis (DEAD(axis) signal). This way, the CNC knows at all times that it will have to apply a dwell before every movement of that axis.

2.

MACHINE PARAMETERS.
Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.18 Radius / diameter.

DIAMPROG

Programming in diameters.

Parameter valid for a linear axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.DIAMPROG.xn

The CNC takes this parameter into consideration only when FACEAXIS = Yes.

On turning machines, the coordinates of the cross axis may be programmed in either radius or diameter. To change the type of coordinates via program, use function G151 or G152.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.19 Home search.

2.

MACHINE PARAMETERS.
Machine parameters for the axes and spindles.

REFDIREC Homing direction.

*Parameter valid for linear and rotary axes and spindles.
Parameter valid for analog drive, Sercos and simulated.
Possible values: Negative / Positive.
Default value: Positive.
Associated variable: (V.)[ch].MPA.REFDIREC.xn*

For analog and simulated axes, parameter REFDIREC indicates the axis moving direction to begin the home search. For a Sercos axis, parameter REFDIREC indicates the motor turning direction, which does not necessary have to be the same as the positive or negative feedback reading (pulse counting) direction of the axis.

This parameter has no effect on spindles that are stopped and have no home switch. If the spindle is turning, the CNC respects its turning direction to start searching home. If the spindle has a home switch and is turning, the CNC stops the spindle and starts the home search in the direction indicated by REFDIREC.

REFMODE Home search mode.

*Parameter valid for linear and rotary axes and spindles.
Parameter valid for a position-Sercos drive.
Possible values: Drive / CNC.
Default value: CNC.
Associated variable: (V.)[ch].MPA.REFMODE.xn*

This parameter indicates who manages the axis home search.

Value.	Meaning.
Drive.	The drive homes the axis as if it were a position-Sercos axis.
CNC.	The drive homes the axis as if it were a velocity-Sercos axis. The CNC executes the parameter matching as if it were a velocity-Sercos axis.

DECINPUT The axis/spindle has a home switch.

*Parameter valid for linear and rotary axes and spindles.
Parameter valid for analog drives and Sercos drives.
Possible values: Yes / No.
Default value: Yes.
Associated variable: (V.)[ch].MPA.DECINPUT.xn*

If the axis has a home search but no distance-coded reference marks, the axis moves up to the home switch, then it reverses the movement and when it releases the home switch, the home switch is enabled. If the axis has a home switch and distance-coded reference marks, the axis needs not reach the home switch in order to find the reference mark.

On spindles with home switch, the home search algorithm runs two passes over the home switch. The first pass is run at the speed set in REFFFEED1 and it serves to calculate the position of the home switch. The second pass is run at the same speed until reaching the home switch, going over it at the speed set in REFFFEED2 and doing the usual home search.

When there is a home switch, the CNC does not search home on the go, depending on whether it is turning in M3 or M4; the home search is always done in the direction set by REFDIREC.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

REFINI

Home search in the first movement.

Parameter valid for spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)[ch].MPA.REFINI.xn

This parameter determines whether the CNC homes the spindle in its first movement or not. The CNC takes this parameter into account only when parameters NPULSES y NPULSES2 have been set with a value other than 0.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.20 Configuration of the probing movement.

PROBEAXIS

The axis can participate in probing movements.

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.PROBEAXIS.xn

This parameter indicates whether the axis is involved in a probing move (G100).

PROBERANGE

Maximum braking distance.

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 99999.9999 mm/min.

Possible values (2): From 0 to 3937.00787 inch/min.

Possible values (3): From 0 to 99999.9999 degrees/min.

Default value: 1.0000 mm / 0.03937 inch / 1.0000 degree.

Associated variable: (V.)[ch].MPA.PROBERANGE.xn

This parameter sets the maximum braking distance for the probe after probing to avoid breaking it (ceramic, etc). The CNC issues an error messages when this distance is exceeded.

PROBEFEED

Maximum probing feedrate.

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 36000000.0000 mm/min.

Possible values (2): From 0 to 1417322.83465 inch/min.

Possible values (3): From 0 to 36000000.0000 degrees/min.

Default value: 100.0000 mm/min / 3.93701 inch/min / 100.0000 degrees/min.

Associated variable: (V.)[ch].MPA.PROBEFEED.xn

The value of this parameter must be smaller than the feedrate needed to brake within the distance set by PROBERANGE with the acceleration and jerk values of the axis. Otherwise, it will show a warning when validating the axis parameters indicating the maximum feedrate that may be reached.

PROBEDELAY

Delay for the "probe 1" signal.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Within ± 100000.0000 ms.

Default value: 0 (no delay).

Associated variable: (V.)[ch].MPA.PROBEDELAY.xn

See axis machine parameters PROBEDELAY2.

PROBEDELAY2

Delay for the "probe 2" signal.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Within ± 100000.0000 ms.

Default value: 0 (no delay).

Associated variable: (V.)[ch].MPA.PROBEDELAY2.xn

Parameter PROBEDELAY corresponds to the probe set by PRBID1 and PROBEDELAY2 to the probe set by PRBID2.

In some types of probes, there is a short delay of a few milliseconds from the probing instant to when the CNC actually receives the signals (infrared communication, etc.). In these cases,

2.

MACHINE PARAMETERS.
Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

it must indicate the time elapsed from when the probing takes place till the CNC receives the signal.

Probe calibration cycle "#PROBE 2" may be used to set this parameter. After it is executed, the cycle returns, in arithmetic parameter P298 the best value to be assigned to parameter PROBEDELAY for the axes and the spindle.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.

2.5.21 Repositioning of the axes in tool inspection.

REPOSFEED

Maximum repositioning feedrate.

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 200000.0000 mm/min.

Possible values (2): From 0 to 7874.01575 inch/min.

Possible values (3): From 0 to 36000000.0000 degrees/min.

Default value: 0.

Associated variable: (V.)[ch].MPA.REPOSFEED.xn

Repositioning feedrate after a tool inspection or an interruption subroutine. If not defined, the CNC assumes as repositioning feedrate the one defined for the continuous jog mode (parameter JOGFEED). The value of parameter REPOSFEED must always be lower than G00FEED, MAXMANFEED and JOGRAPFEED.

2.5.22 Configuration of the independent axis.

POSFEED

Positioning feedrate.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 200000.0000 mm/min.

Possible values (2): From 0 to 7874.01575 inch/min.

Possible values (3): From 0 to 36000000.0000 degrees/min.

Possible values (4): From 0 to 100000.0000 rpm.

Default value: 1000.0000 mm/min / 39.37008 inch/min / 1000.0000 degrees/min / 1000 rpm.

Associated variable: (V.)[ch].MPA.POSFEED.xn

Positioning feedrate of the independent axis.

2.

MACHINE PARAMETERS.
Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.23 Configure the maximum safety limit for the feedrate and for the speed.

FLIMIT

Maximum safety limit for axis feedrate.

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 500000.0000 mm/min.

Possible values (2): From 0 to 19685.03937 inch/min.

Possible values (3): From 0 to 7200000.0000 degrees/min.

Default value: 0 (there is no limitation).

Associated variable: (V.)[ch].MPA.FLIMIT.xn

This parameter sets the maximum safety limit for axis feedrate to be applied, for example, when performing maintenance on the machine, operations with disabled safety (open doors), etc. The set value is activated via PLC (marks FLIMITAC or FLIMITACCHC1) and the CNC applies it on the block being executed. When deactivating the limits, the CNC restores the programmed feedrate.

If the parameter is set with a "0" value, the feedrate is not limited. The feedrate safety limit is applied to the movements in automatic (G0, G1, etc.) and in jog mode (jog, handwheels, etc.). This parameter affects neither the threading operations nor the independent axis movements, which are executed at the programmed feedrate.

When there is a maximum feedrate set by PLC (variable (V.)PLC.G00FEED), the CNC applies the most restrictive feedrate.

SLIMIT

Maximum safety limit for spindle speed.

Parameter valid for spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 200000.0000 rpm.

Default value: 0 (there is no limitation).

Associated variable: (V.)[ch].MPA.SLIMIT.sn

This parameter sets the maximum safety limit for spindle speed to be applied, for example, when performing maintenance on the machine, operations with disabled safety (open doors), etc. The set value is activated via PLC (marks SLIMITAC or SLIMITACSPDL) and the CNC applies it on the block being executed. When deactivating the limits, the CNC restores the programmed speed.

If the parameter is set with a "0" value, the speed is not limited. The safety limit is also applied to the spindles controlled by PLC (PLCCNTL mark), except when the spindle is digital and controlled in position.

It is also possible to set the maximum machining turning speed by program (function G192) or by PLC (variable (V.)PLC.SL.sn). When a maximum machining speed has been set, the CNC applies the most restrictive speed limit; either the one set for machining or the safety one.

Safety speed limit.	Machining speed limit.	Active speed limit.
0	100	100
50	0	50
50	100	50
150	100	100

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.24 JOG mode.

MANUAL**Manual operating mode parameters.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

This parameter shows the parameter table to define the movements in jog mode. On spindles, these parameters will only be applied when the spindle is interpolated with axes during rigid tapping or when the spindle works as C axis.

MANUAL	
JOGFEED	Feedrate in continuous jog mode.
JOGRAPFEED	Rapid feedrate in continuous jog mode.
MAXMANFEED	Maximum feedrate in continuous jog mode.
MAXMANACC	Maximum acceleration in continuous JOG mode.
INCJOGDIST	Distance to travel in incremental jog.
INCJOGFEED	Feedrate in incremental jog.
MPGRESOL	Handwheel resolution.
MPGFILTER	Filter time for the handwheel.
MANPOSSW	Maximum positive travel with G201.
MANEGSW	Maximum negative travel with G201.
MANFEEDP	Maximum % of jog feedrate in G201.
IPOFEEDP	Maximum % of execution feedrate in G201.
MANACCP	Maximum % of jog acceleration in G201.
IPOACCP	Maximum % of execution acceleration in G201.

JOG MODE. CONTINUOUS JOG.**JOGFEED****Feedrate in continuous jog.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 200000.0000 mm/min.

Possible values (2): From 0 to 7874.01575 inch/min.

Possible values (3): From 0 to 36000000.0000 degrees/min.

Default value: 1000.0000 mm/min / 39.37008 inch/min / 1000.0000 degrees/min.

Associated variable: (V.)[ch].MPA.JOGFEED.xn

Parameter included in the JOG table.

This parameter sets the feedrate for movements in continuous jog mode.

JOGRAPFEED**Rapid feedrate in continuous jog.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 200000.0000 mm/min.

Possible values (2): From 0 to 7874.01575 inch/min.

Possible values (3): From 0 to 36000000.0000 degrees/min.

Default value: 10000.0000 mm/min / 393.70079 inch/min / 10000.0000 degrees/min.

Associated variable: (V.)[ch].MPA.JOGRAPFEED.xn

Parameter included in the JOG table.

This parameter sets the rapid feedrate for movements in continuous jog mode.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

MAXMANFEED

Maximum feedrate in continuous jog.

Parameter valid for linear and rotary axes and spindles.
 Parameter valid for analog drive, Sercos and simulated.
 Possible values (1): From 0 to 200000.0000 mm/min.
 Possible values (2): From 0 to 7874.01575 inch/min.
 Possible values (3): From 0 to 36000000.0000 degrees/min.
 Default value: 10000.0000 mm/min / 393.70079 inch/min / 10000.0000 degrees/min.
 Associated variable: (V.)[ch].MPA.MAXMANFEED.xn
 Parameter included in the JOG table.

This parameter sets the maximum feedrate for movements in continuous jog mode.

MAXMANACC

Maximum acceleration in continuous JOG.

Parameter valid for linear and rotary axes and spindles.
 Parameter valid for analog drive, Sercos and simulated.
 Possible values (1): From 0.0010 to 600000000.0000 mm/s².
 Possible values (2): From 0.00004 to 23622047.2440 inch/s².
 Possible values (3): From 0.0010 to 600000000.0000 degrees/s².
 Default value: 10000.0000 mm/s² / 393.70079 inch/s² / 10000.0000 degrees/s².
 Associated variable: (V.)[ch].MPA.MAXMANACC.xn
 Parameter included in the JOG table.

This parameter sets the maximum acceleration for movements in continuous jog mode.

JOG MODE. INCREMENTAL JOG.

INCJOGDIST

Table for incremental jog distances.

Parameter valid for linear and rotary axes and spindles.
 Parameter valid for analog drive, Sercos and simulated.

This parameter shows the table to define the axis movements in each position of the jog selector switch. The table shows five parameters, one per each position of the operator panel.

- INCJOGDIST1 Switch position ·1·.
- INCJOGDIST2 Switch position ·10·.
- INCJOGDIST3 Switch position ·100·.
- INCJOGDIST4 Switch position ·1000·.
- INCJOGDIST5 Switch position ·10000·.

INCJOGDIST

Distance traveled in incremental jog.

Parameter valid for linear and rotary axes and spindles.
 Parameter valid for analog drive, Sercos and simulated.
 Possible values (1): From 0.0001 to 99999.9999 mm.
 Possible values (2): From 0.0000 3937.00787 inch.
 Possible values (3): From 0.0001 to 99999.9999 degrees.
 Associated variable: (V.)[ch].MPA.INCJOGDIST[pos].xn

In each parameter, you must set the distance the axis moves every time the jog key is pressed. The most typical values are those set by default.

Parameter.	Distance.
INCJOGDIST1	0.0010 mm or degrees / 0.00003937 inch.
INCJOGDIST2	0.0100 mm or degrees / 0.00039370 inch.
INCJOGDIST3	0.1000 mm or degrees / 0.00393700 inch.
INCJOGDIST4	1.0000 mm or degrees / 0.03937007 inch.
INCJOGDIST5	10.000 mm or degrees / 0.39370078 inch.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

MACHINE PARAMETERS.
Machine parameters for the axes and spindles.

INCJOGFEED**Table for incremental jog feedrates.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

This parameter shows the table to define the axis feedrate in each position of the jog selector switch. The table shows five parameters, one per each position of the operator panel.

INCJOGFEED1	Switch position ·1·.
INCJOGFEED2	Switch position ·10·.
INCJOGFEED3	Switch position ·100·.
INCJOGFEED4	Switch position ·1000·.
INCJOGFEED5	Switch position ·10000·.

INCJOGFEED n**Feedrate in incremental jog.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 1 to 200000.0000 mm/min.

Possible values (2): From 0.03937 to 7874.01575 inch/min.

Possible values (3): From 1 to 200000.0000 degrees/min.

Default value: 1000.0000 mm/min / 39.37008 inch/min / 1000.0000 degrees/min.

Associated variable: (V.)[ch].MPA.INCJOGFEED[pos].xn

In each parameter, you must set the axis feedrate.

JOG MODE. HANDWHEELS.**MPGRESOL****Table for handwheel resolution.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

This parameter shows the table to define the resolution of the handwheel in each position of the jog selector switch. The table shows three parameters, one per each position of the operator panel.

MPGRESOL1	Switch position ·1·.
MPGRESOL2	Switch position ·10·.
MPGRESOL3	Switch position ·100·.

MPGRESOL n**Handwheel resolution in each switch position.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): Within ± 99999.9999 mm ($\cdot 0$ not admitted).

Possible values (2): Within ± 3937.00787 inch ($\cdot 0$ not admitted).

Possible values (3): Within ± 99999.9999 degrees ($\cdot 0$ not admitted).

Associated variable: (V.)[ch].MPA.MPGRESOL[pos].xn

In each parameter, you must set the distance the axis moves per feedback pulse provided by the handwheel. If the resolution is negative, the moving direction is reversed (movement in the opposite direction to that indicated by the A and B signals of the handwheel). The most typical values are those set by default.

Parameter.	Resolution.
MPGRESOL1	0.0010 mm or degrees / 0.00003937 inch.
MPGRESOL2	0.0100 mm or degrees / 0.00039370 inch.
MPGRESOL3	0.1000 mm or degrees / 0.00393700 inch.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Example:

The handwheel has a graduated disk with 100 lines and we would like a feed of 0.001 mm per line at switch position -1.

- A 100-line handwheel provides 1 pulse per line.
MPGRESOL1 = 0.0010 mm.
- A 200-line handwheel provides 2 pulses per line.
MPGRESOL1 = 0.0005 mm.
- A 25-line handwheel provides 1 pulse every 4 lines.
MPGRESOL1 = 0.0040 mm.

MPGFILTER**Filter time for the handwheel.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 1 to 1000.

Default value: 10.

Associated variable: (V.)[ch].MPA.MPGFILTER.xn

This filter smoothes the handwheel movements avoiding sudden variations. This parameter indicates the number of CNC cycles used to are the handwheel pulses read.

JOG MODE. MANUAL INTERVENTION.**MANPOSSW****Maximum positive travel with G201.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): Within ±99999.9999 mm.

Possible values (2): Within ±3937.00787 inch.

Possible values (3): Within ±99999.9999 degrees.

Default value: 99999.9999 mm / 3937.00787 inch / 99999.9999 degrees.

Associated variable: (V.)[ch].MPA.MANPOSSW.xn

Parameter included in the JOG table.

See axis machine parameters MANNEGSW.

MANNEGSW**Maximum negative travel with G201.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): Within ±99999.9999 mm.

Possible values (2): Within ±3937.00787 inch.

Possible values (3): Within ±99999.9999 degrees.

Default value: -99999.9999 mm / -3937.00787 inch / -99999.9999 degrees .

Associated variable: (V.)[ch].MPA.MANNEGSW.xn

Parameter included in the JOG table.

When using function G201, Manual mode laid over the Automatic mode, these parameters indicate how far the axis may be moved in both directions.

MANFEEDP**Maximum % of jog feedrate in G201.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 100.

Default value: 20.

Associated variable: (V.)[ch].MPA.MANFEEDP.xn

Parameter included in the JOG table.

See axis machine parameters IPOFEEDP.

2.**MACHINE PARAMETERS.**

Machine parameters for the axes and spindles.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

MACHINE PARAMETERS.
Machine parameters for the axes and spindles.

IPOFEEDP**Maximum % of execution feedrate in G201.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 100.

Default value: 80.

Associated variable: (V.)[ch].MPA.IPOFEEDP.xn

Parameter included in the JOG table.

When using function G201, Manual mode laid over the Automatic mode, both parameters indicate the maximum feedrate to be used in each mode. The sum of both parameters should not exceed 100 in order so as not to exceed the dynamic limits of the machine under certain conditions.

When activating function G201, the feedrate assumes instantaneously the value set by IPOFEEDP.

MANACCP**Maximum % of jog acceleration in G201.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 100.

Default value: 20.

Associated variable: (V.)[ch].MPA.MANACCP.xn

Parameter included in the JOG table.

See axis machine parameters IPOACCP.

IPOACCP**Maximum % of execution acceleration in G201.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 100.

Default value: 80.

Associated variable: (V.)[ch].MPA.IPOACCP.xn

Parameter included in the JOG table.

When using function G201, Manual mode laid over the Automatic mode, both parameters indicate the maximum acceleration to be used in each mode. The sum of both parameters should not exceed 100 in order so as not to exceed the dynamic limits of the machine under certain conditions.

When activating function G201, the feedrate assumes instantaneously the value set by IPOACCP.

Considering the following values for the Y axis:

G00FEED: 1000 mm/min.

JOGFEED: 100 mm/min.

MAXMANFEED: 120 mm/min.

IPOFEEDP: 50%

MANFEEDP: 50%

When executing the following blocks:

N10 G201 #AXIS [Y]

N20 G1 Y100 F1000

At block N20, the maximum execution feedrate of the Y axis is not 1000 mm/min (G00FEED), but 500 mm/min due to the 50% limitation of IPOFEED over G00FEED. Therefore, in spite of the programmed feedrate "F1000", the axis will move at 500 mm/min due to the limitation in G201.

If while the execution, the Yaxis is moved via JOG panel, a 100 mm/min feedrate (JOGFEED) should be added. However, the maximum jogging feedrate will be 60 mm/min because it has been limited to 50% of MANFEEDP over MAXMANFEED.

Therefore, the Y axis will move at 560 mm/min when combining the automatic and jog modes.

2.5.25 Leadscrew error compensation.

LSCRWCOMP

Leadscrew error compensation.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.LSCRWCOMP.xn

This parameter indicates whether the axis uses leadscrew error compensation or not.

LSCRWDATA

Leadscrew compensation table.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

This parameter shows the table to define the leadscrew error compensation. The table shows the following parameters to configure it.

LSCRWDATA	
NPOINTS	Number of points in the table.
TYPLSCRW	Type of compensation.
BIDIR	Bi-directional compensation.
REFNEED	Home search required for compensation.
DATA	Table to define the compensation values.

NPOINTS

Number of points in the table.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 1000.

Default value: 0 (there is no table).

Associated variable: (V.)[ch].MPA.NPOINTS.xn

Parameter included in the LSCRWCOMP table.

The leadscrew error compensation table can have up to 1000 points.

TYPLSCRW

Type of compensation (type of coordinates).

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Real / Theoretical.

Default value: Real.

Associated variable: (V.)[ch].MPA.TYPLSCRW.xn

Parameter included in the LSCRWCOMP table.

This parameter determines whether the leadscrew error compensation will be applied on to theoretical or real coordinates.

BIDIR

Bi-directional compensation.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.BIDIR.xn

Parameter included in the LSCRWCOMP table.

This parameter indicates whether the compensation is bidirectional or not; i.e. if the compensation is different in each direction. If the compensation is not bidirectional, it applies the same compensation in both directions.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

REFNEED**Mandatory home search.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.REFNEED.xn

Parameter included in the LSCRWCOMP table.

This parameter indicates whether the axis must be homed or not before applying the compensation.

DATA**Leadscrew error compensation at each point.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

This parameter shows the list of compensation points and values. The CNC only lets access this table when parameter NPOINTS has been set.

The DATA table must indicate the amount of error to be compensated in specific positions of the axis. The number of points is set by parameter NPOINTS. Parameters POSITION, POSERROR and NEGERROR must be set for each point. Parameter NEGERROR only if the table has been defined with bidirectional compensation (BIDIR = YES).

LSCRWCOMP DATA	
POSITION	Axis position.
POSERROR	Amount of error to compensate when moving in the positive direction.
NEGERROR	Amount of error to compensate when moving in the negative direction.

POSITION**Position of each point.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): Within ± 99999.9999 mm.

Possible values (2): Within ± 3937.00787 inch.

Possible values (3): Within ± 99999.9999 degrees.

Default value: 0.

Associated variable: (V.)[ch].MPA.POSITION[pt].xn

Parameter included in the LSCRWCOMP // DATA table.

Each parameter of the table represents a profile point to be compensated. The position that occupies the point in the profile will be referred to machine zero. When defining the various points in the table, the following requirements must be met.

- The points of the table must be ordered by their position on the axis and the table must begin by the most negative point (or least positive) to be compensated. For axis positioning outside this area, the CNC will apply the compensation that was defined for the nearest end.
- The maximum slope of the table is $\cdot 1\cdot$; i.e. the compensation steps must not exceed the position step between two consecutive points.

Import leadscrew compensation tables.

In order to save time and eliminate transcription errors, instead of entering the data manually, it is possible to adapt the format of a text file that contains the result of the measurement and import it later on.



See the operating manual for further information on how to import leadscrew error compensation tables.

(REF: 1709)

POSERROR

Error in the positive direction.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): Within ±99999.9999 mm.

Possible values (2): Within ±3937.00787 inch.

Possible values (3): Within ±99999.9999 degrees.

Default value: 0.

Associated variable: (V.)[ch].MPA.POSERROR[pt].xn

Parameter included in the LSCRWCOMP // DATA table.

This parameter indicates the amount of error when the axis moves in the positive direction. If the table has not been defined with bi-directional compensation, this amount of error will also be applied on movements in the negative direction.

NEGERROR

Error in the negative direction.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): Within ±99999.9999 mm.

Possible values (2): Within ±3937.00787 inch.

Possible values (3): Within ±99999.9999 degrees.

Default value: 0.

Associated variable: (V.)[ch].MPA.NEGERROR[pt].xn

Parameter included in the LSCRWCOMP // DATA table.

This parameter indicates the amount of error when the axis moves in the negative direction. This parameter is only available if the table has been defined with bidirectional compensation.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.5.26 Filters to eliminate resonance frequency.

FILTER

Filter table.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

This parameter shows the tables to configure the frequency filters. Up to 3 different filters may be defined for each axis or spindle, hence being possible to eliminate several resonance frequencies.

Frequency filters may be applied to axes and spindles. The filters defined for the spindle will only be applied when it works as a C axis or it is doing a rigid tapping.

The CNC offers several filters. The "anti-resonance" filter (notch filter) that limits a zone of the passing band and the low-pass filters that eliminate the banding passing from a particular frequency on.

One of them is usually used, although both types of filters can also be applied to the same axis or spindle when the resonance frequency is within the bandwidth of a "low passing" filter.

In order to obtain a good part finish, it is recommended to set all the axes that interpolate with each other with the same filter type and with the same frequency.

FILTER n

Filter configuration.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Each table has the following machine parameters to configure it.

FILTER	
ORDER	Filter order.
TYPE	Filter type.
FREQUENCY	Cut-off or center frequency.
NORBWIDTH	Normal bandwidth.
SHARE	Percentage of signal going through the filter.

ORDER

Filter order.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 10 for low passing filter / from 0 to 5 for anti-resonance filter / from 0 to 50 for FAGOR low passing filter.

Default value: 0 (No filter is applied).

Associated variable: (V.)[ch].MPA.ORDER[nb].xn

Parameter included in the FILTER table.

The down ramp is dampened down; the higher the order the greater the drop. A higher order implies more calculation; therefore, we recommend to consult our Technical support department before modifying this parameter.

TYPE

Filter type.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Low Passing / Anti-resonance (notch filter) / Low passing FAGOR.

Default value: Low Passing.

Associated variable: (V.)[ch].MPA.TYPE[nb].xn

Parameter included in the FILTER table.

Usually, rigid and robust machines have a passing band of up to 30 Hz; i.e. they are capable of responding up to this frequency. The rest of the machines may have resonance at lower frequencies (10 Hz or less) that are generated on usual paths.

2.

MACHINE PARAMETERS.
Machine parameters for the axes and spindles.



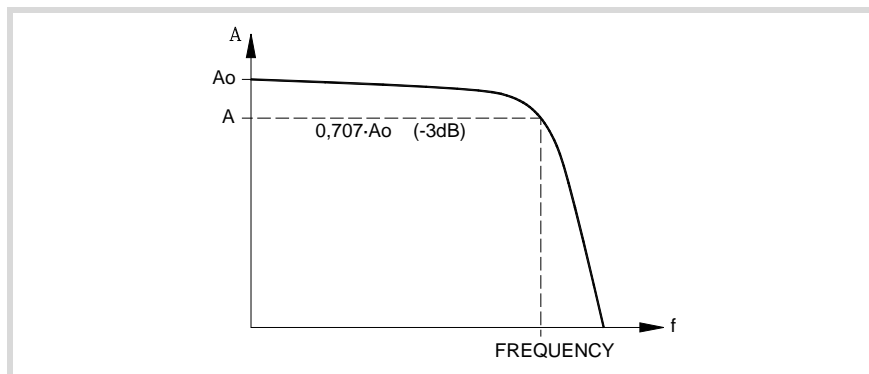
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

"Low passing" filter.

If the machine shows the resonance frequency at the end of its passing band, it will be enough to eliminate it using a "low passing" filter.



These filters are also used to limit a jerk whose parameter has been set too high for the machine but necessary to make it run faster. This way, the CNC smooths the movements out although it has the drawback of slightly rounding the corners off.

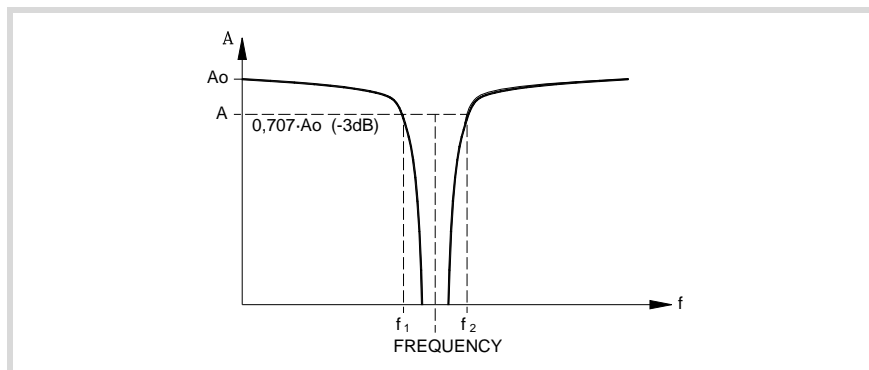
- The "low passing" filter, for the same frequency requires a lower order and consequently adds a shorter delay.

This type of filter inserts a variable (non constant) phase shift depending on the frequencies. This phase shift can cause a change on the path if it is not executed at the same feedrate as, for example, when changing the feedrate override percentage or when moving back and forth on the same path.

- The "FAGOR low passing" filter inserts a constant phase shift regardless of the frequency. This type of filter needs a higher order to apply the same amount of filtering.

Anti-resonance filter (band-rejection, notch filter).

If the machine shows the resonance frequency in the middle of its passing band, it will be a good idea to eliminate it using a "notch" filter.



FREQUENCY

Cut-off or center frequency.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: from 0 to 500.0 Hz.

Default value: 30.0 Hz.

Associated variable: (V.)[ch].MPA.FREQUENCY[nb].xn

Parameter included in the FILTER table.

On "low passing" filters, this parameter indicates the break point frequency or frequency where the amplitude drops 3 dB or it reaches 70% of the nominal amplitude.

$$-3 \text{ dB} = 20 \log (A/A_0) \implies A = 0,707 A_0$$

On the band-rejection filter (notch filter), this parameter indicates the center frequency or the frequency where the resonance reaches its maximum value.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.

NORWIDTH**Normal bandwidth.***Parameter valid for linear and rotary axes and spindles.**Parameter valid for analog drive, Sercos and simulated.**Possible values: From 0 to 100.0**Default value: 1.0**Associated variable: (V.)[ch].MPA.NORWIDTH[nb].xn**Parameter included in the FILTER table.*

This parameter is calculated with the following formula. The f1 and f2 values correspond to the break frequency where the amplitude drops 3 dB or it reaches 70% of the nominal amplitude.

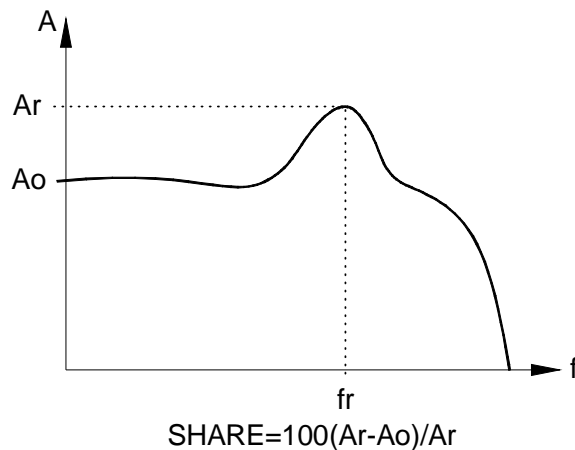
$$-3 \text{ dB} = 20 \log (A/A_0) \implies A = 0,707 A_0$$

$$\text{NORWIDTH} = \frac{\text{FREQUENCY}}{(f_2 - f_1)}$$

SHARE**Percentage of signal going through the filter.***Parameter valid for linear and rotary axes and spindles.**Parameter valid for analog drive, Sercos and simulated.**Possible values: From 0 to 100.**Default value: 100.**Associated variable: (V.)[ch].MPA.SHARE[nb].xn**Parameter included in the FILTER table.*

This parameter indicates the percentage of signal going through the filter. This value must be equivalent to the percentual overshooting of the resonance because it must compensate for it.

Example of a calculation for a particular response of the machine.



2.5.27 Work sets.

NPARSETS

Number of parameter sets available.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 1 to 4.

Default value: 1.

Associated variable: (V.)[ch].MPA.NPARSETS.xn

Up to 4 different parameter sets may be defined to indicate the dynamics of the axis in each one of them (feedrates, gains, accelerations, etc.).

DEFAULTSET

Set of default parameters on power-up.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 4.

Default value: 1.

Associated variable: (V.)[ch].MPA.DEFAULTSET.xn

Axis behavior:

For the axes, the parameter DEFAULTSET indicates the parameter set assumed by the CNC during power-up, after executing an M02 or M30 or after a reset. When defined with a "0" value, the parameter set is always maintained. To select a parameter set on the axis or via the part-program, use function G112.

Spindle behavior.

For the spindles, the parameter DEFAULTSET is not used either in the start-up or in the reset; both circumstances are managed from the PLC maneuver. During power-up, the CNC uses the set indicated by the PLC; if there is no set indicated, then the CNC starts-up without a set. After a reset, the CNC uses the set indicated by the PLC; if there is no set indicated, then the CNC remains as it is.

The parameter DEFAULTSET is only used when moving the spindle for the first time without a set and without S programmed. In this case, the CNC uses the set indicated in the parameter DEFAULTSET. If DEFAULTSET is set at the value of ·0·, the CNC uses the set of 1.

- The CNC starts-up without a set, and just after starting-up the CNC runs an M19 or G74.
- The CNC starts-up without a set, and just after starting-up the spindles that do not have the SYNCSET parameter defined are synchronized.
- If the Sercos ring falls and the drive uses the default set.

If the CNC starts-up without a set, when the spindle starts-up for the first time (M3/M4 with S programmed) the CNC uses the appropriate set, which is dependant on S.

The start-up and reset do not use the parameter DEFAULTSET; both circumstances are managed from the PLC maneuver. During power-up, the CNC uses the set indicated by the PLC; if there is no set indicated, then the CNC starts-up without a set. After a reset, the CNC uses the set indicated by the PLC; if there is no set indicated, then the CNC remains as it is.

To select a parameter set on a spindle from the part-program and make a gear change, use functions M41 through M44.

SET n

Work sets.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

It shows the machine parameter table of the set.

2.

MACHINE PARAMETERS.

Machine parameters for the axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6 Machine parameters for the axes. Work sets.

Up to 4 different parameter sets may be defined to indicate the dynamics of the axis in each one of them (feedrates, gains, accelerations, etc.). Each set only shows the parameters for the selected type of axis and drive.

In the laser model, the parameters related to spindle speed (defined in rpm) define the power (defined in watts).

2.

2.6.1 Feedback resolution.

PITCH

Leadscrew pitch.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 99999.9999 mm.

Possible values (2): From 0 to 3937.00787 inch.

Possible values (3): From 0 to 99999.9999 degrees.

Default value: 5 mm / 0.19685 inch / 360°.

Associated variable: (V.)[ch].MPA.PITCH[set].xn

The meaning of this parameter depends on the type of feedback.

- On a linear axis with a rotary encoder and leadscrew, the parameter defines the leadscrew pitch.
- On a linear axis with a linear encoder (scale), the parameter defines the pitch of the scale.
- On a rotary axis, the parameter sets the number of degrees per turn of the encoder.

Type of axis.	PITCH
Axis with a 5 mm pitch leadscrew.	5 mm.
Axis with a 20 µm pitch Fagor scale.	0.020 mm.
Rotary axis with a 1/10 gear ratio.	36°.

INPUTREV

Turns of the motor shaft.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 1 to 32767.

Default value: 1.

Associated variable: (V.)[ch].MPA.INPUTREV[set].xn

See machine parameter OUTPUTREV.

OUTPUTREV

Turns of machine axis.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 1 to 32767.

Default value: 1.

Associated variable: (V.)[ch].MPA.OUTPUTREV[set].xn

The possible gear ratio between the motor and the encoder may also be entered directly through parameter PITCH; in that case, parameters INPUTREV and OUTPUTREV must be set to ·1. If the gear ratio is not an integer multiple, we recommend to set the real values of the gear ratio in INPUTREV and OUTPUTREV instead of leaving an inaccurate number in PITCH.

NPULSES

Number of encoder pulses.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives, velocity-Sercos drives and simulated drives.

Possible values: From 0 to 1000000.

Default value: 1250.

Associated variable: (V.)[ch].MPA.NPULSES[set].xn

This parameter sets the number of degrees per turn of the encoder. When using a gear reduction on the axis, the whole assembly must be taken into account when defining the number of pulses per turn. With linear encoders (scales), set NPULSES = 0

If parameters NPULSES and NPULSES2 are set with a value other than -0-, parameter REFINI determines whether the CNC homes the spindle on its first movement or not.

PITCH2

Leadscrew pitch (external feedback).

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Sercos drive.

Possible values (1): From 0 to 99999.9999 mm.

Possible values (2): From 0 to 3937.00787 inch.

Possible values (3): From 0 to 99999.9999 degrees.

Default value: 5 mm / 0.19685 inch / 360°.

Associated variable: (V.)[ch].MPA.PITCH2[set].xn

The meaning of this parameter depends on the type of feedback.

- On a linear axis with a rotary encoder and leadscrew, the parameter defines the leadscrew pitch.
- On a linear axis with a linear encoder (scale), the parameter defines the pitch of the scale.
- On a rotary axis, the parameter sets the number of degrees per turn of the encoder.

Type of axis.	PITCH
Axis with a 5 mm pitch leadscrew.	5 mm.
Axis with a 20 µm pitch Fagor scale.	0.020 mm.
Rotary axis with a 1/10 gear ratio.	36°.

INPUTREV2

Turns of the motor shaft (external feedback)

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Sercos drive.

Possible values: From 1 to 32767.

Default value: 1.

Associated variable: (V.)[ch].MPA.INPUTREV2[set].xn

See machine parameter OUTPUTREV2.

OUTPUTREV2

Turns of machine axis (external feedback).

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Sercos drive.

Possible values: From 1 to 32767.

Default value: 1.

Associated variable: (V.)[ch].MPA.OUTPUTREV2[set].xn

Parameters INPUTREV2 and OUTPUTREV2 set the gear ratio for the second feedback.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

NPULSES2**Number of pulses of the encoder (external feedback).***Parameter valid for linear and rotary axes and spindles.**Parameter valid for a Sercos drive.**Possible values: From 0 to 1000000.**Default value: 1250.**Associated variable: (V.)[ch].MPA.NPULSES2[set].xn*

This parameter sets the number of degrees per turn of the encoder for the second feedback. When using a gear reduction on the axis, the whole assembly must be taken into account when defining the number of pulses per turn. With linear encoders (scales), set NPULSES2 = 0.

If parameters NPULSES and NPULSES2 are set with a value other than ·0·, parameter REFINI determines whether the CNC homes the spindle on its first movement or not.

SINMAGNI**Multiplying factor for sinusoidal feedback signal.***Parameter valid for linear and rotary axes and spindles.**Parameter valid for analog drives and simulated drives.**Possible values: From 0 to 255.**Default value: 0.**Associated variable: (V.)[ch].MPA.SINMAGNI[set].xn*

This parameter indicates the multiplying factor applied to the sinewave feedback of the axis. For square feedback signals, set SINMAGNI = 0; the CNC always applies a x4 factor.

Parameters SINMAGNI, PITCH and NPULSES set the axis feedback resolution as shown in the table below.

Type of encoder.	PITCH	NPULSES	SINMAGNI
Rotary axis. Rotary encoder. Square signals.	Rotation of the axis per turn of the encoder	Number of pulses.	0
Rotary axis. Rotary encoder. Sinusoidal signals.	Rotation of the axis per turn of the encoder	Number of pulses.	Multiplying factor.
Linear axis. Rotary encoder. Square signals.	Leadscrew pitch.	Number of pulses.	0
Linear axis. Rotary encoder. Sinusoidal signals.	Leadscrew pitch.	Number of pulses.	Multiplying factor.
Linear encoder. Square signals.	Pitch of encoder.	0	0
Linear encoder. Sinusoidal signals.	Pitch of encoder.	0	Multiplying factor.

ABSFEEDBACK**Absolute feedback system.***Parameter valid for linear and rotary axes and spindles.**Parameter valid for analog drive, Sercos and simulated.**Possible values: Yes / No.**Default value: No.**Associated variable: (V.)[ch].MPA.ABSFEEDBACK[set].xn*

This parameter indicates whether the axis uses absolute feedback or not. With this system, the axis is considered to be homed right on power-up and no movement is generated when programming a home search. The PLC mark REFPOIN is always active.

FBACKAL

Activate feedback alarm.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for an analog drive.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)[ch].MPA.FBACKAL[set].xn

This parameter may be used to activate the feedback alarm. This alarm can tell the difference between the feedback cables being disconnected or broken for differential TTL signals and sinusoidal signals.

When a feedback alarm occurs, the PLC turns the REFPOIN mark off.

If this alarm is canceled, the CNC shows on power-up a message indicating that this safety is disabled. This situation can only be allowed during setup; once setup is completed, this alarm must be enabled.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.2 Loop setting.

LOOPCH

Velocity command sign change.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.LOOPCH[set].xn

See machine parameter AXISCH.

AXISCH

Feedback sign change.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.AXISCH[set].xn

If the axis runs away, the CNC issues a following error message. Change the value of parameter LOOPCH. If it does not run away, but the counting direction is not the desired one, change the values of both parameters AXISCH and LOOPCH.

INPOSW

In-position zone.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0.0001 to 99999.9999 mm.

Possible values (2): From 0.0000 to 3937.00787 inch.

Possible values (3): From 0.0001 to 99999.9999 degrees.

Default value: 0.0100 mm / 0.00039 inch / 0.0100 degrees.

Associated variable: (V.)[ch].MPA.INPOSW[set].xn

The in-position zone is defined as the zone before and after the programmed position where the axis is considered to be in position. Parameter INPOSW defines the width of both zones.

2.6.3 Backlash compensation.

BACKLASH

Amount of backlash to be compensated.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): Within ± 3.2768 mm.

Possible values (2): Within ± 0.12901 inch.

Possible values (3): Within ± 3.2768 degrees.

Default value: 0.

Associated variable: (V.)[ch].MPA.BACKLASH[set].xn

With linear encoders (scales), set BACKLASH = 0.

When an axis has backlash and reverses its moving direction, there is a delay from the instant the motor starts turning to the moment when the axis actually moves. This usually happens on axes with encoder and on old machines whose leadscrew assembly is defective (worn out).

Use a dial indicator to measure this backlash. Move the axis in a direction and set the dial indicator to "0". Move the axis in the opposite direction in incremental mode until detecting that the axis is moving. The amount of backlash is the difference between the commanded distance and what it actually moved.

2.

MACHINE PARAMETERS.
Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.4 Backlash compensation with an additional command pulse.

BAKANOUT

Additional velocity command pulse.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and Sercos drives.

Possible values: Within ± 32767 with an analog drive / within ± 1000 rpm with a Sercos drive.

Default value: 0 (Not applied).

Associated variable: (V.)[ch].MPA.BAKANOUT[set].xn

Additional velocity command pulse to make up for the possible leadscrew backlash when reversing the moving direction. Every time the movement is inverted, the CNC will apply to that axis the velocity command corresponding to the movement plus the additional velocity command pulse set in this parameter. The duration of this additional velocity command depends on parameters BAKTIME and PEAKDISP.

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.

BAKANOUT	Velocity command.
-32767	-10 V.
...	...
-3277	-1 V.
...	...
1	0,3 mV.
...	...
3277	1 V.
...	...
32767	10 V.

On analog axes, when LOOPCH=Yes, BAKANOUT must be set with a negative value so the CNC applies the right reversal peak for movement reversal.

When setting an additional command pulse, parameters BAKTIME, ACTBAKAN and PEAKDISP must also be set.

BAKTIME

Duration of the additional velocity command pulse.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and Sercos drives.

Possible values: From 0 to 100 ms.

Default value: 0.

Associated variable: (V.)[ch].MPA.BAKTIME[set].xn

The CNC only takes this parameter into consideration when working with an additional velocity command pulse; BAKANOUT other than zero. Parameter BAKTIME indicates the duration of the additional velocity command pulse to make up for backlash in movement reversals.

ACTBAKAN

Application of the additional velocity command pulse.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and Sercos drives.

Possible values: Always / G2-G3.

Default value: Always.

Associated variable: (V.)[ch].MPA.ACTBAKAN[set].xn

The CNC only takes this parameter into consideration when working with an additional velocity command pulse; BAKANOUT other than zero. Parameter ACTBAKAN determines when the additional command pulse is applied to compensate for backlash peaks.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

MACHINE PARAMETERS.
Machine parameters for the axes. Work sets.

PEAKDISP**Backlash peak cutoff distance.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and Sercos drives.

Possible values (1): From 0 to 99999.9999 mm.

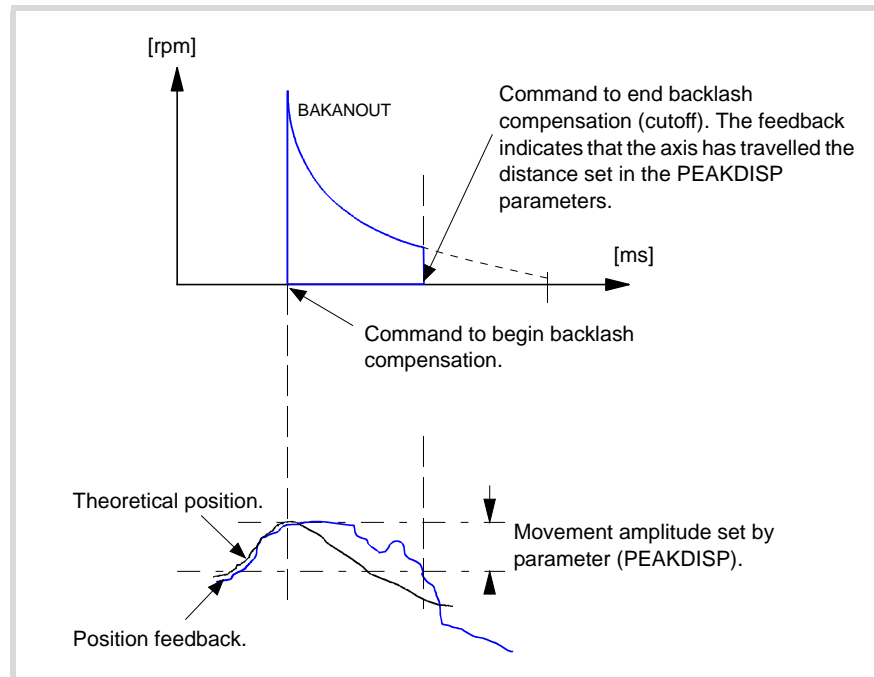
Possible values (2): From 0 to 3937.00787 inch.

Possible values (3): From 0 to 99999.9999 degrees.

Default value: 0.0050 mm / 0.00508 mm / 0.0050 degrees.

Associated variable: (V.)[ch].MPA.PEAKDISP[set].xn

The CNC only takes this parameter into consideration when working with an additional velocity command pulse; BAKANOUT other than zero. Parameter PEAKDISP sets the real distance traveled by the axis after the theoretical movement reversal, where the CNC cuts the reversal peak off on that axis (additional command pulse).

**REVEHYST****Hysteresis for applying the additional command pulse in movement reversals.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and Sercos drives.

Possible values (1): From 0 to 99999.9999 mm.

Possible values (2): From 0 to 3937.00787 inch.

Possible values (3): From 0 to 99999.9999 degrees.

Default value: 0.

Associated variable: (V.)[ch].MPA.REVEHYST[set].xn

This parameter limits the use of backlash compensation when the CNC detects a movement reversal so it is not applied every time the CNC receives a reversal command.

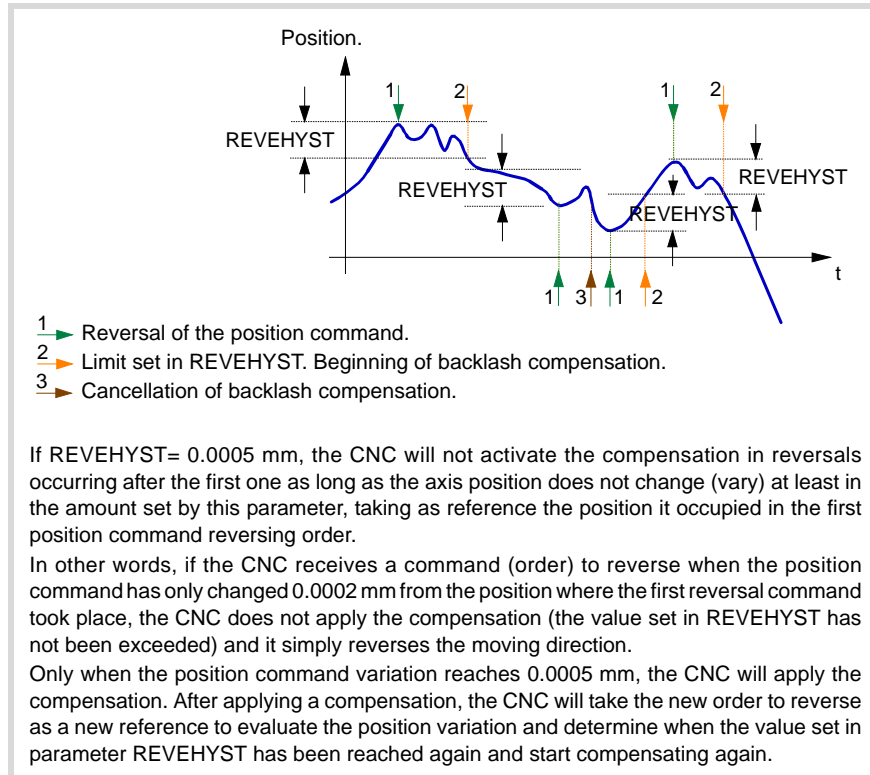
This parameter sets the value that the axis position must vary after the first movement reversal (hysteresis) for the CNC to consider applying backlash compensation. If the axis does not exceed this margin, the CNC does not apply backlash compensation.

Considerations.

- If REVEHYST=0, Backlash compensation by reversal peak will always be applied on each reversal.
- If REVEHYST other than 0, and parameter PEAKDISP is used to cut the backlash peak off, we recommend to set REVEHYST to a lower value than that of PEAKDISP so the CNC applies the backlash peak.

(REF: 1709)

- When having axes set as DRO axes, the value of the BACKLASH parameter will be taken into consideration. In these cases, especially when using sinusoidal feedback, we recommend to set REVEHYST to a value other than 0 in order to apply backlash.



2.

MACHINE PARAMETERS.
 Machine parameters for the axes. Work sets.

2.6.5 Adjustment of rapid traverse G00 and maximum speed.

G00FEED

Feedrate in G00.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 500000.0000 mm/min.

Possible values (2): From 0 to 19685.03937 inch/min.

Possible values (3): From 0 to 72000000.0000 degrees/min.

Possible values (4): From 0 to 200000.0000 rpm.

Default value: 10000.0000 mm/min / 393.70079 inch/min / 1080000 degrees/min / 3000.0000 rpm.

Associated variable: (V.)[ch].MPA.G00FEED[set].xn

Lathe and mill models.

Rapid positioning (traverse, G00) are always carried out at the maximum speed possible. The one indicated by G00FEED.

Laser model.

Maximum generator power. In order for the power control through an analog output to be managed correctly (#PWRCTRL ON[OUT]), the watts defined in the parameter G00FEED as the generator power must match the volt level defined in the parameter MAXVOLT.

How to limit temporarily the maximum feedrate via PLC

The PLC has the variable (V.)[ch].PLC.G00FEED that may be used to limit the feedrate in the channel for any type of movement (G00, G01, etc). This variable limits the feedrate of the path and affects all the axes whether they move interpolated or one axis at a time.

The CNC assumes the change immediately and the change stays active until the variable takes a value of -0- restoring the limits set by machine parameters.

MAXFEED

Maximum axis machining feedrate.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 500000.0000 mm/min.

Possible values (2): From 0 to 19685.03937 inch/min.

Possible values (3): From 0 to 72000000.0000 degrees/min.

Possible values (4): From 0 to 1388.8889 rpm.

Default value: 0.

Associated variable: (V.)[ch].MPA.MAXFEED[set].xn

This parameter sets the maximum machining feedrate (movements in G01/G02/G03) of the axis; when set to -0-, the feedrate is not limited. This parameter cannot be set with a value higher than that of parameter G00FEED.

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 in MAXFEED without showing any error message or warning.

If the machining feedrate is not limited, the CNC assumes for all the movements the one set in machine parameter G00FEED as the maximum feedrate.

Variable.	Machine parameters.		Feedrate.	
	(V.)PLC.G00FEED	G00FEED (axis)	MAXFEED (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

2.

MACHINE PARAMETERS. Work sets. Machine parameters for the axes.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

MAXVOLT

Velocity command to reach G00FEED.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and velocity-Sercos drives.

Possible values: from 0 to 10000.0000 mV.

Default value: 9500 mV (9.5 V).

Associated variable: (V.)[ch].MPA.MAXVOLT[set].xn

Lathe and mill models.

This is the velocity command the CNC must output so the axis can reach its maximum rapid traverse feedrate G00FEED.

Laser model.

This is the command the CNC must provide for the spindle to reach the maximum generator power set in parameter G00FEED.

MAXFREQ

Frequency required to reach G00FEED.

Parameter valid for spindles.

Parameter valid for a Mechatrolink inverter.

Possible values: From 0.0010 to 10000.0000 Hz.

Default value: 50 Hz.

Associated variable: (V.)[ch].MPA.MAXFREQ[set].xn

When the drive is an inverter, this parameter indicates the frequency to be provided by the CNC for the spindle to reach the speed set in parameter G00FEED.

MAXRPM

Maximum motor rpm.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Mechatrolink speed and inverter.

Possible values: From 0 to 100000 rpm.

Default value: 3000 rpm.

Associated variable: (V.)[ch].MPA.MAXRPM[set].xn

Maximum motor rpm.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.6 Rapid traverse for the automatic mode.

FRAPIDEN

Rapid traverse of the axis for movements in automatic mode.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 500000.0000 mm/min.

Possible values (2): From 0 to 19685.03937 inch/min.

Possible values (3): From 0 to 500000.0000 degrees/min.

Possible values (4): From 0 to 1388.8889 rpm.

Default value: 0.

Associated variable: (V.)[ch].MPA.FRAPIDEN[set].xn

Maximum feedrate of the axis for the program being executed, 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. The value of this parameter must always be lower than that of axis parameter G00FEED.

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.

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

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.7 Gain setting.

PROGAIN

Proportional gain.

Parameter valid for linear and rotary axes and spindles.

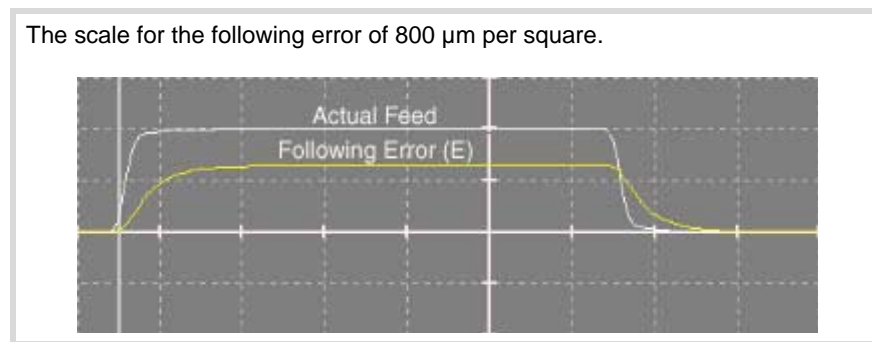
Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0.0 to 100.0 (1000/min).

Default value: 1.

Associated variable: (V.)[ch].MPA.PROGAIN[set].xn

This parameter sets the following error (difference between the theoretical instantaneous position and the actual - real - axis position) for a particular feedrate.



Example:

To obtain a following error (E) of 1 mm for a feedrate (F) of 1000 mm/min. (a gain of 1).

$$F = E \times \text{PROGAIN}$$

$$F / E = 1000 \text{ (mm/min)} / 1 \text{ (mm)} = 1000 / \text{min}$$

$$\text{PROGAIN} = 1$$

FFWTYPE

Pre-control type.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: OFF / Feed Forward / AC-Forward / Feed Forward + AC-Forward.

Default value: OFF

Associated variable: (V.)[ch].MPA.FFWTYPE[set].xn

This parameter indicates the type of pre-control for adjusting the gains.

In order to be able to modify the override while threading (parameter THREADOVR), the feed forward must be active and higher than 90%. Feed forward may be defined with a lower value than 90% in parameter FFGAIN and may be modified later on from the PLC using variables.

FFGAIN

Percentage of Feed-Forward in automatic.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: from 0 to 120 %.

Default value: 0.

Associated variable: (V.)[ch].MPA.FFGAIN[set].xn

The CNC only takes this parameter into account when working with feed forward; parameter FFWTYPE if it is an analog or simulated drive and OPMODEP if it is a Sercos drive. This parameter should only be used when working with non-linear acceleration and deceleration.

An axis that is going to work as an independent axis should have the same feed forward value in automatic and in jog modes; in other words, parameters FFGAIN and MANFFGAIN should be the same.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

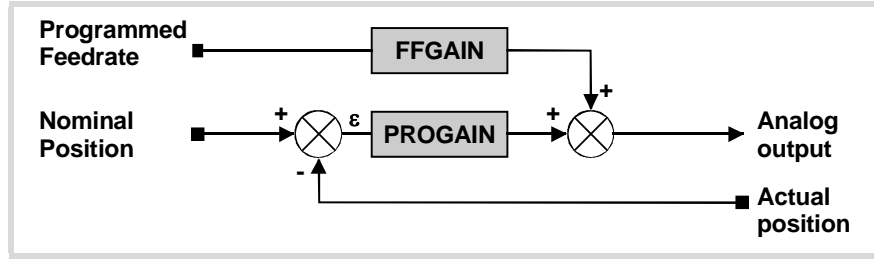
FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

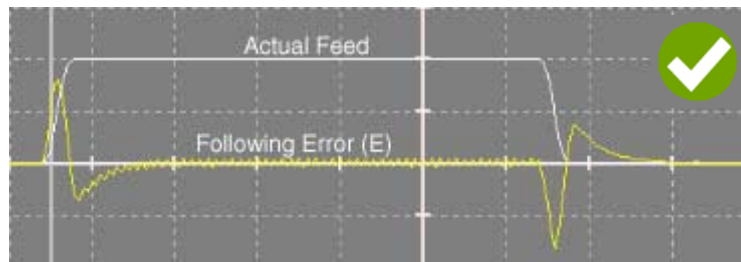
This parameter helps improve the position loop minimizing the amount of following error " ϵ ". This parameter sets the portion of the velocity command is proportional to the programmed feedrate and part of it is proportional to the following error " ϵ ".



The best adjustment is obtained when minimizing the amount of following error as much as possible without reversing its sign, maintaining the moving direction of the axis.

The scale for the following error of 10 μm per square.

- Proper adjustment with Feed forward.



- Wrong adjustment with Feed forward.



MANFFGAIN

Percentage of Feed-Forward in JOG.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and simulated drives.

Possible values: from 0 to 120 %.

Default value: 0.

Associated variable: (V.)[ch].MPA.MANFFGAIN[set].xn

The CNC only takes this parameter into account when working with feed forward; parameter FFWTYPE if it is an analog or simulated drive and OPMODEP if it is a Sercos drive.

Sometimes, the Feed Forward selected for the automatic mode may be too high for the Jog mode, when there is no need to control the following error that closely. In those cases, parameter MANFFGAIN helps adapt the Feed-Forward applied to the jog mode.

An axis that is going to work as an independent axis should have the same feed forward value in automatic and in jog modes; in other words, parameters FFGAIN and MANFFGAIN should be the same.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

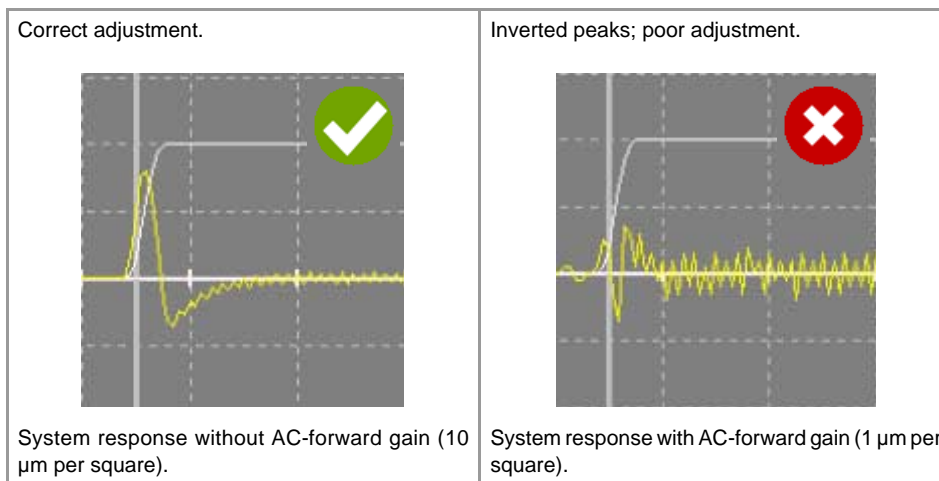
(REF: 1709)

ACFWFACTOR
Acceleration time constant.

Parameter valid for linear and rotary axes and spindles.
 Parameter valid for analog drives, velocity-Sercos drives and simulated drives.
 Possible values: From 0.001 to 1000000.0000 ms.
 Default value: 1000.0000 ms.
 Associated variable: (V.)[ch].MPA.ACFWFACTOR[set].xn

The CNC only takes this parameter into account when working with AC-forward; parameter FFWTYPE if it is an analog or simulated drive and OPMODEP if it is a velocity-Sercos drive. No function in Sercos position.

It is recommended to assign to this parameter a value close to the system's response time. Since the system response time is usually an unknown value that depends on the inertia of the machine and on the drive adjustment, it is recommended to try with several values. The best adjustment is achieved when minimizing the following error as much as possible but without inverting the peaks.



Peculiarities of the laser model.

On laser machines with gap control, the TRANSDELAY parameter of the axis associated with the sensor must have a value of 0 (zero) in order for the machine response to be the same with and without gap control. Delay compensation is implemented with the ACFWFACTOR parameter, which will have a higher value than if the TRANSDELAY parameter is applied (approximately the sum of TRANSDELAY and ACFWFACTOR in an analog or Sercos speed adjustment). In an adjustment without the TRANSDELAY parameter, the following error peaks in accelerations and stops will be greater than with an adjustment using this parameter.

ACFGAIN
Percentage of AC-Forward in automatic.

Parameter valid for linear and rotary axes and spindles.
 Parameter valid for analog drive, Sercos and simulated.
 Possible values: from 0 to 120 %.
 Default value: 0.
 Associated variable: (V.)[ch].MPA.ACFGAIN[set].xn

See machine parameter MANACFGAIN.

MANACFGAIN
Percentage of AC-Forward in jog mode.

Parameter valid for linear and rotary axes and spindles.
 Parameter valid for analog drives and simulated drives.
 Possible values: from 0 to 120 %.
 Default value: 0.
 Associated variable: (V.)[ch].MPA.MANACFGAIN[set].xn

The CNC takes this parameter into account only when working with AC-Forward; parameter ACFWFACTOR.

2.

MACHINE PARAMETERS.
 Machine parameters for the axes. Work sets.



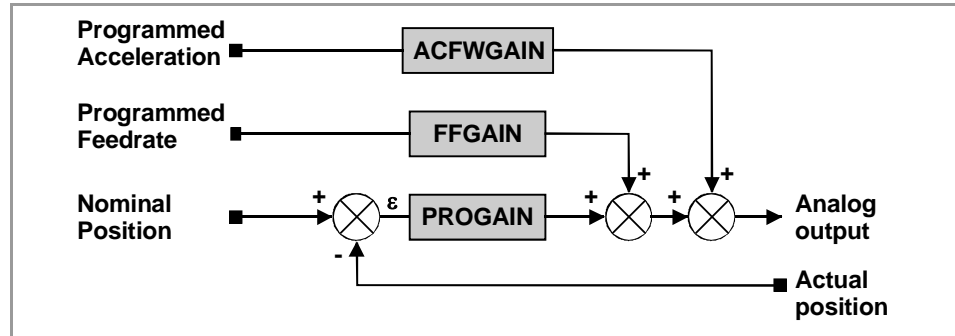
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

An axis that is going to work as an independent axis should have the same AC-forward value in automatic and in jog modes; in other words, parameters ACFGAIN and MANACFGAIN should be the same.

These parameters are similar to parameters FFGAIN and MANFFGAIN; but they affect the AC-Forward. Both parameters improve system response in acceleration changes and minimize the amount of following error "ε" when starting up, braking and reversing the moving direction.



2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.8 Linear acceleration.

LACC1

Acceleration of the first section.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0.0000 to 600000000.0000 mm/s².

Possible values (2): From 0.00000 to 23622047.24409 inch/s².

Possible values (3): From 0.0000 to 600000000.0000 degrees/s².

Default value: 1000.0000 mm/s² / 39.37008 inch/s² / 1000.0000 degrees/s².

Associated variable: (V.)[ch].MPA.LACC1[set].xn

See parameter LACC2.

LACC2

Acceleration of the second section.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0.0000 to 600000000.0000 mm/s².

Possible values (2): From 0.00000 to 23622047.24409 inch/s².

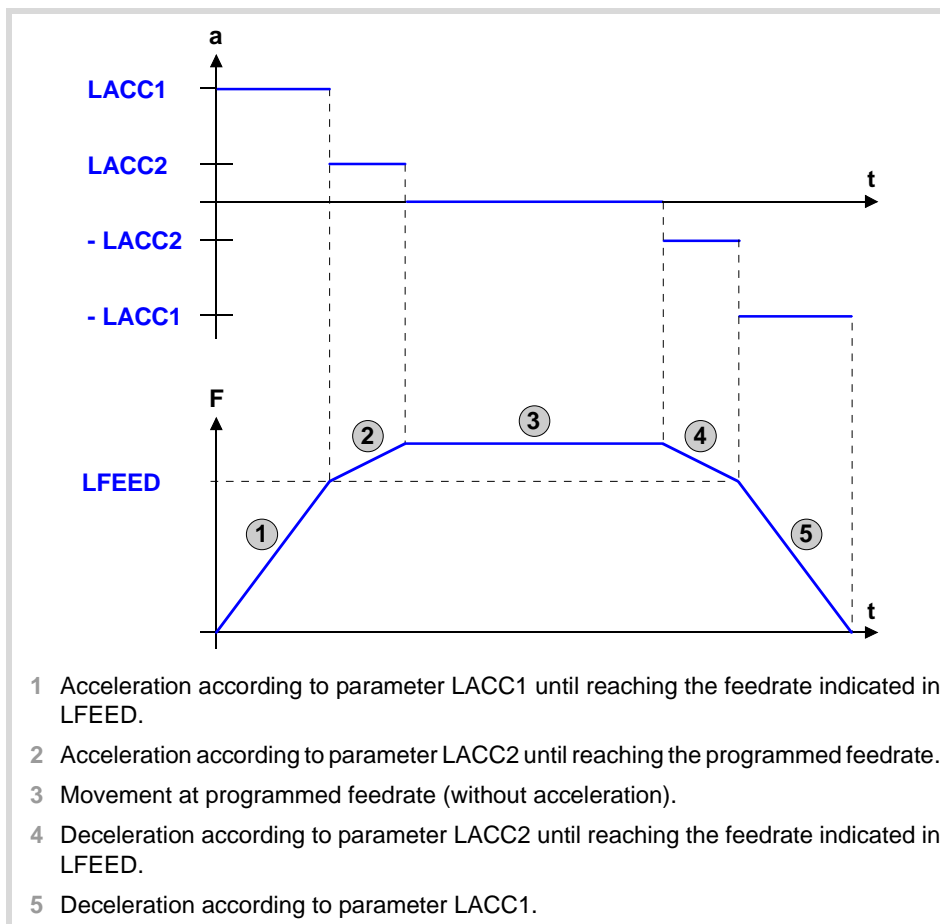
Possible values (3): From 0.0000 to 600000000.0000 degrees/s².

Default value: 1000.0000 mm/s² / 39.37008 inch/s² / 1000.0000 degrees/s².

Associated variable: (V.)[ch].MPA.LACC2[set].xn

Parameters LACC1 and LACC2 set the acceleration values when it is linear (parameter SLOPETYPE) or when working with HSC in FAST mode. The CNC applies parameters LACC1 and LACC2 to the movements in G1, G2 and G3; movements in G0 depend on parameter G0ACDCJERK.

G0ACDCJERK	Meaning.
Yes.	Movements in G0 use parameters LACC1G0, LACC2G0 and LFEEDG0.
No.	Movements in G0 use parameters LACC1, LACC2 and LFEED.



2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

FAGOR

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

LFEED**Acceleration changing speed.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 200000.0000 mm/min.

Possible values (2): From 0 to 7874.01575 inch/min.

Possible values (3): From 0 to 36000000.0000 degrees/min.

Possible values (4): From 0 to 100000.0000 rpm.

Default value: 3000 mm/min / 118.11023 inch/min / 1080000 degrees/min / 3000 rpm.

Associated variable: (V.)[ch].MPA.LFEED[set].xn

While accelerating, when reaching the feedrate defined in this parameter, it changes the acceleration from LACC1 to LACC2. While decelerating, when reaching the feedrate defined in this parameter, it changes the acceleration from LACC2 to LACC1.

2.**MACHINE PARAMETERS.**

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.9 Trapezoidal and square sine acceleration.

ACCEL Acceleration.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0.0000 to 600000000.0000 mm/s².

Possible values (2): From 0.00000 to 23622047.24409 inch/s².

Possible values (3): From 0.0000 to 600000000.0000 degrees/s².

Default value: 3000.0000 mm/s² / 118.12023 inch/s² / 3000 degrees/s².

Associated variable: (V.)[ch].MPA.ACCEL[set].xn

See machine parameter DECEL.

DECEL Deceleration.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0.0000 to 600000000.0000 mm/s².

Possible values (2): From 0.00000 to 23622047.24409 inch/s².

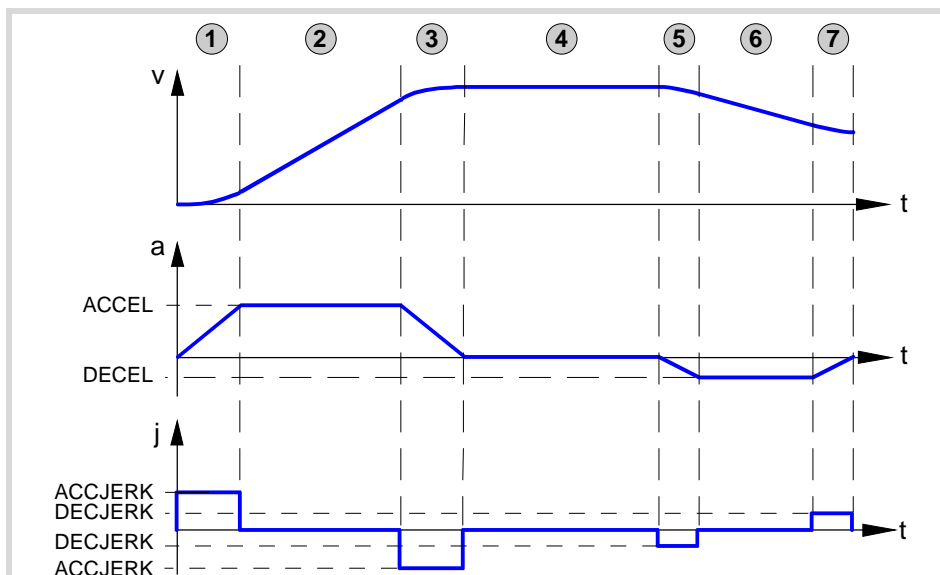
Possible values (3): From 0.0000 to 600000000.0000 degrees/s².

Default value: 3000.0000 mm/s² / 118.12023 inch/s² / 3000 degrees/s².

Associated variable: (V.)[ch].MPA.DECEL[set].xn

Parameters ACCEL and DECEL set the acceleration values when it is trapezoidal or square-sine (bell shaped) (parameter SLOPETYPE). The CNC applies parameters ACCEL and DECEL to the movements in G1, G2 and G3; movements in G0 depend on parameter GOACDCJERK.

GOACDCJERK	Meaning.
Yes.	Movements in G0 use parameters ACCELG0 and DECELG0.
No.	Movements in G0 use parameters ACCEL and DECEL.



- 1 The axis starts moving with a uniformly increasing acceleration, with a slope limited by ACCJERK, until reaching the acceleration indicated in ACCEL.
- 2 The acceleration becomes constant at the ACCEL value.
- 3 Before reaching the programmed feedrate, there is a steady deceleration with a slope limited by ACCJERK.
- 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 DECJERK.
- 6 The deceleration becomes constant at the DECEL value.
- 7 Before reaching the programmed feedrate, or stopping, there is a deceleration limited by DECJERK.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

MACHINE PARAMETERS. Machine parameters for the axes. Work sets.

ACCJERK
Acceleration Jerk.

Parameter valid for linear and rotary axes and spindles.
 Parameter valid for analog drive, Sercos and simulated.
 Possible values (1): From 0.0010 to 6E+011 mm/s³.
 Possible values (2): From 0.00004 to 2.362E+010 inch/s³.
 Possible values (3): From 0.0010 to 6E+011 mm/s³.
 Default value: 45000.000 mm/s³ / 1771.65354 inch/s³ / 45000.000 grados/s³.
 Associated variable: (V.)[ch].MPA.ACCJERK[set].xn

See machine parameter DECJERK.

DECJERK
Deceleration Jerk.

Parameter valid for linear and rotary axes and spindles.
 Parameter valid for analog drive, Sercos and simulated.
 Possible values (1): From 0.0010 to 6E+011 mm/s³.
 Possible values (2): From 0.00004 to 2.362E+010 inch/s³.
 Possible values (3): From 0.0010 to 6E+011 mm/s³.
 Default value: 45000.000 mm/s³ / 1771.65354 inch/s³ / 45000.000 grados/s³.
 Associated variable: (V.)[ch].MPA.DECJERK[set].xn

Parameters ACCJERK and DECJERK set the slope of the acceleration and deceleration. Both parameters help limit the acceleration changes so the machine runs more smoothly in small speed increments or decrements and with FFGAIN values close to 100%. The lower the value assigned to these parameters, the smoother the machine response, but the acc/dec time will increase.

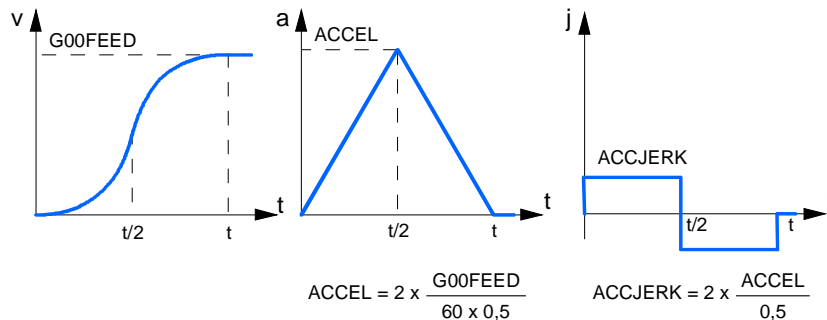
The CNC applies parameters ACCEL and DECEL to the movements in G1, G2 and G3; movements in G0 depend on parameter G0ACDCJERK. The CNC ignores these parameters in threading movements (G33) and with HSC FAST.

G0ACDCJERK	Meaning.
Yes.	Movements in G0 use parameters ACCJERKG0 and DECJERKG0.
No.	Movements in G0 use parameters ACCJERK and DECJERK.

Example. Square sine acceleration.

Here are two examples so a stopped axis reaches G00FEED at maximum acceleration in a particular amount of time (0.5 seconds).

- The axis parameters may be set with maximum acceleration and minimum jerk.

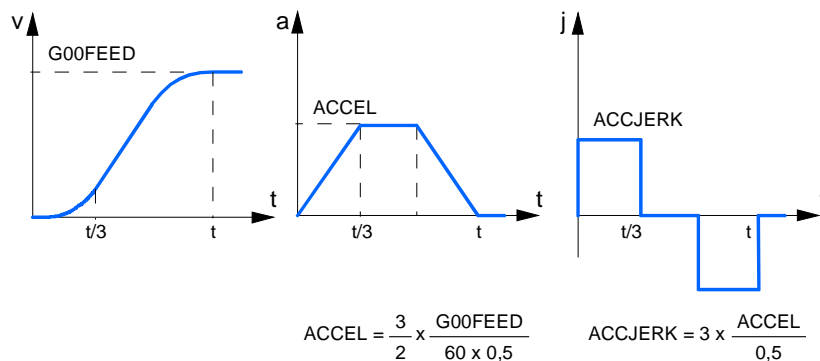


FAGOR AUTOMATION

CNC 8070

(REF: 1709)

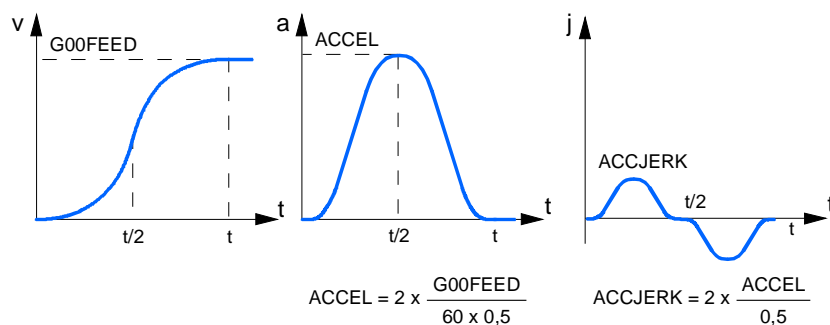
- The axis parameters may be set with less acceleration and greater jerk so the axis can reach the maximum acceleration faster and it stays longer at maximum acceleration.



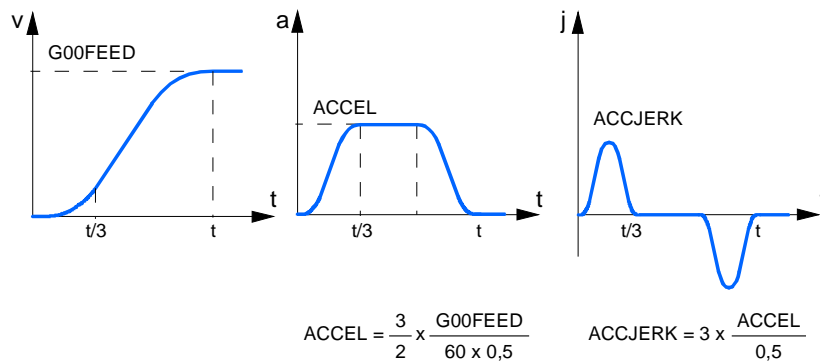
Example. Square sine acceleration.

Here are two examples so a stopped axis reaches G00FEED at maximum acceleration in a particular amount of time (0.5 seconds).

- The axis parameters may be set with maximum acceleration and minimum jerk.



- The axis parameters may be set with less acceleration and greater jerk so the axis can reach the maximum acceleration faster and it stays longer at maximum acceleration.



2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.10 Enable specific acceleration values for movements in G0.

GOACDCJERK

More parameters if G0.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.GOACDCJERK[set].xn

This parameter may be used to set some specific acceleration and jerk values for movements in G0. Otherwise, movements in G0 use the same acceleration and jerk values as movements in G1, G2 and G3.

Parameters to be used in G0 movements.	G0ACDCJERK	
	No	Yes
Linear acceleration (G0 movements). <ul style="list-style-type: none"> Acceleration of the first section. Acceleration of the second section. Acceleration changing speed. 	LACC1 LACC2 LFEED	LACC1G0 LACC2G0 LFEEDG0
Trapezoidal and square sine acceleration (G0 movements). <ul style="list-style-type: none"> Acceleration. Deceleration. Acceleration Jerk. Deceleration Jerk. 	ACCEL DECEL ACCJERK DECJERK	ACCELG0 DECELG0 ACCJERKG0 DECJERKG0

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.11 Linear acceleration (G0 movements).

LACC1G0

Acceleration of the first section (G0 movements).

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0.0000 to 600000000.0000 mm/s².

Possible values (2): From 0.00000 to 23622047.24409 inch/s².

Possible values (3): From 0.0000 to 600000000.0000 degrees/s².

Default value: 1000.0000 mm/s² / 39.37008 inch/s² / 1000.0000 degrees/s².

Associated variable: (V.)[ch].MPA.LACC1G0[set].xn

See parameter LACC2G0.

LACC2G0

Acceleration of the second section (G0 movements).

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0.0000 to 600000000.0000 mm/s².

Possible values (2): From 0.00000 to 23622047.24409 inch/s².

Possible values (3): From 0.0000 to 600000000.0000 degrees/s².

Default value: 1000.0000 mm/s² / 39.37008 inch/s² / 1000.0000 degrees/s².

Associated variable: (V.)[ch].MPA.LACC2G0[set].xn

Parameters LACC1G0 and LACC2G0 set the acceleration values when it is linear (parameter SLOPE TYPE) or when working with HSC in FAST mode. The CNC applies parameters LACC1G0 and LACC2G0 to G0 movements.

LFEEDG0

Acceleration changing speed (G0 movements).

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 200000.0000 mm/min.

Possible values (2): From 0 to 7874.01575 inch/min.

Possible values (3): From 0 to 36000000.0000 degrees/min.

Possible values (4): From 0 to 100000.0000 rpm.

Default value: 3000 mm/min / 118.11023 inch/min / 1080000 degrees/min / 3000 rpm.

Associated variable: (V.)[ch].MPA.LFEEDG0[set].xn

While accelerating, when reaching the feedrate defined in this parameter, it changes the acceleration from LACC1G0 to LACC2G0. While decelerating, when reaching the feedrate defined in this parameter, it changes the acceleration from LACC2G0 to LACC1G0.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.12 Trapezoidal and square sine acceleration (G0 movements).

ACCELGO

Acceleration (G0 movements).

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0.0000 to 600000000.0000 mm/s².

Possible values (2): From 0.00000 to 23622047.24409 inch/s².

Possible values (3): From 0.0000 to 600000000.0000 degrees/s².

Default value: 4000.0000 mm/s² / 157.48031 inch/s² / 4000 degrees/s².

Associated variable: (V.)[ch].MPA.ACCELGO[set].xn

See machine parameter DECELGO.

DECELGO

Deceleration (G0 movements).

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0.0000 to 600000000.0000 mm/s².

Possible values (2): From 0.00000 to 23622047.24409 inch/s².

Possible values (3): From 0.0000 to 600000000.0000 degrees/s².

Default value: 4000.0000 mm/s² / 157.48031 inch/s² / 4000 degrees/s².

Associated variable: (V.)[ch].MPA.DECELGO[set].xn

Parameters ACCEL and DECEL set the acceleration values when it is trapezoidal or square-sine (bell shaped) (parameter SLOPETYPE). The CNC applies parameters ACCEL and DECEL to the movements in G1, G2 and G3; movements in G0 depend on parameter G0ACDCJERK.

G0ACDCJERK	Meaning.
Yes.	Movements in G0 use parameters ACCELGO and DECELGO.
No.	Movements in G0 use parameters ACCEL and DECEL.

ACCJERKGO

Acceleration jerk (G0 movements).

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0.0010 to 6E+011 mm/s³.

Possible values (2): From 0.00004 to 2.362E+010 inch/s³.

Possible values (3): From 0.0010 to 6E+011 mm/s³.

Default value: 55000.000 mm/s³ / 2165.35433 inch/s³ / 55000.000 grados/s³.

Associated variable: (V.)[ch].MPA.ACCJERKGO[set].xn

See machine parameter DECJERKGO.

DECJERKGO

Deceleration jerk (G0 movements).

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0.0010 to 6E+011 mm/s³.

Possible values (2): From 0.00004 to 2.362E+010 inch/s³.

Possible values (3): From 0.0010 to 6E+011 mm/s³.

Default value: 55000.000 mm/s³ / 2165.35433 inch/s³ / 55000.000 grados/s³.

Associated variable: (V.)[ch].MPA.DECJERKGO[set].xn

Parameters ACCJERKGO and DECJERKGO set the slope of the acceleration and deceleration. Both parameters help limit the acceleration changes so the machine runs more smoothly in small speed increments or decrements and with FFGAIN values close to 100%. The lower the value assigned to these parameters, the smoother the machine response, but the acc/dec time will increase.

2.

MACHINE PARAMETERS. Work sets. Machine parameters for the axes.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

The CNC applies parameters ACCEL and DECEL to the movements in G1, G2 and G3; movements in G0 depend on parameter G0ACDCJERK. The CNC ignores these parameters in threading movements (G33) and with HSC FAST.

G0ACDCJERK	Meaning.
Yes.	Movements in G0 use parameters ACCJERKG0 and DECJERKG0.
No.	Movements in G0 use parameters ACCJERK and DECJERK.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.13 Configuration of the HSC mode.

HSC

Parameter table of the HSC mode.

This parameter shows the table to define the HSC work mode.

HSC	
CORNERACC	Maximum acceleration permitted at the corners.
CURVACC	Maximum contouring acceleration permitted.
CORNERJERK	Maximum Jerk permitted at the corners.
CURVJERK	Maximum contouring Jerk permitted.
FASTACC	Maximum acceleration permitted (FAST mode).
MAXERROR	Position error.
CONTERROR	Tolerance per axis.

These parameters help increase or decrease the acceleration and jerk limits on curved path, due to the centripetal acceleration and at path joints, without affecting the way the axis accelerates or decelerates.

CORNERACC

Maximum acceleration permitted at the corners.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 100000.0000 mm/s².

Possible values (2): From 0 to 3937.00787 inch/s².

Possible values (3): From 0 to 100000.0000 grados/s².

Default value: 0.

Associated variable: (V.)[ch].MPA.CORNERACC[set].xn

This parameter sets the maximum acceleration allowed for the axis during block transition. If this parameter is set with a -0· value, the maximum acceleration of the axis is respected.

CURVACC

Maximum contouring acceleration permitted.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 100000.0000 mm/s².

Possible values (2): From 0 to 3937.00787 inch/s².

Possible values (3): From 0 to 100000.0000 grados/s².

Default value: 0.

Associated variable: (V.)[ch].MPA.CURVAACC[set].xn

If this parameter is set with a -0· value, the maximum acceleration of the axis is respected.

CORNERJERK

Maximum Jerk permitted at the corners.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): De 0 a 1E+009 mm/s³.

Possible values (2): De 0 a 39370078.74016 inch/s³.

Possible values (3): De 0 a 1E+009 grados/s³.

Default value: 0.

Associated variable: (V.)[ch].MPA.CORNERJERK[set].xn

If this parameter is set with a -0· value, the maximum jerk of the axis is respected.

2.

MACHINE PARAMETERS.
Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

CURVJERK

Maximum contouring Jerk permitted.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): De 0 a 1E+009 mm/s³.

Possible values (2): De 0 a 39370078.74016 inch/s³.

Possible values (3): De 0 a 1E+009 grados/s³.

Default value: 0.

Associated variable: (V.)[ch].MPA.CURVJERK[set].xn

If this parameter is set with a ·0· value, the maximum jerk of the axis is respected.

FASTACC

Maximum acceleration permitted (FAST mode).

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 100000.0000 mm/s².

Possible values (2): From 0 to 3937.00787 inch/s².

Possible values (3): From 0 to 100000.0000 grados/s².

Default value: 0.

Associated variable: (V.)[ch].MPA.FASTACC[set].xn

If this parameter is set with a ·0· value, the maximum acceleration of the axis is respected.

MAXERROR

Position error.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 9999.99999 mm.

Possible values (2): From 0 to 3937.00787 inch.

Possible values (3): From 0 to 9999.99999 degrees.

Default value: 0.1000 mm / 0.00394 inch / 0.1000 grados.

Associated variable: (V.)[ch].MPA.MAXERROR[set].xn

Maximum axis position error in HSC when working outside the plane/trihedron.

CONTERROR

Tolerance per axis for smoothing the generated n-dimensional path.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Default value: 0.1000 mm⁰.

Associated variable: (V.)[ch].MPA.CONTERROR[set].xn

For 5-axis machining, this parameter may be used to set the tolerance per axis for smoothing the generated n-dimensional path. This parameter does not affect the 3D error when working with RTCP. We recommend setting this parameter on rotary axes.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

2.6.14 Home search.

The home search process depends on the characteristics of the machine.

IOTYPE

Reference mark (I0) type.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Not-distance-coded / increasing distance-coded / decreasing distance-coded.

Default value: Not-distance-coded.

Associated variable: (V.)[ch].MPA.IOTYPE[set].xn

This parameter indicates how the reference marks of the linear encoder are read with respect to the moving direction during the home search.

REFVALUE

Home position.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): Within ± 99999.9999 mm.

Possible values (2): Within ± 3937.00787 inch.

Possible values (3): Within ± 99999.9999 degrees.

Default value: 0.

Associated variable: (V.)[ch].MPA.REFVALUE[set].xn

The machine reference point must be defined in the following cases:

- The feedback system does not have distance-coded marks.
- The feedback system has distance-coded marks and leadscrew error compensation is being applied on that axis.

Set the home position referred to Machine Reference Zero.

REFSHIFT

Offset of the reference point (home).

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): Within ± 99999.9999 mm.

Possible values (2): Within ± 3937.00787 inch.

Possible values (3): Within ± 99999.9999 degrees.

Default value: 0.

Associated variable: (V.)[ch].MPA.REFSHIFT[set].xn

Sometimes, to readjust the machine, it is necessary to take down the feedback device, thus when putting back up, the new home point might not coincide with the previous one. Since the home point must still be the same, the difference between the new point and the old point must be assigned to parameter REFSHIFT. This way, when the axis finds the I0, it moves the distance indicated in REFSHIFT and at that point updates its coordinate to the value of REFVALUE.

The value to be set in REFSHIFT must be measured without leadscrew compensation being active because it is considered to be a correction to the encoder position value.

When the axes are position-Sercos; during parameter matching, the CNC sends the value of parameter REFSHIFT to the drive so it takes it into account; this way the CNC coordinate and that of the drive will be the same.

2.

MACHINE PARAMETERS. Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

REFFEED1**Fast home searching feedrate.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 200000.0000 mm/min.

Possible values (2): From 0 to 7873.992 inch/min.

Possible values (3): From 0 to 200000.0000 degrees/min.

Possible values (4): From 0 to 100000.0000 rpm.

Default value: 1000.0000 mm/min / 39.37001 inch/min / 1000.0000 degrees/min / 100.0000 rpm.

Associated variable: (V.)[ch].MPA.REFFFEED1[set].xn

See machine parameter REFFFEED2.

REFFEED2**Slow home searching feedrate.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 200000.0000 mm/min.

Possible values (2): From 0 to 7873.992 inch/min.

Possible values (3): From 0 to 200000.0000 degrees/min.

Possible values (4): From 0 to 100000.0000 rpm.

Default value: 100.0000 mm/min / 3.93700 inch/min / 100.0000 degrees/min / 10.0000 rpm.

Associated variable: (V.)[ch].MPA.REFFFEED2[set].xn

When the feedback system does not have distance-coded reference marks (I0, the home search is carried out at the feedrate indicated by "REFFEED1" until the home switch is reached. It then reverses its movement at the feedrate indicated by "REFFEED2" and it goes on until the CNC receives the reference marker pulse from the feedback device.

REFPULSE**Type of I0 pulse.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Positive / Negative.

Default value: Positive.

Associated variable: (V.)[ch].MPA.REFPULSE[set].xn

This parameter indicates the type of flank of the I0 signal that is used for home search.

POSINREF**Home search moving the axis to the reference point.**

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)[ch].MPA.POSINREF[set].xn

This parameter indicates whether the axis moves or not to the machine reference point (parameter REFVALUE) while homing. This movement is not necessary when the axis has internal absolute feedback (motor feedback).

On axes with absolute motor feedback, the CNC knows at all times the relative axis position per motor revolution. In these cases, while homing the axis, the CNC knows its position as soon as the home switch is pressed; hence not being necessary to move up to the reference point.

2.**MACHINE PARAMETERS.**

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

MAXDIFREF

Maximum position difference allowed to consider that there is no need to home again.

Parameter valid for linear and rotary axes.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0.0001 to 99999.0000 mm.

Possible values (2): From 0 to 3936.96850 inch.

Possible values (3): From 0.0001 to 99999.0000 degrees.

Default value: 0.0200 mm / 0.00079 inch / 0.0200 degrees.

Associated variable: (V.)[ch].MPA.MAXDIFREF[set].xn

This parameter indicates whether or not the axis must be homed depending on the maximum position difference allowed.

ABSOFF

Offset with respect to the distance-coded ref. mark.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): Within ± 99999.9999 mm.

Possible values (2): Within ± 3937.00787 inch.

Possible values (3): Within ± 99999.9999 degrees.

Default value: 0.

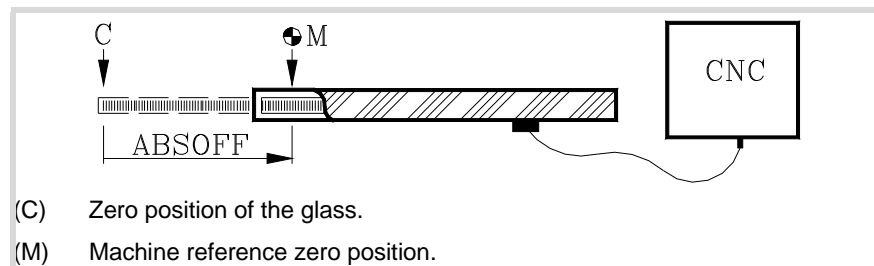
Associated variable: (V.)[ch].MPA.ABSOFF[set].xn

The CNC takes this parameter into account when I0TYPE = Distance coded, when ABSFEEDBACK = Yes or when the absolute feedback is an RCS-S module.

With linear encoders with distance-coded (I0 reference marks, it is possible to know the position of the machine by simply moving the axis a distance of 20 or 100 mm. After reading two consecutive distance-coded reference marks (20 or 100 mm away from each other), it is possible to know the axis position with respect to the zero point of the graduated glass (C).

In order for the CNC to show the position with respect to Machine Zero (M), this parameter must be assigned the position of the machine zero (M) with respect to the glass zero point (C).

The zero point of the glass (beginning of the distance code) may be in or out the scale's measuring length.

**EXTMULT**

External factor for distance-coded reference marks.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and simulated drives.

Possible values: From 0 to 65535.

Default value: 0.

Associated variable: (V.)[ch].MPA.EXTMULT[set].xn

The CNC takes this parameter into account if I0TYPE = Distance coded.

This parameter indicates the relationship between the mechanical period (of the graduation on the glass) and the electrical period (of the feedback signal) being applied to the CNC.

Example with a Fagor linear encoder.

A "FOX" type Fagor linear encoder has a graduation period of 100 μm (gap between lines) and an electrical signal period of 4 μm .

$$\text{EXTMULT} = 100 / 4 = 25$$

Values to be assigned for Fagor encoders with distance-coded I0.

Linear encoder.						EXTMULT
SOP SVOP	GOP	MOP MOC MOT MOVP	COP COC COT COVP		FOP	1
SOX SVOX	GOX	MOX MOVX	COX COVX		FOT	5
		MOY MOVY	COY			10
				LOP		1
				LOX		10
					FOX	25

Rotary encoder.			EXTMULT
HO	SO	90,000 pulses.	5
HO	SO	180.000 pulses.	10
HOP	SOP	18.000 pulses.	1

I0CODDI 1

Pitch between 2 fixed distance-coded reference marks.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 65535.

Default value: 1000.

Associated variable: (V.)[ch].MPA.I0CODDI1[set].xn

See machine parameter I0CODDI2.

I0CODDI 2

Pitch between 2 variable distance-coded reference marks.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 65535.

Default value: 1001.

Associated variable: (V.)[ch].MPA.I0CODDI2[set].xn

The CNC takes this parameter into account if I0TYPE = Distance coded.

The pitch between reference marks is defined in number of waves.

Example with a Fagor linear encoder.

Pitch between 2 fixed reference marks.	20.000 mm
Pitch between 2 variable reference marks.	20.020 mm
Period of the sinusoidal signal	20 µm
Number of waves between fixed reference marks	$20000 / (20 \times \text{EXTMULT}) = 1000$
Number of waves between variable reference marks	$20020 / (20 \times \text{EXTMULT}) = 1001$

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Values to be assigned for Fagor encoders with distance-coded I0.

Linear encoder.					I0CODDI1	I0CODDI2	
SOP SVOP	GOP	MOP MOC MOT MOVP	COP COC COT COVP		FOP	1000	1001
SOX SVOX	GOX	MOX MOVX	COX COVX		FOT	1000	1001
		MOY MOVY	COY			1000	1001
				LOP		2000	2001
				LOX		2000	2001
					FOX	1000	1001
Rotary encoder.					I0CODDI1	I0CODDI2	
HO	SO	90,000 pulses.			1000	1001	
HO	SO	180.000 pulses.			1000	1001	
HOP	SOP	18.000 pulses.			1000	1001	

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

2.6.15 Following error.

Following error is the difference between the theoretical position and the actual (real) position of the axis. The following error decreases when increasing the axis gain.

The more similar (identical) the following errors of interpolating axes are, the better the machining of curved sections will be on circular interpolations.

FLWEMONITOR

Type of monitoring of the following error (axis lag).

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: OFF / Standard / Linear.

Default value: Standard.

Associated variable: (V.)[ch].MPA.FLWEMONITOR[set].xn

This parameter determines how the following error will be monitored.

- If set to "OFF", the following error is not monitored, thus no error message will be issued.
- The "standard" monitoring monitors the following error all the time and an error message will be issued when it exceeds the value of parameter MAXFLWE if the axis is moving or MINFLWE if the axis is stopped.
- Linear monitoring performs a dynamic supervision that allows a percentage of following error. This percentage is set by parameter FEDYNFAC.

If this watch is canceled, the CNC shows on power-up a message indicating that this safety is disabled. This situation can only be allowed during setup; once setup is completed, this watch must be enabled.

MINFLWE

Maximum following error when stopped.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and Sercos drives.

Possible values (1): From 0.0001 to 99999.9999 mm.

Possible values (2): From 0 to 3937.00787 inch.

Possible values (3): From 0.0001 to 99999.9999 degrees.

Default value: 1.0000 mm / 0.03937 inch / 1.0000 degrees.

Associated variable: (V.)[ch].MPA.MINFLWE[set].xn

The CNC takes this parameter into account only if FLWEMONITOR other than OFF.

This parameter indicates the maximum amount of following error allowed when the axis is stopped. The MINFLWE value cannot be greater than 1/4 of the total axis travel (LIMIT+ to LIMIT-).

MAXFLWE

Maximum following error when moving.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and Sercos drives.

Possible values (1): From 0.0001 to 99999.9999 mm.

Possible values (2): From 0 to 3937.00787 inch.

Possible values (3): From 0.0001 to 99999.9999 degrees.

Default value: 30.0000 mm / 1.18110 inch / 30.0000 degrees.

Associated variable: (V.)[ch].MPA.MAXFLWE[set].xn

The CNC takes this parameter into account only if FLWEMONITOR other than OFF.

The meaning of this parameter depends on the value of FLWEMONITOR.

- With FLWEMONITOR = Standard, MAXFLWE indicates the maximum amount of following error allowed when the axis is moving.
- With FLWEMONITOR = Linear, MAXFLWE indicates the value starting at which the following error is dynamically monitored.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

FEDYNFAC**Percentage of following error deviation allowed.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and Sercos drives.

Possible values: from 0 to 100 %.

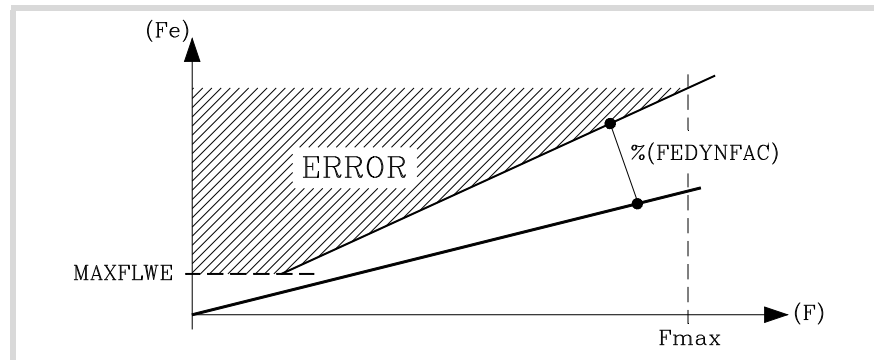
Default value: 50.

Associated variable: (V.)[ch].MPA.FEDYNFAC[set].xn

The CNC takes this parameter into account only if FLWEMONITOR other than OFF.

This parameter indicates the permitted percentage error; in other words the deviation of the real following error with respect to the theoretical one.

The CNC calculates the maximum and minimum following error (Fe) at all times depending on feedrate (F). If it is not within the permitted zone (shaded area of the figure), the CNC will issue the relevant error message.



Parameter MAXFLWE indicates the value starting at which the following error will be dynamically monitored.

ESTDELAY**Following error delay.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and Sercos drives.

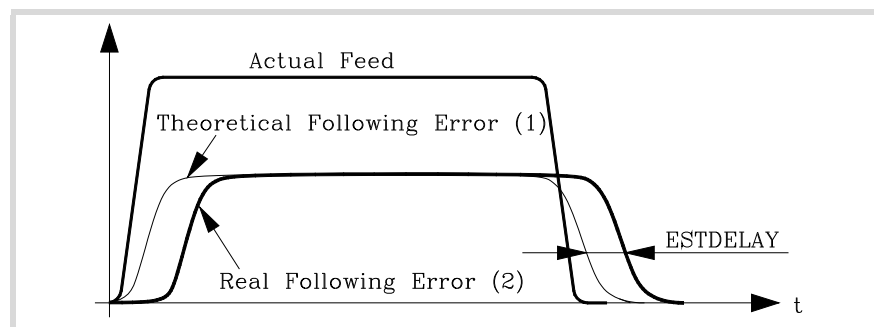
Possible values: From 0 to 1000000 ms.

Default value: 0.

Associated variable: (V.)[ch].MPA.ESTDELAY[set].xn

The CNC takes this parameter into account if FLWEMONITOR=Linear.

This parameter is used to define a delay applied when estimating the following error so the theoretical value (1) comes closer to the real one (2) thus avoiding undesired following error messages. This parameter is also used as the time required to issue the tendency error (run-away protection).



Setting parameter ESTDELAY requires activating the estimate of linear following error and using the oscilloscope to check variables (V.)A.FLWE.xn and (V.)A.FLWEST.xn. The value of parameter ESTDELAY must be slightly higher than the maximum delay between the two signals. When calculating this parameter, bear in mind that the worst scenario is when reversing the movement in G00.

2.

MACHINE PARAMETERS.
Machine parameters for the axes. Work sets.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

INPOMAX

Time to get in position.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 1000000 ms.

Default value: 0.

Associated variable: (V.)[ch].MPA.INPOMAX[set].xn

Parameter INPOMAX limits (maximum time) the time the axis needs to get in position. This parameter provides the possibility to watch the positioning of the axis ensuring that it gets in position in a specific amount of time and issuing an error if otherwise.

INPOTIME

Minimum time to stay in position.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 0 to 1000000 ms.

Default value: 0.

Associated variable: (V.)[ch].MPA.MPA.INPOTIME[set].xn

Parameter INPOTIME sets the time the axis must stay in the in-position zone so the CNC considers it to be "in position".

Parameters INPOMAX and INPOTIME ensure that when working with dead axes (axes only controlled while moving), the movement will be completed when they are in position.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

2.6.16 Axis lubrication.

DISTLUBRI

Distance to move to lubricate the axis.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values (1): From 0 to 2000000000 mm.

Possible values (2): From 0 to 78739920 inch.

Possible values (3): From 0 to 2000000000 degrees.

Default value: 0 (no lubrication).

Associated variable: (V.)[ch].MPA.DISTLUBRI[set].xn

The PLC reads this parameter in mm instead of doing it in tenths of a micron (0.0001 mm).

The lubrication signal is activated after travelling the distance indicated in this parameter. The CNC logic inputs and outputs: LUBR(axis), LUBRENA(axis) and LUBROK(axis) must be used in order for the PLC lubricates the axes and gears.

- 1 The LUBRENA(axis) mark indicates whether this feature is to be used or not.
- 2 When the axis has traveled the distance set by parameter DISTLUBRI, the CNC turns the LUBR(axis) mark on to let the PLC know that it must lubricate the axis.
- 3 After lubricating the axis, the PLC sets the LUBROK(axis) mark high (=1) to let the CNC know that the axis has been lubricated.
- 4 The CNC sets the LUBR(axis) mark low (=0) and resets its count to "0".

2.

MACHINE PARAMETERS.
Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.17 Module configuration (rotary axes and spindle).

MODUPLIM

Upper limit of the module.

Parameter valid for rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 360.

Associated variable: (V.)[ch].MPA.MODUPLIM[set].xn

See machine parameter MODLOWLIM.

MODLOWLIM

Lower limit of the module.

Parameter valid for rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

Associated variable: (V.)[ch].MPA.MODLOWLIM[set].xn

The CNC takes these parameters into consideration only when AXISMODE = Module.

For a reading within $\pm 180^\circ$, set MODUPLIM = 180° and MODLOWLIM = -180° .

MODNROT

Module error. Number of turns.

Parameter valid for rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: From 1 to 32767 turns.

Default value: 1.

Associated variable: (V.)[ch].MPA.MODNROT[set].xn

See machine parameter MODERR.

MODERR

Module error. Number of increments.

Parameter valid for rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Within ± 32767 .

Default value: 0.

Associated variable: (V.)[ch].MPA.MODERR[set].xn

The CNC takes these parameters into consideration only when AXISMODE = Module and MODCOMP = Yes. These parameters indicate the compensation to be applied to compensate for an inexact axis resolution. The CNC applies module compensation throughout the entire revolution.

Parameter MODERR indicates the amount of error to be compensated when the axis has rotated the revolutions indicated in parameter MODNROT. This correction is necessary, for example, when using a 1024 line encoder whose parameter has been set to 1000 pulses.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

2.6.18 Spindle speed.

SZERO

Speed considered "0 rpm".

Parameter valid for spindles.

Parameter valid for analog drives and simulated drives.

Possible values: From 0 to 100000 rpm.

Default value: 1.

Associated variable: (V.)[ch].MPA.SZERO[set].xn

This parameter indicates the rpm value below which the spindle is considered to be stopped. The CNC uses this parameter to switch the spindle to C axis mode and also to accept the velocity command programmed with SANALOG when the spindle is controlled from the PLC. This parameter can also be used from the PLC to permit opening the machine doors.

POLARM3

Sign of the velocity command for M3.

Parameter valid for spindles.

Parameter valid for analog drives, velocity-Sercos drives and simulated drives.

Possible values: Positive / Negative.

Default value: Positive.

Associated variable: (V.)[ch].MPA.POLARM3[set].xn

See machine parameter POLARM3.

POLARM4

Sign of the velocity command for M4.

Parameter valid for spindles.

Parameter valid for analog drives, velocity-Sercos drives and simulated drives.

Possible values: Positive / Negative.

Default value: Negative.

Associated variable: (V.)[ch].MPA.POLARM4[set].xn

Lathe and mill models.

Parameters POLARM3 and POLARM indicate the direction of the velocity command and, therefore, the spindle turning direction for functions M3 and M4 respectively.

The CNC offers the following variable to reverse, via part-program or PLC, the direction of the velocity command set in this parameter.

Variable.	Meaning.
(V.)[ch].A.POLARITY.sn (V.)[ch].SP.POLARITY.sn	This variable may be used to reverse the meaning of parameters POLARM3 and POLARM4. The variable does not modify the values of the machine parameters.

Laser model.

Parameters POLARM3 and POLARM indicate the direction of the velocity command for functions M3 and M4 respectively. Both parameters must be set to "Positive".

2.

MACHINE PARAMETERS. Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.19 Analog command setting.

SERVOOFF

Offset compensation.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for an analog drive.

Possible values: Within ± 32767 .

Default value: 0.

Associated variable: (V.)[ch].MPA.SERVOOFF[set].xn

Velocity command applied as offset to the drive. The velocity command is given in D/A converter units that could be any integer within ± 32767 ; a value of ± 32767 corresponds to an analog velocity command of $\pm 10V$.

SERVOOFF	Velocity command.
-32767	-10 V.
...	...
-3277	-1 V.
...	...
1	0,3 mV.
...	...
3277	1 V.
...	...
32767	10 V.

MINANOUT

Minimum velocity command.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for an analog drive.

Possible values: From 0 to 32767.

Default value: 0.

Associated variable: (V.)[ch].MPA.MINANOUT[set].xn

The velocity command is given in D/A converter units that could be any integer between 0 and 32767; a value of ± 32767 corresponds to an analog velocity command of 10V.

MINANOUT	Velocity command.
1	0,3 mV.
...	...
3277	1 V.
...	...
32767	10 V.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

2.6.20 Number of the analog output and of the feedback input associated with the axis.

ANAOUTTYPE

Type of analog output associated with the axis.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for an analog drive.

Possible values: Remote CAN / Drive / Sercos Counter.

Default value: Remote CAN.

Associated variable: (V.)[ch].MPA.ANAOUTTYPE[set].xn

This parameter is only valid for analog axes and indicates where the analog output is located; in the remote CAN modules, in a Sercos drive or in an RCS-S module (Sercos counter).

ANAOUTID

Number of the analog output associated with the axis.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for an analog drive.

Possible values: From 1 to 16 for remote modules / Sercos drives, from 101 to 132 or from 201 to 232 / Sercos counter, rom 1 to 32.

Default value: 0.

Associated variable: (V.)[ch].MPA.ANAOUTID[set].xn

The velocity command for an analog axis may be taken from an analog output of the remote CAN modules or of a Sercos drive. or of an RCS-S module (Sercos counter). In either case, this parameter indicates the analog output used for the command.

When two axes will not be in the configuration at the same time, the CNC allows using the same analog output for both axes.

Managing the analog command from CAN remote modules.

In this case, this parameter indicates the number of the analog output used for the command. The analog output modules are numbered following the logic order of the remote groups (rotary switch of the power supply). If there are several analog-output modules in each group, the order is from top to bottom and from left to right. The analog outputs of the first module will be 1 through 4, those of the second module 5 through 8 and so on.

Managing the analog command from Sercos drives.

In this case, this parameter indicates the number of the analog output and the drive to be used. The parameter is set with a three-digit number; the first digit indicates the number of the analog output to be used (1 or 2) and the other two digits indicate the logic address of the drive (1 through 32). The "address" rotary selector switch sets the logic address.

For example, if the parameter is with a value of ·107·, it indicates that the CNC uses analog input ·1· of the drive whose logic address is ·7· In this case, to consult the voltage at this output, the variable must be written as V.G.ANAO[107].



The parameters OP1 and OP2 of the drive must be set to 0.

Managing the analog command from the RCS-S module.

In this case, this parameter indicates the number of the analog output used for the command. The RCS-S modules are numbered according to their logic order (parameter SERCOUNTID). The analog outputs of the first module will be 1 through 4, those of the second module 5 through 8 and so on.

2.

MACHINE PARAMETERS. Work sets. Machine parameters for the axes.

COUNTERTYPE**Type of feedback input for the axis.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and velocity-Sercos drives.

Possible values: Remote CAN / Drive / Sercos Counter / Local.

Default value: Remote.

Associated variable: (V.)[ch].MPA.COUNTERTYPE[set].xn

Meaning for an analog axis.

This parameter indicates the location of the axis feedback input; at the remote modules, at the free feedback of a Sercos drive, in an RCS-S module (Sercos counter) or at the local feedback inputs of the central unit. When the feedback input is at a drive, the CNC does not allow homing the axis.



When the axis feedback is at a Sercos drive, at this drive set parameter PP5=-0,0001.

Meaning for a velocity-Sercos axis.

This parameter indicates the location of the axis external feedback (direct) input; at the remote modules, at the free feedback of a Sercos drive or at the local feedback inputs of the central unit.

The axis can use the second free feedback input of that drive or of another one. When the axis uses the second feedback input of another drive, the CNC does not allow homing the axis.



When the internal (motor) feedback and the external (direct) feedback of an axis are at different drives, set parameter PP5=-0,0001 at the drive that receives the external feedback.

COUNTERID**Number of the feedback input for the axis.**

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and velocity-Sercos drives.

Possible values: Remote modules, from 1 to 40 / Sercos drive, from 1 to 32 / RCS-S module, from 1 to 32 / Local inputs, from 1 to 2.

Default value: 0.

Associated variable: (V.)[ch].MPA.COUNTERID[set].xn

The feedback input of an analog axis and the input for the external (direct) feedback of a Sercos axis may be located at the remote modules, at the free feedback input of a Sercos drive, in an RCS-S module (Sercos counter) or at the local feedback inputs of the central unit. Depending on the location of the feedback input, this parameter will indicate which feedback input will be used (either local or remote) or the drive/counter number. When two axes will not be in the configuration at the same time, the CNC allows using the same feedback input for both axes.

Feedback input in CANfagor remote modules.

A counter module is used as feedback input in the remote modules. This parameter indicates the number of the feedback input being used.

The counter modules are numbered following the logic order of the remote groups (rotary switch of the power supply). If there are several Counter modules in each group, the order is from top to bottom and from left to right. The counters of the first module will be 1 through 4, those of the second module 5 through 8 and so on.

Feedback input in Sercos drives.

In this case, the second feedback input of the drive will always be used. The parameter indicates the logic address of the drive (from 1 to 32). The "address" rotary selector switch sets the logic address.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Feedback input of an RCS-S module.

This parameter indicates the number of the feedback input being used.

The RCS-S modules are numbered according to their logic order (parameter SERCOUNTID). The feedback inputs of the first module will be 1 through 4, those of the second module 5 through 8 and so on.

Local feedback input .

Central units ICU and MCU have two feedback inputs. This parameter indicates the feedback input being used.

Laser model.

In the laser model, the spindle feedback input will be 0, since there is no real feedback.

2.**MACHINE PARAMETERS.**

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.21 Set the drive associated with the axes of a multi-axis group.

DRIVESET

Set of the drive to be used.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for a Sercos drive.

Possible values: From 0 to 8.

Default value: 0.

Associated variable: (V.)[ch].MPA.DRIVESET[set].xn

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). The CNC sends the changes of Kv, feed-forward (parameter FFGAIN) and AC-forward (parameter ACFGAIN) to the set indicated in this parameter.

Being parameter matching active, the CNC sends the parameters of set ·1· of the master axis to the set ·0· and to the gear ·1· (first) of the drive. For the rest of the sets of the master axis, the CNC only sends the gear data to the drive; set ·2· of the CNC to the gear 2 of the drive and so on. At the master axis of multi-axis group, in order for parameter matching to work properly, this parameter must coincide with the set number where it is; i.e. DRIVESET=1 in set ·1·, DRIVESET=2 in ·2· and so on. At the slave axis of a multi-axis group, this parameter indicates the gear to be used at the drive. If the axis belongs to a multi-axis group, the parameter does not admit a ·0· (zero) value.

2.6.22 Feedback type.

FEEDBACKTYPE

Feedback type of the RCS-S module.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and velocity-Sercos drives.

Possible values (1): Local inputs; TTL / differential TTL / Vpp / SSI.

Possible values (2): Sercos counter; TTL / differential TTL / Vpp / SSI / FeeDat / EnDat.

Default value: Differential TTL.

Associated variable: (V.)[ch].MPA.FEEDBACKTYPE[set].xn

Cuando la entrada de captación del eje se encuentra en la entrada local o en un módulo RCS-S (contadora Sercos), este parámetro define el tipo de señal. El parámetro muestra las opciones posibles para cada caso; entrada local o módulo RCS.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.23 Encoder Information.

ENCODERINFO Encoder Information.

When the type of feedback is FeeDat or EnDat (FEEDBACKTYPE parameter), this parameter displays a parameter table with the encoder information.

Parameter.	Meaning.
NAME	Encoder nomenclature.
TYPE	Type of encoder.
RESOLUTION	Encoder resolution.
SERIALNUMBER	Encoder serial number.
DATALength	Number of data bits in the transmission.



The CNC obtains this information from the encoder itself, and therefore these parameters cannot be modified. If the data read from the encoder do not match those saved in the parameter table, the CNC will display a new table with both groups of values. At the same time, it will display a softkey to save the new data in the table. The CNC will only display this softkey if the data read from the encoder are valid (the encoder is connected and correctly identified) and they do not match those already saved.

NAME Encoder nomenclature.

Associated variable (value saved in the table): (V.)[ch].MPA.DE_NAME[set].xn

Associated variable (value read on the encoder): (V.)[ch].A.CDE_NAME[set].xn

Parameter cannot be modified; the CNC reads this parameter from the encoder.

- If it is a Fagor encoder, the name of the encoder.
- If it is an EnDat encoder, the "EnDat" text.

TYPE Encoder type.

Associated variable (value saved in the table): (V.)[ch].MPA.DE_TYPE[set].xn

Associated variable (value read on the encoder): (V.)[ch].A.CDE_TYPE[set].xn

Parameter cannot be modified; the CNC reads this parameter from the encoder.

Encoder type (multi-turn absolute rotary, absolute rotary, absolute linear, etc).

RESOLUTION Encoder resolution.

Associated variable (value saved in the table): (V.)[ch].MPA.DE_RESOLUTION[set].xn

Associated variable (value read on the encoder): (V.)[ch].A.CDE_RESOLUTION[set].xn

Parameter cannot be modified; the CNC reads this parameter from the encoder.

- If the encoder is linear, nanometer resolution.
- If the encoder is angular, the number of pulses per turn.

SERIALNUMBER Encoder serial number.

Associated variable (value saved in the table): (V.)[ch].MPA.DE_SERIALNUMBER[set].xn

Associated variable (value read on the encoder): (V.)[ch].A.CDE_SERIALNUMBER[set].xn

Parameter cannot be modified; the CNC reads this parameter from the encoder.

2.

MACHINE PARAMETERS.
Machine parameters for the axes. Work sets.

DATALENGTH

Number of data bits in the transmission.

Associated variable (value saved in the table): (V.)[ch].MPA.DE_DATALENGTH[set].xn

Associated variable (value read on the encoder): (V.)[ch].A.CDE_DATALENGTH[set].xn

Parameter cannot be modified; the CNC reads this parameter from the encoder.

2.6.24 EnDat transmission format.

ENDAT

EnDat parameter table.

When the feedback type (FEEDBACKTYPE parameter) is EnDat, this parameter displays the parameter table to configure the transmission.

Parameter.	Meaning.
ENDATCLKFREQ	EnDat communication frequency.

ENDATCLKFREQ

EnDat communication frequency.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drives and velocity-Sercos drives.

Possible values: 250kHz / 500 kHz / 1 MHz / 2 MHz / 4 MHz.

Default value: 2 MHz.

Associated variable: (V.)[ch].MPA.ENDATCLKFREQ[set].xn

When the feedback type is EnDat (FEEDBACKTYPE parameter), this parameter defines the communication frequency. The frequency depends on the cable length; the shorter the cable, the greater the frequency can be.

It is recommended to use the default frequency (2 MHz), which is viable for cables of up to 100 m in length. To select the speed for 4 MHz, the encoder must be capable of operating this speed and the cable must be shorter than 50 m in length.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.25 SSI Feedback type.

SSITYPE

SSI feedback type connected to the corresponding feedback.

Possible values: Fagor SSITYPE / G user / User.

Default value: User.

Associated variable: (V.)[ch].MPA.SSITYPE[set].xn

When FEEDBACKTYPE=SSI, this parameter is used to set the format of the SSI transmission. The parameter offers a number of preset types for connecting the most common linear and rotary encoders. If the encoder is not listed, the user option may be selected to set the properties of the SSI communication.

Value.	Meaning.
Fagor LA	Fagor linear encoder.
Fagor GA Fagor SA Fagor SVA	Fagor linear encoder.
Fagor HA-27-D200	Fagor encoder.
Fagor HA-23-D90 Fagor SA-23-D90 Fagor SA-23-D170	Fagor encoder.
ABSIND (inductosyn LIN+ABS)	Absolute linear feedback.
ABSIND (inductosyn ROT+ABS)	Absolute rotary feedback in one encoder revolution.
ABSIND (resolver)	Absolute rotary feedback in one revolution.
ABSIND (inductosyn LIN)	Absolute linear feedback in the pitch. Homing the linear encoder required.
ABSIND (inductosyn ROT)	Absolute rotary feedback in the pitch. Homing the linear encoder required.
G user	User configuration.
User	User configuration.

SSI

SSI (Synchronous Serial Interface) parameter table.

This parameter shows the parameter table to set the SSI transmission format. Depending on the type of linear encoder selected, (parameter SSITYPE), some of these values will be preset and cannot be changed. All values may be used for the "user" type linear encoder.

Example for a Fagor absolute linear encoder (Fagor LA).

Example of parameter setting for connecting the Fagor LA absolute linear encoder, set with default parameters (factory set).

SSICLKREQ = 1 / T = 400.	400 kHz.
SSIDATALENGTH = n	Bit length of the position value (coordinate).
SSICRCBITS = 0	There are no CRC bits.
SSISTARTBITS = 0	There are no start bits.
SSIPACKFORMAT = 3	Data + Alarm.
SSIALARMBITS = 1	1 alarm/acknowledge bit.
SSIDATAMODE = 1	MSB First.
SSIALARMLEVEL = 1	The acknowledge is indicated with data at 0; therefore, an alarm signal is produced when the bit is at 1.

2.

MACHINE PARAMETERS.
Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

SSICLKREQ

SSI communication frequency.

Possible values: From 59 to 7500 kHz.

Default value: 150 kHz.

Associated variable: (V.)[ch].MPA.SSICLKREQ[set].xn

Frequency of the clock for SSI communication. The recommended maximum frequency depends on the length of the cable.

Length.	Maximum frequency.
Up to 20 meters.	600 kHz.
Up to 50 meters.	400 kHz.
Up to 75 meters.	300 kHz.
Up to 100 meters.	250 kHz.

SSIDATAFORMAT

Parameters to set the format of the SSI transmission reading data.

The position read via SSI transmission may be received in different formats.

- The following parameters set the amount of SSI data that will be read.

Parameter.	Meaning.
SSIDATALENGTH	Number of bits of the SSI transmission that make up the position value (coordinate).
SSICRCBITS	Number of bits, if any, for some kind of transmission validity check (CRC, checksum, parity).
SSIALARMBITS	Number of bits, if any, for transmission OK check. The feedback device can send one or more bits indicating an alarm condition if something goes wrong or one or more acknowledgment bits if the transmission is successful.
SSISTARTBITS	Number of bits, if any, to wait for before receiving position data.

- The following parameters set the order and format used for the system to send the data.

Parameter.	Meaning.
SSIPACKFORMAT	Order in which the encoder sends the data. Whether the position data, the CRC or the alarm is received first and in which order. If start bits have been programmed, it is assumed that they will be received first.
SSIDATAMODE	Order in which the most significant bit is transmitted. <ul style="list-style-type: none"> • SSIDATAMODE = 0; the LSB (Least Significant Bit) is the first one. • SSIDATAMODE = 1; the MSB (Most Significant Bit) is the first one.

- The following parameters set how to interpret the bits that determine that the SSI transmission has been successful.

Parameter.	Meaning.
SSIALARMLEVEL	Values that may make the encoder transmit one or more bits indicating some kind of error.
SSICRCTYPE	Type of calculation to do when the encoder transmits extra data besides the position value in order to run a data coherence check.

SSIDATALENGTH

Number of bits of the SSI transmission that make up the position value (coordinate).

Possible values: From 0 to 255.

Default value: 0.

Associated variable: (V.)[ch].MPA.SSIDATALENGTH[set].xn

Number of bits of the SSI transmission that make up the position value (coordinate). For example, 32 for Fagor absolute linear encoders with the default parameters.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

SSIPACKFORMAT**SSI package format.**

Possible values: Data / Data-CRC / CRC-Data / Data-Alarm / Alarm-Data / Data-CRC-Alarm / Alarm-Data-CRC / Data-Alarm-CRC / Alarm-CRC-Data / CRC-Data-Alarm / CRC-Alarm-Data.

Default value: Data.

Associated variable: (V.)[ch].MPA.SSIPACKFORMAT[set].xn

This parameter indicates the order in which the different data types of the SSI transmission will be received. Depending on parameters SSICRCBITS and SSIALARMBITS, only the possible options will be displayed. The order indicated by the parameter is the one that corresponds to the SSI transmission sequence. If start bits have been programmed, the CNC assumes that they will be received first.

Example:

SSICRCBITS = 5

SSIALARMBITS = 1

SSIDATABITS = 23

SSIPACKFORMAT = Alarm - Data - CRC

The CNC expects the SSI transmission to be a sequence of bits where the first one is the alarm bit followed by 23 position bits or data bits and finally the 5 CNC bits.

SSICRCTYPE**CRC type.**

Possible values: Do not calculate CRC / Fagor Checksum / INDUCTOSYN Checksum.

Default value: Do not calculate CRC

Associated variable: (V.)[ch].MPA.SSICRCTYPE[set].xn

When the encoder transmits extra data besides the position data, this parameter sets the type of calculation to do to check data coherence. This parameter only makes sense when SSICRCBITS is other than 0.

Value.	Meaning.
Do not calculate CRC.	Even if the CNC receives the CRC bits, they are not processed at all and, therefore, no errors are reported when the transmission is not correct. It is not a recommendable option and it must only be used during setup.
Fagor checksum.	Data checking algorithm used in Fagor Automation feedback systems.
INDUCTOSYN checksum.	Data checking algorithm used when connecting to an INDUCTOSYN module.

SSICRCBITS**Number of CRC bits of the SSI transmission to check that the transmission is valid.**

Possible values: From 0 to 31.

Default value: 0.

Associated variable: (V.)[ch].MPA.SSICRCBITS[set].xn

Number of bits, if any, for the transmission validity check (CRC, checksum, parity).

For example, Fagor absolute linear encoders may be configured to transmit 5 CNC bits together with the position value. If a linear encoder is configured in this mode, program SSIDATALENGTH=27 and SSICRCBITS =5.

SSISTARTBITS**Number of start bits.**

Possible values: From 0 to 3.

Default value: 0.

Associated variable: (V.)[ch].MPA.SSISTARTBITS[set].xn

Number of bits, if any, to wait for before receiving position data.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

SSIALARMBITS

Number of alarm bits.

Possible values: From 0 to 2.

Default value: 0.

Associated variable: (V.)[ch].MPA.SSIALARMBITS[set].xn

The feedback device can send one or more bits indicating an alarm condition if something goes wrong or one or more acknowledgment bits if the transmission is successful.

SSIALARMLEVEL

Level of the alarm bits to issue an error.

Possible values: From 0 to 3.

Default value: 0.

Associated variable: (V.)[ch].MPA.SSIALARMLEVEL[set].xn

Value that the alarm/acknowledge bits for an error condition to occur. This parameter only makes sense when SSIALARMBITS is other than 0.

Example:

If an encoder indicates an error condition with 2 bits where the first one is a 1 and the second a 0
 SSIALARMBITS = 2
 SSIALARMLEVEL = 2.

Example:

If an encoder has a transmission-OK acknowledgment bit and it indicates it with a logic 0.
 SSIALARMBITS = 1
 SSIALARMLEVEL = 1. Because a 0 means that everything is OK.

SSIDATAMODE

Transmission mode.

Possible values: From 0 to 1.

Default value: 0.

Associated variable: (V.)[ch].MPA.SSIDATAMODE[set].xn

Order in which the most significant bit is transmitted.

Value.	Meaning.
0	The LSB (Least Significant Bit) is the first one.
1	The MSB (Most Significant Bit) is the first one.

STARTDELAY

Number of clocks to wait between the first down flank and the first up flank.

Possible values: From 0 to 255.

Default value: 0.

Associated variable: (V.)[ch].MPA.STARTDELAY[set].xn

This parameter sets the number of SSI clocks to wait for since the first SSI down flank until the first up flank is generated. This value is needed to implement the wait time to convert the signal in some encoders.

Example:

CLOCKFREQ = 400 kHz
 STARTDELAY = 3
 Waiting time = $(1/400 \cdot 1000) \cdot 3 = 7,5$ clocks.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

SSIRESOL**Digital counting (feedback) resolution.**

Possible values: From 1 to 999999999.

Default value: 10000.

Associated variable: (V.)[ch].MPA.SSIRE SOL[set].xn

Number of digital counting units contained in a pitch.

Example:

Fagor absolute linear encoder with a pitch of 20 microns and a digital resolution of 1 μ .

$$\text{SSIRESOL} = 20 \mu / 0.1 \mu = 200.$$

Example:

For a 8192 ppt encoder and a 10 mm pitch screw.

$$\text{PITCH} = 10 \text{ mm.}$$

$$\text{SSIRESOL} = 8192.$$

The resolution of the axis will be; $10 / 8192 = 0.0012 \text{ mm.}$

Example:

For SSITYPE = ABSIND (inductosyn ROT+ABS), for a 2 or 4-degree turn and depending on whether it is a high resolution or low resolution encoder, the counting increment is 10000 units.

$$\text{PITCH} = 2^\circ \text{ or } 4^\circ$$

$$\text{SSIRESOL} = 10000$$

The resolution of the axis will be; $2^\circ \text{ or } 4^\circ / 10000 = 0.0002 \text{ or } 0.0004 \text{ degrees.}$

2.**MACHINE PARAMETERS.**

Machine parameters for the axes. Work sets.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.6.26 Delay estimate at the drive.

AXDELAY

Parameter value at the drive.

Parameter valid for linear and rotary axes and spindles.

Parameter valid for analog drive, Sercos and simulated.

Possible values: Standard / From 0 to 127 cycles.

Default value: Standard.

Associated variable: (V.)[ch].MPA.AXDELAY[set].xn

Parameter AXDELAY is an estimate of the drive's delay when applying the velocity command sent by the CNC. The CNC uses this parameter to compensate for the delay difference between the axes of a channel so the command reaches all the axes at the same time and the movement of the axes starts and ends at the same time. Depending on the delay set, the CNC calculates how far in advance it must send the velocity command to the drive.

AXDELAY	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.

The CNC also takes this delay into account when calculating the anticipation time (ADVINPOS mark), in thread repair and when calculating the braking distance for the probe.

2.6.27 Correction of bus and drive delay.

TRANSDELAY

Correction of the delay inserted by the bus and the drive.

Parameter valid for linear and rotary axes.

Parameter valid for analog drives, velocity-Sercos drives and simulated drives.

Possible values: From 0 to 4*LOOPTIME μ s.

Default value: 0.

Associated variable: (V.)[ch].MPA.TRANSDELAY[set].xn

The TRANSDELAY parameter enables the bus and drive delay to be compensated.

- Analog axes (feedback through Can). About 2.5 sampling periods.
- Velocity Sercos axes. About 1.2 sampling periods.
- Position-Sercos axes Not used.

This parameter affects the adjustment of the ACFWFACTOR and ACFGAIN and may be used to obtain better results in speed variations with less following error (axis lag).

Peculiarities of the laser model.

On laser machines with gap control, the TRANSDELAY parameter of the axis associated with the sensor must have a value of 0 (zero) in order for the machine response to be the same with and without gap control. The compensation for the delays is implemented through the ACFWFACTOR parameter, which will have a higher value than if the TRANSDELAY parameter is applied (approximately the sum of TRANSDELAY and ACFWFACTOR in an analog or Sercos speed adjustment). In an adjustment without the TRANSDELAY parameter, the following error peaks in accelerations and stops will be greater than with an adjustment using this parameter.

2.

MACHINE PARAMETERS.

Machine parameters for the axes. Work sets.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.7 Machine parameters for JOG mode.

2.7.1 Handwheel configuration.

NMPG

Number of handwheels connected to the CNC.

Possible values: From 0 to 12.

Default value: 0 (there are no handwheels).

Associated variable: (V.)MPMAN.NMPG

Electronic handwheels may be used to move the axes. Depending on the type of handwheel, The CNC may have general handwheels to move any axis or individual handwheels that will only move their associated axes. Several individual and general handwheels may be used at the same time, up to a total of 12 handwheels. The portable operator panel HBL5 does not count as handwheel when setting this parameter.

To set the resolution of each handwheel, the distance it moves at each switch position, set the parameter MPGRESOL.

MANPG

Table of handwheels.

This parameter shows the list of available handwheels.

MANPG n

Handwheel configuration.

This parameter shows the table to configure the handwheels. Each table shows the following parameters to configure it.

MANPG n	
COUNTERTYPE	Handwheel feedback type.
COUNTERID	Feedback input for the handwheel.
HWFBTYPE	Feedback type associated with the handwheel input.
MPGAXIS	Name of the axis associated with the handwheel.

COUNTERTYPE

Type of feedback input for the handwheel.

Possible values: Remote / Keyboard / Local / Sercos Counter.

Default value: Remote.

Associated variable: (V.)MPMAN.COUNTERTYPE[hw]

Parameter included in the MANPG table.

This parameter indicates the location of the handwheel feedback input; at the counter modules of the remote CAN groups, at the RCS-S modules (Sercos counter) at the keyboard or at the local feedback inputs of the central unit.

COUNTERID

Feedback input associated with the handwheel.

Possible values: From 1 to 40 if remote CAN feedback / from -1 to -9 if feedback at the keyboard / from 1 to 2 if local feedback / from 1 to 32 if feedback at RCS-S module.

Default value: 0.

Associated variable: (V.)MPMAN.COUNTERID[hw]

Parameter included in the MANPG table.

The handwheels may be connected through the keyboards (3 per keyboard) , and through the counters modules of the remote CAN groups (4 per module), through the RCS-S counter modules (4 per module) or through the local feedback inputs.

2.

MACHINE PARAMETERS.
Machine parameters for JOG mode.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Handwheels connected through the keyboard.

Handwheels are numbered from -9 to -1. Handwheels are numbered following the logic order of the keyboards (rotary switch of the Power Supply element).

Keyboard order.	Numbering of the handwheels.
First keyboard in the bus.	-1 -2 -3
Second keyboard in the bus.	-4 -5 -6
Third keyboard in the bus.	-7 -8 -9

Handwheels connected through the remote CAN groups.

Handwheels are numbered from 1 to 40. The handwheels are numbered following the logic order of the remote groups (rotary switch of the Power Supply element). If there are several Counter modules in each group, the order is from top to bottom and from left to right.

Handwheels connected through the local feedback inputs.

Handwheels are numbered from 1 to 2. The central unit has two feedback inputs and this parameter indicates which one of them is used.

Handwheels connected through the RCS-S modules (Sercos counter).

Handwheels are numbered from 1 to 32. Handwheels are numbered according to the logical order of the RCS-.S modules (parameter SERCOUNTID). The feedback inputs of the first module will be 1 through 4, those of the second module 5 through 8 and so on.

HWFBTYPE**Feedback type associated with the handwheel input.**

Possible values: TTL / TTLDIFF.

Default value: TTLDIFF.

Associated variable: (V.)MPMAN.HWFBTYPE[hw]

Parameter included in the MANPG table.

Type of feedback associated with the handwheel input when it is connected to the local inputs or to an RCS-S module (Sercos counter).

MPGAXIS**Name of the axis associated with the handwheel.**

Possible values: Any axes defined in AXISNAME.

Associated variable: (V.)MPMAN.MPGAXIS[hw]

Parameter included in the MANPG table.

This parameter sets whether the handwheel is associated with a particular axis (individual handwheel) or it can move any axis of the machine (general handwheel). To set an individual handwheel, define the name of the axis it is associated with. To set a general handwheel, do not assign any value to this parameter, leave it blank.

An axis may be moved indistinctly with its individual handwheel or with a general handwheel. To move an axis with a general handwheel, use the jog keypad to select the axis to be moved. To move an axis with an individual handwheel, no prior axis selection is needed.

- If there are several axes selected in handwheel mode, the general handwheel will move all of them.
- If an axis has been selected which has an individual handwheel selected with it, this axis may be moved with the general handwheel, with the individual one or with both at the same time. When using both handwheels simultaneously, the CNC will add or subtract the pulses provided by both handwheels depending on which direction they are turned.
- If the CNC has several general handwheels, any of them can move the axes selected in handwheel mode. When using several handwheels simultaneously, each axis involved will be applied the sum of the increments of all the handwheels.

2.**MACHINE PARAMETERS.**

Machine parameters for JOG mode.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.7.2 Configure the jog keys.

JOGKEYDEF
 JOGKEYBD2DEF
 ..

JOGKEYBD8DEF
Tables to configure the jog keys.

These parameters may be used to configure the jog keypad of each panel. Parameter JOGKEYDEF corresponds to the first jog panel, JOGKEYBD2DEF to the second one and so on.

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.

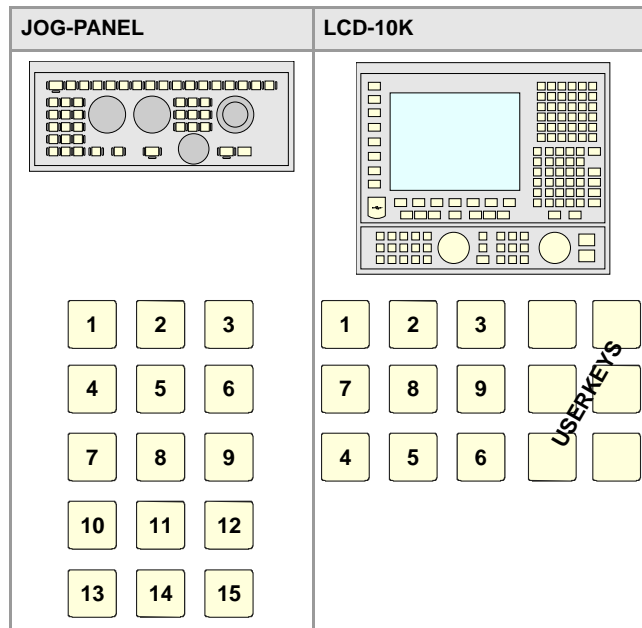
JOGKEYDEF n
 JOGKEYBD2DEF n
 ..

JOGKEYBD8DEF n
Axis and moving direction of each jog key.

*Associated variable: (V.)MPMAN.JOGKEYDEF[jk] / (V.)MPMAN.JOGKEYBDkbDEF[jk]
 Parameter included in the JOGKEYDEF table.*








There is one table for each jog keypad. Each one of these parameter sets the function of each JOG key. The number of jog keys available depends on the type of keyboard that may sometimes have up to 15 keys. The CNC always offers 15 parameters; if the jog keypad has fewer keys, the parameters that do not have any keys associated with them will be ignored.

The relationship between these parameters and the jog keys is the following.



2.
MACHINE PARAMETERS.
 Machine parameters for JOG mode.

The JOG keypad may consist of the following types of keys. Both types of keys may be defined at the same JOG keypad. To define the behavior of each key, assign to them one of the following values.

Keys.	Meaning.
 	Keys to define the axis and the jogging direction. The parameter is set with a value between -1 and +16 (signed). The sign indicates the positive direction (+) or the negative direction (-) and the number corresponds to the logic axis, according to parameter AXISNAME.
 	Keys to define the axis to be jogged. The parameter is set with a value between 1 and 16 (unsigned) that corresponds to the logic axis according to parameter AXISNAME.
 	Keys to define the direction of the movement. The parameter is set with one of the "+" and "-" values to indicate the moving direction.
	Rapid key. The parameter is set with the "R" value.

JOGTYPE

Behavior of the jog keys.

Possible values: Pressed axis / Maintained axis.

Default value: Pressed axis.

Associated variable: (V.)MPMAN.JOGTYPE

This parameter is applied when the jog keyboard has different keys for selecting the axis and the jogging direction. In this case, jogging an axis requires activating both the axis key and the moving direction.

There are two options, depending on how the jog keyboard has been configured.

- With the "pressed axis" option, the axis will move while both keys are pressed, the axis key and the direction key.
- With the "maintained axis" option, pressing the axis key will select it. The axis will move while the direction key is kept pressed. To de-select the axis, press [ESC] or [STOP].

2.

MACHINE PARAMETERS.
Machine parameters for JOG mode.

2.7.3 Configure the user keys as jog keys.

USERKEYDEF
USERKEYBD2DEF

..

USERKEYBD8DEF

Tables to configure the user keys as jog keys.

These parameters may be used to configure the user keys of each panel as jog keys. Parameter USERKEYDEF corresponds to the first jog panel, USERKEYBD2DEF to the second one and so on.

USERKEYDEF n
USERKEYBD2DEF n

..

USERKEYBD8DEF n

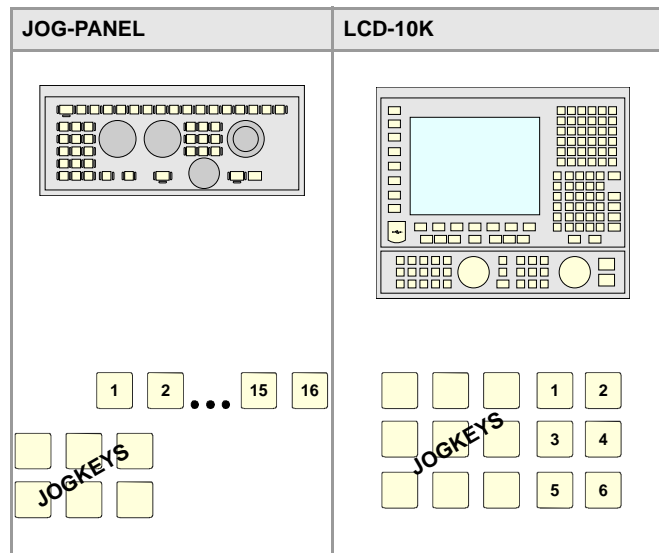
Axis and moving direction of each user key.

Associated variable: (V.)MPMAN.USERKEYDEF[uk] / (V.)MPMAN.USERKEYBDkbDEF[uk]

Parameter included in the USERKEYDEF table.

There is one table for each jog keypad. Each one of these parameter sets the function of each user key. The number of user keys available depends on the type of keyboard that may sometimes have up to 16 keys. The CNC always offers 16 parameters; if the jog keypad has fewer keys, the parameters that do not have any keys associated with them will be ignored.

The relationship between these parameters and the user keys is the following.



The meaning of this parameter is similar to that of machine parameter JOGKEYBDkbDEF. To define the behavior of each key, assign to them one of the following values:

- They are defined with a value between 1 and 16 (signed) for the keys that define the axis and the direction. The sign indicates the positive direction (+) or the negative direction (-) and the number corresponds to the logic axis, according to parameter AXISNAME.
- They are defined with a value between 1 and 16 (unsigned) for the keys that only define the axis.
- They are defined with the "+" and "-" values for the keys that only define the moving direction.
- They are defined with the "R" value for the rapid key.

The user keys defined this way behave like the jog keys whether they've been defined signed or unsigned and also respect the setting of machine parameter JOGTYPE.

(REF: 1709)

2.

MACHINE PARAMETERS.
Machine parameters for JOG mode.



FAGOR AUTOMATION

CNC 8070

2.7.4 HBSL portable operator panel.

HBSL

Activate the portable operator panel HBSL.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)MPMAN.HBSL

This parameter indicates whether there is a portable panel HBSL connected to the CNC through the serial line or not. In order to be able to use the portable operator panel HBSL, the serial line must be configured as RS422 (parameter RSTYPE). The portable operator panel HBSL must not be considered when counting the handwheels of the system (parameter NMPG).

2.

MACHINE PARAMETERS.

Machine parameters for JOG mode.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.7.5 Example of how to set the handwheels and jog keys.

Handwheel setting.

On a machine with X, Y, Z and A axes, we would like an individual handwheel for the X axis, an individual handwheel for the Y and a general handwheel for the Z and A axes.

- Individual handwheel for the X axis.
Feedback input: keyboard (MPG1).
Disk graduated with 100 lines.
Pulses/ turn: 100
Resolutions (X): 0.001, 0.01, 0.1
- Individual handwheel for the Y axis.
Feedback input: keyboard (MPG2).
Disk graduated with 100 lines.
Pulses/ turn: 200
Resolutions (Y): 0.001, 0.01, 0.1
- General handwheel for the rest of the axes (Z, A).
Feedback input: Counter (X1).
Disk graduated with 100 lines.
Pulses/ turn: 100
Resolutions (Z): 0.001, 0.01, 0.1
Resolutions (A): 0.01, 0.1, 1

Parameter setting:

Machine parameter.	Value.	Meaning.
NMPG	3	3-handwheel system.

X axis handwheel (MANPG 1).

Machine parameter.	Value.	Meaning.
COUNTERTYPE	Keyboard.	Handwheel connected to the keyboard.
COUNTERID	-1	MPG1 connector.
MPGAXIS	X	Associated axis.
MPGRESOL 1	0.001	Resolution 0.001
MPGRESOL 10	0.01	Resolution 0.01
MPGRESOL 100	0.1	Resolution 0.1

Y axis handwheel (MANPG 2).

Machine parameter.	Value.	Meaning.
COUNTERTYPE	Keyboard.	Handwheel connected to the keyboard.
COUNTERID	-2	MPG2 connector.
MPGAXIS	Y	Associated axis.
MPGRESOL 1	0.0005	Resolution 0.001
MPGRESOL 10	0.005	Resolution 0.01
MPGRESOL 100	0.05	Resolution 0.1

2.

MACHINE PARAMETERS.
Machine parameters for JOG mode.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Handwheel associated with the rest of the axes (Z + A).

Machine parameter.	Value.	Meaning.
COUNTERTYPE	Remote.	Handwheel connected to the remote modules.
COUNTERID	1	Feedback input X1.
MPGAXIS		Handwheel associated with the rest of the axes.
Z- MPGRESOL 1	0.001	Z axis. Resolution 0.001
Z- MPGRESOL 10	0.01	Z axis. Resolution 0.01
Z- MPGRESOL 100	0.1	Z axis. Resolution 0.1
A- MPGRESOL 1	0.01	A axis. Resolution 0.01
A- MPGRESOL 10	0.1	A axis. Resolution 0.1
A- MPGRESOL 100	1	A axis. Resolution 1

2.

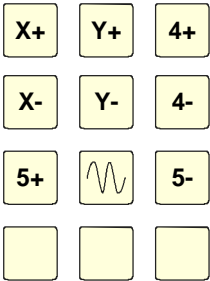
MACHINE PARAMETERS.

Machine parameters for JOG mode.

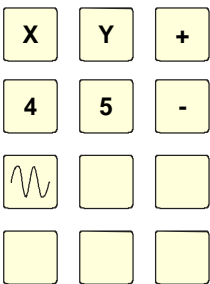
Setting the jog keys.

On a machine with the X, Y, U, V axes defined as AXISNAME 1, 2, 3, 4, we would like to jog the X and Y axes with the keys that have the same names, the U axis with the keys for the 4th axis and the V axis with those of the 5th axis.

Example 1: JOG-PANEL keyboard.

Jog keyboard.	JOGKEYDEF	Key.	Value.
	1	[X+]	1+
	2	[Y+]	2+
	3	[4+]	3+
	4	[X-]	1-
	5	[Y-]	2-
	6	[4-]	3-
	7	[5+]	4+
	8	[R]	R
	9	[5-]	4-
	10 - 15	---	

Example 2: JOG-PANEL keyboard.

Jog keyboard.	JOGKEYDEF	Key.	Value.
	1	[X]	1
	2	[Y]	2
	3	[+]	+
	4	[4]	3
	5	[5]	4
	6	[-]	-
	7	[R]	R
	8 - 15	---	



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Example 3: JOG-PANEL keyboard.

Jog keyboard.	JOGKEYDEF	Key.	Value.
	1	[+]	+
	2	[R]	R
	3	[-]	-
	4	[X]	1
	5	[4+]	3+
	6	[4-]	3-
	7	[Y]	2
	8	[5+]	4+
	9	[5-]	4-
	10 - 15	---	

Example 4: LCD-10K keyboard.

Jog keyboard.	JOGKEYDEF	Key.	Value.
	1	[X+]	1+
	2	[Y+]	2+
	3	[4+]	3+
	4	[5+]	4+
	5	[R]	R
	6	[5-]	4-
	7	[X-]	1-
	8	[Y-]	2-
	9	[4-]	3-








Example 5: LCD-10K keyboard.

Jog keyboard.	JOGKEYDEF	Key.	Value.
	1	[X]	1
	2	[Y]	2
	3	[+]	+
	4	[R]	R
	5	---	
	6	---	
	7	[4]	3
	8	[5]	4
	9	[-]	-

2.

MACHINE PARAMETERS.
 Machine parameters for JOG mode.

Example 6: LCD-10K keyboard.

Jog keyboard.					JOGKEYDEF	Key.	Value.
JOGKEYS		USERKEYS			1	[X+]	1+
X+	Y+		4	5	2	[Y+]	2+
X-	Y-		+	-	3	[R]	R
					4 - 6	---	
					7	[X-]	1-
					8	[Y-]	2-
					9	---	
					USERKEYDEF	Key.	Value.
					1	[4]	4
					2	[5]	5
					3	[+]	+
					4	[-]	-
					5 - 6	---	

2.

MACHINE PARAMETERS.

Machine parameters for JOG mode.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.8 Machine parameters for the M function table.

2.8.1 M function table.

MTABLESIZE

Number of table elements.

Possible values: from 0 to 200.

Default value: 50.

Associated variable: (V.)MPM.MTABLESIZE

This parameter indicates the number of M functions to be defined in the table. Each function may be assigned a subroutine and defined the type of synchronization.

It must be borne in mind that some auxiliary functions, besides what is indicated in this table, have a specific meaning when used in a CNC program. These functions are M00, M01, M02, M03, M04, M05, M06, M08, M09, M19, M30, M41, M42, M43 and M44.

DATA

M function table.

It shows the "M" function setting table. The following fields must be defined for each M function.

DATA	
MNUM	M function number.
SYNCHTYPE	Type of synchronism.
MPROGRAMNAME	Name of subroutine associated with M function.
MTIME	Estimated time for an M function.
MPLC	Send the M function to the PLC during block search.
COMMENT	Description of the M function.

MNUM

M function number.

Possible values: from 0 to 65535.

Associated variable: (V.)MPM.MNUM[pos]

Parameter included in the DATA table.

This parameter indicates the number of the M function.

SYNCHTYPE

Type of synchronization.

Possible values: Without synchronization / Before-Before / Before-After / After-After.

Default value: Before - Before.

Associated variable: (V.)MPM.SYNCHTYPE[pos]

Parameter included in the DATA table.

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).

Type of synchronism.	Meaning.
Without synchronization.	M function not synchronized.
Before - Before.	The M function is sent to the PLC and synchronized before the movement.
Before - After.	The M function is sent to the PLC before the movement and synchronized after the movement.
After - After.	The M function is sent to the PLC and synchronized after the movement.

2.

MACHINE PARAMETERS.
Machine parameters for the M function table.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

The M functions may be sent and/or synchronized before or after the movement.

- If an M function is used to turn on a lamp, it will be set without synchronization because there is no need to check that the lamp has indeed turned on.
- Functions M03 and M04 to start up the spindle, they should be executed and synchronized before the movement.
- Function M5 to stop the spindle, should be executed and synchronized after the movement.

MPROGNAME

Name of subroutine associated with M function.

Possible values: any text with up to 64 characters.

Default value: Without subroutine.

Associated variable: (V.)MPM.MPROGNAME[pos]

Parameter included in the DATA table.

The subroutines associated with the M functions must be located in the "C:\CNC8070\MTB\SUB" folder.

When associating a subroutine with an M function, the execution of the M function executes the associated subroutine without executing the M function itself. To send the M function to the PLC, it must be programmed in the subroutine.

The type of synchronization of the M functions that have an associated subroutine must be "without synchronization" or "After-After". The CNC executes the associated subroutine after executing the programmed movement (if any).



To have different procedures in the subroutines associated with certain M functions, the code of each channel may be differentiated within the subroutine using the variable (V.)G.CNCHANNEL.

MTIME

Estimated time for an M function.

Possible values: From 0 to 1000000 ms.

Default value: 0 ms.

Associated variable: (V.)MPM.MTIME[pos]

Parameter included in the DATA table.

In EDISIMU mode, there is an option that allows calculating the time required to execute a part with the machining conditions established in the program. This parameter may be set to fine tune that calculation.

MPLC

Send the M function to the PLC during block search.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)MPM.MPLC[pos]

Parameter included in the DATA table.

This field indicates whether the M function must be sent to the PLC during block search.

COMMENT

Description of the M function.

Associated variable: (V.)MPM.COMMENT[pos]

Parameter included in the DATA table.

This field offers the possibility to associate a short description with the M function. This field is for information only; it is not used by the CNC.

The comments are saved in the file MComments.txt and it is possible to have one file per language. These files are saved in the folder C:\CNC8070\MTB\data\Lang.

2.

MACHINE PARAMETERS.

Machine parameters for the M function table.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.9 Machine parameters for kinetics.

Up to 6 different kinetics may be set for a machine. The type of kinetics to be applied must be defined for each of them. Channel machine parameter KINID indicates the kinematics number (not type) assumed by the CNC in that channel on power-up. To select a kinematics from the part-program, use the #KIN ID instruction.

Types of kinematics (predefined by Fagor or integrated by the OEM).

The CNC offers a number of pre-defined kinematics that may be easily configured by machine parameters. Besides these kinematics, the OEM can integrate 6 additional kinematics.

The OEM kinematics are integrated through a generic API and are then configured using these machine parameters. To include the kinematics for your machine, contact Fagor Automation.

Axes of the kinematics.

One kinematics may be active per channel. A kinematics may be configured by between 3 and 8 axes. All the axes making up the kinematics must belong to the same channel and must occupy the first positions in the following order.

Axis order.	Meaning.
1st axis	First main axis of the plane (abscissa).
2nd axis	2nd main axis of the plane (ordinate).
3rd axis	Longitudinal axis.
4th axis	Four axes of the kinematics.
5th axis	Fifth axis of the kinematics.
6th axis	Sixth axis of the kinematics.
7th axis	Seventh axis of the kinematics.
8th axis	Eighth axis of the kinematics.
9th axis and the next ones.	Rest of the axes.

The first 3 axes must be linear; spindle compensation will be applied on to them. The rest of the axes may be either rotary or linear, depending on the type of kinematics.

2.

MACHINE PARAMETERS.
Machine parameters for kinetics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.9.1 Kinematics configuration.

NKIN

Number of kinematics defined.

Possible values: From 0 to 6.

Default value: 0.

Associated variable: (V.)MPK.NKIN

Up to 6 different kinetics may be set for a machine. The type of kinetics to be applied must be defined for each of them. Channel machine parameter KINID indicates the kinematics number (not type) assumed by the CNC in that channel on power-up. To select a kinematics from the part-program, use the #KIN ID instruction.

KINEMATIC n

Kinematics table.

It shows the table to define the kinematics of the system. The kinematics preset by Fagor have the following machine parameters to configure them.

DATA	
TYPE	Kinematics type.
TDATA n	Numeric parameter in decimal format.
TDATA_I n	Numeric parameter in integer format.

The OEM kinematics have the following machine parameters to configure them.

DATA	
TYPE	Kinematics type.
NKINAX	Number of axes of the kinematics.
PARAM_D_SIZE	Number of numeric parameters in decimal format.
TDATA n	Numeric parameter of the kinematics in decimal format.
PARAM_I_SIZE	Number of parameters of the kinematics in integer format.
TDATA_I n	Numeric parameter of the kinematics in integer format.
AUXCTE_SIZE	Size of the area of auxiliary variables.
KINDATA_SIZE	Size of the area for general purpose data.

TYPE

Kinematics type.

Possible values: From 0 to 99 (kinematics predefined by Fagor) / from 100 to 105 (OEM kinematics).

Default value: 0.

Associated variable: (V.)MPK.TYPE

The first 100 of the 106 possible kinematics are predefined by Fagor and the other 6 may be integrated into the CNC by the manufacturer (OEM).

Kinematics predefined by Fagor.

TYPE	Kinematics type.
TYPE = 1	Orthogonal or spherical spindle head YX.
TYPE = 2	Orthogonal or spherical spindle head ZX.
TYPE = 3	Orthogonal or spherical spindle head XY.
TYPE = 4	Orthogonal or spherical spindle head ZY.
TYPE = 5	Swivel (angular) spindle XZ.
TYPE = 6	Swivel (angular) spindle YZ.
TYPE = 7	Swivel (angular) spindle ZX.
TYPE = 8	Swivel (angular) spindle ZY.
TYPE = 9	Rotary table AB.
TYPE = 10	Rotary table AC.

2.

MACHINE PARAMETERS.

Machine parameters for kinematics.

FAGOR 

FAGOR AUTOMATION

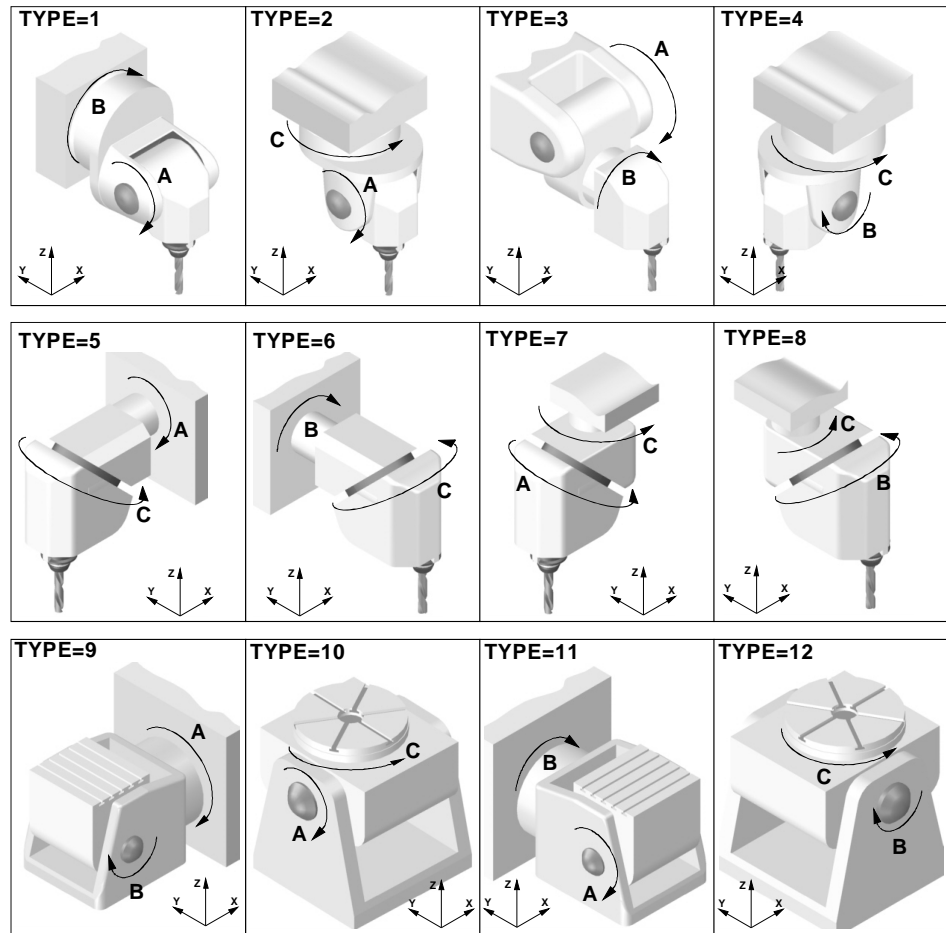
CNC 8070

(REF: 1709)

2.

MACHINE PARAMETERS.
Machine parameters for kinematics.

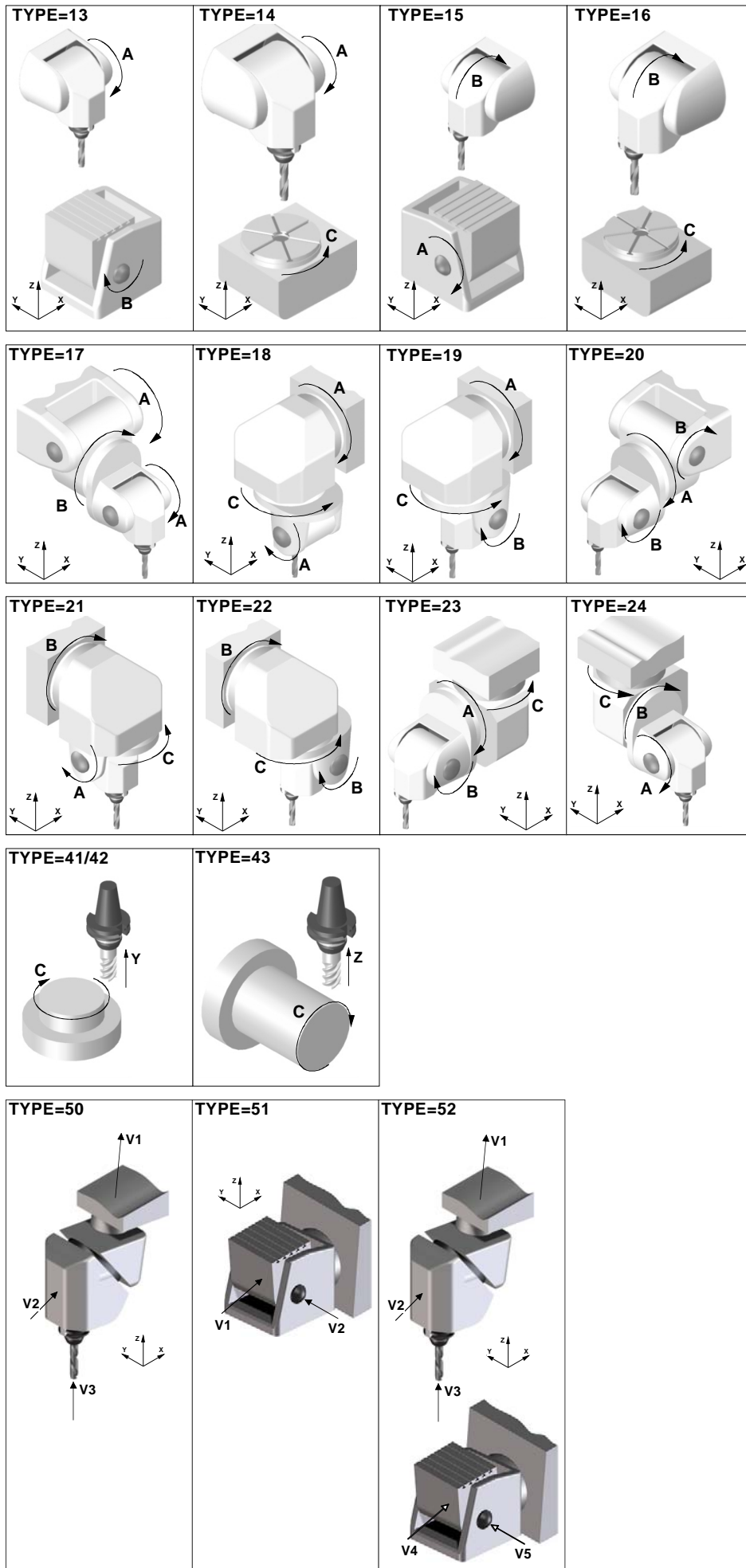
TYPE	Kinematics type.
TYPE = 11	Rotary table BA.
TYPE = 12	Rotary table BC.
TYPE = 13	Spindle - table AB.
TYPE = 14	Spindle - table AC.
TYPE = 15	Spindle - table BA.
TYPE = 16	Spindle - table BC.
TYPE = 17	Orthogonal spindle with three rotary axes ABA.
TYPE = 18	Orthogonal spindle with three rotary axes ACA.
TYPE = 19	Orthogonal spindle with three rotary axes ACB.
TYPE = 20	Orthogonal spindle with three rotary axes BAB.
TYPE = 21	Orthogonal spindle with three rotary axes BCA.
TYPE = 22	Orthogonal spindle with three rotary axes BCB.
TYPE = 23	Orthogonal spindle with three rotary axes CAB.
TYPE = 24	Orthogonal spindle with three rotary axes CBA.
TYPE = 41	"C" axis. Machining on the face of the part when ALIGNC = YES.
TYPE = 42	"C" axis. Machining on the face of the part when ALIGNC = NO.
TYPE = 43	"C" axis. Machining of the turning side of the part.
TYPE = 50	Vectorial definition of spindle kinematics.
TYPE = 51	Vectorial definition of table kinematics.
TYPE = 52	Vectorial definition of spindle-table kinematics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)



2.

MACHINE PARAMETERS.
Machine parameters for kinetics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TDATA n**Numeric parameter in decimal format.***Possible values: Within ± 999999999.0000* *Default value: 0.**Associated variable: (V.)MPK.TDATAkin[nb]**Associated variable: (V.)MPK.MAXOFTDATAkin[nb]*

Numeric parameter in decimal format. The following fields must be defined for each parameter.

DATA	
VALUE	To be defined by the OEM. Value of the parameter.
MAX OFFSET	To be defined by the OEM. Maximum offset that may be defined by the user.
OFFSET	To be defined by the user. Offset of the parameter.

On the "Value" column, the OEM sets the parameter value that may be modified by the user using the "OFFSET" column or its associated variable.

On the "MAX OFFSET" column, the OEM determines the maximum offset that the user may set. This column will always be a positive value and the offset set by the user will be a value between "- MAX OFFSET" and "+ MAX OFFSET". If he sets a maximum offset of zero, the user will not be able to set any offset.

The offset values set by the user will be added to its corresponding TDATA parameter. The new value will be effective when activating the kinematics (#KIN ID) or RTCP.

TDATA_I n**Numeric parameter in integer format.***Possible values: Within ± 2147483647* *Default value: 0.**Associated variable: (V.)MPK.TDATA_ikin[nb]**Associated variable: (V.)MPK.MAXOFTDATA_ikin[nb]*

Numeric parameter in integer format. The following fields must be defined for each parameter.

DATA	
VALUE	To be defined by the OEM. Value of the parameter.
MAX OFFSET	To be defined by the OEM. Maximum offset that may be defined by the user.
OFFSET	To be defined by the user. Offset of the parameter.

On the "Value" column, the OEM sets the parameter value that may be modified by the user using the "OFFSET" column or its associated variable.

On the "MAX OFFSET" column, the OEM determines the maximum offset that the user may set. This column will always be a positive value and the offset set by the user will be a value between "- MAX OFFSET" and "+ MAX OFFSET". If he sets a maximum offset of zero, the user will not be able to set any offset.

The offset values set by the user will be added to its corresponding TDATA parameter. The new value will be effective when activating the kinematics (#KIN ID) or RTCP.

2.

MACHINE PARAMETERS.
Machine parameters for kinematics.



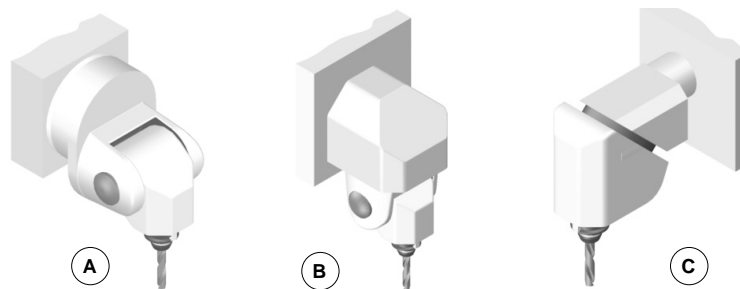
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.9.2 Definition of the spindle kinetics (types 1 through 8)

It is possible to control spheric spindle heads orthogonal and angular.

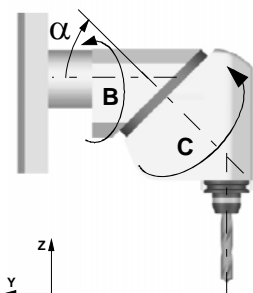
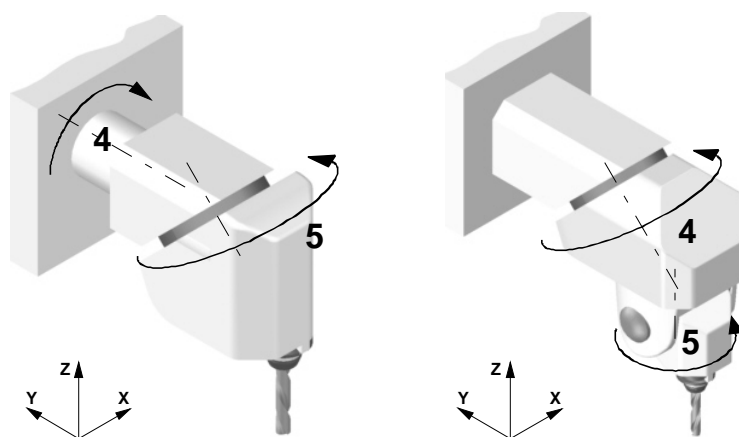


A Spherical spindle.

B Orthogonal spindle.

C Swivel (angular) spindle.

When having an angular spindle head, the main rotary axis (4) must rotate around one of the main axes (X, Y, Z) and the secondary or dragged axis (5) will form a particular angle. The left figure meets this condition, whereas in the right one the main rotary axis (4) does not rotate around the Y axis (it forms an angle with it).



From now on, all the explanations will assume that the main axes are X Y and Z and their associated rotary axes A, B and C.

TDATA1..TDATA7 Spindle dimensions.

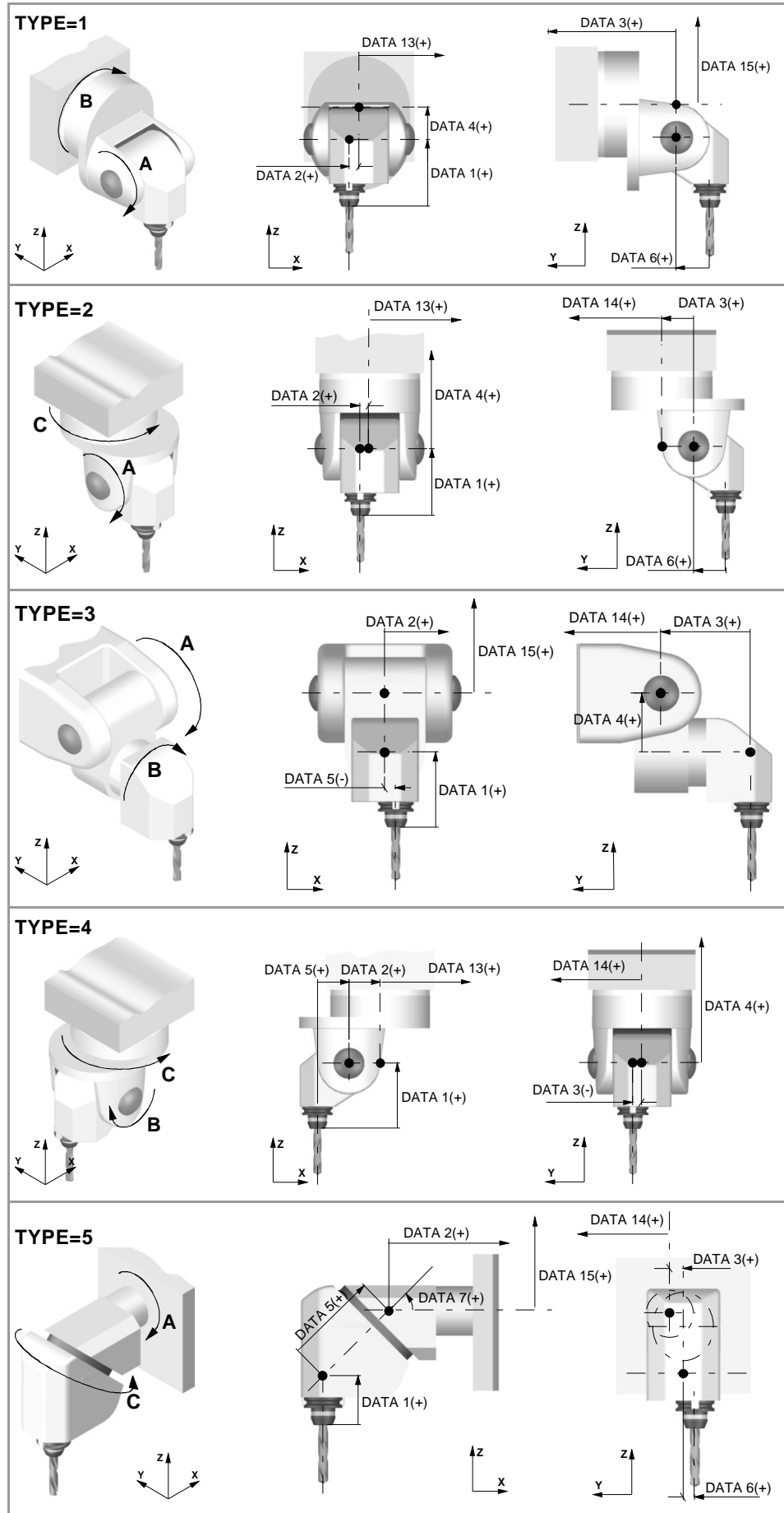
All the parameters need not be set. The parameters to be defined for each model and their meanings are listed here below. All the parameters may be set with a positive or negative value. The (+) sign in the illustrations indicates that the direction is assumed as positive.

- TDATA1 It indicates the distance between the tip of the quill and the secondary rotary axis along the Z axis.
- TDATA2 It indicates the distance between the secondary rotary axis and the main axis along the X axis.
- TDATA3 It indicates the distance between the secondary rotary axis and the main axis along the Y axis.
- TDATA4 It indicates the distance between the secondary rotary axis and the main axis along the Z axis.
- TDATA5 It indicates the distance between the tool axis and the secondary rotary axis along the X axis.

- TDATA6 It indicates the distance between the tool axis and the secondary rotary axis along the Y axis.
- TDATA7 It indicates the angle between the main and secondary rotary axes on rotary spindles.

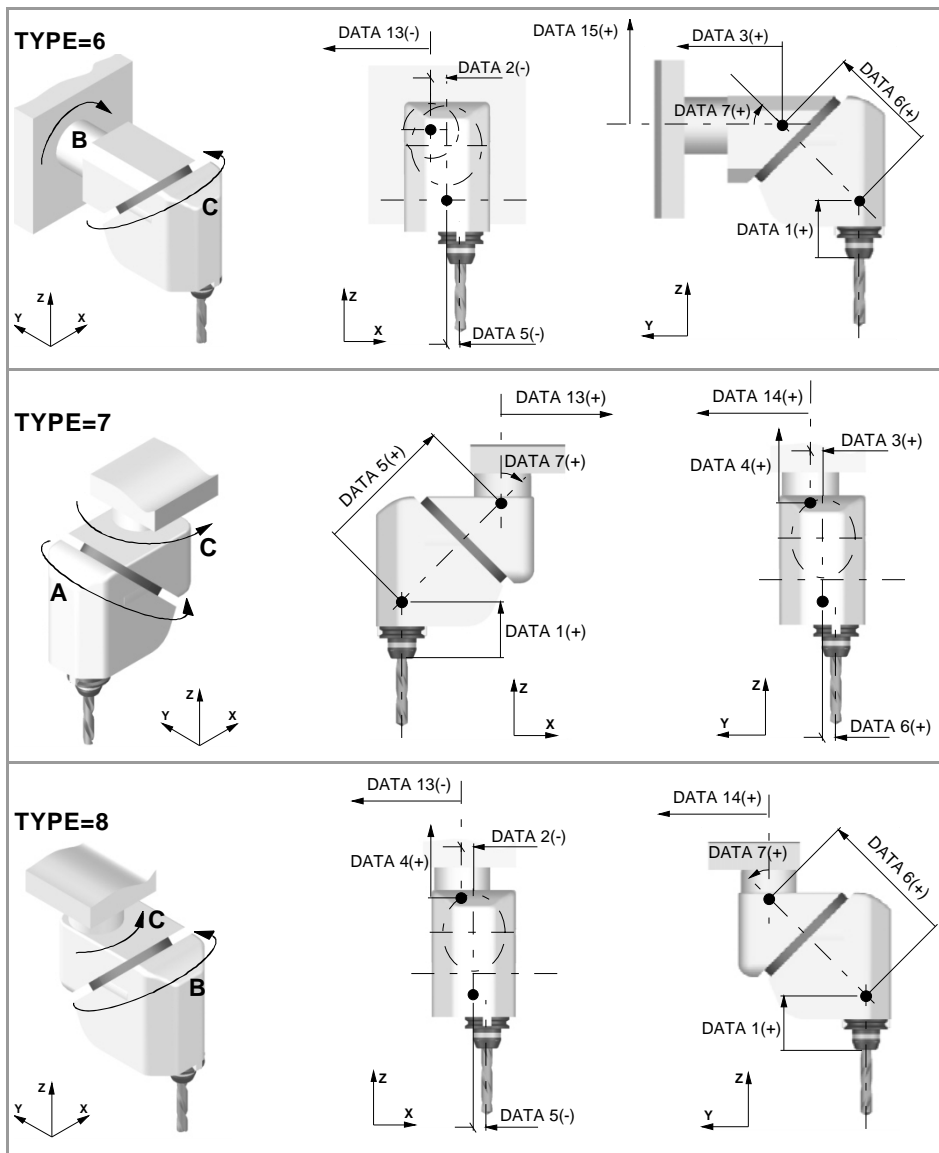
2.

MACHINE PARAMETERS.
Machine parameters for kinetics.



CNC 8070

(REF: 1709)



2.

MACHINE PARAMETERS.
Machine parameters for kinetics.

TDATA8
Rest position of the main rotary axis.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

See parameter TDATA9.

TDATA9
Rest position of the secondary rotary axis.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

The rest position is when the tool is perpendicular to the work plane (parallel to the longitudinal axis).

TDATA10
Turning direction of the main rotary axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

See parameter TDATA11.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

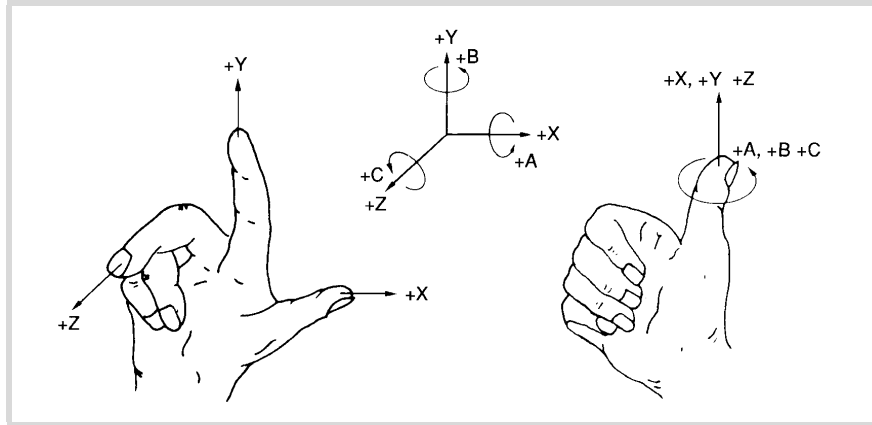
TDATA11

Turning direction of the secondary rotary axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

The direction of the XYZ axes, according to the DIN 66217 standard is easy to remember using the rule of the right hand. On rotary axes, the turning direction is established when bending your fingers (closing your hand) around the associated linear axis while your thumb is pointing in the positive direction of the linear axis.



TDATA12

Manual rotary axes or servo-controlled.

Possible values: 0 to 3.

Default value: 0.

This parameter indicates whether the rotary axes are manual or servo-controlled.

TDATA12	Meaning.
0	The two axes are servo-controlled.
1	Main axis manual and secondary axis servo-controlled.
2	Main axis servo-controlled and secondary axis manual.
3	The two axes are manuals.

TDATA 13..TDATA15

Spindle position referred to the machine reference point.

- TDATA13 Distance between the main rotary axis and the reference point, along the X axis.
- TDATA14 Distance between the main rotary axis and the reference point, along the Y axis.
- TDATA15 Distance between the main rotary axis and the reference point, along the Z axis.

2.

MACHINE PARAMETERS.
Machine parameters for kinetics.



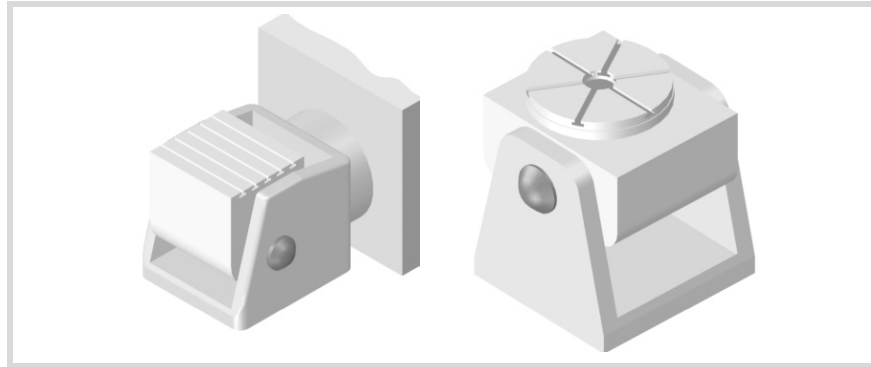
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.9.3 Definition of the table kinetics (types 9 through 12).

The following types of rotary tables may be controlled.



TDATA2..TDATA5 Table dimensions.

The dimensions may be defined with a positive or negative value. The (+) sign in the illustrations indicates that the direction is assumed as positive.

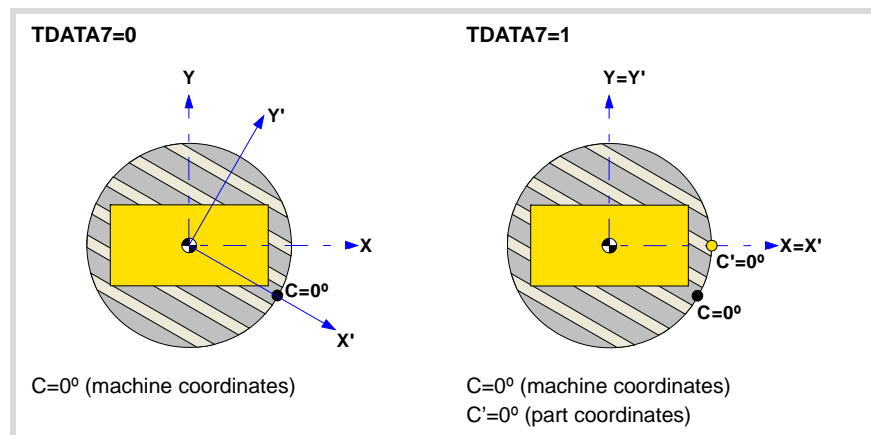
- TDATA2 It indicates the position of the secondary rotary axis or the intersection with the primary axis along the X axis.
- TDATA3 It indicates the position of the secondary rotary axis or the intersection with the primary axis along the Y axis.
- TDATA4 It indicates the position of the secondary rotary axis or the intersection with the primary axis along the Z axis.
- TDATA5 It indicates the distance between the secondary and the main rotary tables.

TDATA7 Origin for applying the RTCP.

Possible values: 0 (machine zero) / 1 (part zero).

Default value: 0.

This parameter indicates whether the CNC applies the RTCP referred to machine zero or to part zero. For this type of kinematics it is recommended to set this parameter with a value of ·1·.



TDATA8 Rest position of the main rotary axis.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

See parameter TDATA9.

TDATA9

Rest position of the secondary rotary axis.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

The rest position is when the tool is perpendicular to the work plane (parallel to the longitudinal axis).

TDATA10

Turning direction of the main rotary axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

See parameter TDATA11.

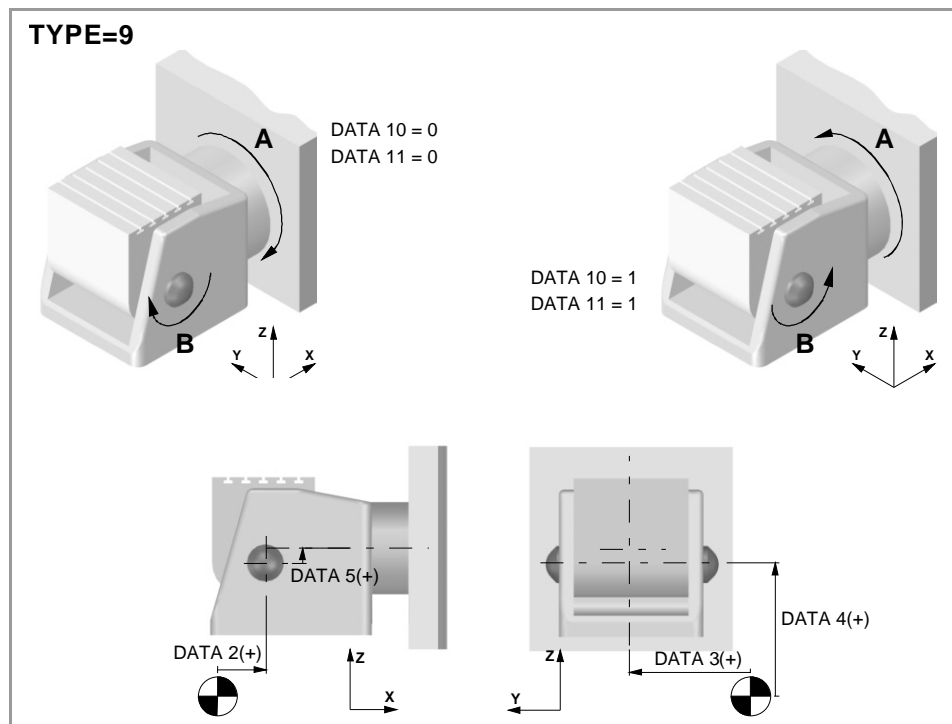
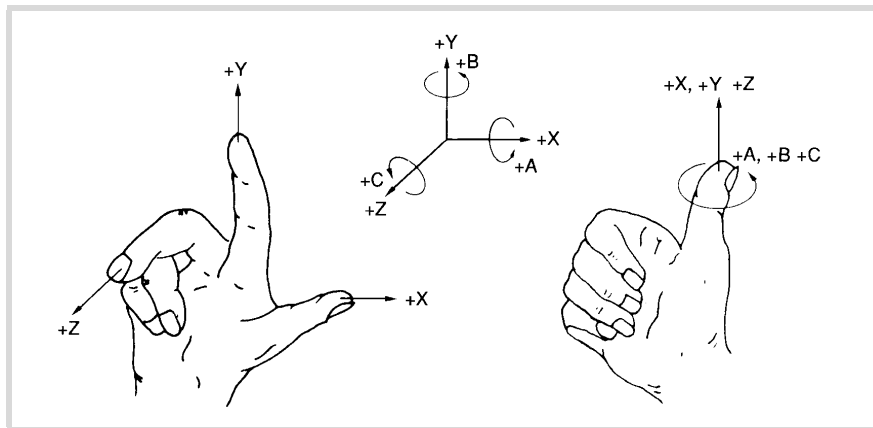
TDATA11

Turning direction of the secondary rotary axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

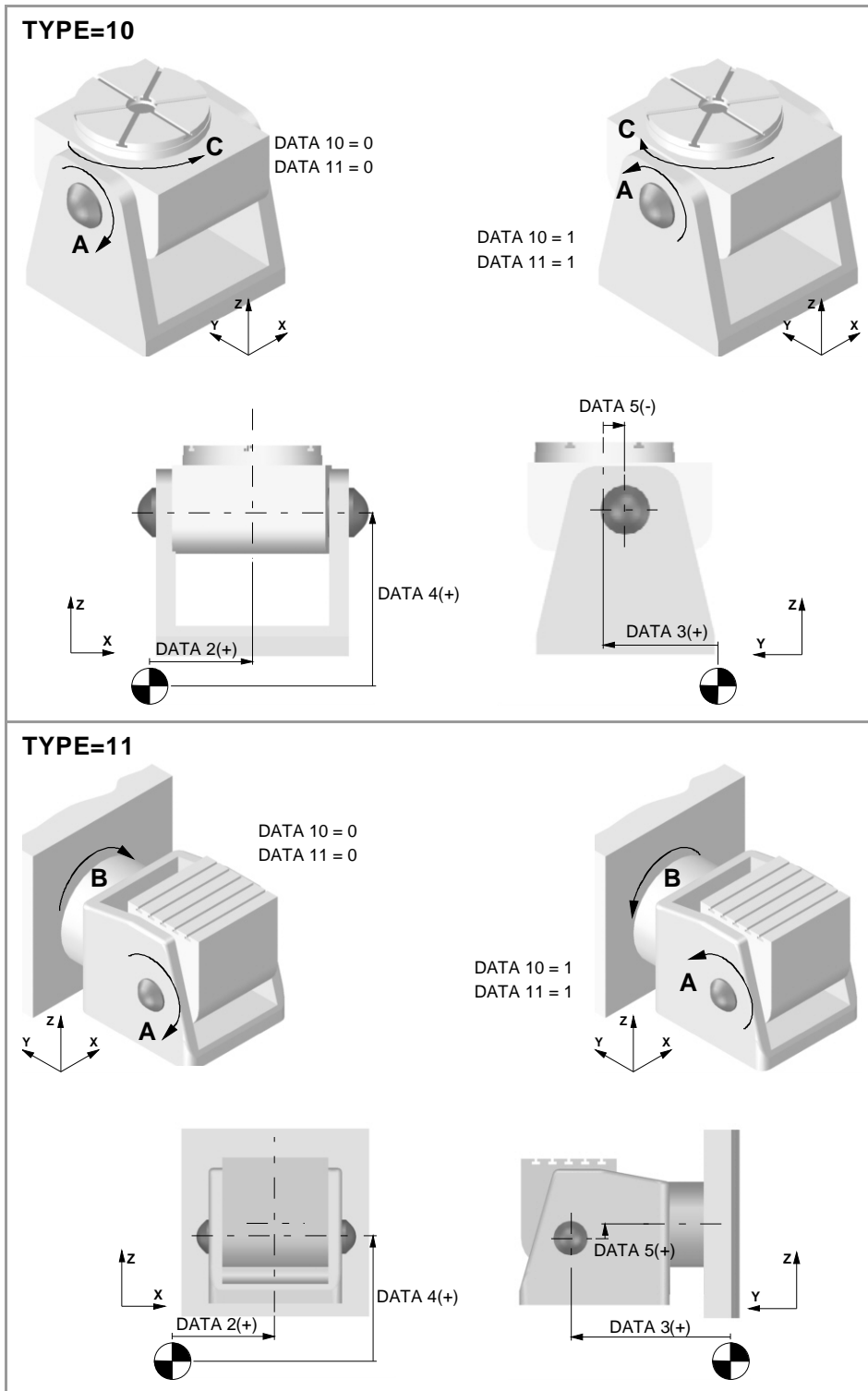
Default value: 1.

The direction of the XYZ axes, according to the DIN 66217 standard is easy to remember using the rule of the right hand. On rotary axes, the turning direction is established when bending your fingers (closing your hand) around the associated linear axis while your thumb is pointing in the positive direction of the linear axis. On kinematics with rotary axes on the table, the turning direction is set from the point of view of the tool; therefore, TDATA10 and TDATA11 must be set to 1.



2.

MACHINE PARAMETERS.
Machine parameters for kinematics.



2.

MACHINE PARAMETERS.
Machine parameters for kinetics.



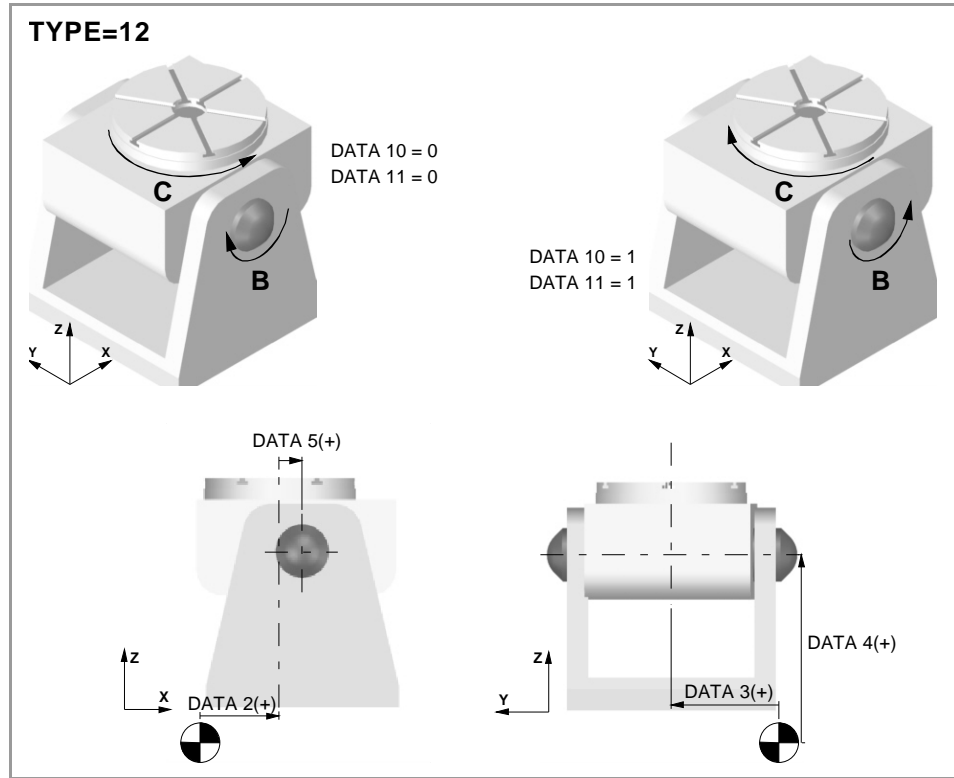
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

MACHINE PARAMETERS.
Machine parameters for kinetics.



TDATA12
Manual rotary axes or servo-controlled.

Possible values: 0 to 3.
Default value: 0.

This parameter indicates whether the rotary axes are manual or servo-controlled.

TDATA12	Meaning.
0	The two axes are servo-controlled.
1	Main axis manual and secondary axis servo-controlled.
2	Main axis servo-controlled and secondary axis manual.
3	The two axes are manuals.

TDATA16
Rotate the part coordinate system when rotating the table.

Possible values: 0 / 1.
Default value: 0.

Having the RTCP mode active, this parameter determines whether the part coordinate system is fixed on the part and rotates with it or it stays parallel to the machine coordinate system.

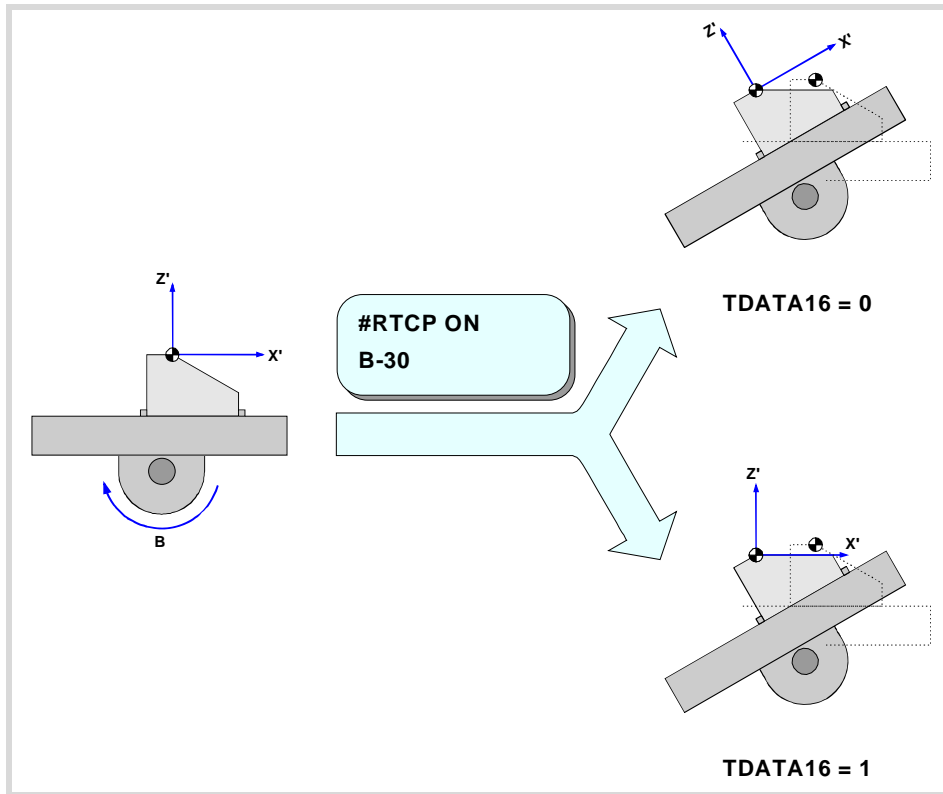
TDATA16	Meaning.
0	When changing the part orientation, the position occupied by the tool tip on the part does not change. The coordinate system is fixed on the part and rotates with it.
1	When changing the part orientation, the position occupied by the point set as part zero does not change. The coordinate system stays parallel to the machine coordinate system.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)



2.

MACHINE PARAMETERS.
Machine parameters for kinetics.



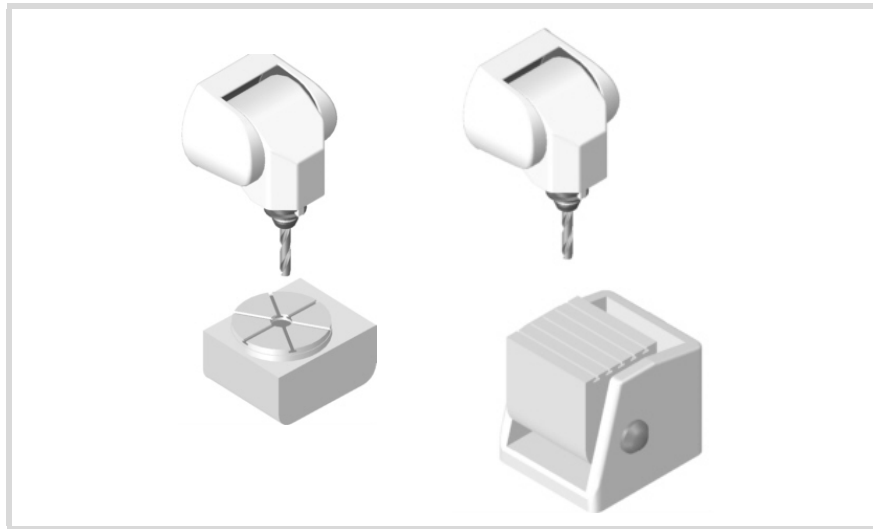
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.9.4 Definition of the kinematics of the spindle - table (types 13 through 16).

In this type of kinematics, one rotating axis is at the spindle and the other one at the table. The one at the spindle orients the tool and that of the work table orients the part.



The order of the axes in the channel where the kinematics is applied is:

- The first two axes correspond to the work plane.
- The third axis corresponds to the tool axis.
- The fourth axis corresponds to the rotary axis of the spindle.
- The fifth axis corresponds to the rotary axis of the table.

The type of kinematics is defined being the tool parallel to the third axis of the channel and the work plane perpendicular to the tool.

TDATA1..TDATA6

Spindle dimensions and table placement.

All of them need not be defined. The parameters to be defined for each kinematics and their meanings are listed here below. They may be defined with a positive or negative value. The (+) sign in the illustrations indicates the direction it assumes as positive.

TDATA1	It indicates the distance between the tip of the quill and the spindle rotary axis along the Z axis.
TDATA2	It indicates the distance between the tool axis and the spindle rotary axis along the X axis.
TDATA3	It indicates the distance between the tool axis and the spindle rotary axis along the Y axis.
TDATA4	It indicates the position of the table's rotary axis along the X axis.
TDATA5	It indicates the position of the table's rotary axis along the Y axis.
TDATA6	It indicates the position of the table's rotary axis along the Z axis.

2.

MACHINE PARAMETERS.
Machine parameters for kinematics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

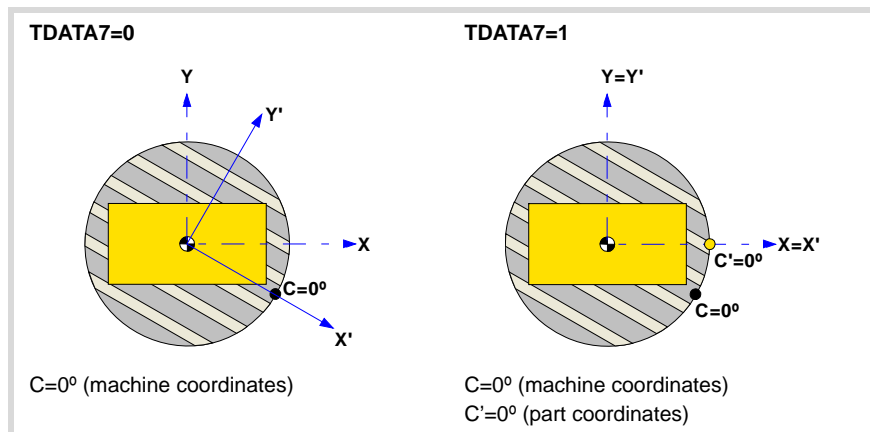
TDATA7

Origin for applying the RTCP.

Possible values: 0 (machine zero) / 1 (part zero).

Default value: 0.

This parameter indicates whether the CNC applies the RTCP referred to machine zero or to part zero.



TDATA8

Rest position of the main rotary axis.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

See parameter TDATA9.

TDATA9

Rest position of the secondary rotary axis.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

The rest position is when the tool is perpendicular to the work plane (parallel to the longitudinal axis).

TDATA10

Turning direction of the main rotary axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

See parameter TDATA11.

TDATA11

Turning direction of the secondary rotary axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

The direction of the XYZ axes, according to the DIN 66217 standard is easy to remember using the rule of the right hand. On rotary axes, the turning direction is established when bending your fingers (closing your hand) around the associated linear axis while your thumb is pointing in the positive direction of the linear axis. On kinematics with rotary axes on the table, the turning direction is set from the point of view of the tool; therefore, TDATA10 and TDATA11 must be set to 1.

2.

MACHINE PARAMETERS.
Machine parameters for kinetics.



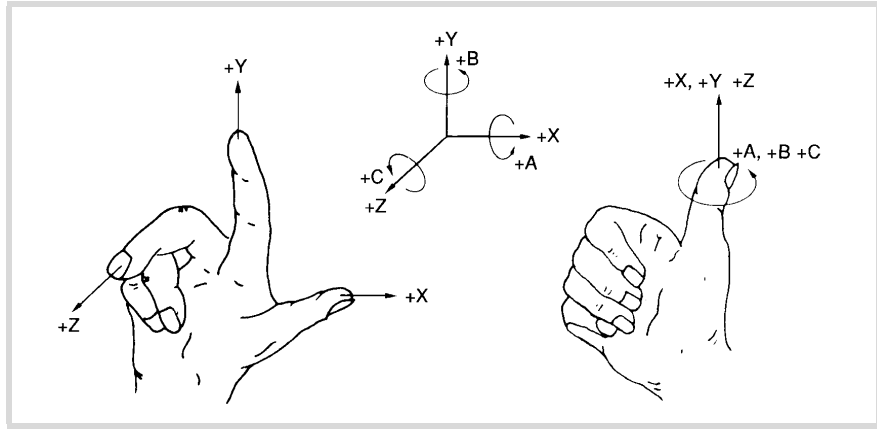
FAGOR AUTOMATION

CNC 8070

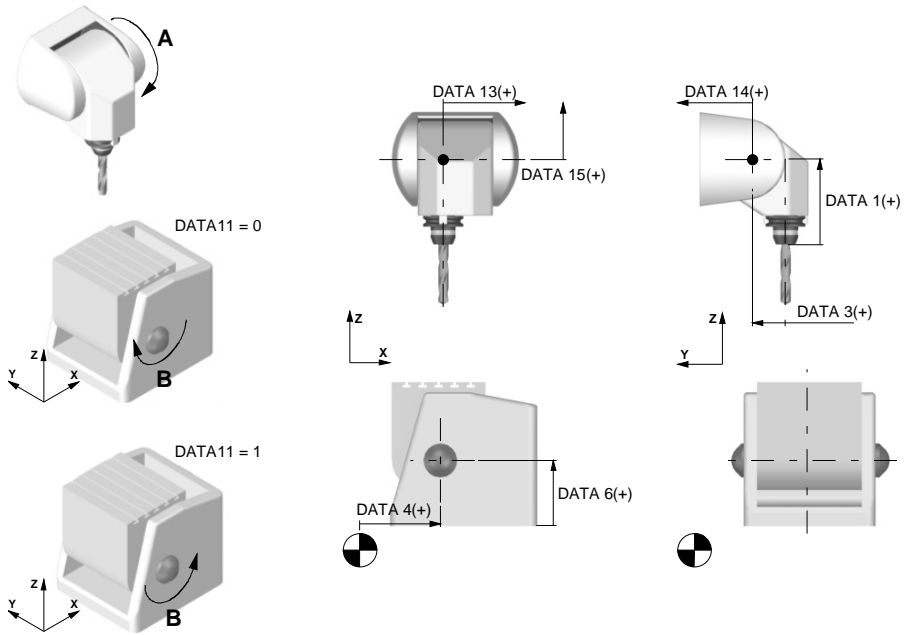
(REF: 1709)

2.

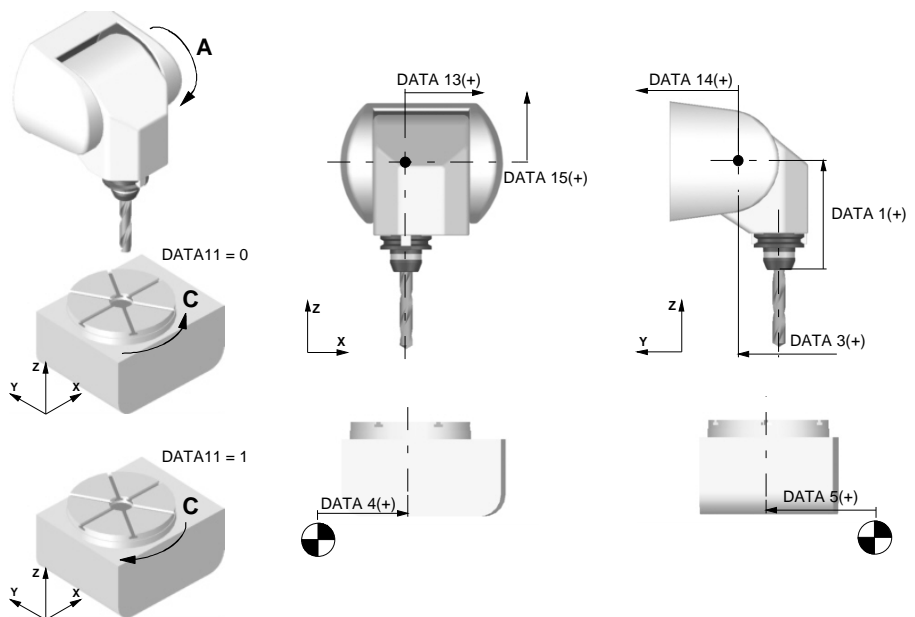
MACHINE PARAMETERS.
Machine parameters for kinematics.



TYPE=13

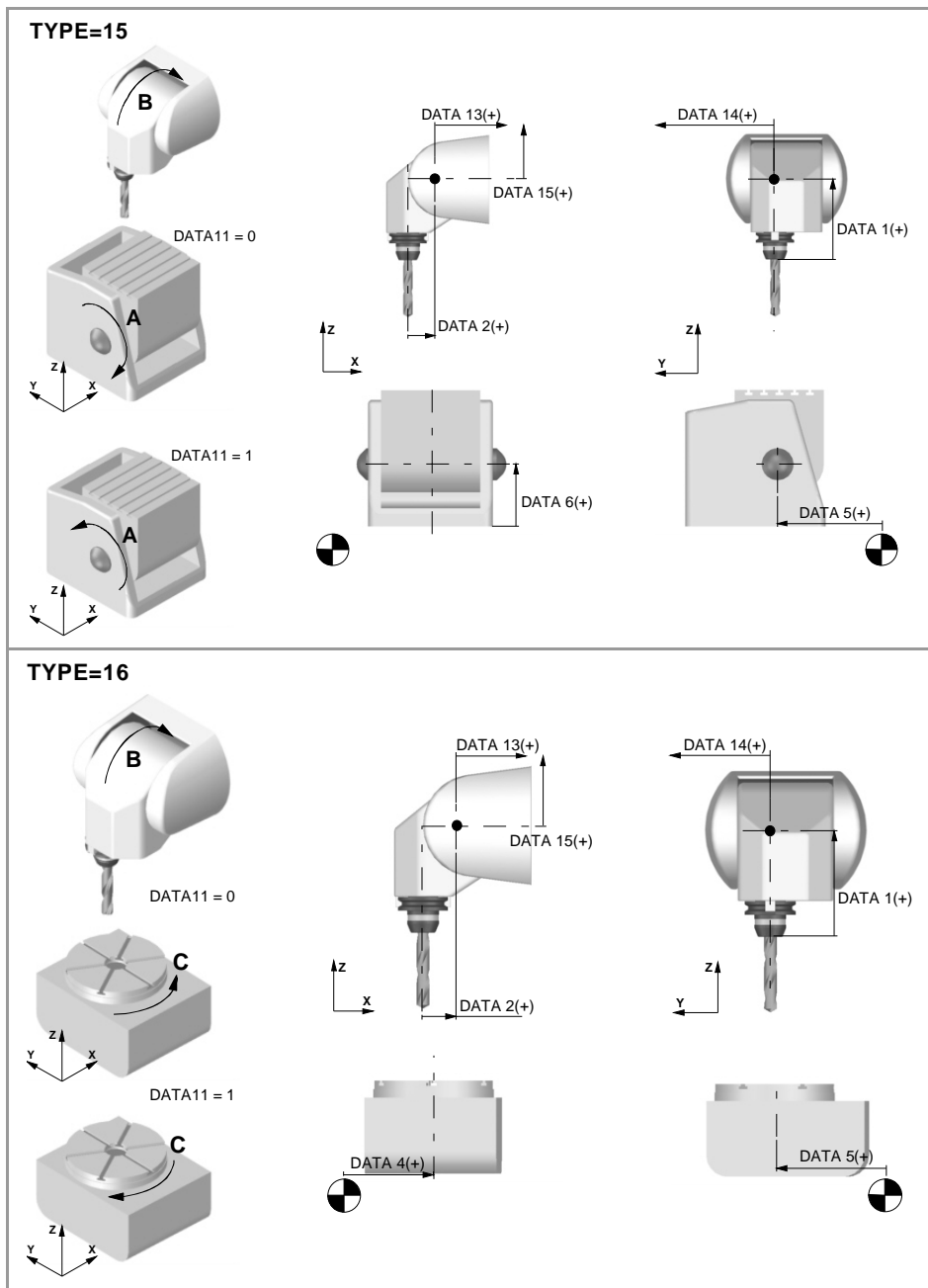


TYPE=14



CNC 8070

(REF: 1709)



2.

MACHINE PARAMETERS.
Machine parameters for kinetics.

TDATA12
Manual rotary axes or servo-controlled.

Possible values: 0 to 3.

Default value: 0.

This parameter indicates whether the rotary axes are manual or servo-controlled.

TDATA12	Meaning.
0	The two axes are servo-controlled.
1	Main axis manual and secondary axis servo-controlled.
2	Main axis servo-controlled and secondary axis manual.
3	The two axes are manuals.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TDATA13..TDATA15
Spindle placement.

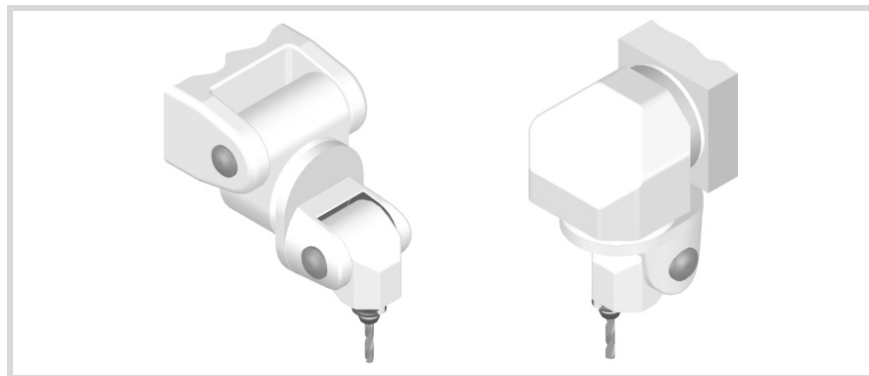
- TDATA13 Distance defining the spindle placement, from the rotary axis, along the X axis.
- TDATA14 Distance defining the spindle placement, from the rotary axis, along the Y axis.
- TDATA15 Distance defining the spindle placement, from the rotary axis, along the Z axis.

2.**MACHINE PARAMETERS.**

Machine parameters for kinetics.

2.9.5 Definition of the spindle kinetics (types 17 through 24)

It is possible to control orthogonal spindles with three rotary axes.



TDATA1..TDATA12 Spindle dimensions.

All the parameters need not be set. The parameters to be defined for each spindle model and their meanings are listed here below.

They may be defined with a positive or negative value. The (+) sign in the illustrations indicates that the direction is assumed as positive, numbers 1st, 2nd and 3rd indicate the rotation centers.

TDATA1	It indicates the distance between the tip of the quill and the third rotation center, the third rotary axis along the Z axis.
TDATA2	It indicates the distance of the spindle from the 1st rotation center to its fixture on the machine along the X axis.
TDATA3	It indicates the distance of the spindle from the 1st rotation center to its fixture on the machine along the Y axis.
TDATA4	It indicates the distance of the spindle from the 1st rotation center to its fixture on the machine along the Z axis.
TDATA5	It indicates the distance from the 2nd rotation center to the 1st rotation center along the X axis.
TDATA6	It indicates the distance from the 2nd rotation center to the 1st rotation center along the Y axis.
TDATA7	It indicates the distance from the 2nd rotation center to the 1st rotation center along the Z axis.
TDATA8	It indicates the distance from the 3rd rotation center to the 2nd rotation center along the X axis.
TDATA9	It indicates the distance from the 3rd rotation center to the 2nd rotation center along the Y axis.
TDATA10	It indicates the distance from the 3rd rotation center to the 2nd rotation center along the Z axis.
TDATA11	It indicates the distance from the shaft of the tool to the 3rd rotation center along the X axis.
TDATA12	It indicates the distance from the shaft of the tool to the 3rd rotation center along the Y axis.

The main rotary axis is the rotary axis of the spindle that when it rotates it drags the other two rotation axes. The secondary rotary axis is the rotary axis of the spindle that when it rotates it drags one rotation axis. The third rotary axis is the rotary axis of the spindle that when it rotates it does not drag any other rotation axis, only the tool.

2.

MACHINE PARAMETERS.
Machine parameters for kinetics.

FAGOR 

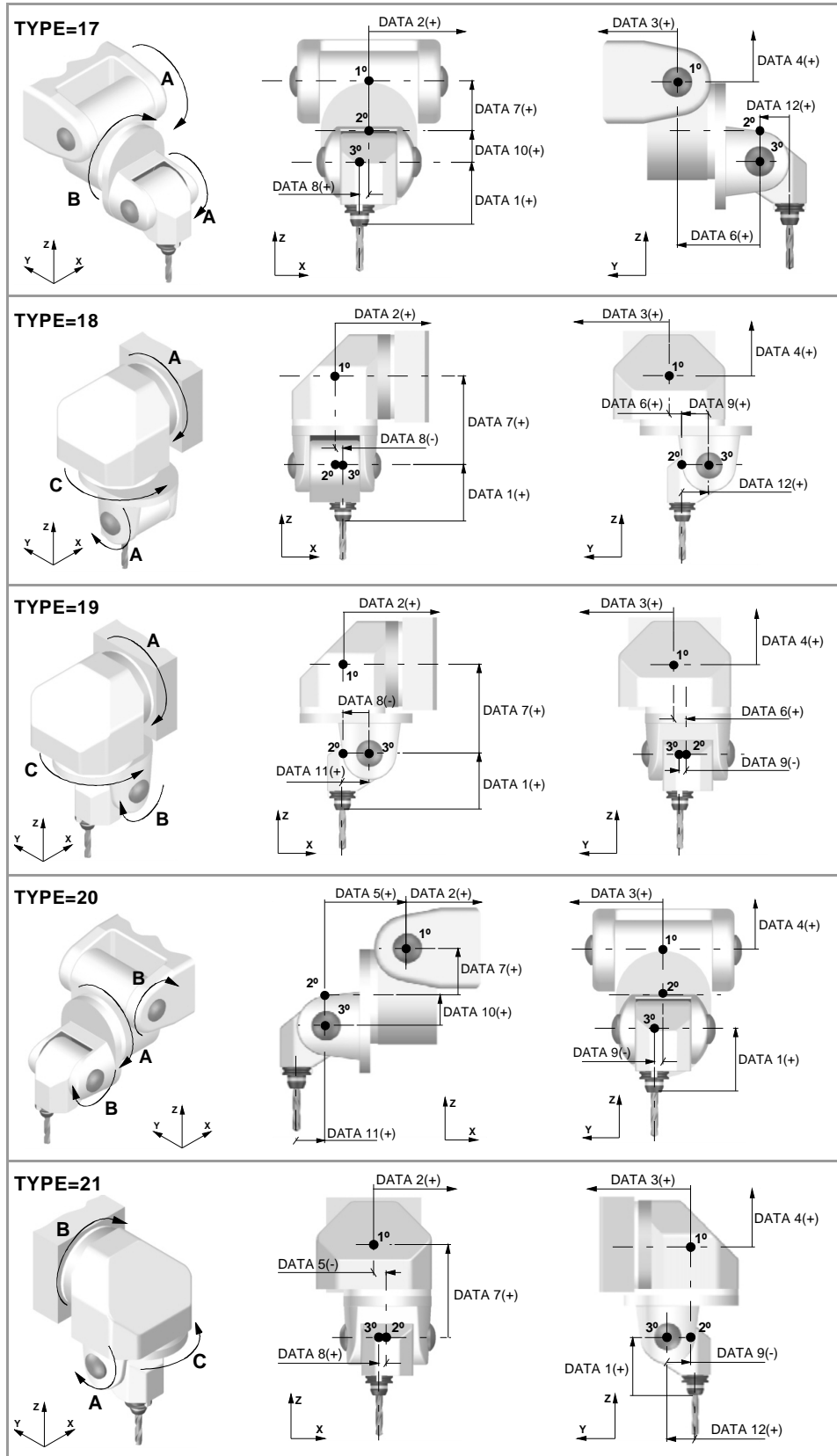
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

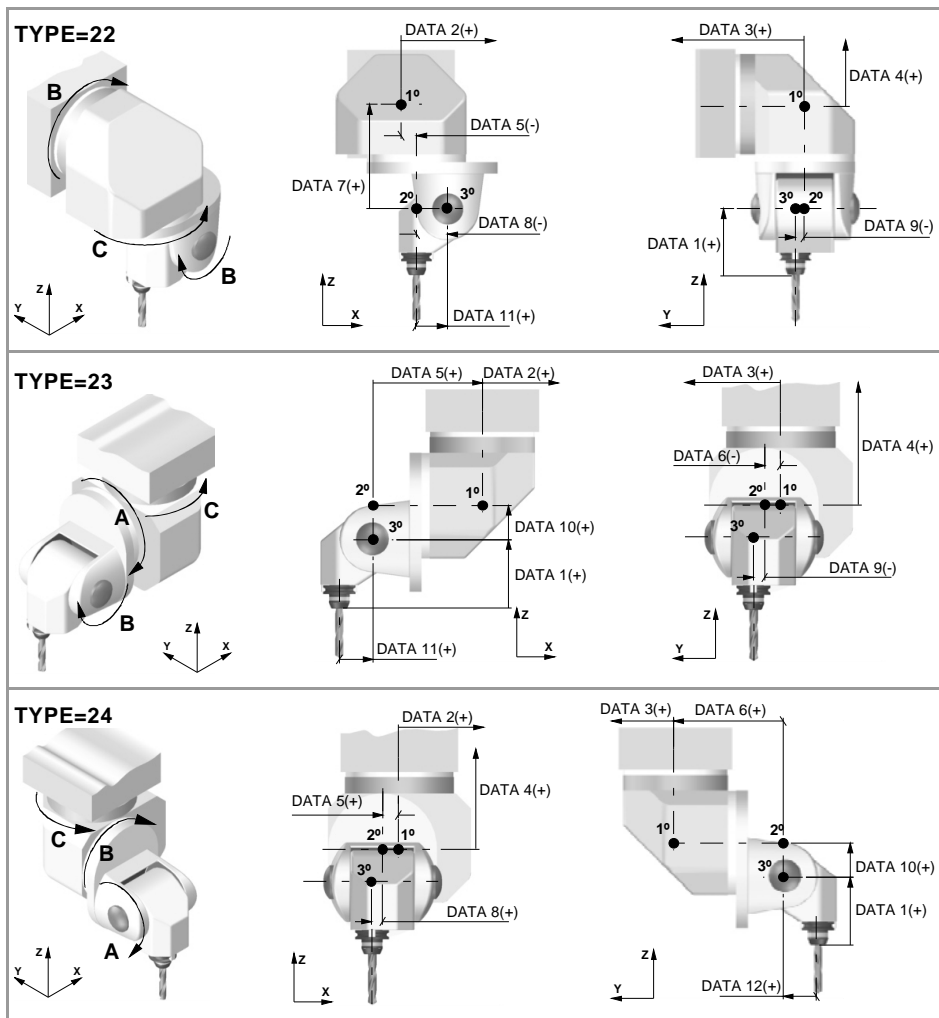
MACHINE PARAMETERS.
Machine parameters for kinematics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)



TDATA13
Rest position of the main rotary axis.

Possible values: Within $\pm 999999999.0000^\circ$.
 By default: 0° .

See parameter TDATA13.

TDATA14
Rest position of the secondary rotary axis.

Possible values: Within $\pm 999999999.0000^\circ$.
 By default: 0° .

See parameter TDATA14.

TDATA15
Rest position of the third rotary axis.

Possible values: Within $\pm 999999999.0000^\circ$.
 By default: 0° .

The rest position is when the tool is perpendicular to the work plane (parallel to the longitudinal axis).

TDATA16
Turning direction of the main rotary axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).
 Default value: 0.

See parameter TDATA18.

2.

MACHINE PARAMETERS.
 Machine parameters for kinetics.



CNC 8070

(REF: 1709)

TDATA17

Turning direction of the main rotary axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

See parameter TDATA18.

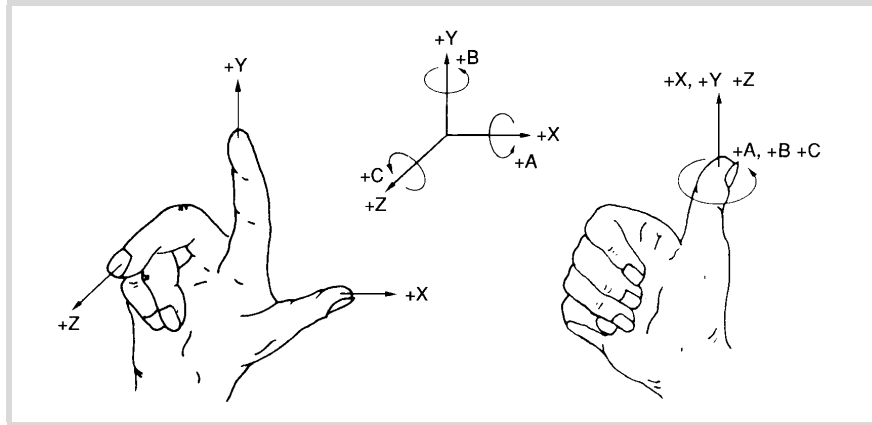
TDATA18

Turning direction of the secondary rotary axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

The direction of the XYZ axes, according to the DIN 66217 standard is easy to remember using the rule of the right hand. On rotary axes, the turning direction is established when bending your fingers (closing your hand) around the associated linear axis while your thumb is pointing in the positive direction of the linear axis.



TDATA19

Manual rotary axes or servo-controlled.

Possible values: 0 to 7.

Default value: 0.

This parameter indicates whether the rotary axes are manual or servo-controlled.

TDATA12	Meaning.
0	The three axes are servo-controlled.
1	Main axis manual and the rest servo-controlled.
2	Secondary axis manual and the rest servo-controlled.
3	Main axis and secondary axis manual, the third axis servo-controlled.
4	Third axis manual and the rest servo-controlled.
5	Main axis and third axis manual, the secondary axis servo-controlled.
6	Secondary axis and third axis manual, the main axis servo-controlled.
7	All the axes are manual.

2.

MACHINE PARAMETERS.
Machine parameters for kinetics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.9.6 Definition of the –C– axis kinematics (types 41 through 42).

In this type of kinematics, one must define the physical location of the rotary axis with respect to the linear axes. If these kinematics are defined, it assumes that the rotary axis coincides with the linear axis (e.g. the spindle of a lathe).

These kinematics are selected via part-program with function #FACE. If when executing this function the kinematics is not selected, the CNC will take it from the first kinematics type 41 or 42 defined in the table.

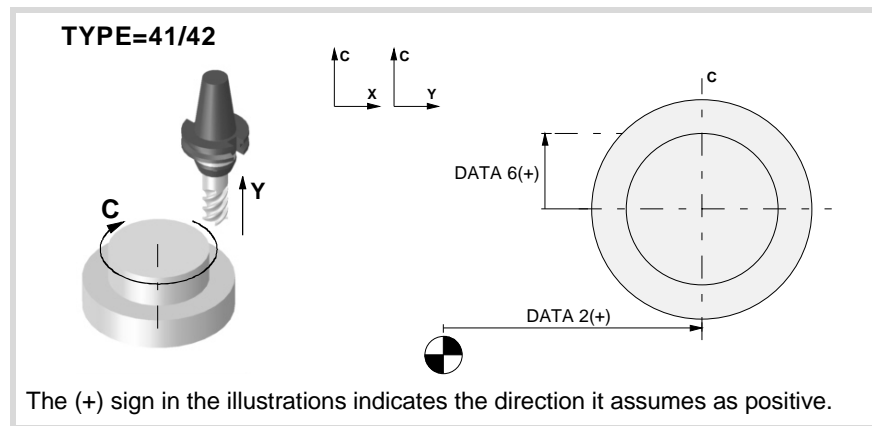
TDATA2

Rotary axis position.

Possible values: Within ± 999999999.0000

Default value: 0.

It indicates the distance from the rotary axis to the linear axis on which it unrolls. When defined with a $\cdot 0\cdot$ value, it assumes that the rotary axis coincides with the linear axis (e.g. the spindle of a lathe).



TDATA4

Angular offset of the rotary axis.

Possible values: Within $\pm 999999999.0000^\circ$.

Default value: 0.

TDATA5

Position of the rotation axis.

Possible values: 0 (the rotation axis is in part-zero) / 1 (the position of the rotating axis is given by DATA2).

Default value: 0.

TDATA6

Tool misalignment with the C axis.

Possible values: Within ± 999999999.0000

Default value: 0.

This parameter may be used to correct the misalignment of the tool. A tool is out of alignment when its zero position does not coincide with the rotation axis of the axis. The misalignment of the tool results in a circular area with a TDATA6 radius that cannot be machined.

2.

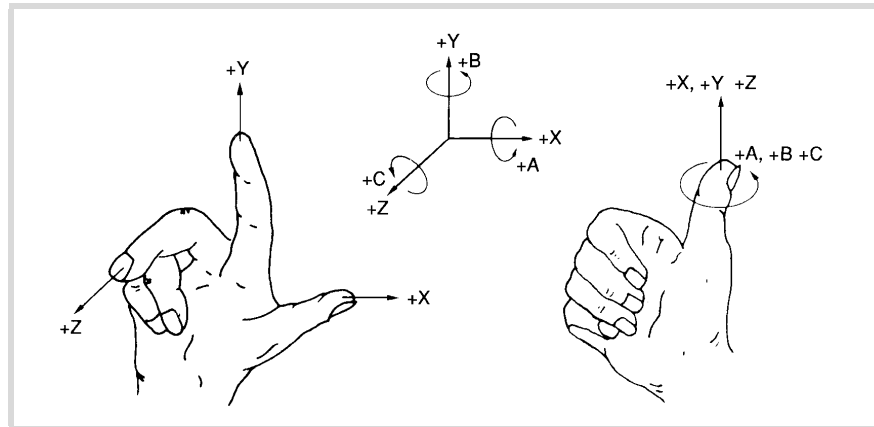
MACHINE PARAMETERS.
Machine parameters for kinematics.

TDATA10**Turning direction of the rotation axis.**

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

On rotary axes, the turning direction is established when bending your fingers (closing your hand) around the associated linear axis while your thumb is pointing in the positive direction of the linear axis.

**2.****MACHINE PARAMETERS.**

Machine parameters for kinematics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.9.7 Definition of the -C- axis kinematics (type 43) .

In this type of kinematics, one must define the physical location of the rotary axis with respect to the linear axes. If these kinematics are defined, it assumes that the rotary axis coincides with the linear axis (e.g. the spindle of a lathe).

These kinematics are selected via part-program with function #CYL. If when executing this function the kinematics is not selected, the CNC will take it from the first kinematics type 43 defined in the table.

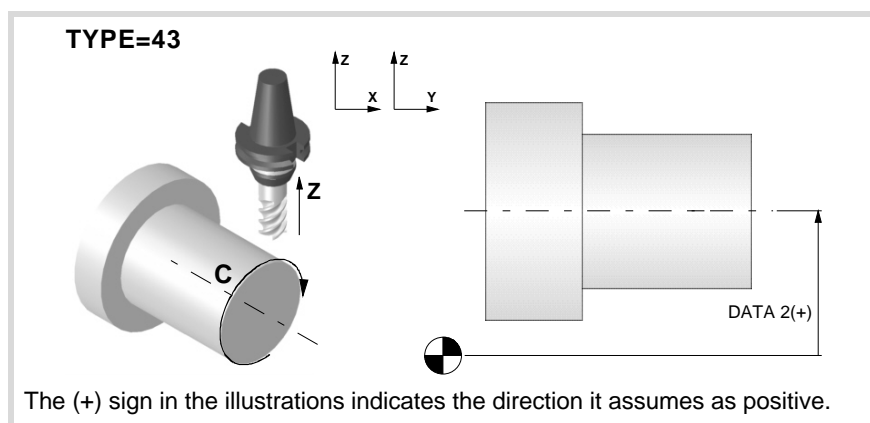
TDATA2

Rotary axis position.

Possible values: Within ± 999999999.0000

Default value: 0.

It indicates the position of the rotary axis along the tool axis. When defined with a $\cdot 0$ value, it assumes that the rotary axis coincides with the linear axis (e.g. the spindle of a lathe).



TDATA4

Angular offset for the rotary axis.

Possible values: Within $\pm 999999999.0000^\circ$.

Default value: 0.

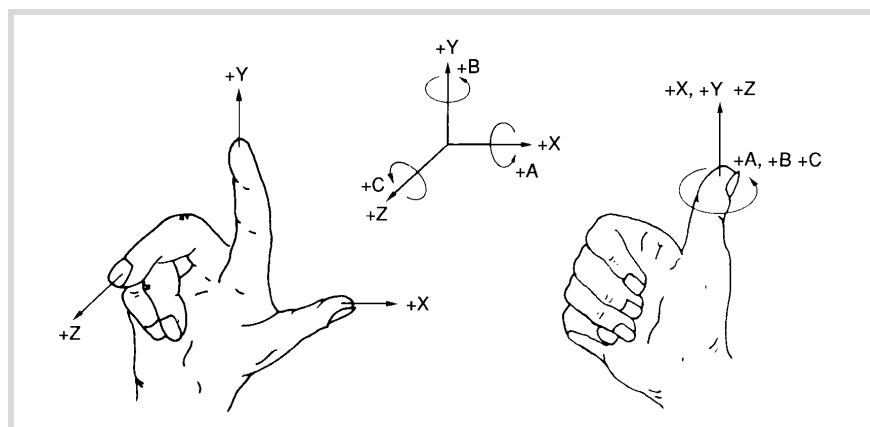
TDATA10

Turning direction of the rotation axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

On rotary axes, the turning direction is established when bending your fingers (closing your hand) around the associated linear axis while your thumb is pointing in the positive direction of the linear axis.



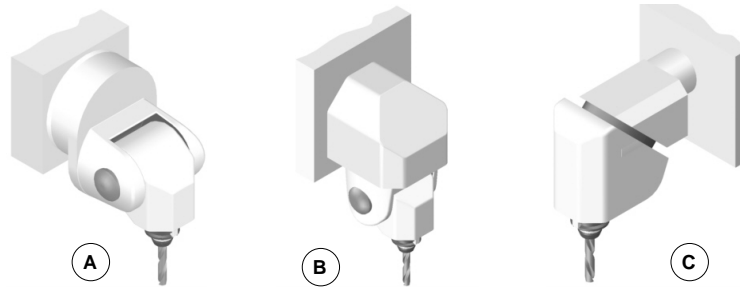
2.

MACHINE PARAMETERS.

Machine parameters for kinematics.

2.9.8 Vectorial definition of spindle kinematics (type 50).

This kinematics may be used to control any type of spindle that has a maximum of two rotary axes.

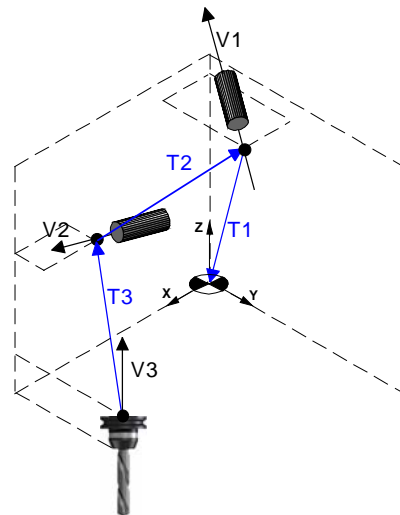


A Spherical spindle. B Orthogonal spindle. C Swivel (angular) spindle.

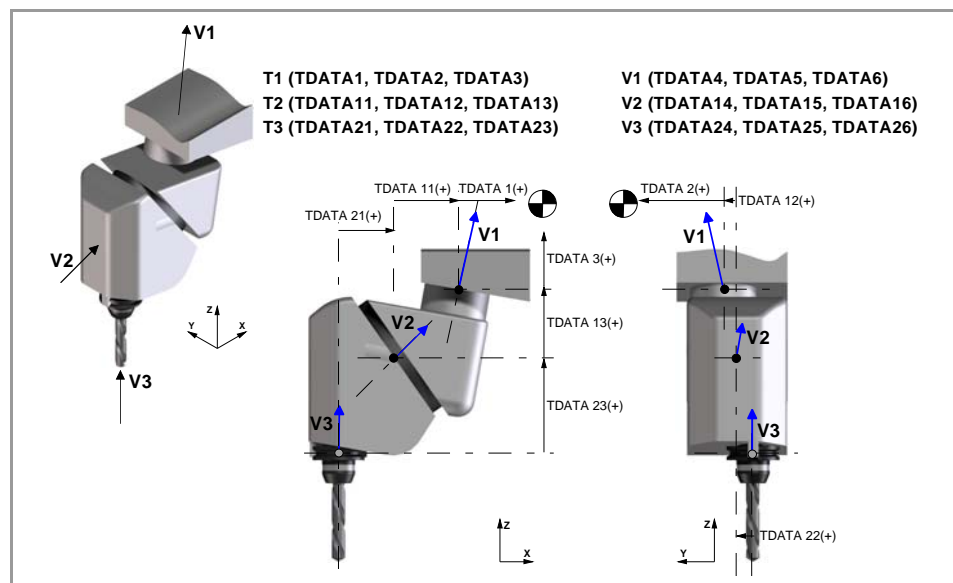
Vectorial definition of the kinematics.

This type of kinematics may be used to define the rotary axes that make it up using vectors. Each rotary axis of the kinematics is defined with a translation vector (position of the axis) and a direction vector (direction of the axis). The position of the tool holder is defined the same way, with a translation vector and a direction vector.

The translation vector may have its origin anywhere on the rotation axis of the rotary axis (except the translation value of the tool holder, whose origin is at its base). This reference point will be the same for all the translation vector associated with that axis.



- T1: Translation vector of the first rotary axis.
- T2: Translation vector of the second rotary axis.
- T3: Translation vector of the tool holder.
- V1: Direction vector of the first rotary axis.
- V2: Direction vector of the second rotary axis.
- V3: Direction vector of the tool holder.



2.

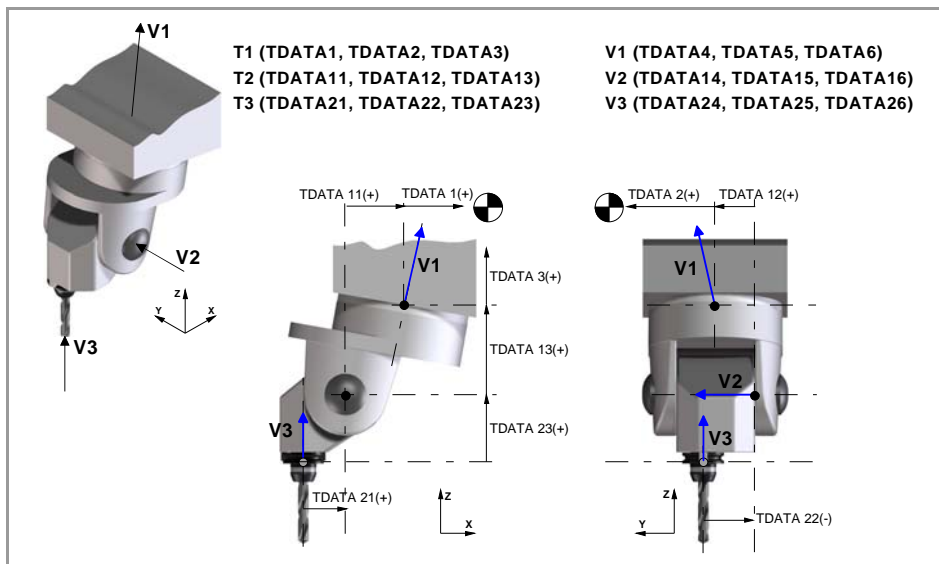
MACHINE PARAMETERS.
 Machine parameters for kinematics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)



2.

MACHINE PARAMETERS.
Machine parameters for kinetics.

TDATA 1..TDATA3

Translation vector of the main rotary axis.

Possible values: From 0 to 99999.9999 mm / from 0 to 3937.00787 inches.

Default value: 0.

This translation vector indicates the distance from the reference point of the main rotary axis to the machine reference point.

- TDATA1 Distance from the main rotary axis to the machine reference point (X axis).
- TDATA2 Distance from the main rotary axis to the machine reference point (Y axis).
- TDATA3 Distance from the main rotary axis to the machine reference point (Z axis).

TDATA 4..TDATA6

Direction vector of the main rotary axis.

- TDATA4 Component of the direction vector of the main rotary axis (X axis).
- TDATA5 Component of the direction vector of the main rotary axis (Y axis).
- TDATA6 Component of the direction vector of the main rotary axis (Z axis).

TDATA7

Rest position of the main rotary axis.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

The rest position is when the tool is perpendicular to the work plane (parallel to the longitudinal axis).

TDATA8

Turning direction of the main rotary axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

The direction of the XYZ axes, according to the DIN 66217 standard is easy to remember using the rule of the right hand. On rotary axes, the turning direction is established when bending your fingers (closing your hand) around the associated linear axis while your thumb is pointing in the positive direction of the linear axis.



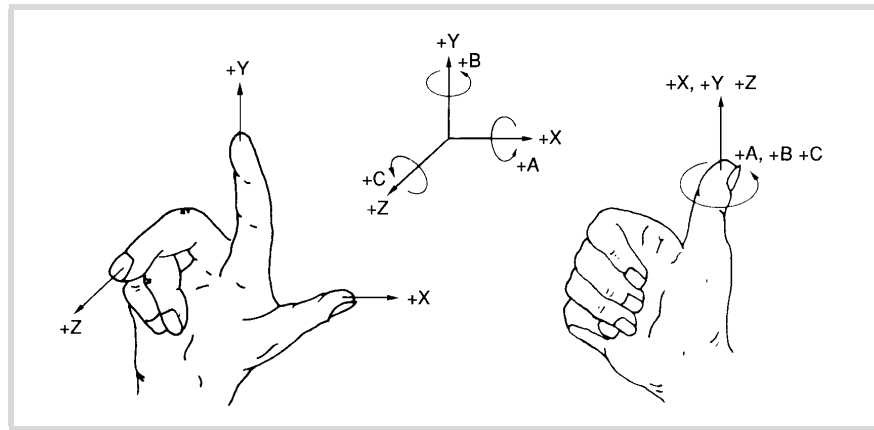
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

MACHINE PARAMETERS.
Machine parameters for kinematics.

**TDATA9**

Main rotary axis, manual or servo-controlled.

Possible values: 0 (servo-controlled) / 1 (manual).

Default value: 0.

This parameter indicates whether the rotary axis is manual or servo-controlled.

TDATA11..TDATA13

Translation vector of the secondary rotary axis.

Possible values: From 0 to 99999.9999 mm / from 0 to 3937.00787 inches.

Default value: 0.

This translation vector indicates the distance from the reference point of the secondary rotary axis to the reference point of the main rotary axis.

TDATA11 Distance from the secondary rotary axis to the main axis (X axis).

TDATA12 Distance from the secondary rotary axis to the main axis (Y axis).

TDATA13 Distance from the secondary rotary axis to the main axis (Z axis).

TDATA 14..TDATA16

Direction vector of the secondary rotary axis.

TDATA14 Component of the direction of the secondary rotary axis (X axis).

TDATA15 Component of the direction of the secondary rotary axis (Y axis).

TDATA16 Component of the direction of the secondary rotary axis (Z axis).

TDATA17

Rest position of the secondary rotary axis.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

The rest position is when the tool is perpendicular to the work plane (parallel to the longitudinal axis).

TDATA18

Turning direction of the secondary rotary axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

See parameter TDATA8.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TDATA19

Secondary rotary axis, manual or servo-controlled.

Possible values: 0 (servo-controlled) / 1 (manual).

Default value: 0.

This parameter indicates whether the rotary axis is manual or servo-controlled.

TDATA21..TDATA23

Translation vector of the tool holder.

Possible values: From 0 to 99999.9999 mm / from 0 to 3937.00787 inches.

Default value: 0.

This translation vector indicates the distance from the base of the tool holder to the reference point of the secondary rotary axis.

TDATA21 Distance from the tool base to the secondary rotary axis (X axis).

TDATA22 Distance from the tool base to the secondary rotary axis (Y axis).

TDATA23 Distance from the tool base to the secondary rotary axis (Z axis).

TDATA 24..TDATA26

Direction vector of the tool holder.

TDATA24 Component of the direction of the tool holder (X axis).

TDATA25 Component of the direction of the tool holder (Y axis).

TDATA26 Component of the direction of the tool holder (Z axis).

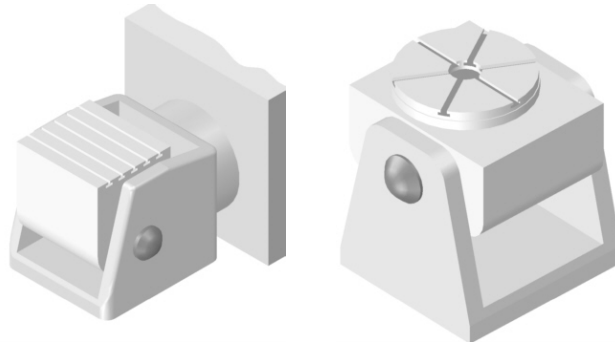
2.

MACHINE PARAMETERS.

Machine parameters for kinetics.

2.9.9 Vectorial definition of table kinematics (type 51).

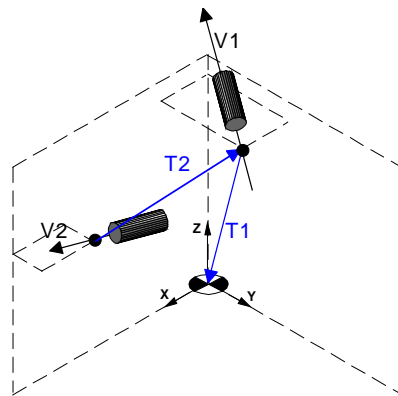
This kinematics may be used to control any type of table that has a maximum of two rotary axes.



Vectorial definition of the kinematics.

This type of kinematics may be used to define the rotary axes that make it up using vectors. Each rotary axis of the kinematics is defined with a translation vector (position of the axis) and a direction vector (direction of the axis).

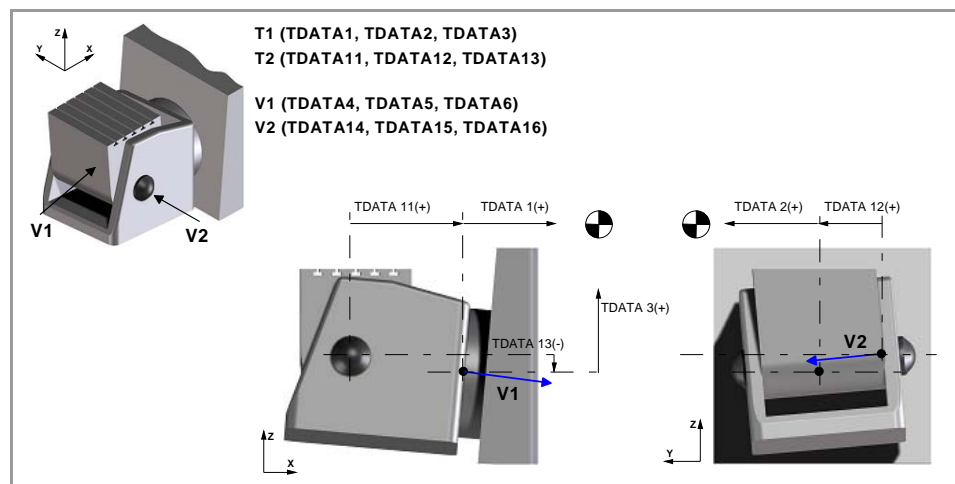
The translation vector may have its origin anywhere on the rotation axis of the rotary axis. This reference point will be the same for all the translation vector associated with that axis.



From now on, all the explanations assume that the main axes are X Y Z.

T1: Translation vector of the first rotary axis.
T2: Translation vector of the second rotary axis.

V1: Direction vector of the first rotary axis.
V2: Direction vector of the second rotary axis.



2.

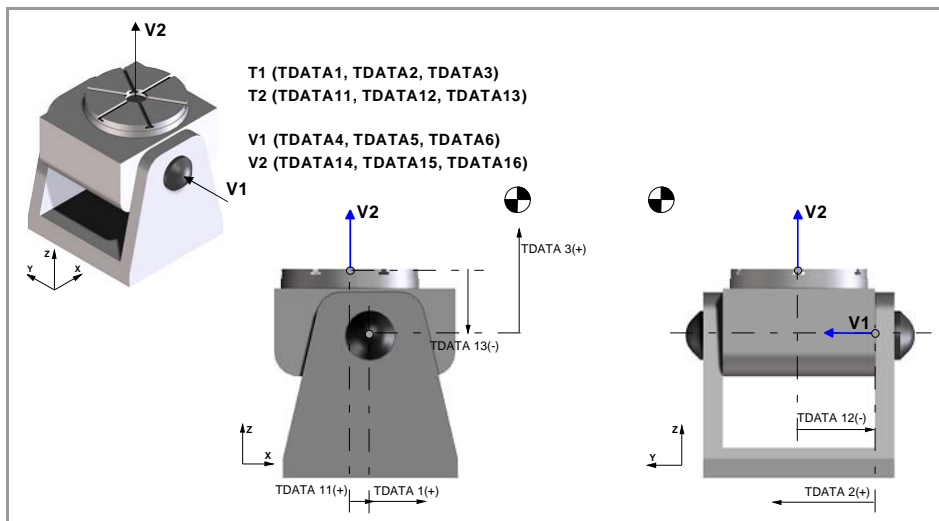
MACHINE PARAMETERS.
Machine parameters for kinematics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)



TDATA 1..TDATA3

Translation vector of the main rotary axis.

Possible values: From 0 to 99999.9999 mm / from 0 to 3937.00787 inches.

Default value: 0.

This translation vector indicates the distance from the reference point of the main rotary axis to the machine reference point.

- TDATA1 Distance from the main rotary axis to the machine reference point (X axis).
- TDATA2 Distance from the main rotary axis to the machine reference point (Y axis).
- TDATA3 Distance from the main rotary axis to the machine reference point (Z axis).

TDATA 4..TDATA6

Direction vector of the main rotary axis.

- TDATA4 Component of the direction vector of the main rotary axis (X axis).
- TDATA5 Component of the direction vector of the main rotary axis (Y axis).
- TDATA6 Component of the direction vector of the main rotary axis (Z axis).

TDATA7

Rest position of the main rotary axis.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

The rest position is the one where the direction vector of the rotary axes of the kinematics have been defined.

TDATA8

Turning direction of the main rotary axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

The direction of the XYZ axes, according to the DIN 66217 standard is easy to remember using the rule of the right hand. On rotary axes, the turning direction is established when bending your fingers (closing your hand) around the associated linear axis while your thumb is pointing in the positive direction of the linear axis.

2.

MACHINE PARAMETERS.
Machine parameters for kinetics.



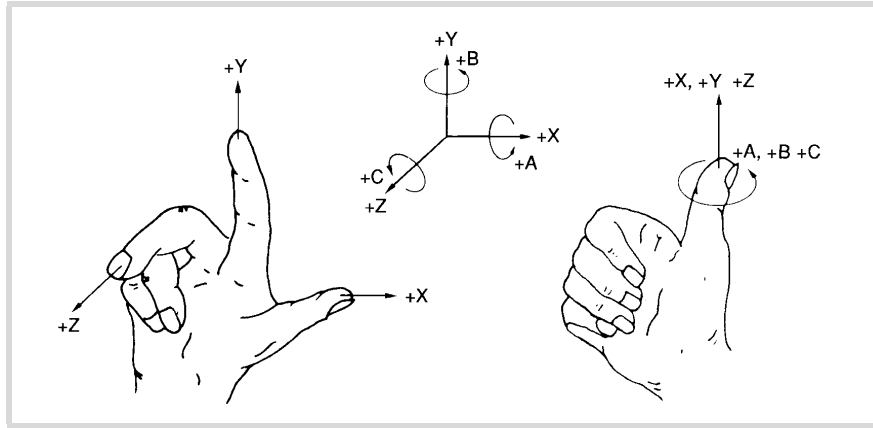
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.

MACHINE PARAMETERS.
Machine parameters for kinematics.



TDATA9

Main rotary axis, manual or servo-controlled.

Possible values: 0 (servo-controlled) / 1 (manual).

Default value: 0.

This parameter indicates whether the rotary axis is manual or servo-controlled.

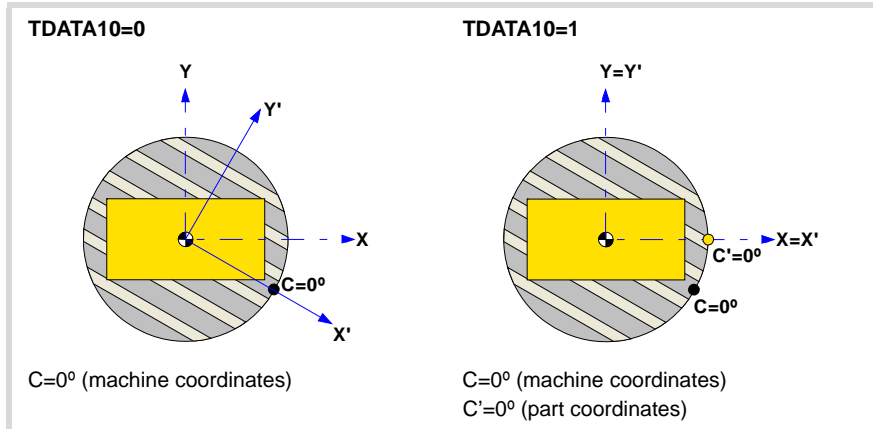
TDATA10

Origin for applying the RTCP.

Possible values: 0 (machine zero) / 1 (part zero).

Default value: 0.

This parameter indicates whether the CNC applies the RTCP referred to machine zero or to part zero.



TDATA11..TDATA13

Translation vector of the secondary rotary axis.

Possible values: From 0 to 99999.9999 mm / from 0 to 3937.00787 inches.

Default value: 0.

This translation vector indicates the distance from the reference point of the secondary rotary axis to the reference point of the main rotary axis.

- TDATA11 Distance from the secondary rotary axis to the main axis (X axis).
- TDATA12 Distance from the secondary rotary axis to the main axis (Y axis).
- TDATA13 Distance from the secondary rotary axis to the main axis (Z axis).

TDATA 14..TDATA16

Direction vector of the secondary rotary axis.

- TDATA14 Component of the direction of the secondary rotary axis (X axis).
- TDATA15 Component of the direction of the secondary rotary axis (Y axis).
- TDATA16 Component of the direction of the secondary rotary axis (Z axis).

(REF: 1709)



CNC 8070

TDATA17
Rest position of the secondary rotary axis.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

The rest position is the one where the direction vector of the rotary axes of the kinematics have been defined.

TDATA18
Turning direction of the secondary rotary axis.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

See parameter TDATA8.

TDATA19
Secondary rotary axis, manual or servo-controlled.

Possible values: 0 (servo-controlled) / 1 (manual).

Default value: 0.

This parameter indicates whether the rotary axis is manual or servo-controlled.

TDATA20
Origin for applying the RTCP.

Possible values: 0 (machine zero) / 1 (part zero).

Default value: 0.

See parameter TDATA10.

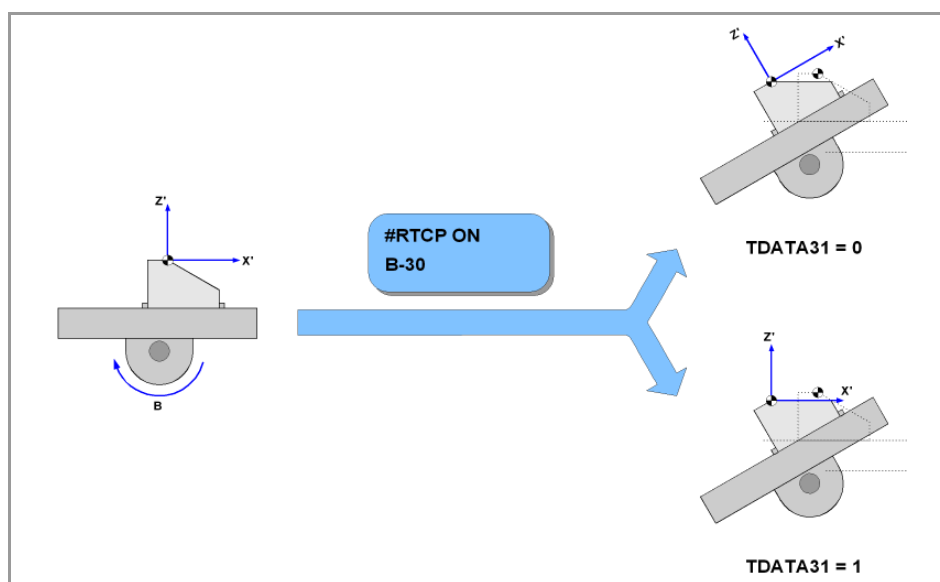
TDATA31
Rotate the part coordinate system when rotating the table.

Possible values: 0 (yes) / 1 (no).

Default value: 0.

Having the RTCP mode active, this parameter determines whether the part coordinate system is fixed on the part and rotates with it or it stays parallel to the machine coordinate system.

TDATA31	Meaning.
0	When changing the part orientation, the position occupied by the tool tip on the part does not change. The coordinate system is fixed on the part and rotates with it.
1	When changing the part orientation, the position occupied by the point set as part zero does not change. The coordinate system stays parallel to the machine coordinate system.



2.

MACHINE PARAMETERS.
 Machine parameters for kinematics.



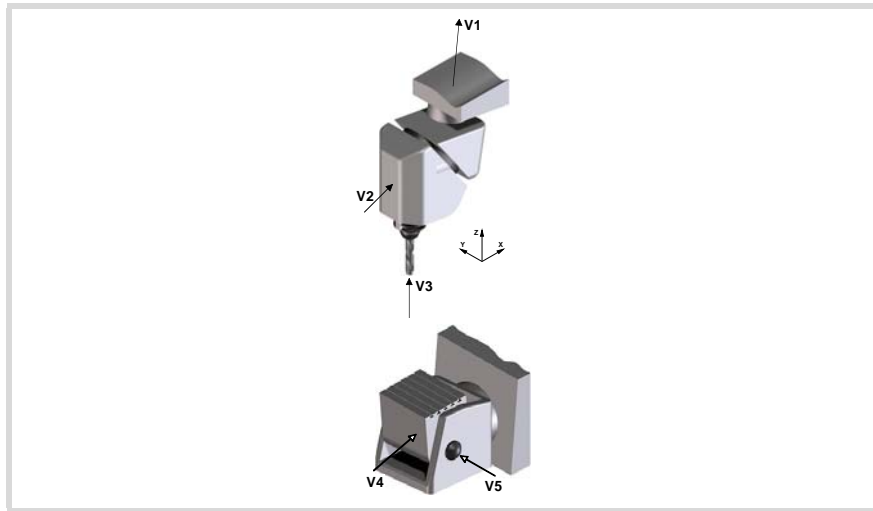
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.9.10 Vectorial definition of spindle-table kinematics (type 52).

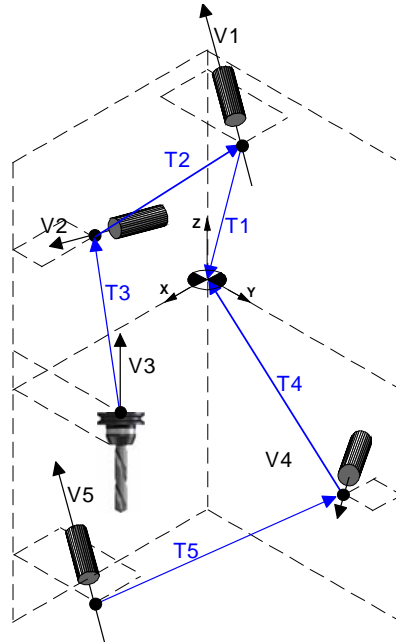
This kinematics may be used to control any type of kinematics consisting of a spindle and a table, having a maximum of two rotary axes on the spindle and two on the table.



Vectorial definition of the kinematics.

This type of kinematics may be used to define the rotary axes that make it up using vectors. Each rotary axis of the kinematics is defined with a translation vector (position of the axis) and a direction vector (direction of the axis). The position of the tool holder is defined the same way, with a translation vector and a direction vector.

The translation vector may have its origin anywhere on the rotation axis of the rotary axis (except the translation value of the tool holder, whose origin is at its base). This reference point will be the same for all the translation vector associated with that axis.



Spindle:
 T1: Translation vector of the first rotary axis.
 T2: Translation vector of the second rotary axis.
 T3: Translation vector of the tool holder.
 Table:
 T4: Translation vector of the third rotary axis.
 T5: Translation vector of the fourth rotary axis.

Spindle:
 V1: Direction vector of the first rotary axis.
 V2: Direction vector of the second rotary axis.
 V3: Direction vector of the tool holder.
 Table:
 V4: Direction vector of the third rotary axis.
 V5: Direction vector of the fourth rotary axis.

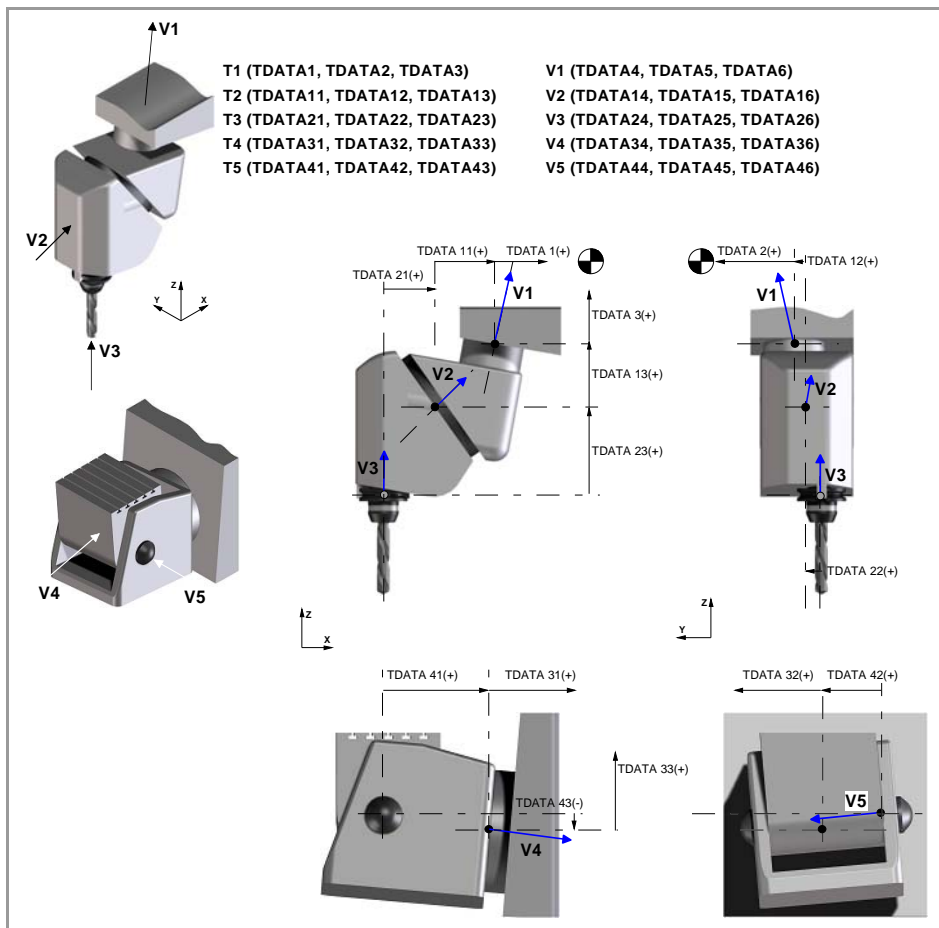
Definition of the axes that do not exist in the kinematics.

When one of the axes of the kinematics is not physically on the machine, its parameters are set as follows.

- Translation vector of the axis: 0,0,0
- Direction vector of the axis: 0,0,1
- Rest position: 0°
- Manual or servo-controlled rotary axis: 1 (manual)
- The variable of the corresponding component: V.G.POSROTn = 0

2.

MACHINE PARAMETERS.
 Machine parameters for kinematics.



2.

MACHINE PARAMETERS.
Machine parameters for kinematics.

TDATA 1..TDATA3

Translation vector of the main rotary axis of the spindle.

Possible values: From 0 to 99999.9999 mm / from 0 to 3937.00787 inches.

Default value: 0.

This translation vector indicates the distance from the reference point of the main rotary axis to the machine reference point.

TDATA1 Distance from the main rotary axis to the machine reference point (X axis).

TDATA2 Distance from the main rotary axis to the machine reference point (Y axis).

TDATA3 Distance from the main rotary axis to the machine reference point (Z axis).

If the axis does not exist in the kinematics, it must be defined as TDATA1=0, TDATA2=0, TDATA3=0.

TDATA 4..TDATA6

Direction vector of the main rotary axis of the spindle.

TDATA4 Component of the direction vector of the main rotary axis (X axis).

TDATA5 Component of the direction vector of the main rotary axis (Y axis).

TDATA6 Component of the direction vector of the main rotary axis (Z axis).

If the axis does not exist in the kinematics, it must be defined as TDATA4=0, TDATA5=0, TDATA6=1.

TDATA7

Rest position of the main rotary axis of the spindle.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

The rest position is when the tool is perpendicular to the work plane (parallel to the longitudinal axis). If the axis does not exist in the kinematics, it must be defined as TDATA7=0.



FAGOR AUTOMATION

CNC 8070

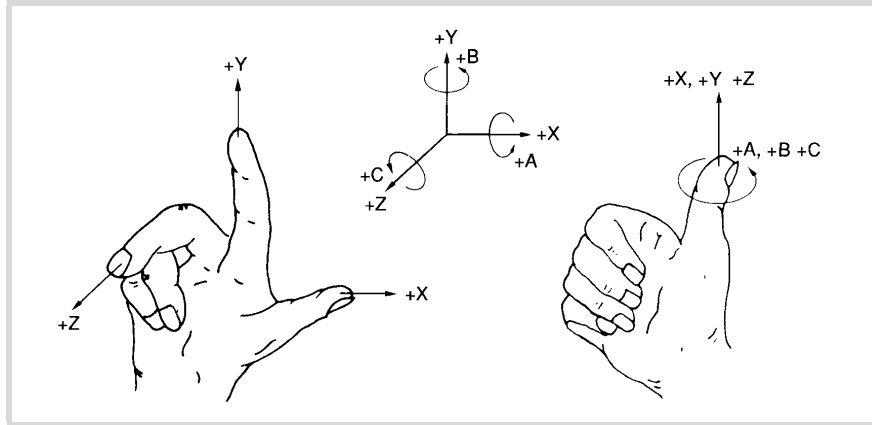
(REF: 1709)

TDATA8**Turning direction of the main rotary axis of the spindle.**

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

The direction of the XYZ axes, according to the DIN 66217 standard is easy to remember using the rule of the right hand. On rotary axes, the turning direction is established when bending your fingers (closing your hand) around the associated linear axis while your thumb is pointing in the positive direction of the linear axis.

**TDATA9****Manual or servo-controlled main rotary axis of the spindle.**

Possible values: 0 (servo-controlled) / 1 (manual).

Default value: 0.

This parameter indicates whether the rotary axis is manual or servo-controlled. If the axis does not exist in the kinematics, it must be defined as TDATA9=1.

TDATA11..TDATA13**Translation vector of the secondary rotary axis of the spindle.**

Possible values: From 0 to 99999.9999 mm / from 0 to 3937.00787 inches.

Default value: 0.

This translation vector indicates the distance from the reference point of the secondary rotary axis to the reference point of the main rotary axis.

- TDATA11 Distance from the secondary rotary axis to the main axis (X axis).
- TDATA12 Distance from the secondary rotary axis to the main axis (Y axis).
- TDATA13 Distance from the secondary rotary axis to the main axis (Z axis).

If the axis does not exist in the kinematics, it must be defined as TDATA11=0, TDATA12=0, TDATA13=0.

TDATA 14..TDATA16**Direction vector of the secondary rotary axis of the spindle.**

- TDATA14 Component of the direction of the secondary rotary axis (X axis).
- TDATA15 Component of the direction of the secondary rotary axis (Y axis).
- TDATA16 Component of the direction of the secondary rotary axis (Z axis).

If the axis does not exist in the kinematics, it must be defined as TDATA14=0, TDATA15=0, TDATA16=1.

TDATA17**Rest position of the secondary rotary axis of the spindle.**

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

The rest position is when the tool is perpendicular to the work plane (parallel to the longitudinal axis). If the axis does not exist in the kinematics, it must be defined as TDATA17=0.

2.

MACHINE PARAMETERS.
Machine parameters for kinematics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TDATA18

Turning direction of the secondary rotary axis of the spindle.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

See parameter TDATA8.

TDATA19

manual or servo-controlled secondary rotary axis of the spindle.

Possible values: 0 (servo-controlled) / 1 (manual).

Default value: 0.

This parameter indicates whether the rotary axis is manual or servo-controlled. If the axis does not exist in the kinematics, it must be defined as TDATA19=1.

TDATA21..TDATA23

Translation vector of the tool holder.

Possible values: From 0 to 99999.9999 mm / from 0 to 3937.00787 inches.

Default value: 0.

This translation vector indicates the distance from the base of the tool holder to the reference point of the secondary rotary axis.

- TDATA21 Distance from the tool base to the secondary rotary axis (X axis).
- TDATA22 Distance from the tool base to the secondary rotary axis (Y axis).
- TDATA23 Distance from the tool base to the secondary rotary axis (Z axis).

TDATA 24..TDATA26

Direction vector of the tool holder.

- TDATA24 Component of the direction of the tool holder (X axis).
- TDATA25 Component of the direction of the tool holder (Y axis).
- TDATA26 Component of the direction of the tool holder (Z axis).

TDATA 31..TDATA33

Translation vector of the main rotary axis of the table.

Possible values: From 0 to 99999.9999 mm / from 0 to 3937.00787 inches.

Default value: 0.

This translation vector indicates the distance from the reference point of the main rotary axis to the machine reference point.

- TDATA31 Distance from the main rotary axis to the machine reference point (X axis).
- TDATA32 Distance from the main rotary axis to the machine reference point (Y axis).
- TDATA33 Distance from the main rotary axis to the machine reference point (Z axis).

If the axis does not exist in the kinematics, it must be defined as TDATA31=0, TDATA32=0, TDATA33=0.

TDATA 34..TDATA36

Direction vector of the main rotary axis of the table.

- TDATA34 Component of the direction vector of the main rotary axis (X axis).
- TDATA35 Component of the direction vector of the main rotary axis (Y axis).
- TDATA36 Component of the direction vector of the main rotary axis (Z axis).

If the axis does not exist in the kinematics, it must be defined as TDATA34=0, TDATA35=0, TDATA36=1.

2.

MACHINE PARAMETERS.
Machine parameters for kinetics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TDATA37

Rest position of the main rotary axis of the table.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

The rest position is the one where the direction vector of the rotary axes of the kinematics have been defined. If the axis does not exist in the kinematics, it must be defined as TDATA37=0.

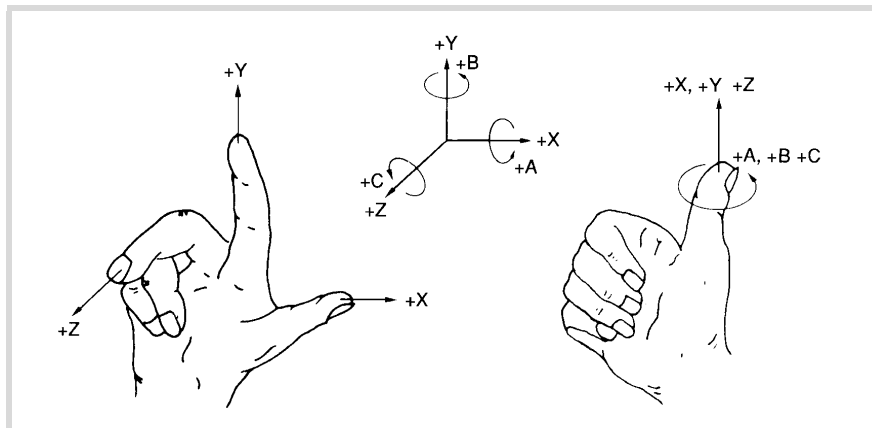
TDATA38

Turning direction of the main rotary axis of the table.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

The direction of the XYZ axes, according to the DIN 66217 standard is easy to remember using the rule of the right hand. On rotary axes, the turning direction is established when bending your fingers (closing your hand) around the associated linear axis while your thumb is pointing in the positive direction of the linear axis.



TDATA39

Manual or servo-controlled main rotary axis of the table.

Possible values: 0 (servo-controlled) / 1 (manual).

Default value: 0.

This parameter indicates whether the rotary axis is manual or servo-controlled. If the axis does not exist in the kinematics, it must be defined as TDATA39=1.

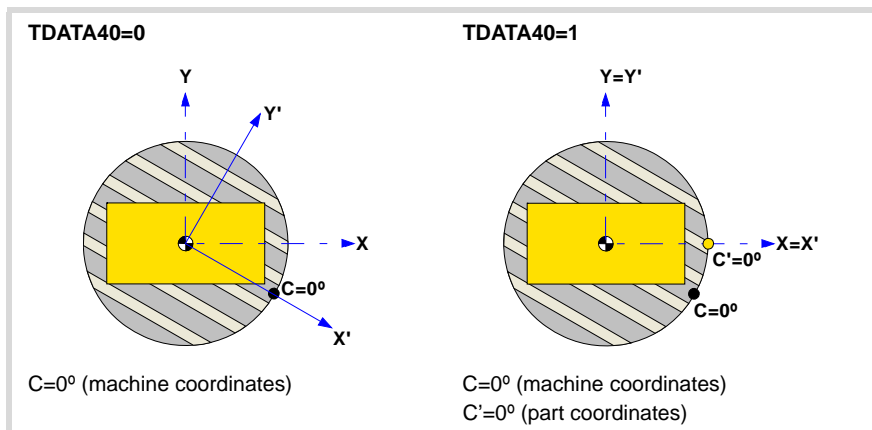
TDATA40

Origin for applying the RTCP.

Possible values: 0 (machine zero) / 1 (part zero).

Default value: 0.

This parameter indicates whether the CNC applies the RTCP referred to machine zero or to part zero.



2.

MACHINE PARAMETERS.
Machine parameters for kinematics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TDATA41..TDATA43

Translation vector of the secondary rotary axis of the table.

Possible values: From 0 to 99999.9999 mm / from 0 to 3937.00787 inches.

Default value: 0.

This translation vector indicates the distance from the reference point of the secondary rotary axis to the reference point of the main rotary axis.

TDATA41 Distance from the secondary rotary axis to the main axis (X axis).

TDATA42 Distance from the secondary rotary axis to the main axis (Y axis).

TDATA43 Distance from the secondary rotary axis to the main axis (Z axis).

If the axis does not exist in the kinematics, it must be defined as TDATA41=0, TDATA42=0, TDATA43=0.

TDATA 44..TDATA46

Direction vector of the secondary rotary axis of the table.

TDATA44 Component of the direction of the secondary rotary axis (X axis).

TDATA45 Component of the direction of the secondary rotary axis (Y axis).

TDATA46 Component of the direction of the secondary rotary axis (Z axis).

If the axis does not exist in the kinematics, it must be defined as TDATA44=0, TDATA45=0, TDATA46=1.

TDATA47

Rest position of the secondary rotary axis of the table.

Possible values: Within $\pm 99999.9999^\circ$.

Default value: 0.

The rest position is the one where the direction vector of the rotary axes of the kinematics have been defined. If the axis does not exist in the kinematics, it must be defined as TDATA7=0.

TDATA48

Turning direction of the secondary rotary axis of the table.

Possible values: 0 (by the standard DIN 66217) / 1 (opposite to the standard DIN 66217).

Default value: 0.

See parameter TDATA38.

TDATA49

Manual or servo-controlled secondary rotary axis of the table.

Possible values: 0 (servo-controlled) / 1 (manual).

Default value: 0.

This parameter indicates whether the rotary axis is manual or servo-controlled. If the axis does not exist in the kinematics, it must be defined as TDATA49=1.

TDATA50

Origin for applying the RTCP.

Possible values: 0 (machine zero) / 1 (part zero).

Default value: 0.

See parameter TDATA40.

2.

MACHINE PARAMETERS.
Machine parameters for kinematics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TDATA51

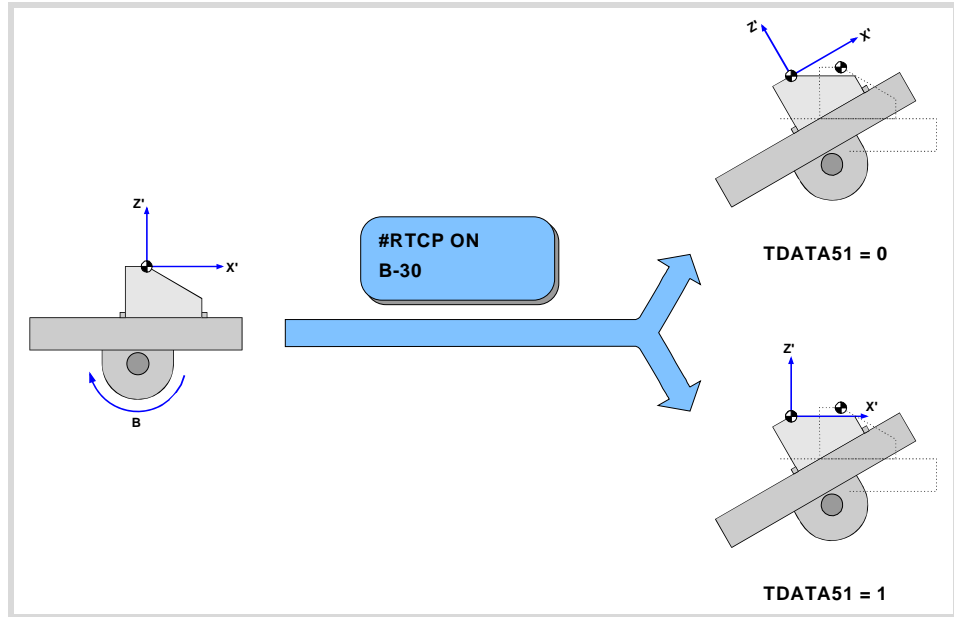
Rotate the part coordinate system when rotating the table.

Possible values: 0 (yes) / 1 (no).

Default value: 0.

Having the RTCP mode active, this parameter determines whether the part coordinate system is fixed on the part and rotates with it or it stays parallel to the machine coordinate system.

TDATA51	Meaning.
0	When changing the part orientation, the position occupied by the tool tip on the part does not change. The coordinate system is fixed on the part and rotates with it.
1	When changing the part orientation, the position occupied by the point set as part zero does not change. The coordinate system stays parallel to the machine coordinate system.



TDATA52

Apply full RTCP.

Possible values: 0 (yes) / 1 (no).

Default value: 0.

This parameter determines whether the CNC applies the RTCP to the whole kinematics or only to the spindle side.

TDATA52	Meaning.
0	Full RTCP, table + spindle.
1	RTCP considering only the spindle side.

2.

MACHINE PARAMETERS.
Machine parameters for kinematics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.9.11 Definition of the OEM kinematics (types 100 through 105).

In OEM kinematics, you must indicate the number of parameters, auxiliary variables and general purpose data being used.

NKINAX

Number of axes of the kinematics.

Possible values: From 0 to 8.

Default value: 0.

Associated variable: (V.)MPK.NKINAX[kin]

PARAM_D_SIZE

Number of parameters in decimal format.

Possible values: From 0 to 100.

Default value: 0.

Associated variable: (V.)MPK.PARAM_D_SIZE[kin]

TDATA1..100

Parameters for configuring the kinematics (decimal format).

There are 100 kinematics predefined by Fagor. For OEM kinematics, the number of parameters may be configured.

PARAM_I_SIZE

Number of parameters in integers.

Possible values: From 0 to 100.

Default value: 0.

Associated variable: (V.)MPK.PARAM_I_SIZE[kin]

TDATA_I1..100

Parameters for configuring the kinematics (integer format).

There are 100 kinematics predefined by Fagor. For OEM kinematics, the number of parameters may be configured.

AUXCTE_SIZE

Size of the area of auxiliary variables.

Possible values: From 0 to 1000 bytes.

Default value: 0.

Associated variable: (V.)MPK.AUXCTE_SIZE[kin]

KINDATA_SIZE

Size of the area for general purpose data.

Possible values: From 0 to 100000 bytes.

Default value: 0.

Associated variable: (V.)MPK.KINDATA_SIZE[kin]

2.

MACHINE PARAMETERS.
Machine parameters for kinematics.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

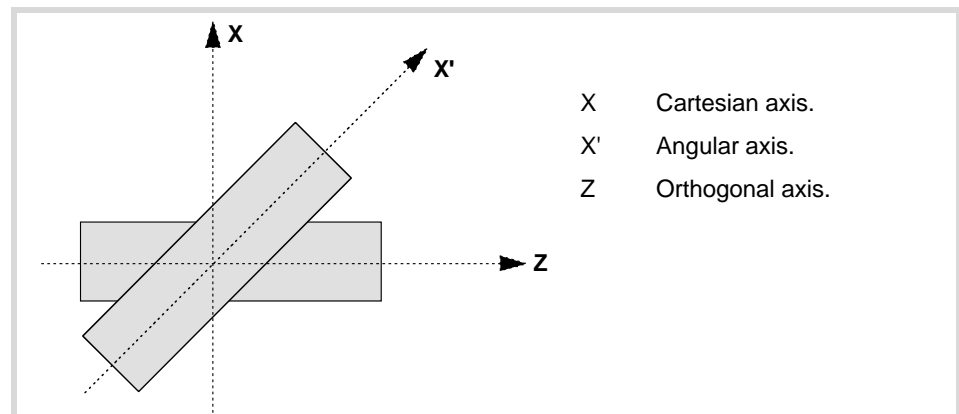
2.9.12 Configuration of angular transformations.

Up to 14 different angular transformations may be set for the same machine. The CNC assumes no transformation on power-up; the angular transformations are activated via part-program using the instruction #ANGAX ON. The angular transformation of an incline axis is kept active after a RESET or an M30.

What is an angular transformation of an incline axis?

The angular transformation of an incline axis may be used to program movements in Cartesian coordinates and execute them in a non-Cartesian plane; in other words, in a plane where the axes are not at 90° from each other.

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.

Considerations for the angular transformation of an incline axis.

The axes that make up the angular transformation must meet the following requirements:

- The angular transformation may be defined with any pair of axes of the system, but both axes must belong to the same channel in order to activate the transformation.
- 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.

2.

MACHINE PARAMETERS.
Machine parameters for kinetics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.9.13 Configuration of angular transformations (parameters).

NANG

Number of angular transformations defined.

Possible values: From 0 to 14.

Default value: 0.

Associated variable: (V.)MPK.NANG

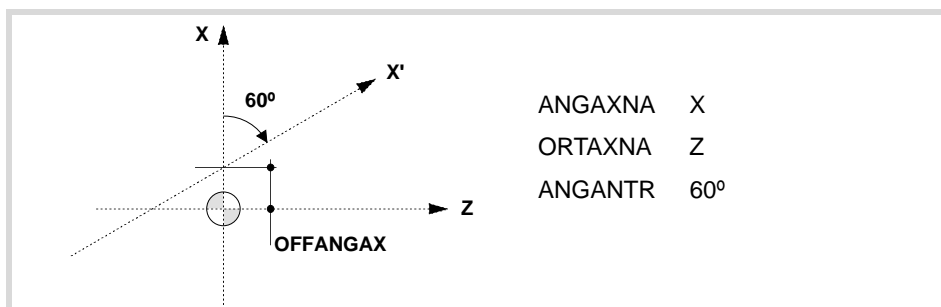
Up to 14 different angular transformations may be set for the same machine. The CNC assumes no transformation on power-up; the angular transformations are activated via part-program using the instruction #ANGAX ON. The angular transformation of an incline axis is kept active after a RESET or an M30.

ANGTR

Number of angular transformations.

It shows the configuration tables for angular transformations. The following parameters must be defined for each kinematics:

DATA	
ANGAXNA	Name of the angular axis (incline axis).
ORTAXNA	Name of the orthogonal axis.
ANGANTR	Angle between the Cartesian axis and the incline axis.
OFFANGAX	Offset of the origin of the angular transformation.



ANGAXNA

Name of the angular axis (incline axis).

Associated variable: (V.)MPK.ANGAXNA[ang]

Any axis defined in the parameter "AXISNAME".

ORTAXNA

Name of the orthogonal axis.

Associated variable: (V.)MPK.ORTAXNA[ang]

Name of the axis perpendicular to the Cartesian axis, used in the angular transformation.

Any axis defined in the parameter "AXISNAME".

ANGANTR

Angle between the Cartesian axis and the incline axis.

Possible values: Within $\pm 360.0000^\circ$.

Default value: 30 degrees.

Associated variable: (V.)MPK.ANGANTR[ang]

Angle between the Cartesian angle and the angular axis it is associated with. If its value is 0° , there is no need to do an angular transformation.

Positive angle when the angular axis has been rotated clockwise and negative if counterclockwise.

2.

MACHINE PARAMETERS.

Machine parameters for kinematics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

OFFANGAX

Offset of the origin of the angular transformation.

Possible values: Within ± 99999.9999 mm / within ± 3937.00787 inch.

Default value: 0.

Associated variable: (V.)MPK.OFFANGAX[ang]

Distance between machine zero and the origin of the coordinate system of the incline axis.

2.

MACHINE PARAMETERS.

Machine parameters for kinetics.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.10 Machine parameters for the magazine.

2.10.1 Tool magazine configuration.

NTOOLMZ

Number of tool magazines.

Possible values: From 0 to 4.

Default value: 1.

Associated variable: (V.)TM.NTOOLMZ

Number of system's tool magazines.

Although each channel has its own tool management, the tool magazines are not associated with any particular channel. They are not associated with any particular spindle either.

GROUND

Ground tools are permitted (manual load).

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)TM.MZGROUND[mz]

They are the ones not located in the magazine. When programming them, the CNC requests them to be inserted in the spindle.

MAGAZINE n

Tool magazine table.

This parameter shows the tables to define the magazine data. Each table shows the following machine parameters to configure it.

MAGAZINE n	
STORAGE	Parameters related to storage.
MANAGEMENT	Magazine management related parameters.

2.

MACHINE PARAMETERS.

Machine parameters for the magazine.

2.10.2 Magazine data.

STORAGE

Parameters related to magazine.

This parameter shows or hides the magazine data. Each table shows the following machine parameters to configure it.

MAGAZINE n	
SIZE	Size of the tool magazine (number of pockets).
RANDOM	Random magazine.

SIZE

Size of the magazine (number of pockets).

Possible values: From 0 to 1000.

Default value: 20.

Associated variable: (V.)TM.MZSIZE[mz]

Parameter included in the MAGAZINE // STORAGE table.

This parameter indicates the capacity of the magazine without including the ground tools.

RANDOM

Random magazine.

Possible values: Yes / No.

Default value: No.

Associated variable: (V.)TM.MZRANDOM[mz]

Parameter included in the MAGAZINE // STORAGE table.

This parameter indicates whether the tools must always occupy the same position (non-random) or they may occupy any position (random).

2.

MACHINE PARAMETERS.
Machine parameters for the magazine.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.10.3 Tool magazine management.

MANAGEMENT

Magazine management related parameters.

This parameter shows the data to configure the magazine management. Each table shows the following machine parameters to configure it.

MANAGEMENT	
TYPE	Type of magazine.
CYCLIC	Cyclic tool changer.
OPTIMIZE	Tool management.
M6ALONE	Action when executing an M06 without selecting a T.

TYPE

Magazine type.

Possible values: Asynchronous / Synchronous / Turret / Synchronous + 2 Arms / Synchronous + 1 Arm.

Default value: Synchronous.

Associated variable: (V.)TM.MZTYPE[mz]

Parameter included in the MAGAZINE // MANAGEMENT table.

This parameter indicates the type of magazine. See "[7.1 Types of tool magazine.](#)" on page 451.

CYCLIC

Cyclic tool changer.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)TM.MZCYCLIC[mz]

Parameter included in the MAGAZINE // MANAGEMENT table.

A cyclic tool changer requires a tool change command (function M06) after searching a tool and before searching the next one. With a non-cyclic tool changer, it is possible to search for several tools in a row without necessarily having to make the actual tool change (M06 function).

OPTIMIZE

Tool management.

Possible values: Yes / No.

Default value: Yes.

Associated variable: (V.)TM.MZOPTIMIZED[mz]

Parameter included in the MAGAZINE // MANAGEMENT table.

When programming several T's in a row without an M6, this parameter indicates whether all the programmed tools are selected (OPTIMIZE = No) or just the ones involving a tool change (OPTIMIZE = Yes). Optimizing only works when executing a program. In MDI mode, this parameter is ignored and all the blocks are executed.

T2	It looks for the tool if OPTIMIZE = No.
T3 M6	It always looks for the tool. The M6 implies a tool change.
T5	It always looks for the tool. M6 comes next.
M6	

2.

MACHINE PARAMETERS.

Machine parameters for the magazine.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

RESPECTSIZES

In random magazine, look for pockets of the same size.

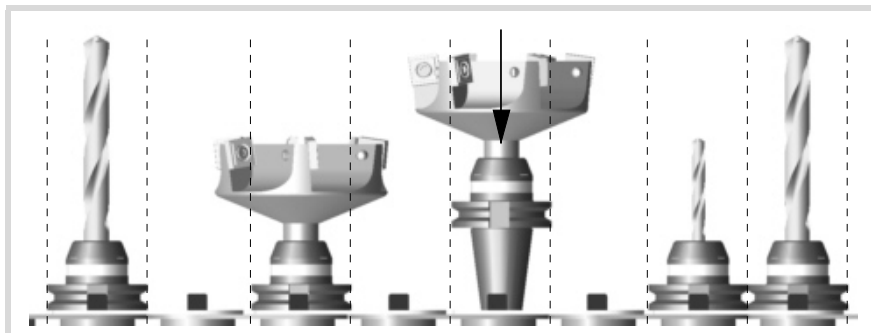
Possible values: Yes / No.

Default value: No.

Associated variable: (V.)TM.RESPECTSIZES

Parameter included in the MAGAZINE // MANAGEMENT table.

Depending on the size of the tool, it may occupy more than one magazine position. In random magazines, this parameter indicates that the tools must always be saved in positions of the same size.

**M6ALONE**

Action when executing an M06 without selecting a tool.

Possible values: Nothing / Issue a warning / Issue an error.

Default value: Show an error.

Associated variable: (V.)TM.MZM6ALONE[mz]

Parameter included in the MAGAZINE // MANAGEMENT table.

The M06 function implies a tool change. This parameter indicates what happens when executing an M06 without having selected a tool first.

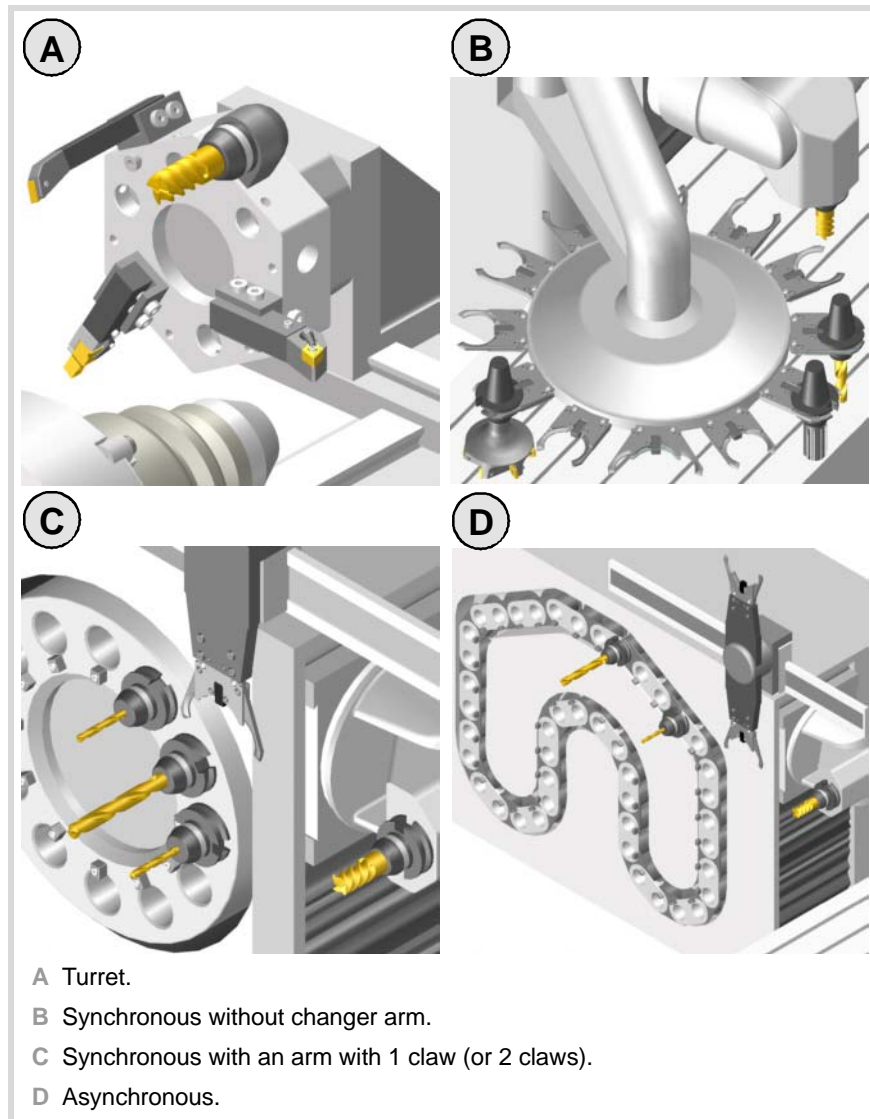
2.

MACHINE PARAMETERS.

Machine parameters for the magazine.

2.10.4 Types of tool magazine.

The CNC can manage the following types of magazine:



Turret type.

It is a typical magazine for lathes. The tool cannot be changed while the part is being machined.

Synchronous magazine without changer arm.

In a synchronous tool changer without arm, the magazine must move up to the spindle to change the tool. The tool cannot be changed while the part is being machined.

The tool change is carried out as follows:

- 1 It ends the movement of the axes.
- 2 The magazine approaches the spindle to grab the tool.
- 3 It selects the new tool and puts it in the spindle.
- 4 The magazine retracts.
- 5 The CNC resumes program execution.

Asynchronous magazines with changer arm (1 or 2 claws).

Synchronous magazines with tool changer arm (1 or 2 holders) have the magazine close to the spindle. The tool cannot be changed while machining the part because the arm would collide.

2.

MACHINE PARAMETERS.
 Machine parameters for the magazine.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

The tool change is carried out as follows: Example with 2 holders:

- 1 The new tool is selected in the magazine.
- 2 It ends the movement of the axes.
- 3 The arm takes a tool on each holder (from the magazine and from the spindle) and swaps them.
- 4 The arm retracts.
- 5 The CNC resumes program execution.

Asynchronous magazines.

An asynchronous magazine is located away from the spindle. Most of the movements may be carried out while machining the part, thus minimizing machining time.

The tool change is carried out as follows:

- 1 While machining, a new tool is selected at the magazine, the changer arm picks it up and brings it closer to the spindle.
- 2 It ends the movement of the axes.
- 3 The other holder picks up the tool that is in the spindle and makes the change.
- 4 The program resumes execution and the changer arm returns to the magazine to leave the tool.

2.

MACHINE PARAMETERS.

Machine parameters for the magazine.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.11 Machine parameters for HMI (Interface).

These parameters are used to define the communication environment (interface) between the operator and the CNC. To validate the changes made to these parameters, the CNC must be reset.

2.11.1 Main window dimensions and resolution.

WINDOW

Main window dimensions and resolution.

This parameter shows the data to configure the main window. The table has the following machine parameters.

WINDOW	
POSX	X coordinate of the top left corner.
POSY	Y coordinate of the top left corner.
RESOLUTION	Window resolution.
WIDTH	Width of the window.
HEIGHT	Height of the window.

POSX

X coordinate of the top left corner.

Default value: 0.

Parameter included in the WINDOW table.

Consult the machine parameter of HMI POSY.

POSY

Y coordinate of the top left corner.

Default value: 0.

Parameter included in the WINDOW table.

These parameters define the X and Y coordinate to position the upper left corner of the interface. The interface can be moved by clicking with the mouse on the icon showing the program status of the active channel. Double click on this icon and the interface is positioned in the upper left corner (POSX=0 POSY=0).



RESOLUTION

Screen resolution.

Possible values: CUSTOM / 10" / 15" / 19" / 21" LANDSCAPE / 21" PORTRAIT.

Default value: 10".

Parameter included in the WINDOW table.

Screen resolution. The CUSTOM option enables the width (WIDTH parameter) and the height (HEIGHT parameter) of the window to be customized. The predefined sizes are set to the following resolutions.

RESOLUTION. (value)	Interface size (pixels).
CUSTOM	This option enables the width (WIDTH parameter) and the height (HEIGHT parameter) of the screen to be customized.
10"	800 × 600.
15"	1024 × 768.

2.

RESOLUTION. (value)	Interface size (pixels).
19"	1280 × 1024.
21" LANDSCAPE Horizontal widescreen.	1300 × 1080. The interface does not cover the entire screen, allowing room for a virtual keyboard.
21" PORTRAIT Vertical widescreen.	1080 × 1400. The interface does not cover the entire screen, allowing room for a virtual keyboard.

WIDTH**Width of the window.**

Possible values: From 800 to 1680 pixels.

Default value: 800 pixels.

Parameter included in the WINDOW table.

Consult the machine parameter of HMI HEIGHT.

HEIGHT**Height of the window.**

Possible values: From 480 to 1050 pixels.

Default value: 600 pixels.

Parameter included in the WINDOW table.

The WIDTH and HEIGHT parameters define the window size. These parameters can only be modified when the RESOLUTION parameter has the value "CUSTOM".

2.11.2 Customizing the softkeys.**HMENUNSOFTKEY****Number of softkeys on the horizontal menu.**

Possible values: From 7 to 12.

Default value: 7.

Number of softkeys on the horizontal menu.

VMENUNSOFTKEY**Number of softkeys on the vertical menu.**

Possible values: From 5 to 12.

Default value: 5.

Number of softkeys on the vertical menu.

SOFTKEYMETRICS**Dimensions of the softkeys.**

Possible values: Standard / Advanced.

Default value: Standard.

This parameter makes it possible to optimize the width of the vertical softkeys and the height of the horizontal softkeys, so that the space they occupy is more proportionate.

Value.	Meaning.
Standard.	The interface maintains a proportional value between the height and width of the softkeys.
Advanced.	The interface optimizes the space occupied by the softkeys. Recommended option for panoramic monitors, where the interface occupies the entire screen.

2.

MACHINE PARAMETERS.
Machine parameters for HMI (Interface).

2.11.3 Interface setting.

HMITYPE

Interface type of the CNC.

Possible values: *Standard / Classic / Advanced.*

Default value: *Standard.*

This parameter sets how to operate with the CNC, through the interface.

Value.	Meaning.
Standard.	Default way to operate at the CNC that may be classic or advanced depending on the CNC model.
Classic.	Hotkey-based method to operate to select the work mode (automatic, jog, editor, etc.). This is the standard way of the 8070.
Advanced.	Operating method based on a main menu to access the various work modes (automatic, jog, editor, etc.) through softkeys. Hotkeys for direct access can also be used.

SFTYPE

Type of tree of softkeys.

Possible values: *Standard / Popup.*

Default value: *Standard.*

Depending on the selected value, the softkey tree appears as follows.

Value.	Meaning.
Standard.	Softkey tree based on menus and submenus, i.e. there are different softkey levels within the same work mode.
Popup.	Softkey tree based on popup menus so there is only one softkey level. There are no softkey submenu, which simplifies the operation.

VMENU

Position of the vertical softkey-menu.

Possible values: *Left / Right.*

Default value: *Right.*

Depending on hardware, the vertical softkeys F8 through F12 appear on the left or on the right of the monitor. This parameter may be used to put the vertical softkey menu on the proper side.

LANGUAGE

Work language.

Default value: *English.*

Select a language among those available.

ENGLISH	SPANISH	ITALIAN	GERMAN
FRENCH	BASQUE	PORTUGUESE	CHINESE
RUSSIAN	CZECK	KOREAN	DUTCH

FFORMAT

Display format for the programmed F.

Default value: *5.2.*

This parameter sets the numerical format (integers and decimals) for displaying the feedrate value. If the format is 0.0, the interface assumes the format defined by means of the FGUIM application.

2.

MACHINE PARAMETERS.

Machine parameters for HMI (Interface).



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

SFORMAT**Display format for the programmed S.**

Default value: 5.1.

This parameter sets the numerical format (integers and decimals) for displaying the spindle speed value. If the format is 0.0, the interface assumes the format defined by means of the FGUIM application.

MMINCHSOFTKEY**Show the softkey for toggling the display format between mm and inches.**

Possible values: Yes / No.

Default value: No.

This parameter indicates whether or not the CNC shows the softkey for toggling between mm and inches. This parameter is linked to the MMInchSoftkey input of the mmc8070.ini file.

2.

MACHINE PARAMETERS.
Machine parameters for HMI (Interface).



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.11.4 Keyboard configuration.

USERKEY

Customizing the user key.

This table shows the parameters to configure the user key. The table has the following machine parameters.

USERKEY	
FUNCTION	Function of the user key.
COMPONENT	Access a component of the CNC.
APPLICATION	Execute a PC application.

FUNCTION

Function of the user key.

Possible values: Windows / Component / Application / CNC OFF / Nothing.

Default value: Nothing.

Parameter included in the USERKEY table.

Depending on the function selected, it is possible to carry out one of the following tasks.

Value.	Meaning.
Windows	Minimize the CNC and show the Windows screen.
Component	Access a component (work mode) without CNC hotkey.
Application	Execute an external application, for example the FGUIM.
CNC OFF	It turns the CNC application off.
Nothing.	Disable the key.

COMPONENT

Access a component of the CNC.

Possible values: Diagnosis / PLC / Machine parameters / DDSSETUP / TUNING

Parameter included in the USERKEY table.

This parameter shows the list of components (work modes) of the CNC that have no preset key on the operator panel. Besides these components, it will also display the components created with the tool FGUIM.

APPLICATION

Execute a PC application.

Parameter included in the USERKEY table.

When setting parameter USERKEY=Application, this parameter allows to select the application. The whole application path must be indicated; for example C:\CNC8070\FAGOR \RELEASE\FGUIM.EXE.

CHANGEKEY

Customizing the change key.

This table shows the parameters to configure the change key. The table has the following machine parameters.

CHANGEKEY	
FUNCTION	Function of the change key.
SYSTEMMODE	Behavior of the system menu.
SYSHMENU	Horizontal system menu.
SYSVMENU	Vertical system menu.

2.

MACHINE PARAMETERS.

Machine parameters for HMI (Interface).

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

FUNCTION**Function of the change key.**

Possible values: Next page / Next channel / Menu.

Default value: Next page.

Parameter included in the CHANGEKEY table.

Depending on the function selected, it is possible to carry out one of the following tasks.

Value.	Meaning.
Next page.	The key selects the next page of the active work mode.
Next Channel.	The key selects the next channel.
Menu.	The key shows the list of channels and pages on the softkey menus.

SYSTEMUMODE**Behavior of the system menu.**

Possible values: Volatile / Fixed.

Default value: Volatile.

Parameter included in the CHANGEKEY table.

This parameter determines when the system menu is disabled.

Value.	Meaning.
Volatile.	The softkey menu is disabled when selecting a menu option or when changing the active component.
Fixed.	The softkey menu remains until the change key is pressed again.

SYSHMENU**Horizontal system-menu.**

Possible values: Disabled / Pages / Channels / Components.

Default value: Disabled.

Parameter included in the CHANGEKEY table.

See parameter SYSHMENU.

SYVMENU**Vertical system-menu.**

Possible values: Disabled / Pages / Channels / Components.

Default value: Disabled.

Parameter included in the CHANGEKEY table.

This parameter sets the options that will appear on each softkey-menu.

Value.	Meaning.
Disabled.	The menu will be disabled.
Screens.	The menu shows the various pages or screens of the active work mode.
Channels.	The menu shows the available channels.
Components.	The menu shows the components or work modes of the CNC.

ESCAPEKEY**Customizing the escape key.**

This table shows the parameters to configure the escape key. The table has the following machine parameters.

ESCAPEKEY	
FUNCTION	Function associated with the escape key.
NPREVIOUS	Maximum number of previous components stored.

2.

MACHINE PARAMETERS.
Machine parameters for HMI (Interface).

FUNCTION

Function associated with the escape key.

Possible values: Previous menu / Previous component / Previous menu-component.

Default value: Previous menu.

Parameter included in the ESCAPEKEY table.

This parameter sets the behavior of the [ESC] key.

Value.	Meaning.
Previous menu.	Every time the [ESC] key is pressed, the CNC goes up on the menu until it reaches the main menu of the component. From then on, the CNC changes to the main menu.
Previous component.	Every time the [ESC] key is pressed, the CNC goes up on the menu until it reaches the main menu of the component. From then on, the CNC toggles between the previous last work modes (parameter NPREVIOUS). The [ESC] key does not change to the main menu. The CNC does not change softkey menu of the previous work modes.
Previous menu and component.	Every time the [ESC] key is pressed, the CNC goes up on the menu until it reaches the main menu of the component. From then on, the CNC toggles between the previous last work modes (parameter NPREVIOUS). The [ESC] key does not change to the main menu. In each component, the [ESC] key first goes up on the menu until it reaches the main menu of the component.

NPREVIOUS

Maximum number of previous components stored.

Possible values: From 1 to 5.

Default value: 1.

Parameter included in the ESCAPEKEY table.

Number of components stored by the CNC to be displayed when pressing the escape key.

2.

MACHINE PARAMETERS.

Machine parameters for HMI (Interface).



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.11.5 Simulated jog keyboard.

SIMJOGPANEL **Simulated JOG panel.**

Possible values: Yes / No.

Default value: No.

This parameter indicates whether the simulated panel is available or not. To select or deselect it, press [CTRL][J].

A simulated jog keyboard is a window that lays over the CNC screen and may be used to simulated the jog keys and the keys for accessing the work modes. It may have to be used when working with telediagnosis (remote control of the CNC).

2.11.6 CNC shut down.

WINEXIT **Exit Windows when closing the CNC.**

Possible values: Yes / No.

Default value: No (Windows is not closed).

This parameter indicates whether Windows is closed or not when exiting the CNC using [ALT][F4].

2.

MACHINE PARAMETERS.
Machine parameters for HMI (Interface).



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.11.7 Graphics configuration.

GRAPHTYPE

List of graphics in each channel.

On the lathe model, this parameter shows the table to define the configuration of the graphics in each channel.

GRAPHTYPE	
GRAPHTYPECH n	Type of graphics in each channel.

GRAPHTYPECH n

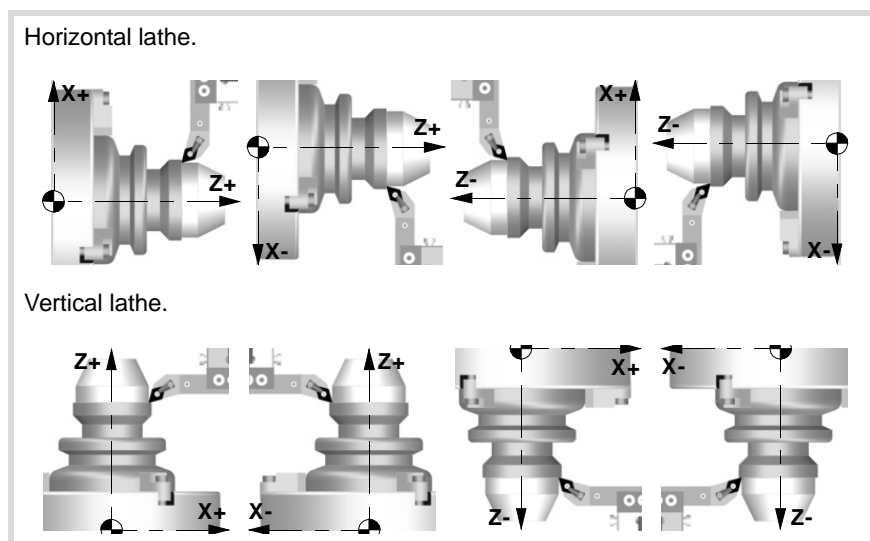
Type of graphics in each channel.

Possible values: Horizontal or vertical lathe; X+ Z+ / X- Z+ / X+ Z- / X- Z-.

Default value: Horizontal X+ Z+.

Parameter included in the GRAPHTYPE table.

In the lathe model, this parameter sets the configuration of the graphics of the channel.



It is also possible to display the graphics of a dual-turret lathe (TT lathe). To do that, use the FGUIM application to change the Channel1 and Channel2 properties of the graphics window to display the execution of both channels on a single graphics.

DIAGPSW
(Reserved)

2.

MACHINE PARAMETERS.

Machine parameters for HMI (Interface).

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.12 OEM machine parameters.

2.12.1 Reading drive (Sercos or Mechatrolink) variables.

DRIVEVAR

Drive variables table.

Parameter DRIVEVAR may be used to define the list of DRV variables that grant access to the variables and/or parameters of the digital drives via part-program, MDI/MDA mode, PLC or interface.

- 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.

The table has the following machine parameters.

DRIVEVAR	
SIZE	Number of elements of the drive variables table
DATA	Variables of the drives.

Characteristics and limitations of Mechatrolink devices.

The access to Mechatrolink devices using DRV variables has the following characteristics and limitations.

- Access to variables and/or parameters of the devices is only available for the Mlink-II mode with 32 bytes.
- The variables can only be read, whereas the parameters may be read or written.
- Access to the variables of the drive is asynchronous (managed through the service channel) whereas access to the parameters is synchronous (managed through the cyclic channel).
- Either the parameters or the variables may be defined for the same axis, not both.
- If variables have been defined, it is not possible to access the DDSSetup or the setup assistance.
- The maximum number of variables per axis or spindle is 2. There is no limit for parameters per axis.

SIZE

Number of variables to be consulted at the drive.

Possible values: From 0 to 99.

Default value: 0.

Associated variable: (V.)DRV.SIZE

Parameter included in the DRIVEVAR table.

This parameter sets the number of variables to be consulted at the drive.

DATA

Variables of the drives.

This parameter shows the table to define the list of DRV variables.

For Sercos drives, the following fields must be defined for each DRV variable.

Field.	Meaning.
MNEMONIC	Mnemonic assigned to the DRV variable. The access to the variable from the CNC will be as follows: (V.)DRV.{mnemonic}.{axis} (V.)DRV.{mnemonic}.{spindle}
AXIS	Name of the axis or spindle being accessed by the variable; defining it as "*" means all axes and spindles.

2.

MACHINE PARAMETERS.
OEM machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Field.	Meaning.
ID	ID Sercos identifier of the variable to be consulted at the drive.
TYPE	Access type, synchronous or asynchronous. Variables of synchronous access are managed through the cyclic channel. Variables of asynchronous access are managed through the service channel, even if the default option at the drive is the opposite. Not all the variables may be defined with synchronous access; only the ones that can use the Sercos cyclic channel. Refer to the drive manual. If possible, we recommend to define the variables with synchronous access because the read-write requests of these variables through the bus are faster and reduces traffic through the bus.
MODE	Access mode. These variables may be read-only or read and written.

For Mechatrolink devices, the following fields must be defined for each DRV variable.

Field.	Meaning.
MNEMONIC	Mnemonic assigned to the DRV variable. The access to the variable from the CNC will be as follows: (V.)DRV.{mnemonic} .{axis} (V.)DRV.{mnemonic} .{spindle}
AXIS	Name of the axis or spindle being accessed by the variable; defining it as "*" means all axes and spindles.
ID	Identifier of the parameter (0000-FFFF) or of the variable (0-F) to be consulted at the device.
TYPE	Type of consultation; variable or parameter.
MODE	Access mode. The variables of the drive can only be read, whereas the parameters may be read or written.

2.

MACHINE PARAMETERS.
OEM machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

2.12.2 Generic OEM parameters.

The values of the table may be modified at any time. The new values are assumed immediately without having to validate the parameters. This means that the values and permissions of the table may be modified while executing a program.

The simulation environment has a copy of this table. On CNC power-up, the values of the parameters of the real table are copied into the simulation table and from there on, they become different in the writing of the variables of both tables.

In the simulation table, only the parameter values may be modified, not the rest of the permissions. The values of the simulation table can only be read or modified through their variable.

MTBPAR OEM-parameter table.

This table offers 1000 generic parameters that the OEM can use as machine parameters. The table has the following machine parameters.

MTBPAR	
SIZE	Number of OEM parameters.
DATA	List of OEM parameters.

SIZE Number of OEM parameters.

Possible values: From 0 to 1000.

Default value: 0.

Associated variable: (V.)MTB.SIZE

This parameter sets the number of OEM parameters that will be used.

DATA List of OEM parameters.

This parameter shows the table of OEM parameters. The following fields must be defined for each parameter.

DATA	
VALUE	Value of the parameter.
MODE	Mode for accessing the parameter from its variable.
INCHES	The parameter is affected by the change of units.
COMMENT	OEM parameter description.

VALUE Parameter value.

Possible values: Within ± 99999.9999 .

Default value: 0.

If the parameter is affected by the change of units (INCHES field), the value is entered in the table in the units selected by machine parameter INCHES.

Each parameter has its own variables for reading or modifying (if it has a write permission) their value from the part-program, PLC or interface. The access to these parameters using variables will be as follows.

(V.)MTB.P[i] Value of the OEM parameter [i].

(V.)MTB.PF[i] Value of the OEM parameter [i]. Value per 10000.

Mnemoni.	Value.	V.MTB.P[i]	V.MTB.PF[i]
P0	7	V.MTB.P[0] = 7	V.MTB.PF[0] = 70000
P8	12,5	V.MTB.P[8] = 12,5	V.MTB.PF[8] = 125000

2.

MACHINE PARAMETERS.
OEM machine parameters.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

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.

MODE

Mode for accessing the parameter from its variable.

Possible values: Read / Write.

Default value: Read.

The access to the parameters from the variables may be read-only or read-write. Both reading and writing interrupt block preparation.

If a read-only access has been defined, the parameter cannot be modified from its variable. If a write access has been defined, the parameter can be modified through its variable.

The value may always be written directly into the table, regardless of the value assigned to this field.

INCHES

The parameter is affected by the change of units.

Possible values: Yes / No.

Default value: Yes.

This field indicates whether the value of the parameter is affected by the change of units, mm or inches. For example, when the parameter represents a coordinate.

COMMENT

OEM parameter description.

This field offers the possibility to associate a short description with the parameter. This field is for information only; it is not used by the CNC.

The comments are saved in the file MTBComments.txt and it is possible to have one file per language. These files are saved in the folder C:\CNC8070 \MTB \data \Lang".

2.

MACHINE PARAMETERS.
OEM machine parameters.

2.12.3 Cam editor.

The cam editor is an graphic assistance element for designing cams. The user must make sure that the design is coherent with the required specifications.

CAMTABLE

Table of electronic cams.

This table shows the parameters to define the electronic cams. The table has the following machine parameters.

CAMTABLE	
SIZE	Number of electronic cams.
DATA	List of cams.
CAM n	Access the cam editor.

SIZE

Number of electronic cams.

Possible values: From 0 to 16.

Default value: 0.

Parameter included in the CAMTABLE table.

This parameter sets the number of electronic cams of the system.

DATA

List of electronic cams.

This parameter shows the list of available cams.

CAM n

Electronic-cam editor.

Cam editor with friendly assistance to analyze the behavior of the cam projected through graphically assisted data entry for speed, acceleration and jerk.



This function has a specific manual.

Refer to the documentation included in the CD-ROM that comes with the product for further information on the requirements and operation of the electronic cam.

2.

MACHINE PARAMETERS.
OEM machine parameters.

The PLC program may be either edited at the front panel or copied from a peripheral or PC. The PLC program has a modular structure and may combine files in "C" language, contact (ladder) language or mnemonic language.

For the program to be executed, its object file (executable) must be generated. On power-up, the CNC executes the executable PLC program stored in memory; if there isn't one, the CNC will show the relevant error code.

Data exchange takes place in automatic mode between the CNC and the PLC. The following is possible from the PLC:

- Control physical inputs and outputs (remote modules).
- Consult and/or modify CNC-PLC exchange variables.
- Consult and/or modify the internal CNC variables.
- Display messages or errors at the CNC.

The following is possible from the CNC:

- Transfer auxiliary functions M, H and S.
- Access PLC resources from any part-program.

Abbreviations used in this chapter.

(=0)	Low logic level.
(=1)	High logic level.
(g.m.p.)	General machine parameter.

3.1 PLC program.

The PLC program may combine several files in mnemonic language (extension: ".plc"), several files in "C" language (extension: ".c") and a file in contact (ladder) language (extension: ".ld"). All the files making up the PLC program must be in the folder \MTB \PLC \PROJECT.

It is recommended to use the mnemonic language files or the contact (ladder) language files as the main PLC program and the "C" language files for auxiliary tasks (for example, temperature compensation).

Subroutines in the PLC program.

The subroutines of the program in mnemonic or contact (ladder) language must be defined outside of the modules, for example at the end of the program after the END instruction.

The subroutines of the "C" language file must be defined as external at the beginning of the mnemonic language program (extension: ".plc") or contact (ladder) language program (extension ".ld"). The name of the subroutine must be written in capital letters in both files.

"C" language programming offers a math library (trigonometric, logarithmic, etc.) that allows performing operations with tables, arrays, float type variables, etc.

PLC program with mnemonic language file.

```
Mnemonic.plc
PRG
() = MOV 1234 R201 = MOV 2345 R202
() = CAL ADD
...
END

SUB SUMA
() = ADS R201 R202 R203
END
```

PLC program with a mnemonic language file and a "C" language file.

```
Mnemonic.plc
EXTERN SUMA
PRG
() = MOV 1234 R201 = MOV 2345 R202
() = CAL ADD
...
END

Languagec.c
#include "plclib.h"
void ADD(void)
(
R203=R201+R202
)
```

3.

INTRODUCTION TO THE PLC.
PLC program.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

3.2 Modular structure of the PLC program.

The PLC program may comprise the following modules. Every module must begin with the directive instruction that defines it (CY1, PRG, PE) and end with the directing instruction END.

- First Cycle module (CY1).
- Main module (PRG).
- Periodic module (PE).

First Cycle module.

It is an optional module. The module begins with the directing instruction CY1 and ends with the directive instruction END.

The PLC executes the first cycle module only once when starting up the PLC program. This module is used to initialize the various resources and variables before executing the main program.

Main module.

The main module begins with the directing instruction PRG and ends with the directive instruction END.

The PLC executes the main module cyclically with the frequency set by parameter PRGFREQ. This module is in charge of analyzing and modifying the CNC inputs, outputs and variables. The PRG execution takes about 100 μ s.

Parameter PRGFREQ indicates how often (every how many CNC cycles) the PLC executes a full cycle of the main module. Hence, with a sampling period of 4 ms (LOOPTIME=4) and a frequency of 2 cycles (PRGFREQ=2), the PLC executes the main module every $4 \times 2 = 8$ ms.

Periodic module.

It is an optional module. The periodic module begins with the directing instruction PE and ends with the directive instruction END.

The PLC executes the periodic module cyclically with the frequency set by the directive instruction PE, between 1 and 2147483647 ms and never less than the loop time (parameter LOOPTIME). This module could be used to execute tasks that do not need to be executed at every PLC cycle.

For example, a task to be performed every 30 seconds could be defined in a periodic module using the instruction (PE 30000).

3.

INTRODUCTION TO THE PLC.
Modular structure of the PLC program.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

3.3 PLC program execution.

Main module (PRG)

The main module is processed as follows:

- 1 The PLC assigns the current value of the physical inputs (remote modules) to the I resources.
- 2 The PLC assumes the current values of the internal CNC variables (CNCREADY, START, FHOUT,).
- 3 The PLC executes the main program (PRG).
- 4 The PLC updates the internal CNC variables (EMERGEN, STOP, FEEDHOL, ...) with the current values of the associated PLC resources.
- 5 The PLC assigns the current value of the PLC's "O" resources to the physical outputs (remote modules).
- 6 The PLC concludes the cycle and is ready for the next scan.

Periodic module (PE)

The periodic module is processed as follows:

- 1 The PLC takes into account the current values of the physical inputs (remote modules) at the beginning of the module.
- 2 The PLC executes the periodic module.
- 3 The PLC assigns the current value of the PLC's "O" resources to the physical outputs (remote modules).
- 4 The PLC concludes the execution of the periodic Module.

3.

INTRODUCTION TO THE PLC.
PLC program execution.

3.4 PLC resources.

The PLC has the following resources.

- Inputs (I1-I1024) and outputs (O1-O1024).
- Local inputs (LI1-LI16) and local outputs (LO1-LO8).
- Marks (M1-M8192).
- Messages (MSG1-MSG1024).
- Errors (ERR1-ERR1024)
- Clocks (CLK).
- Registers (R1-R1024).
- Timers (T1-T512).
- Counters (C1-C256).
- Registers and marks for CNC-PLC communication.

The MSG, ERR, CLK and T resources are initialized (=0) when starting up the PLC. M, C and R resources maintain their value between CNC start-ups.

Inputs (I1-I1024) and outputs (O1-O1024).

The inputs are elements that provide information to the PLC on the signals they receive from the outside world. They are represented by the letter "I" followed by an input number between "I1" and "I1024".

The outputs are elements that let the PLC activate or deactivate the various devices of the electrical cabinet. They are represented by the letter "O" followed by an output number between "O1" and "O1024".

Numbering of the physical inputs and outputs.

There are two different ways to number the inputs and outputs. Depending on the order of the remote modules or via machine parameters. See ["3.4.1 Numbering of the physical inputs and outputs."](#) on page 326.

Local inputs (LI1-LI16) and local outputs (LO1-LO8).

Central unit ICU and MCU have a set of 8 local digital inputs and 8 local digital signals that may be configured both as input and output. They are called local inputs and outputs because they are located in the central unit, not in the remote modules. The local inputs are represented with the letter LI followed by the input number, from LI1 to LI16. The local outputs are represented with the letter LO followed by the output number, from LO1 to LO8.

Local digital I/O are ignored when setting the number of remote I/O or when saving the configuration of the CAN bus. Local I/O will always be referred to in the same way regardless of the system configuration.

Local I/O will be refreshed with the same frequency as the remote I/O and also from the PE module.

Marks (M1-M8192).

They are elements capable of memorizing in a bit (like an internal relay) the value set by the user. If the mark is (=0), it will be referred to as being set low. If the mark is (=1), it will be referred to as being set high.

They are represented by the letter "M" followed by a mark number between M1 and M8192.

3.

Messages (MSG1-MSG1024).

When activated (=1), they display a message on the CNC screen. The texts associated with the messages must be previously defined in the message and error table of the PLC.

They are represented by the letters "MSG" followed by a message number between MSG1 and MSG1024.

All of them are initialized (=0) when starting up the PLC.

Errors (ERR1-ERR1024)

When activated (=1), they cause an error, they show an error message and interrupt the execution of the CNC. The texts associated with the errors must be previously defined in the message and error table of the PLC. PLC errors may be configured so they activate the emergency signal of the PLC (_EMERGEN).

They are represented by the word "ERR" followed by an error number between ERR1 and ERR1024.

All of them are initialized (=0) when starting up the PLC.

The errors do not activate the alarm of the CNC, (_ALARM) signal.

Clocks (CLK).

They are internal clocks with different time periods to be used in the PLC program.

They are represented by the word "CLK" followed by a clock number .

All of them are initialized (=0) when starting up the PLC.

These are the clock marks available. Their half-period (or after how long their state 0/1 changes) is shown next to them.

Clock.	Period.	Clock.	Period.	Clock.	Period.
CLK1	1 ms	CLK100	100 ms	CLK1000	1 s
CLK2	2 ms	CLK200	200 ms	CLK2000	2 s
CLK4	4 ms	CLK400	400 ms	CLK4000	4 s
CLK8	8 ms	CLK800	800 ms	CLK8000	8 s
CLK16	16 ms	CLK1600	1.6 s	CLK16000	16 s
CLK32	32 ms	CLK3200	3.2 s	CLK32000	32 s
CLK64	64 ms	CLK6400	6.4 s	CLK64000	64 s
CLK128	128 ms	CLK12800	12.8 s	CLK128000	128 s

Registers (R1-R1024).

They are elements that can store a numeric variable in 32 bits. The value stored in each register is considered as a signed integer between $\pm 2.147.483.647$. It can be processed as decimal or hexadecimal number (preceded by the "\$" sign). For example:

156 (Decimal)

\$9C (Hexadecimal)

They are represented by the letter "R" followed by a register number between R1 and R1024. It is also possible to refer to a register bit with the letter "B" and a bit number (0/31). The PLC takes bit 0 as the least significant bit and as bit 31 as the most significant bit.

B7R155 Refers to bit 7 of register 155.

(REF: 1709)

3.

INTRODUCTION TO THE PLC.
PLC resources.



FAGOR AUTOMATION

CNC 8070

Timers (T1-T512).

They are elements capable of maintaining their output at the same logic level (state) for a preset time period (time constant) after which their output changes states.

They are represented by the letter "T" followed by a time number between T1 and T512.

All timers are initialized (=0) when starting up the PLC.

See "[3.5 Operation of a timer.](#)" on page 328.

Counters (C1-C256).

They are elements capable of counting up or down a preset amount of events.

They are represented by the letter "C" followed by a counter number between C1 and C256.

See "[3.6 Operation of a counter.](#)" on page 338.

Registers and marks for CNC-PLC communication.

The PLC has access to a some internal CNC data.

The PLC can consult and/or modify certain CNC signals (marks and registers).

- Consultation signals: CNCREADY, START, FHOUT, ...
- Modifiable signals: _EMERGEN, _STOP, _FEEDHOL, ...

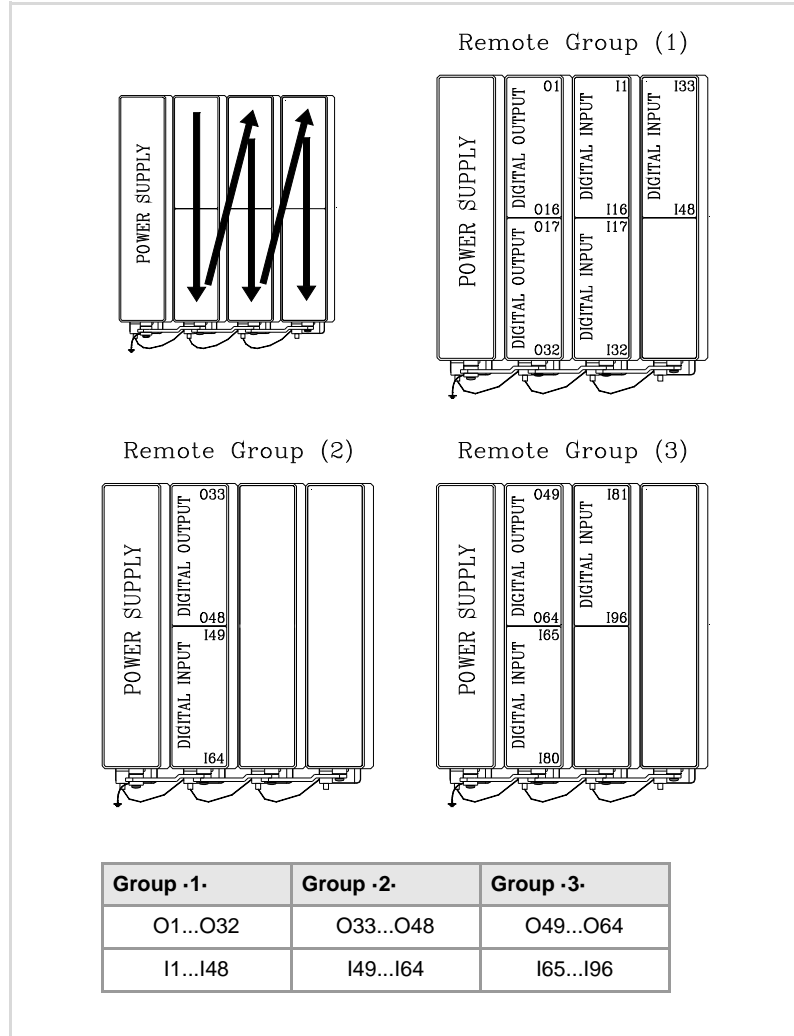
See chapter "[6 Logic CNC inputs and outputs.](#)".

3.4.1 Numbering of the physical inputs and outputs.

The numbering of the I/O modules may be set using the machine parameters. If these parameters are not defined, the CNC numbers the modules automatically according to the order of the remote modules.

Numbering according to the order of the remote groups.

They are numbered following the order of the remote groups (rotary switch of the Power Supply element). Within each group, they are ordered from top to bottom and from left to right.



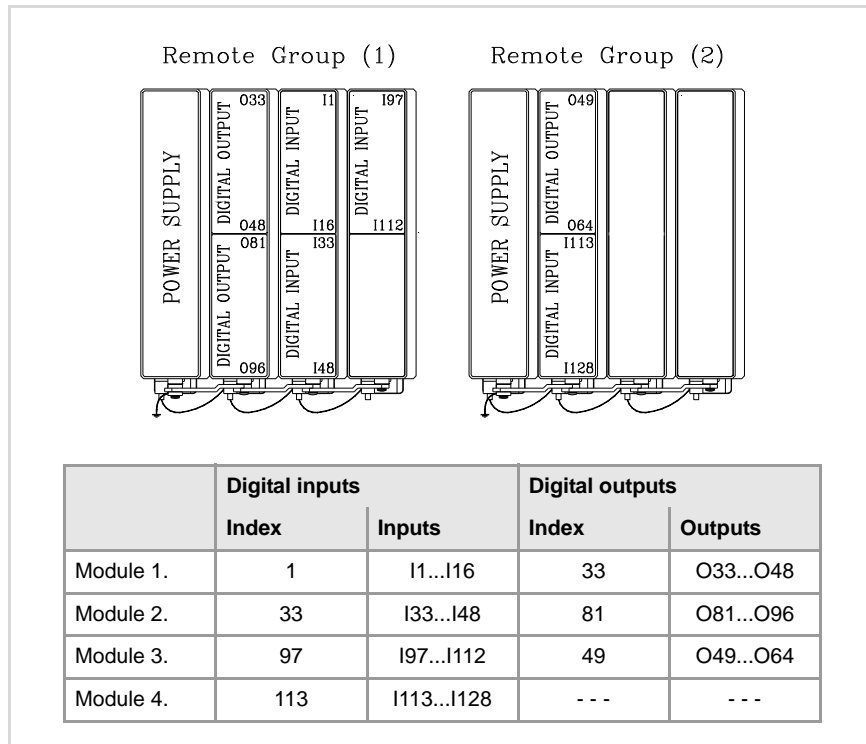
Numbering by machine parameters.

When the numbering is set by machine parameters, each module is assigned a base index and the inputs or outputs of that module are numbered after it. The values of the base index must be multiple of 16, plus 1 (i.e. 1, 17, 33, etc.). The base indexes may follow any order, they do not have to be sequential.

When inserting a new module, the first modules will be assigned the numbering of the table and the last one will be assigned the next valid base index after the highest one assigned until then.

3.

INTRODUCTION TO THE PLC.
PLC resources.



3.

INTRODUCTION TO THE PLC.
PLC resources.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

3.5 Operation of a timer.

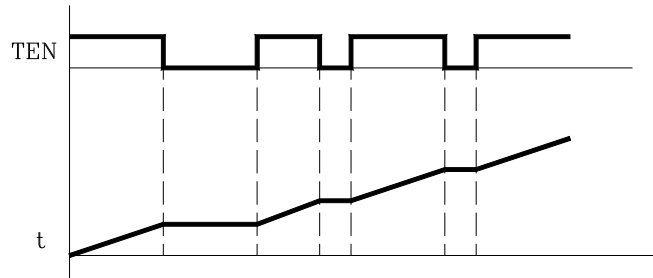
All the timers have a status output "T" and the inputs: TEN, TRS, TG1, TG2, TG3 and TG4. It is also possible to check, at any time, the elapsed time t since the timer was triggered.

When starting up the PLC, all the timers are initialized by setting their status "T" and their time count to "0".

(TEN) Enable input.

It can be used to interrupt and resume the timing. It is referred to by the words "TEN" followed by the timer number. For example TEN 1, TEN 25, TEN 102, etc.

Once the timer is triggered, if input TEN is set low (=0), the PLC stops timing; input TEN must be set back high (=1) to resume timing.



By default, every time a timer is triggered, the PLC sets this input high (=1).

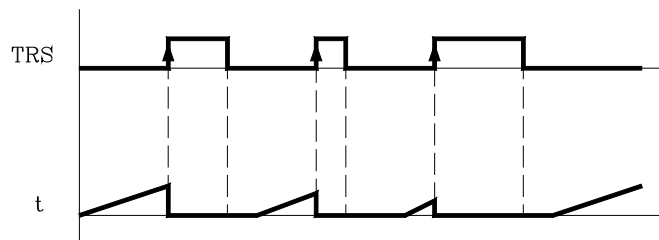
I2 = TEN 10

Input I2 controls the Enable input of timer T10.

(TRS) Reset input.

It is used to initialize the timer by setting its status "T" and its timing to "0". It is referred to by the word "TRS" followed by the timer number, for example TRS 1, TRS 25, TRS 102, etc.

Once the timer has been activated, at an up-flank (0 to 1 transition) of the TRS input, the PLC resets the timer. The timer is deactivated and its trigger input must be activated to turn the timer back on.



By default and every time a timer is triggered, the PLC sets this input high (=0).

I3 = TRS 10

Input I3 controls the Reset input of timer T10.

(TG1, TG2, TG3, TG4) Trigger inputs.

They are used to activate the different work modes of the timer.

- TG1 triggers the mono-stable mode.
- TG2 triggers the delayed activation mode.
- TG3 triggers the delayed deactivation mode.
- TG4 triggers the signal limiting signal.

They are referred to by the words TG1, TG2, TG3, TG4 followed by a timer number and the initial timing value (time constant). For example TG1 1 100, TG2 25 224, TG3 102 0, etc.

3.

INTRODUCTION TO THE PLC.
Operation of a timer.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Set the time constant.

The time constant is defined with a numeric value or with an internal value of a register R. Its value must be within the 0 to 4294967295 ms range, which is the equivalent of 1193 hours (almost 50 days).

TG1 20 100

Triggers timer T20 in Mono-stable mode (TG1) with a time constant of 100 ms.

TG2 22 R200

Triggers timer T22 in delayed activation mode (TG2) with the time constant stored in register R200 in ms.

Activating the timer.

The timer is activated according to the selected input number at an up-flank (0 to 1 transition) or at a down-flank (1 to 0 transition).

Later on, this same section shows how to operate in each of these modes.

(T) Status output.

It indicates the logic state of the timer. It is referred to by the letter "T" followed by the timer number. For example: T1, T25, T102, etc.

As the timer's logic state depends on the selected work mode (TG1, TG2, TG3 and TG4) it will be explained later on.

(T) Elapsed time.

It indicates the time elapsed at the timer since it was triggered. It is referred to by the letter "T" followed by the timer number. It is represented by T123 which is the same as the status output, but is used in different types of instructions.

In binary instructions, it refers to the logic state of the timer.

T123 = M100

Assigns the state (1/0) of T123 to M100.

In arithmetic and comparison instructions, it refers to the elapsed time.

I2 = MOV T123 R200

Transfers the time elapsed at T123 to register R200.

CPS T123 GT 1000 = M100

Compares the time elapsed at T123 is greater than 1000. If so, it activates mark M100.

3.

INTRODUCTION TO THE PLC.

Operation of a timer.

FAGOR 

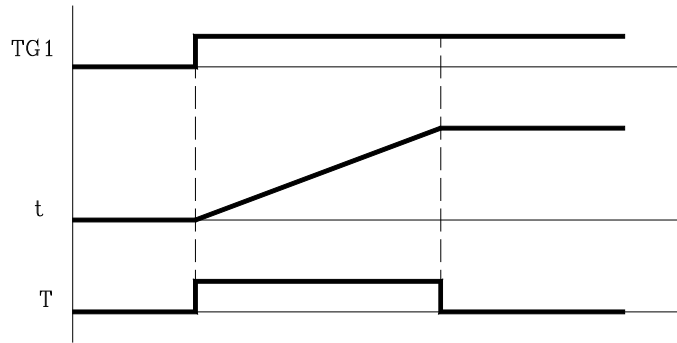
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

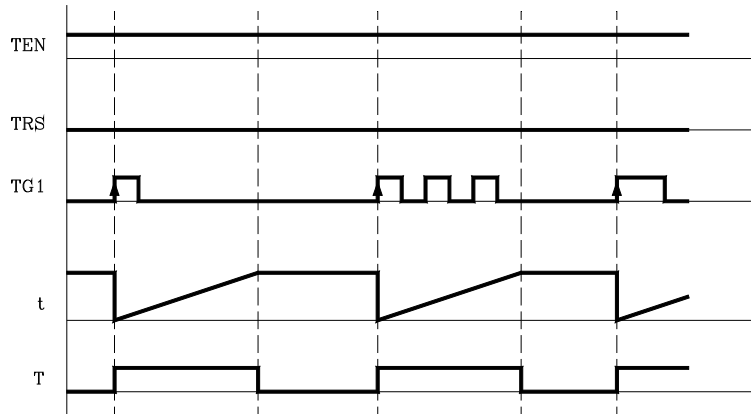
3.5.1 Monostable mode. TG1 input.

In this operating mode, the status of the timer is maintained high ($T=1$) from when the TG1 input is activated until the indicated time period (constant) has elapsed.



With $TEN=1$ and $TRS=0$, the timer is activated with an up-flank at trigger input TG1. At that moment, the timer status output ($T=1$) and the timing starts from $\cdot 0$.

Once the time period indicated by the time constant has elapsed, the timing is over. The status output (T) changes ($T=0$) and the elapsed time t is maintained.

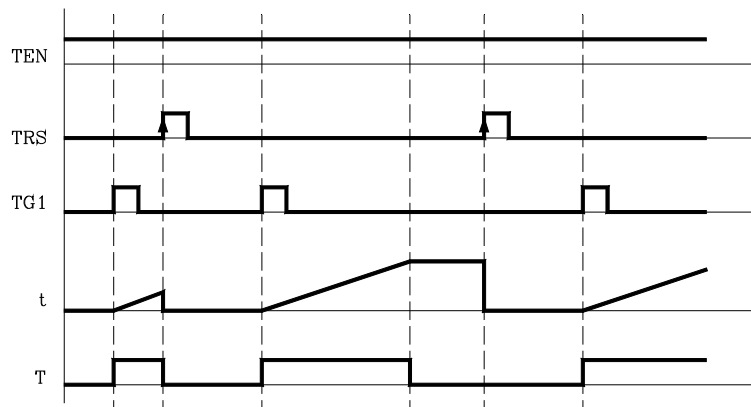


Any changes at the TG1 input (up or down-flank) while timing, has no effect.

Once the timing is over, an up-flank at trigger input TG1 is required to reactivate the timer.

Operation of the TRS input in this mode.

If an up-flank occurs at the TRS input while timing or after it, the PLC resets the timer setting its status output low ($T=0$) and resetting its timing ($t=0$).



Since the timer is reset, its trigger input must be activated again to turn it back on.

3.

INTRODUCTION TO THE PLC.
Operation of a timer.



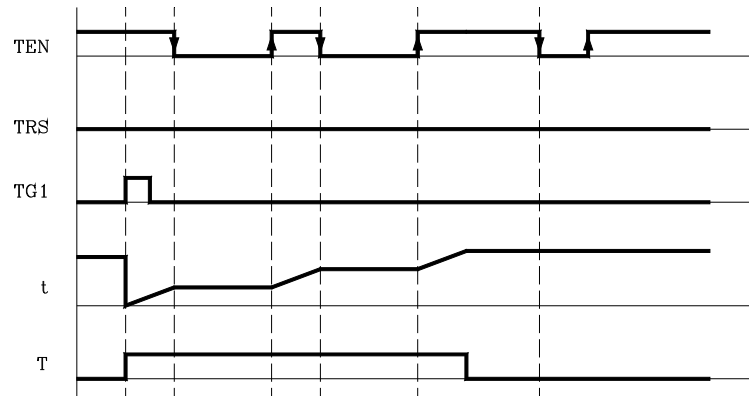
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Operation of the TEN input in this mode.

If once the timer has been activated, $TEN = 0$, the PLC interrupts the timing and TEN must be set to "1" to resume timing.

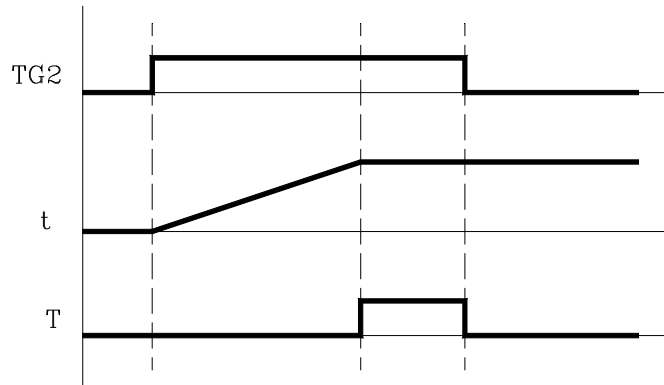


3.

INTRODUCTION TO THE PLC.
Operation of a timer.

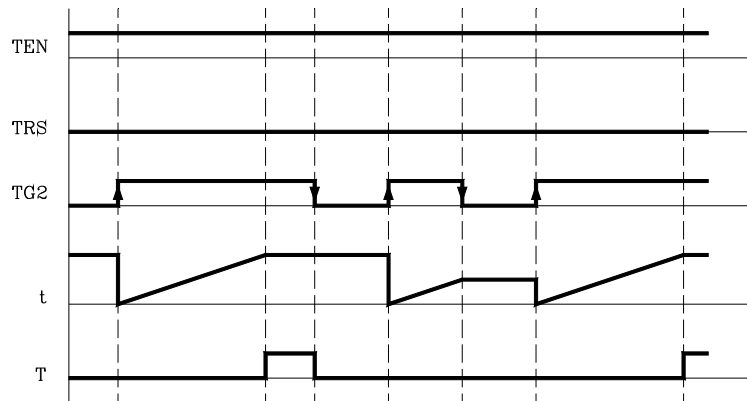
3.5.2 Delayed activation mode. TG2 input.

This mode applies a delay between the activation of the trigger input TG2 and that of the timer status output "T". The time delay is set by the time constant.



With T_{EN}=1 and TR_S=0, the timer is activated with an up-flank at the TG2 input. At that instant, the timing "t" begins from "0".

Once the time indicated with the time constant has elapsed, the timing is over, it activates the timer status output (T=1) which remains high that until a down-flank at trigger input TG2 occurs.

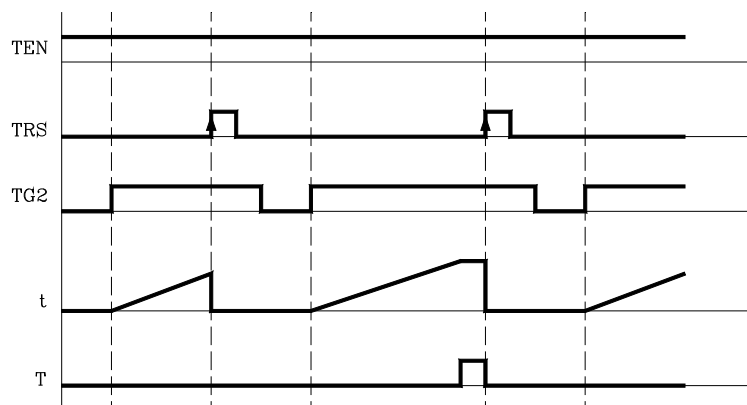


The elapsed time (t) is maintained until a new up-flank occurs at trigger input TG2.

If the down-flank at input TG2 occurs before the indicated time has elapsed, the PLC stops timing and it keeps the "t" value it has at the time.

Operation of the TRS input in this mode.

If an up-flank occurs at the TRS input while timing or after it, the PLC resets the timer setting its status output low (T=0) and resetting its timing (t=0).



Since the timer is reset, its trigger input must be activated again to turn it back on.

3.

INTRODUCTION TO THE PLC.
Operation of a timer.



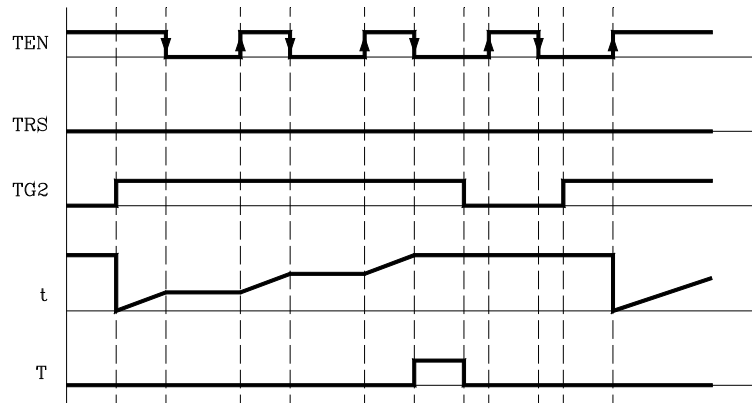
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Operation of the TEN input in this mode.

If once the timer has been activated, $TEN = 0$, the PLC interrupts the timing and TEN must be set to "1" to resume timing.

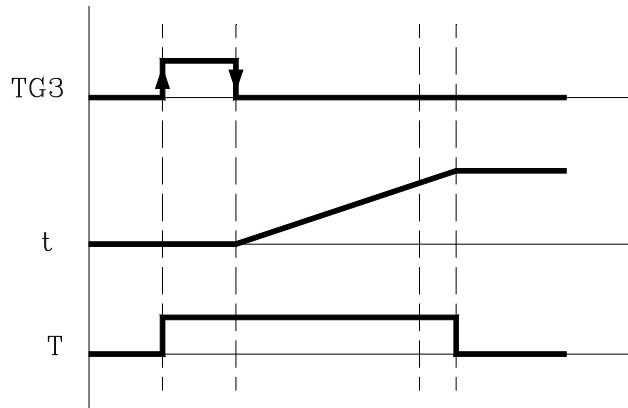


3.

INTRODUCTION TO THE PLC.
Operation of a timer.

3.5.3 Delayed deactivation mode. TG3 input.

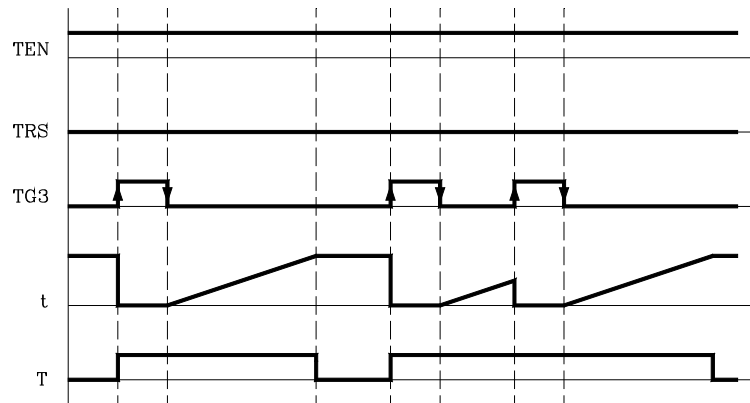
This operating mode is used to apply a delay between the deactivation of trigger input TG3 and that of the "T" output of the timer. The time delay is set by the time constant.



With T_{EN}=1 and T_{RS}=0, the timer is activated with an up-flank at the TG3 input. At that instant, the status output of the timer goes high (T=1).

The timer waits for a down-flank at input TG3 to start the "t" timing from "0".

Once the time indicated by the time constant has elapsed, the timing stops and the status output of the timer goes low (T=0).



The elapsed time "t" is maintained until a new up-flank occurs at input TG3.

If the up-flank at input TG3 takes place before the indicated time has elapsed, the PLC takes it as a new trigger and sets its status output high (T=1) and starts timing again from "0".

3.

INTRODUCTION TO THE PLC.
Operation of a timer.



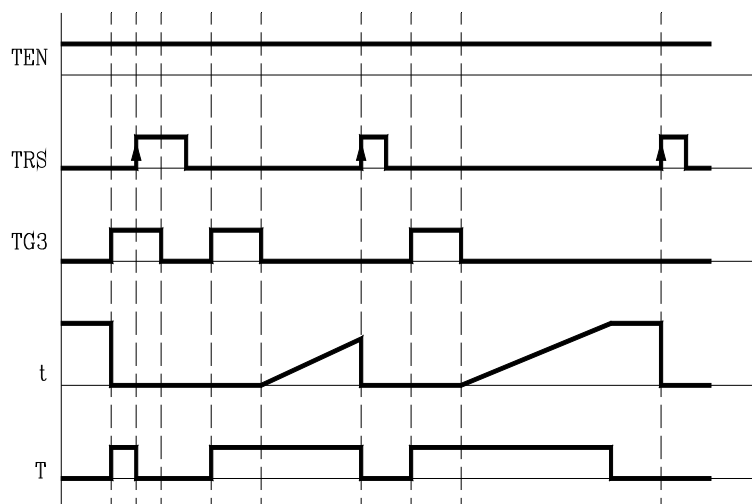
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Operation of the TRS input in this mode.

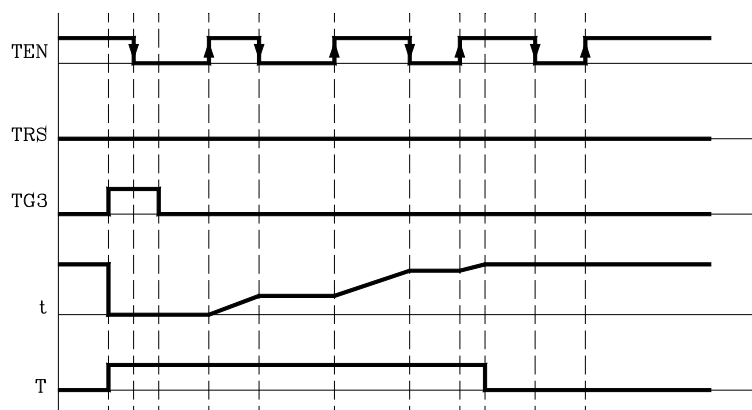
If an up-flank occurs at the TRS input while timing or after it, the PLC resets the timer setting its status output low ($T=0$) and resetting its timing ($t=0$).



Since the timer is reset, its trigger input must be activated again to turn it back on.

Operation of the TEN input in this mode.

If once the timer has been activated, $TEN = 0$, the PLC interrupts the timing and TEN must be set to "1" to resume timing.



3.

INTRODUCTION TO THE PLC.
Operation of a timer.



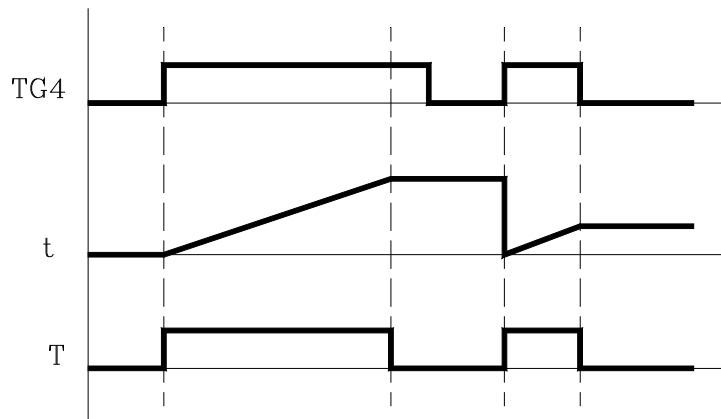
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

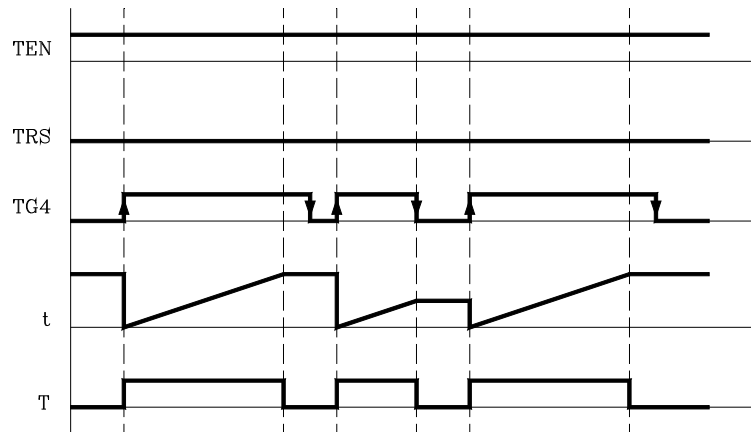
3.5.4 Signal limiting mode. TG4 Input.

In this operating mode, the timer status is kept high ($T=1$) from the moment the TG4 input is activated until the time indicated by the time constant has elapsed or a down-flank occurs at input TG4.



With $TEN=1$ and $TRS=0$, the timer is activated with an up-flank at input TG4. At that moment, the timer status output ($T=1$) changes states ($T=1$) and the timing starts from "0".

Once the time indicated by the time constant has elapsed, the timing stops and the status output of the timer goes low ($T=0$).



The elapsed time (t) is maintained until a new up-flank takes place at input TG4.

If a down-flank occurs at trigger input TG4 before the time indicated by the time constant has elapsed, the PLC stops the timing, brings the status output low ($T=0$) and it keeps the current timing value (t).

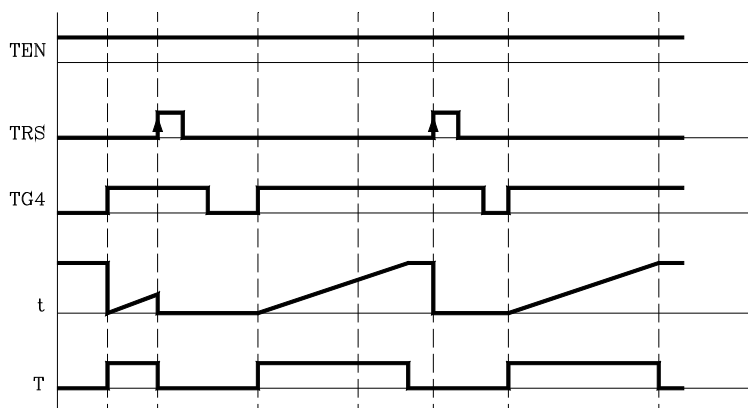
To trigger the timer again, a new up-flank is required at input TG4.

3.

INTRODUCTION TO THE PLC.
Operation of a timer.

Operation of the TRS input in this mode.

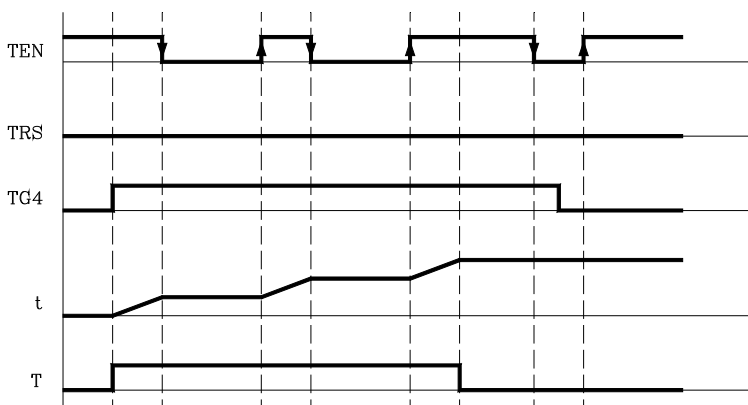
If an up-flank occurs at the TRS input while timing or after it, the PLC resets the timer setting its status output low ($T=0$) and resetting its timing ($t=0$).



Since the timer is reset, its trigger input must be activated again to turn it back on.

Operation of the TEN input in this mode.

If once the timer has been activated, $TEN = 0$, the PLC interrupts the timing and TEN must be set to "1" to resume timing.



3.

INTRODUCTION TO THE PLC.
Operation of a timer.

3.6 Operation of a counter.

All the counters have a status output "C" and the inputs: CUP, CDW, CEN and CPR. Its internal count can also be checked at any time.

The counter's count is stored in a 32-bit variable. Consequently, its value will be in the ± 2147483647 range.

(CUP) Count-up input.

Every time an up-flank occurs at this input, the internal count of the counter increases one unit.

It is referred to by the letters CUP followed by the counter number, for example: CUP 1, CUP 25, CUP 102, etc.

I2 = CUP 10

Every time an up-flank occurs at I2, the count of the C10 counter increases one unit.

(CDW) Countdown input.

Every time an up-flank occurs at this input, the internal count of the counter decreases one unit.

It is referred to by the letters CDW followed by the counter number, for example CDW 1, CDW 25, CDW 102, etc.

I3 = CDW 20

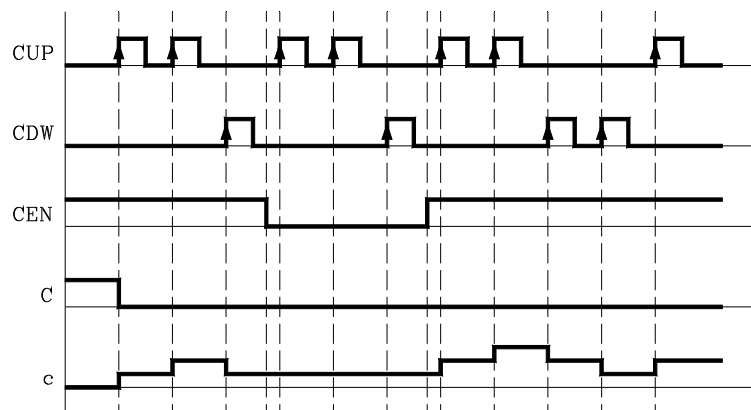
Every time an up-flank occurs at I3, the count of the C10 counter decreases one unit.

(CEN) Enable input.

It enables the internal count of the counter.

It is referred to by the letters CEN followed by the counter number, for example: CEN 1, CEN 25, CEN 102, etc.

To change the internal count (CUP and CDW), the CEN input must be high (=1). Setting CEN = 0 stops the counter's count and ignores the CUP and CDW inputs.



(CPR) Preset input.

To preset the counter with the desired value.

It is referred to with the letters CPR followed by the counter number and the preset count value. The counter is preset with the indicated value with an up-flank at the CPR input.

The count value may be indicated by a numeric value or by the internal value of a register "R". Its value must be between 0 and $\pm 2,147,483.647$.

CPR 20 100
Preset counter C20 with a value of 100.

CPR 22 R200
Preset counter C22 with the value of register R200

(C) Status output.

It indicates the logic state of the counter. It is referred to by the letter "C" followed by the counter number, for example: C1, C25, C102, etc.

The logic status of the counter will be C=1 when its count value is "0" and C=0 if otherwise.

(C) Count value.

It indicates the value of the internal count of the counter.

It is referred to by the letter "C" followed by the counter number, for example: C1, C25, C102, etc. Its representation C123 is the same as that of the status output, but different type instructions are used for them.

In binary instruction, it refers to the logic status of the counter.

C123 = M100
Assigns the status (0/1) of C123 to M100

In arithmetic and comparison instructions, it refers to the internal count of the counter.

I2 = MOV C123 R200
Transfers the count of C123 to register R200.

CPS C123 GT 1000 = M100
Compares whether the count of C123 is greater than 1000. If so, it activates mark M100.

The PLC has a 32-bit variable to store the count of each counter.

3.

INTRODUCTION TO THE PLC.
Operation of a counter.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

3.

INTRODUCTION TO THE PLC.

Operation of a counter.



CNC 8070

(REF: 1709)

The PLC program may comprise the following modules. Every module must begin with the directive instruction that defines it (CY1, PRG, PE) and end with the directing instruction END.

- First Cycle module (CY1).
- Main module (PRG).
- Periodic module (PE).

All of them consist of a series of instructions that depending on their function may be either directing or executable instructions.

Directing instructions.

The directing instructions provide the PLC with information on the type of module (PRG, CY1, ...) and on how it must be executed (REA, IMA, ...).

Executable instructions.

With the executable instructions, it is possible to check and/or change the status of the PLC resources. They consist of:

- Logic or Boolean instructions (I28 AND I30).
- Action instructions (=O25).

Logic expressions consist of:

- Consulting instructions (I28, O25).
- Operators (AND).

Logic expressions can be written in two or more different lines without having to include any dividing character. However, to make the program easier to understand, the "\" character may be inserted at the end of each line, but it is not necessary.

The following programming examples are equivalent.

Option 1.	I32 AND I36 AND M111 = O25
Option 2.	I32 AND I36 AND M111 = O25
Option 3.	I32 AND I36 AND M111 = O25
Option 4.	I32 \ AND I36 \ AND M111 = O25

Comments.

All comments must begin with ";". Lines beginning with a ";" are considered comments and are not executed.

Empty lines are also possible.

Programming example:

```
PRG ; Directing instruction
; Example ; Comment
I100 = M102 ; Executable instruction
I28 AND I30 ; Logic expression
= O25 ; Action instruction
I32 \ ; Consulting instruction (1st part of expression)
AND I36 ; Consulting instruction (2nd part of expression)
= M300 ; Action instruction
END ; Directing instruction
```

4.

PLC PROGRAMMING.

4.1 Directing instructions.

They provide the PLC with information on the type of program module and on how it must be executed. The available directing instructions are:

PRG, PE t, CY1	Type of module.
END	End of module.
REA, IMA	Real or image values.
L	Label.
SUB	Subroutine definition.
DEF, PDEF	Symbol definition.
NOMONIT	No monitoring.
EXTERN	External subroutine definition.

PRG, PE t, CY1 Type of module.

The PLC program is structured by modules. Every module must begin with its defining instruction (PRG, PE, CY1) and end with the END instruction. See ["3.2 Modular structure of the PLC program."](#) on page 321.

CY1	First cycle module.
PRG	Main module.
PE t	Periodic module. It is executed every "t" milliseconds.

END End of module or subroutine.

It must be defined for each module or subroutine.

```

CY1                                ; Beginning of the CY1 module
...
END                                ; End of the CY1 module
PRG                                ; Beginning of the PRG module
...
END                                ; End of the PRG module

```

A carriage return (empty line) is required after the last END.

REA, IMA Real or image values.

They indicate whether the following consultations are carried out using the real values (REA) or image values (IMA) or the I, O, M resources. The rest of the resources have no image values, only real.

The real value is the one the resource has at that time and the image value is the one it had at the end of the previous cycle scan. Image values (IMA) and real values (REA) may be combined in the same instruction.

```
IMA I3 AND REA M4 = O2
```

By default, all the modules (PRG, CY1, PEt) operate with real resource values. Action instructions (=O32) always update the real values of the PLC resources.

4.

Understanding how real and image values work.

The following example shows how the PLC acts when working with real or image values. For the given PLC program and with the resources initialized to zero, it shows the status of all the resources at the end of each scan or cycle.

		REA				IMA			
		M1	M2	M3	O5	M1	M2	M3	O5
()=M1		0	0	0	0	0	0	0	0
M1 = M2	Scan 1	1	1	1	1	1	0	0	0
M2 = M3	Scan 2	1	1	1	1	1	1	0	0
M3 = O5	Scan 3	1	1	1	1	1	1	1	0
	Scan 4	1	1	1	1	1	1	1	1

With real values (REA), output O5 goes high (=1) at the end of the first cycle scan, whereas it needs 4 cycle scans when using image values (IMA).

In the first cycle, ()=M1 sets the real value of M1=1, but its image value is 0. Only at the end of this cycle scan will it be 1.

The system is faster when working with real values (REA); whereas with image values (IMA), it is possible to analyze the same resource throughout the whole program with the same value, regardless of its current value.

**L
Label.**

It is used to identify a program line. It may be defined in two ways:

- With L followed by up to 7 digits (L1 - L9999999).
- With L_ followed by 8 characters (L_GEAR).

If it is defined within a module (CY1, PRG or PE), it identifies a program line and allows making references or jumps.

If it is defined outside the modules, for example at the end of the program after END, it indicates the beginning of a subroutine. It is the same as the SUB directing instruction.

If a program has more than one label with the same name or number, the PLC program will issue the corresponding error message when generating the executable program.

**SUB
Subroutine definition.**

It indicates the beginning of a subroutine. A subroutine is a portion of the program that may be called upon by any executable instruction.

It is defined with SUB followed by a blank space and up to 24 characters. A subroutine must always end with an END instruction.

```
SUB A22
...
END
```

They must be defined outside the modules (PRG, CY1, PE), for example at the end of the program after the directing instruction END.

A subroutine can also start with the L instruction and end with the END instruction.

(REF: 1709)

4.

PLC PROGRAMMING.
Directing instructions.

EXTERN**External subroutine definition.**

The subroutines defined in the "C" language file used by the program must be defined as external at the beginning of the program before the DEF instructions and the modules: CY1, PRE and PEt. With the EXTERN instruction, those subroutines may be defined one by one.

It is defined with EXTERN followed by a blank space and the name of the subroutine with up to 24 characters.

```
EXTERN SUMA
EXTERN TEMPERATURE
```

DEF, PDEF**Symbol definition.**

Symbols are always programmed at the beginning of the program, before the modules CY1, PRE and PEt. Since the PLC project may consist of several files and the symbols must be defined before they are used, we recommend to define them in the first file of the PLC project. However, the symbols may be defined in any file as long as they are used in that file or in later files.

The PLC allows to define a number of symbols for easier programming and later understanding of the PLC program. These symbols are programmed using the DEF or PDEF instruction followed by the name of the symbol, a constant or PLC resource. The PLC allows associating a symbol may be associated with any decimal or hexadecimal number or PLC resource such as inputs (I), outputs (O), marks (M), registers (R), register bits, counters (C) and timers (T).

DEF The DEF instruction may be used to define an unlimited number of symbols to be used only from the PLC.

PDEF The PDEF instruction may be used to define up to 100 symbols that can be used at the PLC itself, in a part-program or in an external application. Symbols exceeding this limit will be ignored and the CNC will issue the corresponding warning.

The symbols associated with a constant can only be used inside the PLC program; they can be accessed neither from the part-program nor from an external application. The symbols associated with a constant cannot be monitored, used in a trace of the logic analyzer or accessed from external variables.

```
PDEF COOL I12
PDEF CONSTANT $FFFF3
DEF DATA_D1 372893
DEF DATA_D3 -437289
DEF /FAN I23
```

Symbols will consist of a sequence of up to 20 characters with capital letters (A..Z) and digits (0..9). Symbols may also begin with the "/" character; in that case, the next character must be a letter. The name may have a "_" character, but it cannot be its first character. Words reserved for instructions cannot be used in symbols.

Duplicate symbols cannot be defined; but several symbols may be assigned to the same resource. Once a symbol has been associated with a resource or numeric value, it is possible to use the name of the resource, the number or its associated symbol.

Accessing the PDEF symbols from a part-program or from an application.

Accessing the symbols defined with the PDEF instruction from a part-program, MDI or external application is done using variables as follows. Consulting this variable from the part-program interrupts block preparation.

V.PLC.symbol	Access from a part-program or MDI.
PLC.symbol	Access from an external application.

The variables may be read or written depending on the resource assigned to the symbol defined with PDEF.

4.

PLC PROGRAMMING.
Directing instructions.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

NOMONIT

No monitoring.

When programming this directing instruction, it does not generate information necessary to monitor the PLC program. In other words, the program is not monitored.

It must always be programmed at the beginning of the program, before the DEF instruction and the modules: CY1, PRG and PEt.

This instruction should only be used when the PLC program execution time is very critical. Define it after debugging the PLC program.

Programming example.

```

;No monitoring
  NOMONIT
;External subroutine
  EXTERN TEMPERATURE
;Symbol definition
  DEF COOL I12
  DEF /FAN I23
;CY1 module
  CY1
  ...
  END
;PRG module
  PRG
  ...
  IMA I3 AND REA M4 = O2
  ...
  L_GEAR
  ...
  END
;PEt module
  PE 100
  ...
  END
;Subroutine
  SUB A22
  ...
  END

```

4.

PLC PROGRAMMING.

Directing instructions.

4.2 Consulting instructions.

They may be used to check the status of PLC resources as well as the marks and registers for CNC-PLC communication. There are the following consulting instructions.

- Simple consulting instructions.
- Flank detection instructions.
- Comparing instructions.

4.2.1 Simple consulting instructions.

They test the status of the resources and they return their logic state.

- Inputs (I1-I1024)
- Outputs (O1-O1024)
- Local inputs (LI1-LI16) (only central units "MCU" and "ICU")
- Local outputs (LO1-LO8) (only central units "MCU" and "ICU")
- Marks (M1-M8192)
- Messages (MSG1-MSG1024)
- Errors (ERR1-ERR1024)
- Clocks (CLK)
- Registers (R1-R1024)
- Register bits (B0-B31 R1-R1024)
- Timer status (T1-T512)
- Counter status (C1-C256)
- CNC-PLC communication marks.

I12

Returns a "1" if the I12 input is active and a "0" if otherwise.

START

Returns a "1" when the CYCLE START key of the front panel is pressed and a "0" if otherwise.

4.

PLC PROGRAMMING.
Consulting instructions.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.2.2 Flank detection instructions.

They check whether the state of a resource has changed since the last time this consultation was made. This consultation may be made on real or image values. The instructions available are:

DFU	Detect an up flank.
DFD	Detect a down flank.

DFU Detect an up flank.

It detects an up-flank (0-to-1 change) at the indicated resource. It returns a "1" if it happened.

DFD Detect a down flank.

It detects an down-flank (0-to-1 change) at the indicated resource. It returns a "1" if it happened.

The programming format for DFU and DFD is:

DFU	I1..1024
DFD	O1..1024
	M1..8192
	MSG1..1024
	ERR1..1024
	B0..31 R1..1024
	CLK
	CNC-PLC communication marks

DFU I23
DFU B3R120
DFU AUXEND
DFD O32
DFD M45

4.

PLC PROGRAMMING.
Consulting instructions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.2.3 Comparing instructions.

CPS

Compare two operands.

With the CPS instruction, it is possible to make comparisons between two operands, checking whether the first one is greater than (GT), greater than or equal to (GE), equal to (EQ), different from (NE), less than or equal to (LE) or less than (LT) the second one.

It is possible to use as operands, timers (internal count), counters (internal count), registers, registers for CNC-PLC communication and decimal (#) or hexadecimal numbers within ± 2147483647 or between 0 and \$FFFFFFFF.

If the required condition is met, the consulting instruction returns a logic value "1" and a "0" if otherwise.

The programming format is:

CPS	T1..512	GT	T1..512
	C1..256	GE	C1..256
	R1..1024	EQ	R1..1024
	R CNC-PLC	NE	R CNC-PLC
	#	LE	#
		LT	

CPS C12 GT R14 = M100

If the internal count of counter "C12" is GREATER than the value of register R14, the PLC will assign the value of "1" to mark M100 and a "0" if otherwise.

CPS T2 EQ 100 = TG1 5 2000

When the time elapsed at timer "T2" is equal to 100 milliseconds, it will trigger timer "T5" in monostable mode and with time constant of 2 seconds.

4.

PLC PROGRAMMING.
Consulting instructions.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.3 Operators and symbols.

They are used to group and operate with different consulting instructions.

The available operators are NOT, AND, OR, XOR. The operators are associated from left to right and the priorities ordered from the highest to the lowest are: NOT AND XOR OR.

The available symbols are: \, (,).

4.

PLC PROGRAMMING.
Operators and symbols.

NOT

Negates the result of the consultation.

NOT I2 = O3

Output "O3" will be active when input I2 is not.

AND

Logic function "AND".

I4 AND I5 = O6

Output "O6" will be active when both inputs (I4, I5) are active.

OR

Logic function "OR".

I7 OR I8 = O9

Output "O9" will be active when either one (or both) inputs are active.

XOR

Logic "Exclusive OR" function.

I10 XOR I11 = O12

Output "O12" will be active when both inputs I10 and I11 have different logic states.

\

Line feed.

It is used to write a logic expression in more than one line The following may be programmed:

DFU MSTROBE AND CPS MFUN* EQ 3 = M1003

or also:

DFU MSTROBE \
AND CPS MFUN* EQ 3
= M1003

()

Open and close parenthesis.

They help clarify and select the order the logic expression is evaluated.

(I2 OR I3) AND (I4 OR (NOT I5 AND I6)) = O7

A consulting instruction consisting of only these two operators always has a value of "1".

() = O2

Output O2 will always be high "1".

4.4 Action instructions.

Action instructions allow changing the status of PLC resources and CNC-PLC communication marks depending on the result of the logic expression.

Logic expression = Action instruction

There may be several action instructions associated with a single logic expression. All the action instructions must be preceded by the "=" sign.

All the action instructions admit a prior NOT that inverts the result of the expression for that action.

```
I2 = O3 = NOT M100 = NOT TG1 2 100 = CPR 1 100
```

Output O3 will show the status of input I2.

Mark M100 will show the negated state of input I2.

A down-flank at input I2 will activate the trigger input TG1 of timer T2.

An up-flank at I2 will preset counter C1 with a value of 100.

The action instructions are divided into:

- Assignment binary action instructions.
- Conditional binary action instructions.
- Sequence breaking action instructions.
- Arithmetic action instructions.
- Logic action instructions.
- Specific action instructions.

Action instructions may change the status of all the PLC resources except for the physical inputs being used. When seeing the field "I 1/1024", one must understand that only the status of the unused inputs may be changed.

For example, if physical inputs I1 through I32 are used, only inputs I33 through I1024 may be changed.

4.

PLC PROGRAMMING.
Action instructions.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.4.1 Assignment binary instructions.

They assign the value (0/1) resulting from a logic expression to the indicated resource.

= I 1/1024	= O 1/1024	= M 1/8192
= MSG 1/1024	= ERR 1/1024	= TEN 1/512
= TRS 1/512	= TGn 1/512 #/R	= CUP 1/256
= CDW 1/256	= CEN 1/256	= CPR 1/256 #/R
= B 0/31 R 1/499	= CNC-PLC mark	

I3 = TG1 4 100

Assigns the status of input I3 to the trigger input TG1 of timer T4. Thus, an up-flank at I3 will trigger the TG1 input of timer T4.

(I2 OR I3) AND (I4 OR (NOT I5 AND I6)) = M111

Assigns to mark M111 the result of evaluating the logic expression: (I2 OR I3) AND (I4 OR (NOT I5 AND I6)).

4.

PLC PROGRAMMING.
Action instructions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.4.2 Conditional binary instructions.

There are 3 instructions: SET, RES and CPL, that are used to change the status of the indicated resource.

The programming format is:

= SET	I 1/1024
= RES	O 1/1024
= CPL	M 1/8192
	MSG 1/1024
	ERR 1/1024
	B 0/31 R 1/1024
	CNC-PLC mark

= SET

If expression = "1", it sets the resource to "1".

If the result of the logic expression is a "1", it sets the indicated resource to "1". If the result is "0", it does not change the resource.

```
CPS T2 EQ 100 = SET B0R100
```

When the timing of timer T2 reaches 100 milliseconds, it sets bit 0 of register R100 to "1".

= RES

If expression = "1", it sets the resource to "0".

If the result of the logic expression is a "1", it sets the indicated resource to "0". If the result is "0", it does not change the resource.

```
I12 OR NOT I22 = RES M55 = NOT RES M65
```

When the result of the logic expression is a "1", the PLC sets "M55 = 0"; but does not change M65. If the result of the logic expression is a "0", the PLC does not change M55; but it sets "M65 = 0".

= CPL

If expression = "1", it complements the resource.

If the result of the logic expression is a "1", it complements the status of the indicated resource. If the result is "0", it will not change the resource.

```
DFU I8 OR DFD M22 = CPL B12R35
```

Every time an up-flank is detected at input I8 or down-flank of mark M22, the PLC complements the state of bit 12 of register R35.

4.

PLC PROGRAMMING.
Action instructions.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.4.3 Sequence breaking action instructions.

These actions interrupt the sequence of a program, resuming it somewhere else in the program.

= **JMP**

Unconditional jump.

If the result of the logic expression is a "1", it jumps to the indicated label. If the result is a "0", it goes on to the next program line.

Its syntax depends on how the label to jump to was defined

= JMP L123 If the label was defined as L123.

= JMP L_ASA2 If the label was defined as L_ASA2.

I8 = JMP L12

If I8=1 the program continues at L12 and it does not execute the intermediate blocks.

NOT M14 AND NOT B7R120 = O8

CPS T2 EQ 2000 = O12

L12

(I12 AND I23) OR M54 = O6

= **CAL**

Call to a subroutine.

If the result of the logic expression is a "1", this action executes the indicated subroutine. If the result of the logic expression is a "0", the PLC will ignore this action and the program will go on without executing that subroutine.

Once the subroutine execution is over, the PLC will continue at the action instruction or executable instruction programmed after CAL.

Its syntax depends on how its associated subroutine was defined.

= CAL OILING If it was defined as SUB OILING.

= CAL L234 If it was defined as L234.

= CAL L_GEAR If it was defined as L_GEAR.

I2 = CAL L5 = O2

With I2=1, subroutine L5 will be executed and once executed, the PLC will set O2 to the value of input I2 (=1). If I2=0, the subroutine is not executed and the PLC sets output O2 to the status of input I2 (=0).

= **RET**

Return or end of a subroutine.

If the result of the logic expression is a "1", the PLC will treat this action like an END instruction. If the result is a "0", the PLC will ignore it.

If while executing a subroutine, the PLC detects a validated RET, it will conclude the subroutine.

4.

PLC PROGRAMMING.
Action instructions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.4.4 Arithmetic action instructions.

= MOV
Move.

It is used to move data from one PLC resource to another.

The programming format is:

	Origin	Destination	Origin code	Destination code	Bits to transmit
= MOV	I1/1024	I1/1024	0(Bin)	0(Bin)	32
	O1/1024	O1/1024	1(BCD)	1(BCD)	28
	M1/8192	M1/8192			24
	MSG1/1024	MSG1/1024			20
	ERR1/1024	ERR1/1024			16
	T1/512	R1/1024			12
	C1/256	R CNC-PLC			8
	R1/1024				4
	R CNC-PLC				
	#				

The Origin and Destination codes indicate which format (binary or BCD) they have and in which format they will be deposited in the destination resource. 4, 8, 12, 16, 20, 24, 28 or 32 bits may be transmitted.

If the codes and number of bits to be moved are not indicated, 32 binary bits will be moved bit to bit (0032).

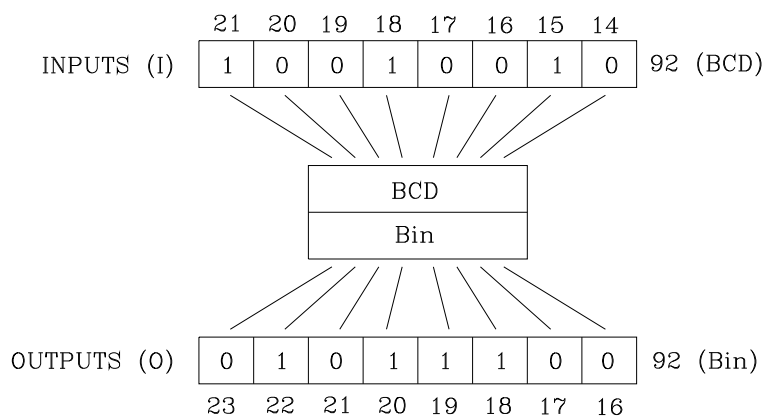
MOV	I12	M100	0032	binary to binary in 32 bits.
MOV	O21	R100	0012	binary to binary in 12 bits.
MOV	C22	O23	0108	binary to BCD in 8 bits.
MOV	T10	M112	1020	BCD to binary in 20 bits.

If the number to be converted from binary to BCD is larger than the maximum BCD, its value will be truncated ignoring the most significant bits.

The maximum BCD value that can be converted is: 9 (with 4 bits), 99 (with 8), 999 (with 12), 9999 (with 16), 99999 (with 20), 999999 (with 24), 9999999 (with 28) and 99999999 (con 32). In these cases, it is recommended to make the move increasing the number of bits by using, if necessary, registers or marks in intermediate steps.

I11 = MOV I14 O16 108

If input I11 is "1", the PLC moves the logic states of the 8 inputs (I14 plus the next 7) in BCD code to the 8 outputs (O16 and the next 7) in binary code.



4.

PLC PROGRAMMING.
Action instructions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

= NGU R 1/1024
Complements the bits of a register.

It changes the state of each one of the 32 bits of a register.

I15 = NGU R152

If input "I15 is =1", the PLC changes the state of the 32 bits of register R152.

R152 before: 0001 0001 0001 0001 0001 0001 0001 0001

R152 after: 1110 1110 1110 1110 1110 1110 1110 1110

= NGS R 1/1024
Register sign change.

Changes the sign of the register.

I16 = NGS R89

If input "I16 = 1", the PLC changes the sign of the contents of register R89.

R89 before: 0001 0001 0001 0001 0001 0001 0001 0001

R89 after: 1110 1110 1110 1110 1110 1110 1110 1111

= ADS, = SBS, = MLS, = DVS, = MDS
Arithmetic operations.

for adding (ADS), subtracting (SBS), multiplying (MLS), dividing (DVS) and calculating the module or remainder of a division (MDS).

Its programming format is:

"Operation" "1st operand" "2nd operand" "Result".

- The operands may be: Registers, CNC-PLC communication registers and numbers (#) in the ± 2147483647 range or between 0 and \$FFFFFFFF.
- The result of the operation may be stored in a register or in a CNC-PLC communication register.

= ADS R1/1024 R1/1024 R1/1024
 = SBS R CNC-PLC R CNC-PLC R CNC-PLC
 = MLS # #
 = DVS
 = MDS

Examples with R100=1234 and R101=100.

() = ADS R100 R101 R102 ; R102 = 1234 + 100 = 1334
 () = SBS R100 R101 R103 ; R103 = 1234 - 100 = 1134
 () = MLS R100 R101 R104 ; R104 = 1234 x 100 = 123400
 () = DVS R100 R101 R105 ; R105 = 1234 : 100 = 12
 () = MDS R100 R101 R106 ; R106 = 1234 MOD 100 = 34
 () = ADS 1563 R101 R112 ; R112 = 1563 + 100 = 1663
 () = SBS R100 1010 R113 ; R113 = 1234 - 1010 = 224
 () = MLS 1563 100 R114 ; R114 = 1563 x 100 = 156300
 () = MLS SANALOG 10000 R115
 = DVS R115 32767 R115 ; Spindle speed command in mV.

4.

PLC PROGRAMMING.
 Action instructions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.4.5 Logic action instructions.

- = AND
- = OR
- = XOR

Logic operations

To perform logic operations: AND, OR and XOR between register contents or between a register content and a number. The result is always stored in a register.

Its programming format is:

AND	R1/1024	R1/1024	R1/1024
OR	R CNC-PLC	R CNC-PLC	R CNC-PLC
XOR	#	#	

Examples with R200 = B1001 0010 and R201 = B0100 0101.

() = AND	R200	R201	R202	; R202 = B0
() = OR	R200	R201	R203	; R203 = B11010111
() = XOR	R200	R201	R204	; R204 = B11010111
() = AND	B1111	R201	R205	; R205 = B00000101
() = OR	R200	B1111	R206	; R206 = B10011111
() = XOR	B1010	B1110	R207	; R207 = B00000100

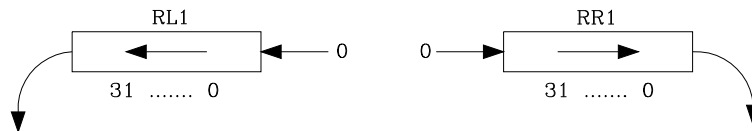
- = RR1
- = RR2
- = RL1
- = RL2

Register rotation.

Register contents may be rotated to the right (RR) or to the left and there are two types of rotations: type 1 (RR1 or RL1) and type 2 (RR2 or RL2).

Rotation type 1 (RL1 or RR1):

It inserts a "0" at the least significant bit (RL1) or at the most significant bit (RR1), pushing the other bits of the register. The value of the last bit disappears.



Rotation type 2 (RL2 or RR2):

Circular rotation of the register in the indicated direction.



Programming format:

The programming format is:

	Origin	Number of repetition	Dtarget
RR1	R1/1024	R1/1024	R1/1024
RR2	R CNC-PLC	R CNC-PLC	R CNC-PLC
RL1		0/31	
RL2			

4.

PLC PROGRAMMING.
Action instructions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

The origin and destination registers must always be defined, even when they are both the same. The number of repetitions indicates the consecutive number of times the register will be rotated.

```
RR1 R100 1 R200
```

It makes one type-1 right-hand rotation of R100 leaving the result in R200.

```
RL2 R102 4 R101
```

It makes four type-2 left-hand rotation of R102 leaving the result in R101.

4.

PLC PROGRAMMING.

Action instructions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.4.6 Specific action instructions.

= ERA

Clears a group of resources.

It is used to clear or initialize a group of same-type resources. Indicate the first and last resource to be erased.

The programming format is:

```
= ERA   I1/1024   1/1024
        O1/1024   1/1024
        M1/8192   1/8192
        MSG1/1024 1/1024
        ERR1/1024 1/1024
        T1/512    1/512
        C1/256    1/256
        R1/1024   1/1024
```

When erasing a group of I, O, M, MSG, ERR or R, the PLC sets them all to "0".

Erasing a group of timers is like resetting them and erasing a group of counters is like presetting with a "0" value.

This action is especially handy when executed in the first cycle module (CY1) in order to set the desired resources to their initial work conditions (states).

I10 = ERA O5 12

If input "I10=1", the PLC sets outputs O5 through O12 (both included) to "0".

I23 = ERA C15 18

If input "I23=1", the PLC presets counters C15 through C18 (both included) to "0".

= PAR

Parity of a register.

It analyzes the type of parity of a register. If the register has EVEN parity, this instruction sets the selected mark, message or error to "1" and to "0" if it has ODD parity.

The programming format is:

```
= PAR   R1/1024   M1/8192
        R CNC-PLC MSG1/1024
        ERR1/1024
        M CNC-PLC
```

I15 = PAR R123 M222

If input "I15=1", the PLC analyzes register R123 and sets mark M222 to "1" if EVEN parity or to "0" if ODD parity.

= CNCRD

= CNCWR

Reading (CNCRD) and writing (CNCWR) of CNC variables.

Used to read (CNCRD) and write (CNCWR) the internal CNC variables. Their programming format is:

CNCRD (Variable, Register, Mark)

CNCWR (Register, Variable, Mark)

The CNCRD action loads the contents of the variable into the register and the CNCWR action loads the contents of register into the variable. The mark is set to "1" at the beginning of the operation and it keeps its value until the end of the operation.

When using CNCRD to read the variables of the arithmetic parameters and those of the OEM, it returns the value multiplied by 10000 (float mode reading).

When requesting information on a nonexistent variable (for example the position of a nonexistent axis), it will show the relevant error message. Likewise, when trying to read a value whose range is greater than that of the PLC register, the CNC will return a zero value

4.

and will issue the corresponding error message. Whenever an error occurs when reading a variable, the communication mark will remain at "1".

Syntax of the variables in commands CNCRD and CNCWR.

For these two commands, it is possible to define, in the mnemonic of the variable, the channel number using an integer, a register or a symbol defined with DEF or PDEF.

```
CNCRD ([1].G.FREAL, R10, M1000)
```

```
... = MOV 1 R12
CNCRD ([R12].G.FREAL, R10, M1000)
```

```
DEF CHANNEL 3
CNCRD ([CHANNEL].G.FREAL, R10, M1000)
```

If the mnemonic of the variable contains numerical suffixes, e.g. (V).G.GUP[i], they too can be defined with an integer, a register or a symbol defined with DEF or PDEF.

```
... = MOV 153 R101
CNCWR (G.GUP[R101], R10, M1000)
```

```
DEF PARAM 153
CNCRD (G.GUP[PARAM], R10, M1000)
```

Consulting synchronous and asynchronous variables.

Synchronous variables are the ones resolved immediately whereas asynchronous variables are the ones requiring several cycle scans to be resolved.

- Example of how to access asynchronous variables:

```
<condition> AND NOT M11 = CNCRD (TM.TOOL, R11, M11)
```

do not repeat this consultation until it ends.

```
DFD M11 AND CPS R11 EQ 3 = ...
```

wait for the consultation to end before comparing the data.

- Examples of how to access synchronous variables:

```
<condition> = CNCRD (G.FREAL, R12, M12)
```

```
CPS R12 GT 2000 = ...
```

No need to wait before consulting the data because synchronous variables are resolved immediately.

```
<condition> = CNCWR (R13, PLC.TIMER, M13)
```

It resets the clock enabled by the PLC with the value contained in register R13.

= CNCEX

Execution of a CNC block.

It may be used to execute a CNC block in the indicated channel, including calls to subroutines or to complete programs. It works like when executing a block in MDI. The restrictions for the execution of commands are the same as for the MDI blocks.

Its programming format is:

```
CNCEX (block, mark, channel)
```

The mark is set to "1" at the beginning of the operation and it keeps its value until the end of the operation. If the channel is not indicated, the block is executed in the first or main channel.

```
... = CNCEX (G00 X0 Y0, M99, 2)
... = CNCEX (#CALL sub3.nc, M34)
```

Once the block has been executed, the CNC channel activates the FREE mark to let the PLC know that it is ready to accept a new block. The execution of the CNCEX command may be canceled with the mark PLCABORT.

4.

Executing independent movements from the PLC. Commands MOVEABS, MOVEADD and MOVEINF.

The movements of the independent axes may be programmed directly or with the CNCEX command; however, it is not recommended to use both methods in the same PLC program or subroutine.

The treatment for the execution of the commands is different and the order they are executed might not be the desired one. The CNCEX command is executed through a CNC channel whereas the MOVE* commands are executed directly at the interpolator (usually faster execution).

The independent movements may be executed as follows: The two should not be used in the same program or subroutine.

```
() = CNCEX(#MOVE ADD [X100,F100,NULL], M120,1)
```

```
() = MOVEADD(X,100000,100000,NULL)
```

4.

PLC PROGRAMMING.
Action instructions.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.4.7 Action instruction of the electronic cam.



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 cam.

= CAM ON
= TCAM ON
= CAM OFF

Activate the electronic cam with real coordinates (CAM ON), with theoretical coordinates (TCAM ON) or cancel the electronic cam (CAM OFF).

The programming format for each of them is the following.

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 command involves eliminating the synchronization of the cam. Once this command has been programmed, the cam ends when reaching the end of its profile.

Parameter	Meaning
cam	Cam number.
master	Name of the master axis.
TIME	Time cam. When programming "TIME" instead of an axis name, the cam is interpreted as being a time cam.
slave	Name of the slave axis.
master_off	Offset for the master axis.
slave_off	Offset for the slave axis.
range_master	Master axis activation scale or range.
range_slave	Slave axis activation scale or range.
type	It defines the type of cam; periodic or not periodic. It is programmed using parameters "ONCE" (non-periodic cam) or "CONT" (periodic cam).

Cam mode.

Two types of cams may be activated; cams in time or cams according to the position of the master axis. The activation instruction is the same and it is selected by the call-parameters.

Cam number.

To activate a cam, it must have been previously defined at the cam editor, within the machine parameters.

Master axis activation range.

The cam is activated when the master axis is between the positions "master_off" and "master_off + range_master".

Range for the slave axis.

The cam applies it to the slave axis when the slave axis is between "slave_off" and "slave_off + range_slave".

4.

PLC PROGRAMMING.
Action instructions.

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. It is selected with the type parameter.

- Not periodic It is defined by assigning the "ONCE" value to the type parameter.

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.
- Periodic It is defined by assigning the "CONT" value to the type parameter.

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 command is executed. When reaching that command, the execution of the cam will end the next time the end of the cam profile is reached.

= CAM SELECT
= CAM DESELECT
Select (CAM SELECT) or cancel a file cam (CAM DESELECT).

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.

The following commands only define the location of the cam; to activate it, use the instruction #CAM ON (from the CNC) or the command CAM ON (from the PLC). After selecting a file cam, it stays available until the cam table of the machine parameters is validated or the CNC is turned off.

The programming format for each of them is the following.

CAM SELECT (cam, file)
CAM DESELECT (cam)

Executing the CAM OFF command involves eliminating the synchronization of the cam. Once this command has been programmed, the cam ends when reaching the end of its profile.

Parameter	Meaning
cam	Cam number.
file	Name and path of the file containing the cam data.

4.

PLC PROGRAMMING.
Action instructions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.4.8 Instructions for independent move: positioning.



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.

= MOVE ABS
= MOVE ADD
= MOVE INF

Absolute positioning movement (MOV ABS), incremental (MOVE ADD) or no-end (MOVE INF).

The programming format for each of them is the following.

MOVE ABS (axis, pos, feed, blend)
MOVE ADD (axis, pos, feed, blend)
MOVE INF (axis, direction, feed, blend)

Parameter	Meaning
axis	Axis to position.
pos	Position to reach.
direction	Moving direction. It is programmed with parameters "DIRPOS" (positive direction) or "DIRNEG" (negative direction).
feed	Positioning feedrate
blend	Dynamic blend with the next block It is programmed using parameters "PRESENT", "NULL", "NEXT" or "WAITINPOS".

The programming units are the standard PLC units. The coordinates the feedrates will be given in ten-thousandths if they are in mm or degrees or in hundred-thousandths if they are in inches.

Position to reach.

With MOVE ABS it will be defined in absolute coordinates whereas with MOVE ADD it will be defined in incremental coordinates. For positioning, the zero offset active in the channel is ignored.

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 it is unidirectional, it positions in the preset direction; otherwise, it positions via the shortest path.

Moving direction.

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.

Dynamic blend with the next block

It sets the feedrate used to reach the position (dynamic blend with the next block). It is programmed with one of the following parameters.

PRESENT	The axis reaches the indicated position at the feedrate indicated by the block itself.
NEXT	The axis reaches the indicated position at the feedrate indicated in the next block.
NULL	The axis reaches the indicated position at zero feedrate.
WAITINPOS	The axis reaches the indicated position at zero feedrate and it waits to be in position before executing the next block.

4.

PLC PROGRAMMING.
Action instructions.

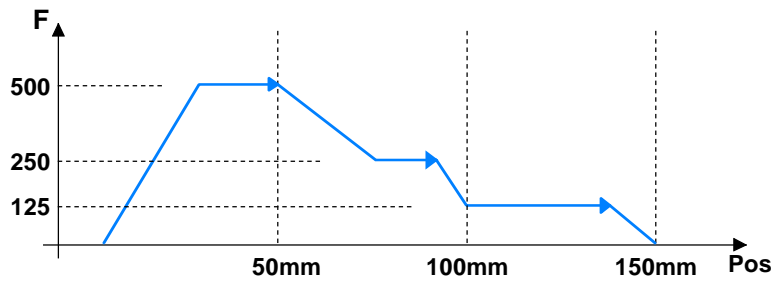


FAGOR AUTOMATION

CNC 8070

(REF: 1709)

```
.. = MOVE ABS (X, 500000, 5000000, PRESENT)  
.. = MOVE ABS (X, 1000000, 2500000, NEXT)  
.. = MOVE ABS (X, 1500000, 1250000, NULL)
```



4.

PLC PROGRAMMING.
Action instructions.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.4.9 Instructions for independent move: synchronization.



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.

- = FOLLOW ON
- = TFOLLOW ON
- = FOLLOW OFF

Activate the synchronized movement with real coordinates (FOLLOW ON), with theoretical coordinates (TFOLLOW ON) or cancel the synchronized movement (FOLLOW OFF).

The programming format for each of them is the following.

FOLLOW ON (master, slave, Nnratio, Ddratio, synctype)
 TFOLLOW ON (master, slave, Nnratio, Ddratio, synctype)
 FOLLOW OFF (slave)

Parameter	Meaning
master	Name of the master axis.
slave	Name of the slave axis.
Nnratio	Numerator of the gear ratio. Turns of the slave axis.
Ddratio	Denominator of the gear ratio. Rotations of the master axis.
synctype	Type of synchronism. It is programmed with parameters "POS" (position synchronism) or "VEL" (velocity synchronism).

The programming units are the standard PLC units. The coordinates the feedrates will be given in ten-thousandths if they are in mm or degrees or in hundred-thousandths if they are in inches.

Example from the PLC program.

```
FOLLOW ON (A1, Z, N3, D1, VEL)
TFOLLOW ON (A1, Z, N3, D1, VEL)
FOLLOW OFF (Z)
```

Velocity synchronization.

Velocity offset.

It may be used to vary the speed of the slave axis independently from that of the master axis. It is defined with the variable V.A.SYNCVELOFF.xn.

Gear ratio.

Ratio (Nslave/Nmaster) between the number of turns of the slave axis (Nslave) and the number of turns of the master axis (Nmaster).

Fine adjustment of the gear ratio.

The gear ratio is determined when programming the instruction and its value remains constant during the whole operation. However, and even if the synchronization is in progress, this ratio may be modified by fine-adjusting it. The fine adjustment of the ratio is defined with the variable GEARADJ.

Position (phase) synchronization.

Position offset.

It may be used to vary the position of the slave axis independently from that of the master axis. It is defined with the variable V.A.SYNCPOSOFF.xn.

4.

PLC PROGRAMMING.
Action instructions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Gear ratio.

Ratio (N_{slave}/N_{master}) between the number of turns of the slave axis (N_{slave}) and the number of turns of the master axis (N_{master}).

The gear ratio is determined when programming the instruction and its value remains constant during the whole operation. In this synchronization mode, it is not possible to change this value while the system is running, because this behavior is more for an electronic cam than for an electronic gear. To solve this matter, an electronic cam may be programmed instead.

4.

PLC PROGRAMMING.
Action instructions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.4.10 Instructions for coordinate latching using a probe or digital input.

Coordinate latching means that the CNC captures the position value of an axis when a given event occurs. The TOUCHPROBE instruction latches the coordinate of an axis by an event of a probe, of a physical digital input or of a logic input. The events to be used will be digital inputs defined in parameters PRBDI1 and PRBDI2.

- If the selected event is a probe and when activating the TOUCHPROBE command the probe has already reached its logic level (the probe acts either by level or by flank), the latched coordinate will be the one captured when the command was activated, not that of the point where the level reaching event occurred.
- If the selected event is a logic input, it may be activated from the PLC program and, therefore, it may be conditioned indirectly by any other event managed by the PLC.

The coordinate may be latched in any position of the axis travel or a latch window may be defined. If there is a latch window, coordinate latching may be conditioned to whether the axis is inside or outside the window.

The following probe-related machine parameters have no effect: PROBEAXIS, PROBERANGE, PROBEFEED, PRB1MIN, PRB2MIN, PRB3MIN, PRB1MAX, PRB2MAX and PRB3MAX.

= TOUCHPROBE Coordinate latching.

The programming format is: Optional parameters are indicated between angle brackets.

TOUCHPROBE (axis,probe<,wintype><,winminpos,winmaxpos>)

Parameter	Meaning
axis	Name of the axis whose coordinate is being latched.
probe	Number of the probe to be used as latching event.
wintype	Type of window for latching. It is programmed with the commands DISABLE (without window), EXCLUSIVE (latch outside the window) and INCLUSIVE (latch inside the window).
winminpos	Lower limit for the latch window.
winmaxpos	Upper limit for the latch window.

The programming units are the standard PLC units. The coordinates the feedrates will be given in ten-thousandths if they are in mm or degrees or in hundred-thousandths if they are in inches.

```
TOUCHPROBE (X, 1, DISABLE)
TOUCHPROBE (X, 1)
TOUCHPROBE (Y, 2, EXCLUSIVE, 1000000, 2300000)
TOUCHPROBE (X3, 1, INCLUSIVE, 500000, 1105000)
```

·axis· Name of the axis whose coordinate is being latched.

Name of the axis whose coordinate is being latched; spindles are not admitted, except if they are active as C axis.

·probe· Number of the probe to be used as latching event.

The names of the probes are set by the order in which the machine parameters have been defined. Probe 1 will be the one assigned to parameter PRBDI1 and probe 2 will be the one assigned to parameter PRBDI2.

Instead of a probe, a physical digital input or a logic input may also be used as an event. In this case, the input being used must also be defined in parameter PRBDI1 or PRBDI2.

(REF: 1709)

4.

PLC PROGRAMMING.
Action instructions.



FAGOR AUTOMATION

CNC 8070

·wintype· Type of window for latching.

This option is programmed using the following commands.

- DISABLE There is no latch window (default option). A coordinate may be latched in any position of the axis travel.
- EXCLUSIVE The coordinate is latched if the axis is outside the window.
- INCLUSIVE The coordinate is latched if the axis is inside the window.

·winminpos· ·winmaxpos· Size of the latch window.

The size of the window must be defined when the latch window is either exclusive or inclusive. Call parameters ·winminpos· and ·winmaxpos· set the lower and upper limits of the latch window.

Coordinate latching operation.

For the same probe, a coordinate latching process and a probing process programmed with G100 cannot be active at the same time, even if it is on different axes; when executing one of them while the other one is active, the CNC will issue an error message. Coordinate latching is not affected by functions G101 and G102.

CNC	PLC	
#PROBE SELECT 1 G100 X100	.. = TOUCHPROBE (X,1)	Wrong.
#PROBE SELECT 1 G100 X100	.. = TOUCHPROBE (Z,1)	Wrong.
#PROBE SELECT 1 G100 X100	.. = TOUCHPROBE (X,2)	Right.
#PROBE SELECT 2 G100 X100	.. = TOUCHPROBE (X,1)	Right.

The TOUCHPROBE instruction may be activated simultaneously for any of the 2 probes and for any of the axes of the system. An active coordinate latching may also be programmed again.

```
.. = TOUCHPROBE (X,1)
.. = TOUCHPROBE (Y,1)
.. = TOUCHPROBE (X,2)
.. = TOUCHPROBE (Y,2)
```

All active latching processes that are assigned to the same probe are canceled when a latch event occurs on any of them. The event cancels all the latching processes assigned to that probe, even on the axes that are positioned outside the latch window. The independent interpolator will only activate the LATCH1DONE(axis) or LATCH2DONE(axis) mark of the axis where the event took place.

Cancelling the latching processes.

The latching process on the axis is canceled when the PLC activates the IRESET(axis) mark. When this mark cancels a latching process, the independent interpolator cancels the LATCH1ACTIVE(axis) or LATCH2ACTIVE(axis) mark associated with the process.

A reset of the channel cancels the latching processes on all axes of the channel.

Influence on functions M02 and M30.

Functions M02 and M30 will not be considered executed (done) until all the latching processes active on the axes of the channel are finished.

4.

PLC PROGRAMMING.
Action instructions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Consultation signals of the PLC associated with coordinate latching.

The independent interpolator has a mark for each probe. Probe 1 will be the one assigned to parameter PRBD11 and probe 2 will be the one assigned to parameter PRBD12.

PROBE1ACTIVE PROBE2ACTIVE

The independent interpolator activates this mark when a latching process is active with the indicated probe and it deactivates when there is no active latching process with the indicated probe.

LATCH1ACTIVE(axis) LATCH2ACTIVE(axis)

The independent interpolator activates this mark when activating a latching process on the axis with the indicated probe and it deactivates when the latching process ends or it is canceled.

The latching process on the axis is canceled when the PLC activates the IRESET(axis) mark. A reset of the channel cancels the latching processes on all axes of the channel.

Functions M02 and M30 will not be considered executed (done) until all the latching processes active on the axes of the channel are finished.

LATCH1DONE(axis) LATCH2DONE(axis)

The independent interpolator activates this mark when the latch event on the axis with the indicated probe occurs; it deactivates it when activating a new latching process on the axis with the same probe.

Variables related to coordinate latching.

The coordinate latching processes do not affect the variables associated with probing G100.

(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.

4.

PLC PROGRAMMING.
Action instructions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

·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·.

(V.)[ch].A.ACCUDIST.xn

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.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

V.MPA.ACCUDIST.Z	Z axis.
V.MPA.ACCUDIST.4	Axis with logic number ·4·.
V.[2].MPA.ACCUDIST.1	Axis with index ·1· in the channel ·2·.

(V.)[ch].A.PREVACCUDIST.xn

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.

V.MPA.PREVACCUDIST.Z	Z axis.
V.MPA.PREVACCUDIST.4	Axis with logic number ·4·.
V.[2].MPA.PREVACCUDIST.1	Axis with index ·1· in the channel ·2·.

4.5 Considerations for some functions.

4.5.1 Work zones.

The work zones may be defined, enabled and disabled from the PLC using the command CNCWR, writing the corresponding variables.

Variable.	Meaning.
[ch].A.ZONELIMITTOL.xn	Safety distance of the limits of the work zones.
[ch].G.ZONEST[k]	Status of work zone [K]. (0=Disabled zone). (1=Work zone enabled as no-entry). (2=Work zone enabled as no-exit).
[ch].A.ZONELOWLIM[k].xn	Lower limit of zone [k].
[ch].A.ZONEUPLIM[k].xn	Upper limit of zone [k].
[ch].G.ZONECIRAX1[k]	Logic axis corresponding to the first coordinate of the center of zone [k].
[ch].G.ZONECIRAX2[k]	Logic axis corresponding to the second coordinate of the center of zone [k].
[ch].G.ZONECIR1[k]	Center coordinate of zone [k] along the first axis that defines the circular zone.
[ch].G.ZONECIR2[k]	Center coordinate of zone [k] along the second axis that defines the circular zone.
[ch].G.ZONER[k]	Radius of zone [k] (circular zone).
[ch].G.ZONETOOLWATCH[k]	Monitor the tool tip or the tool base. (0=Monitor the tool tip). (1=Monitor the tool base). (2=Monitor the tool tip and the tool base).

Syntax of the variables.

- ch· Channel number.
- k· Zone number.
- xn· Name, logic number or index of the axis.

[2].G.ZONEST[1]	Channel ·2·. Zone 1.
A.ZONEUPLIM[1].Z	Z axis. Zone 1.
A.ZONEUPLIM[1].4	Axis with logic number ·4·. Zone 1.
[2].A.ZONEUPLIM[1].1	Axis with index ·1· in the channel ·2·. Zone 1.

Example of how to activate a work zone from the PLC.

The following example shows how to define a forbidden zone of the X axis the area between position 0 and 100 mm (1000000 tenth-thousands of a mm).

```
<condition> = MOV 0 R1 = CNCWR(R1, A.ZONELOWLIM[1].X, M1)
              = MOV 1000000 R1 = CNCWR(R1, A.ZONEUPLIM[1].X, M1)
              = MOV 1 R1 = CNCWR(R1, G.ZONEST[1], M1)
```

The coordinates will be given in ten-thousandths if they are in mm or hundred-thousandths if they are in inches.

4.

PLC PROGRAMMING.
Considerations for some functions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Synchronize changes in the zones from the part-program and from the PLC.

When making changes in the work zone from the PLC while executing a part-program, it must be borne in mind that the CNC has already prepared the program blocks immediately after the one being executed; therefore, checking the new limits and/or the new status may not be correct for some blocks after the change. In order to ensure proper synchronization of the changes between the part-program and the PLC, these changes should be executed via interruption subroutines.

4.

PLC PROGRAMMING.

Considerations for some functions.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

4.6 Summary programming commands.

Resources available at the PLC.

Inputs (I1..1024)
 Outputs (O1..1024)
 Marks (M1..8192)
 Message marks (MSG1..1024)
 Error marks (ERR1..1024)
 Clocks (CLK)
 CNC-PLC marks
 Timers (T1..512)
 Counters (C1..256)
 Registers (R1..1024)
 CNC-PLC registers

The register value may be treated as a decimal or hexadecimal ("S") number.

It is also possible to refer to a register bit using the letter B (0..31) R (1..1024).

Clock	Time	Clock	Time	Clock	Time
CLK1	1 ms	CLK100	100 ms	CLK1000	1 s
CLK2	2 ms	CLK200	200 ms	CLK2000	2 s
CLK4	4 ms	CLK400	400 ms	CLK4000	4 s
CLK8	8 ms	CLK800	800 ms	CLK8000	8 s
CLK16	16 ms	CLK1600	1.6 s	CLK16000	16 s
CLK32	32 ms	CLK3200	3.2 s	CLK32000	32 s
CLK64	64 ms	CLK6400	6.4 s	CLK64000	64 s
CLK128	128 ms	CLK12800	12.8 s	CLK128000	128 s

Directing instructions.

PRG Main module
 PE t Periodic module. It is executed every "t" milliseconds.
 CY1 First Cycle module.
 END End of module.
 L Label.
 SUB Subroutine definition.
 DEF: Symbol definition.
 PDEF External symbol definition.
 REA The consultations will use real values.
 IMA The consultations will use image values.
 NOMONIT No PLC program monitoring.
 EXTERN External subroutine definition.

4.

PLC PROGRAMMING.
 Summary programming commands.

Consulting instructions.

- Simple consulting instructions.

I1..1024	Inputs.
O1..1024	Outputs.
M1..8192	Marks.
MSG1..1024	Messages.
ERR1..1024	Errors.
T1..512	Timers (status).
C1..256	Counters (status).
B0..31 R1..1024	Register bit.
CLK	Clocks.
M <CNC-PLC>	Marks for CNC-PLC communication.
- Flank detection instructions.

DFU	Up flank detection.
DFD	Down flank detection.
- Comparing instructions.

CPS	For comparisons.
-----	------------------

Operators.

NOT	Negates the result of the consultation.
AND	Logic function "AND".
OR	Logic function "OR".
XOR	Logic "Exclusive OR" function.
\	Line feed.
()	Consulting instruction whose value is always "1".

Action instructions.

- Assignment binary action instructions.

= I 1/1024	= O 1/1024	= M 1/8192
= MSG 1/1024	= ERR 1/1024	= TEN 1/512
= TRS 1/512	= TGn 1/512 #/R	= CUP 1/256
= CDW 1/256	= CEN 1/256	= CPR 1/256 #/R
= B 0/31 R 1/499	= CNC-PLC mark	
- Conditional binary action instructions.

= SET	If expression = "1", it sets the resource to "1".
= RES	If expression = "1", it sets the resource to "0".
= CPL	If expression = "1", it complements the resource.
- Sequence breaking action instructions.

= JMP L	Unconditional jump.
= CAL	Call to a subroutine.
= RET	Return or end of a subroutine.
- Arithmetic action instructions.

= MOV	Move.
= NGU R1..1024	Complements the bits of a register.
= NGS R1..1024	Register sign change.
= ADS	Add.

4.

- = SBS Subtract.
- = MLS Multiplication.
- = DVS Division.
- = MDS Module or remainder of a division.
- Logic action instructions.
 - = AND Logic operation "AND".
 - = OR Logic operation "OR".
 - = XOR Logic operation "XOR".
 - = RR 1/2 Right-hand register rotation.
 - = RL 1/2 Left-hand register rotation.
- Specific action instructions.
 - = ERA Erases or resets a group of resources.
 - =PAR Parity of a register.
 - =CNCRD Reading of internal variables.
 - =CNCWR Writing of internal variables.
 - = CNCEX Execution of a CNC block.
- Action instruction of the electronic cam.
 - = CAM ON Activate the electronic cam (real coordinates).
 - = TCAM ON Activate the electronic cam (theoretical coordinates).
 - = CAM OFF Cancel the electronic cam.
 - = CAM SELECT Selet a file cam.
 - = CAM DESELECT Cancel the cam of a file.
- Action instructions for independent axes. Positioning move.
 - = MOVE ABS Absolute positioning move.
 - = MOVE ADD Incremental positioning move.
 - = MOVE INF Infinite (endless) positioning move.
- Action instructions for independent axes. Synchronization movement.
 - =FOLLOW ON Activates the synchronization movement (real coordinates).
 - = TFOLLOW ON Activates the synchronization movement (theoretical coordinates).
 - = FOLLOW OFF Cancels the synchronization movement.
- Action instructions for independent axes. Coordinate latching with the help of a probe or a digital input.
 - = TOUCHPROBE Coordinate latching.

With the data exchange between the CNC and the PLC, it is possible to:

- Control the logic CNC inputs and outputs in peripheral mode using specific PLC marks and registers.
- Transfer auxiliary M, H and S functions from the CNC to the PLC.
- Generate messages and errors at the CNC using PLC marks.
- Read and modify internal CNC variables from the PLC.
- Access all PLC resources from any part-program.
- Monitor PLC resources on the CNC screen.

Abbreviations used in this chapter.

- (=0) Low logic level.
- (=1) High logic level.

–M– and –H– functions with channels.

The M and H functions are exchanged by channel. When using several channels, the marks and registers of these functions must indicate the channel number they refer to. If no channel number is indicated, the marks and registers refer to the first channel.

–S– functions with multiple spindles.

The exchange of S functions is independent from the channel. When using several spindles, the marks and registers of these functions refer to the spindle number. The spindle number is determined by its logic number.

5.1 Auxiliary –M– functions.

Up to 7 M functions in the same block. The CNC indicates to the PLC which auxiliary M functions are programmed in the execution block using 32-bit registers MFUN1 through MFUN7. Each one of them indicates the number of one of the M functions programmed in the block. If all the registers are not used, the CNC assigns \$FFFFFFFF to the unused ones (those with the highest numbers).

This way, if a block contains functions M100, M120 and M135, the CNC will transfer the following information to the PLC.

MFUN1	MFUN2	MFUN3	MFUN4 - MFUN7
100	120	135	\$FFFFFFFF

Command MFUN*. Checking if a function has been programmed in the block.

To know whether a particular M function is programmed in the execution block, use one of the following methods:

- Check all the MFUN registers one by one until that particular M function is found or until one of them has the value of \$FFFFFFFF.
- Use the MFUN* command to check all the registers at the same time.

Example to detect M30. If it has been programmed, it will return a "1", and a "0" if otherwise.

```
CPS MFUN* EQ 30 = ...
```

Sending the function and synchronizing the execution.

Within the CNC machine parameters, the auxiliary M function table indicates when the function is sent and when the PLC execution is synchronized. In either case, it may be before or after the movement.

The sending and synchronizing types may be the following:

M not synchronized.

M sent out and synchronized before the movement.

M sent out before the move and synchronized after the movement.

M sent out and synchronized after the movement.

M functions with different types of synchronization may be programmed in the same block. Each one of them will be sent out to the PLC at the right moment. The transfer of auxiliary M functions is described later on in this chapter. See ["5.4 Transferring auxiliary functions -M-, -H-, -S-"](#) on page 384.

The functions may be set as follows:

M11 not synchronized.

M12 is sent and synchronized before the movement.

M13 is sent before and synchronized after the movement.

M14 is sent and synchronized after the movement.

When executing a block like this:

```
X100 F1000 M11 M12 M13 M14
```

The functions are transferred as follows:

- 1 It sends the M11 M12 and M13 functions out to the PLC.
- 2 waits for the PLC to execute the M12.
- 3 The CNC moves the axis to X100.
- 4 It sends the M14 function to the PLC.
- 5 It waits until the PLC executes the M13 and M14.

5.

CNC-PLC COMMUNICATION.
Auxiliary –M– functions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

5.1.1 Special considerations with the multi-spindle option and channels.

The CNC may have up to 4 channels and each channel can execute a part-program in parallel with the rest. This means that each channel can execute seven auxiliary functions simultaneously. The auxiliary functions executed from each channel are treated independently; to do that, each channel has its own marks and registers.

Since each channel may have four spindles, it is possible to program in the same block 6 non-spindle-related M functions, the startup of all four spindles M3 / M4 and a speed for each of them involving an automatic gear change. This means that, because some functions are generated automatically, it may exceed the maximum of seven auxiliary functions per block. In this case, the CNC will send the M functions out to the PLC in two stages.

Marks and registers in the channels option.

Each channel has 32-bit registers MFUN1 to MFUN7 to indicate to the PLC which auxiliary M functions are programmed in the execution block.

- MFUN1C1 - MFUN7C1 for the first channel.
- MFUN1C2 - MFUN7C2 for the second channel.
- MFUN1C3 - MFUN7C3 for the third channel.
- MFUN1C4 - MFUN7C4 for the fourth channel.

Each one of them indicates the number of one of the M functions programmed in the block. If all the registers are not used, the CNC assigns \$FFFFFFFF to the unused ones (those with the highest numbers).

This way, if functions M100 and M135 are programmed in the first channel and functions M88 and M75 in the second channel, the CNC will transfer the following data.

MFUN1C1	MFUN2C1	MFUN3C1 - MFUN7C1
100	135	\$FFFFFFFF

MFUN1C2	MFUN2C2	MFUN3C2 - MFUN7C2
88	75	\$FFFFFFFF

Commands MFUNC1* - MFUNC4*. Checking if a function has been programmed in the channel.

To know whether a particular M function is programmed in the execution block, use one of the following methods:

- Check all the MFUN registers one by one until that particular M function is found or until one of them has the value of \$FFFFFFFF.
- Use one of the following commands to check all the MFUN registers of the channel at the same time.

- MFUNC1* For channel 1
- MFUNC2* For channel 2.
- MFUNC3* For channel 3.
- MFUNC4* For channel 4.

Example for detecting M04 in channel 1. If it has been programmed, it will return a "1", and a "0" if otherwise.

```
CPS MFUNC1* EQ 4 = ...
```

5.

CNC-PLC COMMUNICATION:
Auxiliary -M- functions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

5.2 Auxiliary –H– functions.

Up to 7 M and 7 H functions may be programmed in a block. The treatment of the auxiliary H functions is similar to the M functions without synchronization.

The CNC indicates to the PLC which auxiliary H functions are programmed in the execution block using 32-bit registers HFUN1 through HFUN7. Each one of them indicates the number of one of the H functions programmed in the block. If all the registers are not used, the CNC assigns \$FFFFFFFF to the unused ones (those with the highest numbers).

This way, if a block contains functions H12, H20 and H35, the CNC will transfer the following information to the PLC.

HFUN1	HFUN2	HFUN3	HFUN4 - HFUN7
12	20	35	\$FFFFFFFF

Command HFUN*. Checking if a function has been programmed in the block.

To know whether a particular H function is programmed in the execution block, use one of the following methods:

- Check all the HFUN registers one by one until that particular H function is found or until one of them has the value of \$FFFFFFFF.
- Use the HFUN* format to check all the HFUN registers at the same time.

Example to detect H77: If it has been programmed, it will return a "1", and a "0" if otherwise.

```
CPS HFUN* EQ 77 = ...
```

Sending and synchronizing the function.

The H functions are not synchronized and are sent out to the PLC at the beginning of block execution.

The transfer of auxiliary H functions is described later on in this chapter. See ["5.4 Transferring auxiliary functions -M-, -H-, -S-." on page 384.](#)

When executing a block like this:

```
X100 F1000 H11 H12
```

The functions are transferred as follows:

- 1 Functions H11 and H12 are sent out to the PLC
- 2 It does not wait for confirmation and the CNC moves the axis to X100.

5.

CNC-PLC COMMUNICATION.
Auxiliary –H– functions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

5.2.1 Special considerations with the multi-spindle option and channels.

The CNC may have up to 4 channels and each channel can execute a part-program in parallel with the rest. This means that each channel can execute seven auxiliary functions simultaneously. The auxiliary functions executed from each channel are treated independently; to do that, each channel has its own marks and registers.

Marks and registers in the channels option.

Each channel has 32-bit registers HFUN1 to HFUN7 to indicate to the PLC which auxiliary H functions are programmed in the execution block.

- HFUN1C1 - HFUN7C1 for the first channel.
- HFUN1C2 - HFUN7C2 for the second channel.
- HFUN1C3 - HFUN7C3 for the third channel.
- HFUN1C4 - HFUN7C4 for the fourth channel.

Each one of them indicates the number of one of the H functions programmed in the block. If all the registers are not used, the CNC assigns \$FFFFFFFF to the unused ones (those with the highest numbers).

This way, if functions H10 and H13 are programmed in the first channel and functions H8 and H10 in the second channel, the CNC will transfer the following data.

HFUN1C1	HFUN2C1	HFUN3C1 - HFUN7C1
10	13	\$FFFFFFFF

HFUN1C2	HFUN2C2	HFUN3C2 - HFUN7C2
8	10	\$FFFFFFFF

Commands HFUNC1* - HFUNC4*. Checking if a function has been programmed in the channel.

To know whether a particular H function is programmed in the execution block, use one of the following methods:

- Check all the HFUN registers one by one until that particular H function is found or until one of them has the value of \$FFFFFFFF.
- Use one of the following commands to check all the HFUN registers of the channel at the same time.

- HFUNC1* For channel 1
- HFUNC2* For channel 2.
- HFUNC3* For channel 3.
- HFUNC4* For channel 4.

5.

CNC-PLC COMMUNICATION.
Auxiliary –H– functions.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

5.3 Auxiliary –S– function.

The auxiliary S function indicates the spindle turning speed with M03 and M04 or the angular position with M19.

The S function with M03 and M04 is always executed at the beginning of the block and the CNC waits for confirmation before going on executing the program. When working with M19, the CNC treats the spindle like a regular linear axis. It only sends the M19 out to the PLC.

The CNC uses 32-bit register SFUN1 to indicate to the PLC the value of the S function programmed in the block. If not programmed, the value of \$FFFFFFFF is sent out. The SFUN command only assumes the value of the programmed S if spindle parameter SPDLTIME is other than zero.

The transfer of the S function is described later on in this chapter. See "[5.4 Transferring auxiliary functions -M-, -H-, -S-](#)" on page 384.

5.

CNC-PLC COMMUNICATION:
Auxiliary –S– function.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

5.3.1 Special considerations with the multi-spindle option and channels.

The CNC may have up to four spindles. All of them may be controlled independently in the same block; in other words, each spindle may be given a different command.

When using channels, the spindles may be distributed indistinctly between them. In this case, a channel can control a spindle of another channel. The marks and registers refer to the spindle regardless of the channel they belong to.

The spindle number is determined by its logic number that is set by the order they were defined in the machine parameter SPDLNAME.

Marks and registers in the multi-spindle version.

The CNC indicates to the PLC which S functions are programmed in the execution block using 32-bit registers SFUN1 through SFUN4. These registers refer to the spindle number; they are independent from the channel where the spindle is.

Each one of them indicates the value of one of the S functions programmed. If all the registers are not used, the CNC assigns \$FFFFFFFF to the unused ones (those with the highest numbers).

This way, if a block contains functions S1000 and S1=550, the CNC will transfer the following information to the PLC:

SFUN1	SFUN2	SFUN3	SFUN4
1000	550	\$FFFFFFFF	\$FFFFFFFF

Commands SP1FUN* - SP4FUN*. Check if an auxiliary function has been programmed for a spindle.

Considering the possible channels/spindles combinations, these functions are available to make it easier to manage the auxiliary M functions associated with each spindle. Each one indicates if any M3, M4, etc. type M function has been programmed in any channel.

- SP1FUN* For the spindle 1.
- SP2FUN* For the spindle 2.
- SP3FUN* For the spindle 3.
- SP4FUN* For the spindle 4.

Checks if the M5 function has been sent to spindle 1 from a channel.

CPS SP1FUN* EQ 5 = ...

5.

CNC-PLC COMMUNICATION.
Auxiliary -S- function.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

5.4 Transferring auxiliary functions -M-, -H-, -S-

The M and H functions are transferred per channel. Transferring S functions does not depend on the channel.

When executing a block that contains M, H, S functions, the following information is transferred to the PLC.

Transferring -M- functions.

The CNC assigns the numbers of the M functions programmed in the block to registers MFUN1 through MFUN7. Some M functions have an associated function (DMxx) that is activated when sending the M to the PLC.

M00	M01	M02	M03	M04
M05	M06	M08	M09	M19
M30	M41	M42	M43	M44

The CNC activates the general logic output MSTROBE to "tell" the PLC that it must execute them. This mark is kept high (=1) for a time period indicated by parameter MINAENDW.

Depending on the type of synchronization, the CNC will either wait or not for the general input AUXEND to be activated indicating the end of the PLC execution. The type of synchronization is defined in the machine parameters

The CNC cancels the general logic output MSTROBE to conclude the execution.

Transferring -H- functions.

The CNC assigns the numbers of the H functions programmed in the block to registers HFUN1 through HFUN7.

The CNC activates the general logic output HSTROBE to "tell" the PLC that it must execute them. This mark is kept high (=1) for a time period indicated by parameter MINAENDW.

After this time period, the CNC considers its execution completed because there is no synchronization.

When sending several blocks in a row just having H functions, the CNC waits twice the time indicated in parameter MINAENDW.

```
N10 H60
N20 H30 H18
N30 H40
```

Transferring -S- functions

The CNC assigns the values of the S programmed in each spindle to registers SFUN1 through SFUN4.

The CNC activates the general logic output SSTROBE to "tell" the PLC that it must execute it. The CNC waits for the general input AUXEND to be activated indicating the end of the PLC execution.

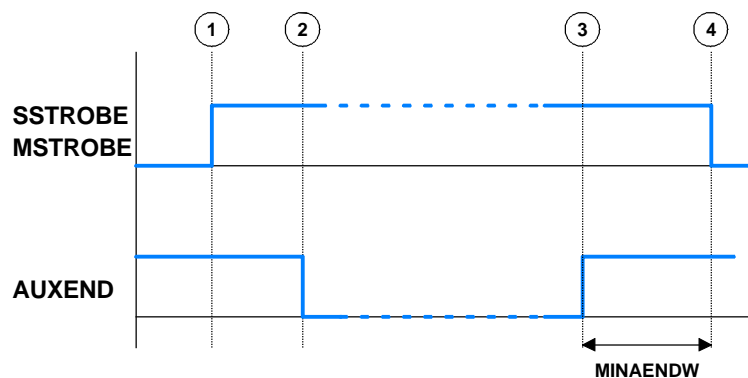
The CNC cancels the general logic output SSTROBE to conclude the execution.

5.

CNC-PLC COMMUNICATION:
Transferring auxiliary functions -M-, -H-, -S-

5.4.1 Synchronized transfer.

This type of transfer takes place with the S function and with the M functions set with synchronization.



When the PLC is requested to execute several M or S functions at the same time, the corresponding SSTROBE or MSTROBE signals are activated; but the CNC waits for a single "AUXEND" signal to end all of them.

Transferring -M- functions

- 1 The CNC indicates in registers MFUN1 to MFUN7 of the channel the M functions programmed in the block and it activates the MSTROBE mark so the PLC executes them.
- 2 The PLC must deactivate the AUXEND mark to let the CNC know that the execution has begun.
- 3 Once the required auxiliary functions have been executed, the PLC must activate the AUXEND mark to let the CNC know that the execution has ended.
The AUXEND mark must be kept high (=1) longer than the time period established by parameter MINAENDW.
- 4 After this time, the CNC deactivates the MSTROBE mark thus ending the execution of the functions.

Transferring -S- functions

- 1 The CNC indicates in register SFUN1 to SFUN4 the S value programmed in the block and activates the SSTROBE mark so the PLC executes it.
- 2 The PLC must deactivate the AUXEND mark to let the CNC know that the execution has begun.
- 3 After selecting the requested S, the PLC must activate the AUXEND mark to let the CNC know that the execution has ended.
The AUXEND mark must be kept high (=1) longer than the time period established by parameter MINAENDW.
- 4 After this time, the CNC deactivates the MSTROBE mark thus ending the execution of the function.

5.

CNC-PLC COMMUNICATION.

Transferring auxiliary functions -M-, -H-, -S-.



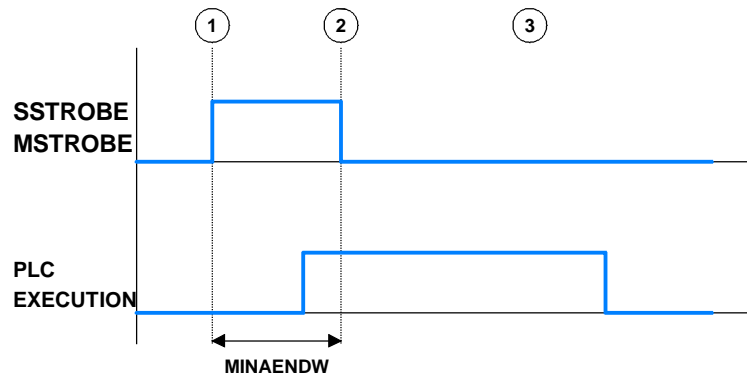
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

5.4.2 Non-synchronized transfer.

This type of transfer takes place with the H function and with the M functions set without synchronization.



Transferring -M- functions

- 1 The CNC indicates in registers MFUN1 to MFUN7 of the channel the M functions programmed in the block and it activates the MSTROBE mark so the PLC executes them.
- 2 The CNC keeps the MSTROBE mark active for a time period indicated by parameter MINAENDW.
- 3 After this time, the CNC goes on executing the program regardless of the time required by the PLC to execute that function.

Transferring -H- functions

- 1 The CNC indicates in registers HFUN1 to HFUN7 of the channel the H functions programmed in the block and it activates the HSTROBE mark so the PLC executes them.
- 2 The CNC keeps the HSTROBE mark active for a time period indicated by parameter MINAENDW.
- 3 After this time, the CNC goes on executing the program regardless of the time required by the PLC to execute that function.

Considerations for transferring these functions.

The value of parameter MINAENDW should be the same or longer than the PLC program execution period parameter PRGFREQ in order to ensure that the PLC detects that signal.

When sending non-synchronized H or M functions corresponding to consecutive blocks of the same program, the CNC waits between blocks for a time period indicated by MINANEDW so the PLC can read all the functions.

5.

CNC-PLC COMMUNICATION:
Transferring auxiliary functions -M-, -H-, -S-



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

5.5 Displaying PLC errors and messages.

The PLC has 1024 marks for displaying messages and another 1024 marks for displaying errors at the CNC. When the mark is high (=1) the message or the error is active.

MSG1 - MSG1024 for displaying messages.

ERR1 - ERR1024 for displaying errors.

There is a message and error table where each message or error may have associated the following:

- A number (ID field).
- A text ("Message" field).
- An additional information file ("associated file" field) that may be a "bmp, txt, jpg, gif, htm, html or avi" type file.
- Whether when activating an error or message the additional help file must be displayed ("Show" field selected) or only the text of the message or of the error.
- For the errors, whether they open the emergency relay of the central unit or not.

MESSAGES AND ERRORS		
MSG1	<input type="checkbox"/>	MSG-1-
MSG2	<input checked="" type="checkbox"/>	MSG-2-
MSG3	<input checked="" type="checkbox"/>	MSG-3-
MSG4	<input checked="" type="checkbox"/>	MSG-4-
MSG5	<input checked="" type="checkbox"/>	MSG-5-
MSG6	<input checked="" type="checkbox"/>	MSG-6-
MSG11	<input type="checkbox"/>	MSG-11-
ERR1	<input checked="" type="checkbox"/>	ERROR-1-
ERR2	<input checked="" type="checkbox"/>	ERROR-2-
ERR3	<input checked="" type="checkbox"/>	ERROR-3-
ERR4	<input checked="" type="checkbox"/>	ERROR-4-
ERR5	<input checked="" type="checkbox"/>	ERROR-5-
ID	5	
SHOW	<input checked="" type="checkbox"/>	
MESSAGE	ERROR-5-	
RELATED FILE	C:\CNC8070\MTB\PLC\Lang\error5.bmp	
EMERGEN	<input checked="" type="checkbox"/>	

PLC RUNNING

For more information on how to edit this table, refer to the operating manual.

PLC messages.

When activating one of the MSG marks, the status bar of the CNC window for PLC messages shows the message number and its associated text. If the message has a file with additional information associated with it, an access icon will appear to the left of the message.



The message does not have a file with additional information.



The message has a file with additional information.

When there are more than one message activated, it always shows the one with the highest priority (the one with the lowest number). The PLC-messages window shows the "+" sign meaning that there are more messages activated by the PLC. To display the whole list, press [CTRL] + [M].

If the "Show" field of the message is selected, the CNC screen shows the additional information file and if there isn't one, a blue window with the text of the message. If the "Show" field is not selected, to show the additional information file, you must expand the list of messages, select a message and press [ENTER] or click on the message. To close the additional data window, press [ESC].

5.

PLC errors.

When activating one of the ERR marks, the CNC interrupts the execution of the part-program and it displays a window in the middle of the screen showing the error number and its associated text. If the error has the "Emergen" field selected, the error will open the emergency relay of the CNC.



If the error has a file with additional information associated with it, an access icon will appear to the right of the error number. If the error has the "Show" field selected, the CNC shows the additional information file directly on the screen. If the "Show" field is not selected, the additional information file will be displayed when pressing the [HELP] key or when clicking on the icon mentioned earlier. To close the additional data window, press [ESC].

External inputs should be used to activate and deactivate error marks, thus preventing the CNC from receiving those errors at every new PLC cycle scan.

Grouping the additional information text files in a single file.

PLC messages and errors can show an additional information file in text format. The PLC allows grouping several or all these files into a single file as follows.

Defining the additional information file.

It must be a text file (extension txt) and may have any name. The information of each message and error must be structured in the following format:

```
[<id>]
<text>
```

The <id> field, keeping the brackets, will be the code that identifies the help text inside the file, which needs not be the same as the number of the error or message it will be associated with. The <text> field will be the information text that may have up to 500 characters including line feeds.

For example, the OEM.txt file will have the following structure.

```
[10]
Help text.
[27]
Help text.
[33]
Help text.
```

Calling the texts from the PLC message or from the PLC error.

To associate the help message with a PLC message or PLC error, the "associated file" field must be defined like <file>#<id>. The <file> field will be the path and the name of the file. The <id> field will be the code that identifies the help text inside the file.

For example, the "associated file" will be defined as follows.

```
C:\CNC8070\MTB\PLC\LANG\OEM.txt#27
```

5.

CNC-PLC COMMUNICATION.
Displaying PLC errors and messages.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

CNC's physical inputs and outputs are the set of system inputs and outputs that are governed by the PLC and communicate with the outside world through CNC connectors.

The CNC also has a number of logic inputs and outputs to exchange the internal data with the marks and registers of the PLC. This way, the PLC has access to some internal CNC data. Each of these logic inputs and outputs may be referred to using its associated mnemonic. The mnemonics that begin with a "_" sign indicate that the signal is active low (0 V).

CNCREADY	_ALARM
AUXEND	_EMERGEN
MANUAL	_STOP

The CNC's logic outputs or PLC consulting signals are grouped into:

- General consulting signals.
- Axis consulting signals.
- Spindle consulting signals.
- Consultation signals of the independent interpolator.
- Tool manager consulting signals.
- Keystroke consulting signals.

The CNC's logic inputs or signals that may be modified via PLC are grouped into:

- General modifiable signals.
- Axis modifiable signals.
- Spindle modifiable signals.
- Modifiable signals of the independent interpolator.
- Tool manager modifiable signals.
- Keystroke modifiable signals.

Abbreviations used in this chapter.

(=0)	Low logic level.
(=1)	High logic level.
(g.m.p.)	General machine parameter.
(a.m.p.)	Machine parameters for axes and spindles.

6.1 General consulting signals.

6.

LOGIC CNC INPUTS AND OUTPUTS.
General consulting signals.

CNCREADY

This mark indicates the CNC status. It is set to "0" when the CNC is in an error state (red status window) and to "1" if otherwise.

Include this mark in the PLC maneuver to enable the drives.

```
; ++++++
; AXIS ENABLE
; ++++++
CNCREADY \ ; CNC without errors
AND NOT B1R2 \ ; FEEDHOLD due to lack of power at the X axis drive
AND NOT PARKEDX \ ; X axis parked
AND NOT PARKEDX \ ; Unpark X axis
AND NOT PARKEDX \ ; Park X axis
= SERVOXON ; X axis servo drive active
= SPENAX ; X axis sercos enable
= TG3 2 500 ; Delay for DRENAX
;
T2\; Delay to the drop of speed enable
= DRENAX \ ; Drive enable via Sercos.
```

READY

There is a mark for each channel. The mnemonics for each channel are the following.

READYC1 READYC2 READYC3 READYC4

This mark indicates the status of the CNC channel. It is set to "0" when the CNC is in an error state (red status window) and to "1" if otherwise.

SERCO SRDY

In Sercos configuration, this mark indicates the ring status. The CNC sets this mark high (=1) to indicate that the Sercos ring has been properly initialized.

All the drive status consultations (DRSTAF DRSTAS) and the read/write (CNCRD CNCWR) operations of the drive variables are conditioned by this mark.

START

There is a mark for each channel. The mnemonics for each channel are the following.

STARTC1 (can also be programmed as START)
STARTC2 STARTC3 STARTC4

The CNC channel activates this mark to indicate to the PLC that the [START] key has been pressed at the operator panel. If the rest of the conditions are met (hydraulic, safety, etc.), the PLC must activate the CYSTART mark in order for the program to start running. See "[CYSTART](#)" on page 419.

```
START AND (rest of conditions) = CYSTART
```

RESETOUT

There is a mark for each channel. The mnemonics for each channel are the following.

RESETOUTC1 (can also be programmed as RESETOUT)
RESETOUTC2 RESETOUTC3 RESETOUTC4

CNC channel reset. The channel activates the RESETOUT (RESETOUT=1) mark to indicate that the reset has finalized. This mark stays active for the time period set in general parameter MINAENDW.

The channel uses this mark whenever performing a reset, either from the [RESET] key on the operator panel or when the PLC activates the RESETIN mark. See "RESETIN" on page 420.

FHOUT

There is a mark for each channel. The mnemonics for each channel are the following.

FHOUTC1 (can also be programmed as FHOUT)
 FHOUTC2 FHOUTC3 FHOUTC4

The CNC channel sets this mark high (=1) when the execution of the part-program is interrupted. To interrupt and later resume the execution of a program, proceed as follows:

- If the PLC has set the mark "_STOP" to (=0), to resume the execution, the PLC must set marks "_STOP" and "CYSTART" to (=1).
- If the PLC has set the mark "_FEEDHOL" to (=0), to resume the execution, the PLC must set the "_FEEDHOL" mark to (=1).

_ALARM

There is a mark for each channel. The mnemonics for each channel are the following.

_ALARMC1 (can also be programmed as _ALARM)
 _ALARMC2 _ALARMC3 _ALARMC4

The CNC channel activates this mark (at "0") when there is an alarm, an emergency generated by the CNC channel or when the PLC activates the emergency signal _EMERGEN. The CNC channel cancels this mark (at "1") when the CNC channel message is removed and the cause of the alarm or emergency has been eliminated.

For moves in automatic mode, if the CNC detects a collision in a positioning or withdrawal movement inside a probing cycle (or even outside of it), it stops the movement, issues the corresponding error message, opens the emergency relay and activates the _ALARM signal ("0" level).

There is no output associated with this mark. The following example shows how to associate output O1.

_ALARM AND (rest of conditions) = O1
 If there are no errors, output O1 will be high (=1)

MANUAL

There is a mark for each channel. The mnemonics for each channel are the following.

MANUALC1 (can also be programmed as MANUAL)
 MANUALC2 MANUALC3 MANUALC4

The CNC sets this mark high (=1) when the Manual Operation (JOG) mode is selected.

AUTOMAT

There is a mark for each channel. The mnemonics for each channel are the following.

AUTOMATC1 (can also be programmed as AUTOMAT)
 AUTOMATC2 AUTOMATC3 AUTOMATC4

The CNC channel sets this mark high (=1) when the automatic operation mode is selected.

MDI

There is a mark for each channel. The mnemonics for each channel are the following.

MDIC1 (can also be programmed as MDI)
 MDIC2 MDIC3 MDIC4

The CNC sets this mark high (=1) when the MDI (Manual Data Input) mode is selected.

- If the MDI mode is selected while in automatic mode, the AUTOMAT and MDI marks will be activated.
- If the MDI mode is selected in JOG mode, the MANUAL and MDI marks are activated.

SBOUT

There is a mark for each channel. The mnemonics for each channel are the following.

SBOUTC1 (can also be programmed as SBOUT)
 SBOUTC2 SBOUTC3 SBOUTC4

The CNC sets this mark high (=1) when the Single Block mode is selected.

- If the –single block– mode is selected while in automatic mode, the AUTOMAT and SBOUT marks are activated.
- If the –single block– mode is selected while in JOG mode, the MANUAL and SBOUT marks.

INCYCLE

There is a mark for each channel. The mnemonics for each channel are the following.

INCYCLEC1 (can also be programmed as INCYCLE)
 INCYCLEC2 INCYCLEC3 INCYCLEC4

The CNC sets this mark high (=1) when executing a block or moving an axis.

- In execution. It is set high (=1) at the beginning of the execution and it stays high until the end, until the [STOP] key is pressed or until the "_STOP" mark is set low (=0).
- In MDI or Single Block execution, it is set low (=0) at the end of the block.
- In JOG mode, it stays high (=1) while any of the JOG keys is pressed.

RAPID

There is a mark for each channel. The mnemonics for each channel are the following.

RAPIDC1 (can also be programmed as RAPID)
 RAPIDC2 RAPIDC3 RAPIDC4

The CNC channel sets this mark high (=1) when executing a rapid traverse movement (G0).

ZERO

There is a mark for each channel. The mnemonics for each channel are the following.

ZEROC1 (can also be programmed as ZERO)
 ZEROC2 ZEROC3 ZEROC4

The CNC sets this mark high (=1) when searching home (G74).

PROBE

There is a mark for each channel. The mnemonics for each channel are the following.

PROBEC1 (can also be programmed as PROBE)
 PROBEC2 PROBEC3 PROBEC4

The CNC channel sets this mark high (=1) when executing a probing movement (G100).

6.

LOGIC CNC INPUTS AND OUTPUTS.
 General consulting signals.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

THREAD

There is a mark for each channel. The mnemonics for each channel are the following.

THREADC1 (can also be programmed as THREAD)
 THREADC2 THREADC3 THREADC4

The CNC sets this mark high (=1) when executing an electronic threading block (G33).

TAPPING

There is a mark for each channel. The mnemonics for each channel are the following.

TAPPINGC1 (can also be programmed as TAPPING)
 TAPPINGC2 TAPPINGC3 TAPPINGC4

The CNC sets this mark high (=1) when executing a tapping canned cycle. The mark stays active during the possible dwell programmed at the bottom of the thread.

In milling, cycle G84, the mark stays active throughout the whole cycle (including the blocks with movement to the starting point, etc.). In lathe, cycle G83, the mark stays active only during the actual threading.

RIGID

There is a mark for each channel. The mnemonics for each channel are the following.

RIGIDC1 (can also be programmed as RIGID)
 RIGIDC2 RIGIDC3 RIGIDC4

The CNC sets this mark high (=1) when executing a rigid tapping block (G63). Tapping (G84 in milling and G83 in lathe) set as rigid tapping set this mark to (=1) during the whole tapping operation, including the dwell at the bottom of the thread.

CSS

There is a mark for each channel. The mnemonics for each channel are the following.

CSSC1 (can also be programmed as CSS)
 CSSC2 CSSC3 CSSC4

The CNC sets this mark high (=1) when Constant Surface Speed is selected (G96).

INTEREND

There is a mark for each channel. The mnemonics for each channel are the following.

INTERENDC1 (can also be programmed as INTEREND)
 INTERENDC2 INTERENDC3 INTERENDC4

The CNC sets the INTEREND mark high (=1) when the theoretical movement of the axes (when it no longer outputs the velocity command).

The INTEREND mark may be used to activate external devices before the axes reach their position. The mark ADVINPOS can also be used. See "[ADVINPOS](#)" on page 396.

INPOSI

There is a mark for each channel. The mnemonics for each channel are the following.

INPOSI1 (can also be programmed as INPOSI or INPOS)
 INPOSI2 INPOSI3 INPOSI4

The CNC channel sets the INPOSI mark to (=1) to indicate that all its active axes and spindles have reached position, except the independent axes programmed from the PLC. The INPOSI mark stays at (=1) while the independent axes are moving.

An axis is in position when it stays within the in-position zone (window) (parameter INPOSW) for a time period indicated by parameter INPOSTIME.

SPN1...SPN7

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.

SPN1C1 SPN1C2 SPN1C3 SPN1C4

The channel uses these registers to indicate to the PLC which spindle of the channel each auxiliary M function selected for execution is addressed to.

Each channel can have up to 7 M functions in a block. If all the registers are not used, the hexadecimal value \$FFFFFFFF is assigned to the unused ones (those with the highest numbers).

This way, if the next block is programmed in the first channel, the CNC will pass on to the PLC the following information.

M3.S1 S1=1000 M4.S2 S2=500

Clockwise rotation of spindle S1 at 1000 rpm and counterclockwise rotation of spindle S2 at 500 rpm.

MFUN1C1	MFUN2C1	MFUN3C1 - MFUN7C1
3	4	\$FFFFFFFF

SPN1C1	SPN2C1	SPN3C1 - SPN7C1
1	2	\$FFFFFFFF

If a function is programmed in the block without mentioning the spindle, it will assume the master spindle of the channel.

Commands SP1FUN* - SP4FUN*. Checking if a spindle receives a function from any channel.

In order to know whether a particular spindle has received a particular function or not, it is possible to check all the registers one by one or use the following commands to check all of them at the same time.

SP1FUN* For spindle 1.
 SP2FUN* For spindle 2.
 SP3FUN* For spindle 3.
 SP4FUN* For spindle 4.

CPS SP1FUN* EQ 5 = ...

Example to check if the first spindle has received an M5 function from any channel. If it has been programmed, it will return a "1", and a "0" if otherwise.

**MFUN1...MFUN7
HFUN1...HFUN7**

There is a register for each channel. The mnemonics for each channel are the following. Here is an example of the mnemonics for MFUN1 and HFUN1; it is the same for the rest of the registers.

MFUN1C1 (can also be programmed as MFUN1)
 MFUN1C2 MFUN1C3 MFUN1C4
 HFUN1C1 (can also be programmed as HFUN1)
 HFUN1C2 HFUN1C3 HFUN1C4

The channel uses these registers to indicate to the PLC the M or H auxiliary functions selected for execution.

Each channel can have up to 7 M and 7 H functions in a block. If all the registers are not used, the hexadecimal value \$FFFFFFFF is assigned to the unused ones (those with the highest numbers).

(REF: 1709)

6.

LOGIC CNC INPUTS AND OUTPUTS.
 General consulting signals.



FAGOR AUTOMATION

CNC 8070

This way, if functions M100 and M135 are programmed in the first channel and functions M88 and M75 in the second channel, the CNC will transfer the following data.

MFUN1C1	MFUN2C1	MFUN3C1 - MFUN7C1
100	135	\$FFFFFFFF

MFUN1C2	MFUN2C2	MFUN3C2 - MFUN7C2
88	75	\$FFFFFFFF

If, then, the M88 function is executed in the first channel, then:

MFUN1C1	MFUN2C1	MFUN3C1 - MFUN7C1
88	\$FFFFFFFF	\$FFFFFFFF

Commands MFUNC1* - MFUNC4* and HFUNC1* - HFUNC4*. Checking if a function has been programmed in the channel.

In order to know whether a particular function is programmed in the block currently being executed, all the registers may be checked one by one or the following commands may be used to check them all at the same time.

- MFUNC1*/HFUNC1* For channel 1. They can also be programmed as MFUN*/HFUN*.
- MFUNC2*/HFUNC2* For channel 2.
- MFUNC3*/HFUNC3* For channel 3.
- MFUNC4*/HFUNC4* For channel 4.

```
CPS MFUNC1* EQ 4 = ...
Example for detecting M04 in channel 1. If it has been programmed, it will return a "1", and a "0" if otherwise.
```

MSTROBE

There is a mark for each channel. The mnemonics for each channel are the following.

- MSTROBEC1 (can also be programmed as MSTROBE)
- MSTROBEC2 MSTROBEC3 MSTROBEC4

The CNC channel sets this mark high (=1) to indicate to the PLC that it must execute the auxiliary M functions indicated in registers MFUN1 through MFUN7 of the channel.

HSTROBE

There is a mark for each channel. The mnemonics for each channel are the following.

- HSTROBEC1 (can also be programmed as HSTROBE)
- HSTROBEC2 HSTROBEC3 HSTROBEC4

The CNC channel sets this mark high (=1) to indicate to the PLC that it must execute the auxiliary H functions indicated in registers HFUN1 through HFUN7.

SFUN1...SFUN4

There is a register for each spindle. The mnemonics for each spindle are the following:

- SFUN1 SFUN2 SFUN3 SFUN4

These registers indicate the programmed speed for each spindle. These registers refer to the spindle number; they are independent from the channel where the spindle is. The CNC only uses the registers of the spindles whose SPDLTIME parameter has a value other than zero.

Each one of them indicates the value of one of the S functions programmed. If all the registers are not used, the CNC assigns \$FFFFFFFF to the unused ones (those with the highest numbers).

6.

LOGIC CNC INPUTS AND OUTPUTS. General consulting signals.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

This way, if a block contains functions S1000 and S1=550 and the value of SPDLTIME parameter of both spindles is other than zero, the CNC will transfer the following information to the PLC.

SFUN1	SFUN2	SFUN3	SFUN4
1000	550	\$FFFFFFFF	\$FFFFFFFF

SSTROBE

There is a mark for each spindle. The mnemonics for each spindle are the following:

SSTROBE1 (can also be programmed as SSTROBE)
 SSTROBE2 SSTROBE3 SSTROBE4

The CNC channel sets this mark high (=1) to indicate to the PLC that a new spindle speed has been selected. The CNC channel only uses this mark on spindles whose SPDLTIME parameter has a value other than zero.

DMxx

This mark is associated with some M auxiliary functions.

The marks associated with functions M00, M01, M02, M06, M08, M09, M30 have 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).

DM00C1 (can also be programmed as DM00)
 DM00C2 DM00C3 DM00C4

The marks associated with functions M03, M04, M05, M19, M41, M42, M43, M44 have 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).

DM03SP1 (can also be programmed as DM03)
 DM03SP2 DM03SP3 DM03SP4

The CNC indicates in these marks the status of the spindle auxiliary M functions. The mark is set to (=1) if the function is active and to (=0) if otherwise.

BLKSEARCH

There is a mark for each channel. The mnemonics for each channel are the following.

BLKSEARCHC1 (can also be programmed as BLKSEARCH)
 BLKSEARCHC2 BLKSEARCHC3 BLKSEARCHC4

The CNC channel sets this mark high (=1) to indicate that the "Block Search" option is active in the Automatic Operation mode.

ADVINPOS

There is a mark for each channel. The mnemonics for each channel are the following.

ADVINPOSC1 (can also be programmed as ADVINPOS)
 ADVINPOSC2 ADVINPOSC3 ADVINPOSC4

The CNC channel sets this signal high for some time before the axes reach position. This time is set by (g.m.p.) ANTIME.

If the total duration of the movement is lower than the value set by (g.m.p.) ANTIME, the mark goes high (=1) immediately.

If (g.m.p.) ANTIME has been set to 0, the mark is always active.

It is used on punch presses that have an eccentric cam as a punching system. This signal may be used to start the movement of the punch before the axes reach the position. This reduces idle time, thus increasing the number of punches per minute.

6.

LOGIC CNC INPUTS AND OUTPUTS.
General consulting signals.

CAXIS

There is a mark for each channel. The mnemonics for each channel are the following.

CAXISC1 (can also be programmed as CAXIS)

CAXISC2 CAXISC3 CAXISC4

The CNC channel sets this mark to (=1) when the spindle is working as C axis. This mark is kept active while any of the functions #CAX, #FACE or #CYL are kept active.

FREE

There is a mark for each channel. The mnemonics for each channel are the following.

FREEC1 FREEC2 FREEC3 FREEC4

The CNC channel sets this signal high (=1) to indicate to the PLC that it is ready to accept a new block, sent using the CNCEX command.

WAITOUT

There is a mark for each channel. The mnemonics for each channel are the following.

WAITOUTC1 WAITOUTC2 WAITOUTC3 WAITOUTC4

It is applied to channel synchronization. The CNC channel sets this signal high (=1) to indicate to the PLC that it is waiting for a synchronization signal. Synchronization signals may be executed from the part-program using the #WAIT or #MEET instructions.

MMCWDG

This mark indicates the status of the operating system. Its value is (=0) when the CNC operating system is working properly and (=1) when the operating system is locked up.

Include this mark in the PLC maneuver to enable the emergencies when the operating system locks up.

RETRAEND

There is a mark for each channel. The mnemonics for each channel are the following.

RETRAENDC1 RETRAENDC2 RETRAENDC3 RETRAENDC4

The CNC channel sets this mark high (=1) to cancel the retrace function. The CNC channel cancels the retrace function in the following cases.

- The CNC channel has executed all the blocks stored for the retrace function.
- The CNC channel has reached the beginning of the program.
- The CNC channel has reached a block that cannot be executed with the retrace function.

In all these cases, the CNC channel sets its mark RETRAEND high (=1) to indicate to the PLC that all possible blocks have been executed and that it must set its RETRACE mark to "0". See "[RETRACE](#)" on page 424.

If the PLC automatically removes the RETRACE mark with the RETRAEND mark, the CNC goes on with the normal execution of the program, forward. Otherwise, if the RETRACE mark stays active, the CNC will issue a warning indicating that this mark must be deactivated in order to resume the execution of the program. The same occurs when trying to execute a new program after executing the retrace function.

The retrace function is also canceled after M30, with a reset or when the PLC sets the channel mark RETRACE low (=0)

6.

TANGACTIV

There is a mark for each channel. The mnemonics for each channel are the following.

TANGACTIVC1 TANGACTIVC2 TANGACTIVC3 TANGACTIVC4

The CNC sets this mark high (=1) to let the PLC know that a tangential control is active in the channel. This mark is not initialized when tangential control is frozen (suspended).

PSWSET

The CNC activates this mark when there is an OEM password.

DINDISTC1
DINDISTC2
DINDISTC3
DINDISTC4

The CNC has three marks per channel. The mnemonics for each channel are the following.

DINDISTC1C2	DINDISTC1C3	DINDISTC1C4
DINDISTC2C1	DINDISTC2C3	DINDISTC2C4
DINDISTC3C1	DINDISTC3C2	DINDISTC3C4
DINDISTC4C1	DINDISTC4C2	DINDISTC4C3

These marks are related to dynamic distribution of machining operations between channels (instruction #DINDIST). The option of synchronized identical passes uses these marks; it also uses the marks associated with synchronizing independent axes.

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. The channel of the CNC cancels the marks after finishing the withdrawal from the last roughing pass in the last channel. During the finishing operation, the CNC channel cancels all these marks.

The first channel indicated in the mnemonic refers to the channel that executes the cycle; the second one refers to the channel involved in the distribution of passes. For example:

- If the cycle is executed in channel ·1· and the rest of the passes are distributed with channel ·2·, the CNC activates the DINDISTC1C2 mark.
- If the cycle is executed in channel ·2· and the rest of the passes are distributed with channels ·1· and ·3·, the CNC activates marks DINDISTC2C1 and DINDISTC2C3.

Use these marks so interrupting the program, resuming it and executing it block by block in one of the channels affects all the channels involved in the distribution of passes.

SERPLCAC

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.

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.

OVERTEMP

This mark indicates the temperature status of the CNC. This mark is deactivated while room temperature is correct. The CNC checks every minute the unit temperature; if in three samples in a row the temperature exceeds 60 °C (140 °F), the CNC activates this mark and issues the warning about this situation. The CNC will deactivate this mark when the temperature of the unit goes below the maximum allowed (60 °C / 140 °F).

Every time [START] is pressed, the CNC checks that the room temperature does not exceed 65 °C (149 °F) and, if it does, inhibits the [START] and issues error E173. The CNC will let complete the execution in progress, but it will not allow to resume it after an interruption, a tool inspection, etc.

6.

LOGIC CNC INPUTS AND OUTPUTS.
General consulting signals.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Include this mark in the PLC routine to restrict the CNC operation in case of over-temperature.



If the PLC activates this mark, there is an over-temperature risk and the integrity of the unit is at risk. Turn the unit off to avoid possible damage.

MLINKRDY

In Mechatrolink configuration, this mark indicates the bus status. The CNC activates this mark to indicate that the Mechatrolink bus has been properly initialized.

All status consultations of the servos and inverters and the read/write (CNCRD/CNCWR) operations of their variables must be conditioned by this mark. The status of the servos and inverters may be consulted using the following variables.

V.A.MSTATUS.xn V.A.MSUBSTAT.xn V.A.MALARM.xn V.A.MIOMON.xn

RETRACT

There is a mark for each channel. The mnemonics for each channel are the following.

RETRACTC1 (can also be programmed as RETRACT)

RETRACTC2 RETRACTC3 RETRACTC4

The CNC uses this mark when it withdraws the axes from the part after interrupting a threading operation ([STOP] key or PLC mark _FEEDHOL). The CNC channel sets this signal high when it starts withdrawing the axes from the part and keeps it high until it completes the movement.

The withdrawal of the axes is enabled in parameter RETRACTTHREAD and it is applied to electronic threading (G33/G34) and the threading canned cycles of the -T- model, both ISO and conversational.

6.

LOGIC CNC INPUTS AND OUTPUTS.
General consulting signals.

FAGOR

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

6.2 Consulting signals for axes and spindles.

When the spindle is working in closed loop (M19 or G63), it behaves like an axis.

The signal names are generic. Replace the word (axis) with the spindle name or with the name or logic number of the axis.

For example, the name of the ENABLE(axis) mark for a machine with the X, Y, Z, Z2, B axes and spindle S.

ENABLEX, ENABLEZ2, ENABLEB, ENABLES

ENABLE3 for the Z axis.

ENABLE5 for the B axis.

ENABLE(axis)

The CNC sets this mark to (=1) to indicate to the PLC that it is going to move the corresponding axis or spindle in closed loop, so the PLC enables the axis or spindle if necessary. This mark stays active until the axis or spindle gets in position, even if the theoretical movement has not finished; i.e. until the CNC activates the INPOS(axis) mark.

This mark is also enabled in the movements of the independent axis, in spindle movements in closed loop (for example M19 or G74) or when a spindle goes into closed loop using the instruction #SERVO ON.

When the movement of an axis implies the movement of a slave axis, the ENABLE(axis) mark of the slave axis will also be enabled. For example, on gantry axes, slaved with #LINK, synchronized with #FOLLOW, electronic cams or spindles synchronized in closed loop with #SYNC even if the slave axis is in another channel.

DIR(axis)

The CNC sets this mark to (=1) to indicate that the axis is moving in the negative direction and to (=0) when moving in the positive direction. When the axis is stopped, it keeps the last value.

If the PLC stops the movement of the axes (mark _FEEDHOL=0) the ENABLE(axis) signal stays at (=1).

REFPOIN(axis)

This mark is set low (=0) on CNC power-up and is set high (=1) after searching home or after setting the machine coordinate (G174). The mark stays at (=1) until turning the CNC off.

This mark is set to (=0) when a feedback alarm occurs on the analog axes.

On axes and spindles without absolute feedback, the mark is set to (=0) in the following cases.

- When a home search fails.
- When parking the axis or spindle.
- On Sercos axes and spindles, if the ring fails.
- On spindles or rotary axes controlled as a spindle, when switching to open loop.
- On analog axes, when a feedback alarm occurs.

6.

LOGIC CNC INPUTS AND OUTPUTS.
Consulting signals for axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

DRSTAF(axis)
DRSTAS(axis)

The CNC uses these marks when communicating with the drive via Sercos® and they indicate the drive's status.

In case of error, both marks stay at (=1) if the axis is moving.

Normal.

- 1 After actuating the main switch at the electrical cabinet, the drive is supplied with 24 Vdc.
- 2 The drive runs an internal test. If OK, it activates the SYSTEM OK output. From that instant on, apply power to the Power Supply.

Marks DRSTAF(*)=0 DRSTAS(*)=1

- 3 When there is power at the bus, the drive is ready to provide torque. To do that, activate the Drive enable and Speed enable inputs.

Marks DRSTAF(*)=1 DRSTAS(*)=0

- 4 Once the Drive enable and Speed enable inputs are activated, the drive is running properly.

Marks DRSTAF(*)=1 DRSTAS(*)=1

Mark consultation.

When consulting the DRSTAF(*) and DRSTAS(*) marks, the following values may result:

DRSTAF(*)=0 DRSTAS(*)=0

The drive is in error or it does not exist.

DRSTAF(*)=0 DRSTAS(*)=1

The DC bus has no power. The drive cannot be enabled; but it is possible to provide power to the drives' power supply.

DRSTAF(*)=1 DRSTAS(*)=0

The drive's DC bus now has power. The drive may be enabled.

DRSTAF(*)=1 DRSTAS(*)=1

The drive is enabled.

INPOS(axis)

The CNC sets this mark high (=1) when the relevant axis or spindle is in position and there is no request for motion (ENABLE(axis) mark deactivated). An axis is in position when it stays within the in-position zone (window) (parameter INPOSW) for a time period indicated by parameter INPOSTIME.

There is an INPOS(axis) mark for each axis and for the spindle and a general INPOS1 mark for the channel that indicates whether all axes and the spindle have reached their position or not, except the independent axes programmed from the PLC. See "*INPOS1*" on page 393.

LUBR(axis)

The CNC sets this mark high (=1) when the relevant axis or spindle needs to be lubricated. See "*LUBRENA(axis) LUBROK(axis)*" on page 431.

(a.m.p.) DISTLUBRI indicates the distance to be moved before being lubricated.

HIRTHON(axis)

This mark is related to Hirth axes. A Hirth axis is a rotary axis that can only be positioned at specific positions, every so many degrees.

The HIRTHON(axis) mark indicates whether it is working as a Hirth axis (=1) or as a regular rotary or linear axis (=0) without having to position at specific positions.

Both linear and rotary axes can be Hirth axes. (a.m.p.) HIRTH indicates whether the axis can work as Hirth axis or not. Functions G170 and G171 indicate whether it is a Hirth axis (G171, by default) or a normal linear or rotary axis (G170).

6.

LOGIC CNC INPUTS AND OUTPUTS.

Consulting signals for axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

MATCH(axis)

The MATCH(axis) mark indicates whether the Hirth axis is properly positioned (=1) or not (=0). A Hirth axis is a rotary axis that can only be positioned at specific positions, every so many degrees.

**PARK(axis)
UNPARK(axis)**

The CNC sets mark PARK(axis) to (=1) to indicate that an axis or spindle is being parked and mark UNPARK(axis) to (=1) to indicate that an axis is being unparked. The mark PARKED(axis) is also used, it may be modified and is described later on. See "*PARKED(axis)*" on page 431.

When parking an axis or spindle, the CNC will not control the axis (it ignores the drive signals, feedback systems, etc.) because, it interprets that the axis is not present in the new machine configuration. When unparking an axis, the CNC will control the axis again because it interprets that the axis is present again in the new machine configuration.

The axes may be parked and unparked from the CNC or from the PLC.

Application example.

There are machines that, depending on the type of machining, may have 2 different axis configurations. For example, a machine that interchanges a regular spindle with an orthogonal one may have the following configurations:

- With a normal spindle, X Y Z axes configuration.
- With an orthogonal spindle, X Y Z A B axes configuration.

When working with a regular spindle, the A and B axes are not present and the CNC issues an error because it takes them into account (drives, feedback systems, etc.). The A and B axes must be parked to avoid this.

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.

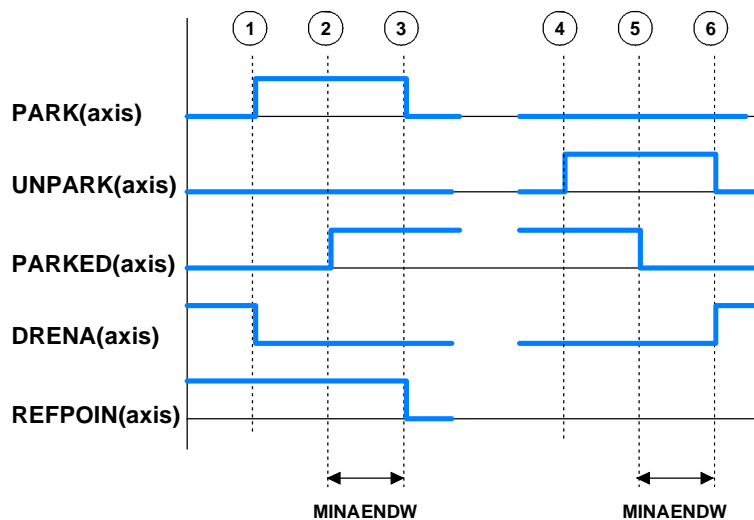
Park/Unpark from the CNC.

This type of maneuver is good for applications that need the axes or spindles to be parked automatically from a part program (for example from a part-program). The parking/unparking maneuver via part-program or MDI is controlled using the programming instructions #PARK and #UNPARK.

6.

The CNC uses the PARK and UNPARK signals to inform the PLC that the parking or unparking process has been engaged.

To park an axis or a spindle, its enable signals must be (=0). Likewise, after unparking the axis, the axis enable signals must be set to (=1).



Maneuver to park an axis or spindle from the CNC.

- 1 When executing the programming instruction #PARK the CNC checks whether the requested axis can be parked or not. If so, the CNC sets the PARK mark high (=1) to let the PLC know that it must park the corresponding axis.

For digital axes, the PLC must first remove the enable of the drive of the axis to be parked (DRENA). Also, the CNC sends to the drive the command to park the axis.

- 2 The PLC, after receiving the PARK signal, parks the requested axis. After verifying that the axis has been parked (presence sensors) the PLC sets mark PARKED to (=1).
- 3 The CNC will recognize that the axis has been parked when detecting that the PARKED signal has been activated. The PARK and REFPOIN signals are reset completing this process.

Maneuver to unpark an axis or spindle from the CNC.

- 4 When executing the programming instruction #UNPARK, the CNC sets the UNPARK mark high (=1) to let the PLC know that it must unpark the corresponding axis.

For digital axes, the CNC sends to the drive the command to unpark the axis.

- 5 The PLC, after receiving the UNPARK signal, unparks the requested axis. After verifying that the axis has been unparked (presence sensors) the PLC sets mark PARKED to (=0).
- 6 The CNC will recognize that the process has been completed when detecting that the PARKED signal has been deactivated. The UNPARK and REFPOIN signals are reset.

For digital axes, the PLC must activate the enable of the drive of the axis (DRENA).

Example for parking and unparking an axis:

Input I15 corresponds to the presence sensor of the "B" axis

PARKB AND NOT I15 = SET PARKEDB

If there is a request to park the "B" axis (PARKB) and the axis is not present (NOT I15), the axis is parked (SET PARKEDB).

UNPARKB AND I15 = RES PARKEDB

If there is a request to unpark the "B" axis (UNPARKB) and the axis is not present (I15), the axis is unparked (RES PARKEDB).

NOT (PARKB OR UNPARKB OR PARKEDB) AND... = DRENAB = SPENAB = SERVOBON

If the axis is neither parked nor being parked and the enabling conditions are met, the axis gets enabled.

6.

LOGIC CNC INPUTS AND OUTPUTS.

Consulting signals for axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Park/Unpark from the PLC.

This type of maneuver is good for applications that need manual parking of the axes either while the machine is off or on (with or without power).

The axis parking/unparking maneuver, handled from the PLC program, is controlled using the PARKED signal. This mark is usually affected by the input for the axis presence sensor. The state of this signal is maintained even if the CNC is turned off.

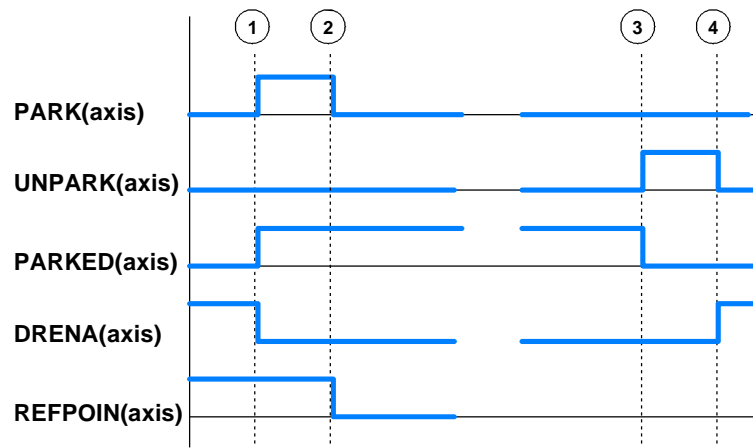
The CNC uses logic signals PARK and UNPARK to inform the PLC that the parking or unparking processes are in execution.

To park an axis, the axis enable signals must be (=0). Likewise, after unparking the axis, the axis enable signals must be set to (=1).

For safety, after parking and unparking an axis, the REFPOIN signal of the axis is set to (=0).



In versions older than V2.00, at the end of the parking or unparking maneuver, the PLC must generate a reset (RESETIN).



Maneuver to park an axis or spindle from the PLC.

- 1 From the PLC program, the PARKED mark is set to (=1) to park the relevant axis. The CNC sets the PARK mark to (=1) and begins parking the axis.

For digital axes, the PLC must first remove the enable of the drive of the axis to be parked (DRENA). Also, the CNC sends to the drive the command to park the axis.

- 2 The CNC considers the operation completed. It resets the PARK signal.

Maneuver to unpark an axis or spindle from the PLC.

- 3 From the PLC program, the PARKED mark is set to (=0) to unpark the relevant axis. The CNC sets the UNPARK mark to (=1) and begins unparking the axis.

For digital axes, the CNC sends to the drive the command to unpark the axis.

- 4 The CNC considers the operation completed. It resets the UNPARK signal.

For digital axes, the PLC must activate the enable of the drive of the axis (DRENA).

Example for parking and unparking an axis:

```
I10 = PARKEDV
```

```
Axis present. "V" axis presence sensor
```

```
NOT (PARKV OR UNPARKV OR PARKEDV) AND... = DRENAV = SPENAV = SERVOVON
```

If the axis is neither parked nor being parked and the enabling conditions are met, the axis gets enabled.

6.

LOGIC CNC INPUTS AND OUTPUTS.
Consulting signals for axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

ACTFBACK(axis)

The CNC uses this mark when the system has external+internal feedback. See machine parameter FBACKSRC.

The CNC sets this mark high (=1) to indicate that the external feedback (direct feedback) is being used . The CNC sets this mark high (=0) to indicate that the internal feedback (motor feedback) is being used .

The feedback type may be swapped from the PLC using the mark FBACKSEL(axis). See "[FBACKSEL\(axis\)](#)" on page 431.

TANGACT(axis)

The CNC sets this mark high (=1) to indicate that tangential control is active in the axis. The CNC sets this mark high (=0) when freezing (suspending) or canceling the tangential control of the axis.

LOPEN(axis)

The CNC activates this mark to inform the PLC that the axis position loop is open.

MAXDIFF(axis)

This mark is used on gantry axes. The CNC activates this mark when it cannot correct the position difference between the master axis and the slave axis because the difference is greater than the value set in machine parameter MAXDIFF. Position correction must be enabled in machine parameter DIFFCOMP.

6.

LOGIC CNC INPUTS AND OUTPUTS.

Consulting signals for axes and spindles.

6.3 Consulting signals for the spindle.

6.

LOGIC CNC INPUTS AND OUTPUTS.
Consulting signals for the spindle.

REVOK

There is a mark for each spindle. The mnemonics for each spindle are the following:

REVOK1 (can also be programmed as REVOK)
REVOK2 REVOK3 REVOK4

It indicates whether the actual (real) spindle rpm match the ones programmed (=1) or not (=0). In other words, whether they are within the percentages set by (a.m.p.) UPSPDLIM and LOSPDLIM.

- When the spindle is stopped, M5, REVOK is (=1).
- With M3 and M4, the CNC sets this mark high (=1) when the actual spindle rpm match the ones programmed.
- When working in closed loop (M19 or G63), the CNC sets this mark low (=0) while the spindle is moving and high (=1) when the spindle is in position.

The REVOK signal may be used to handle the Feedhold signal and avoid machining at lower or higher rpm than desired.

SYNCMaster

There is a mark for each spindle. The mnemonics for each spindle are the following:

SYNCMaster1 SYNCMaster2 SYNCMaster3 SYNCMaster4

This mark is activated at the master spindle and indicates that a synchronization has begun with the #SYNC instruction. When activating a synchronization, the CNC activates the ENABLE signal at both spindles and waits for the SERVOON signal (if DWELL is used).

When a spindle synchronization is active, the PLCCNTL, INHIBIT and SPDLEREV signals of both master and slave are ignored. Likewise, while threading, only the feedback and reference signal of the main spindle are taken into account.

SYNCHRON

There is a mark for each spindle. The mnemonics for each spindle are the following:

SYNCHRON1 SYNCHRON2 SYNCHRON3 SYNCHRON4

This mark is activated at the slave spindle and indicates that a synchronization has begun with the #SYNC instruction. When activating a synchronization, the CNC activates the ENABLE signal at both spindles and waits for the SERVOON signal (if DWELL is used).

When a spindle synchronization is active, the PLCCNTL, INHIBIT and SPDLEREV signals of both master and slave are ignored. Likewise, while threading, only the feedback and reference signal of the main spindle are taken into account.

SYNCHRONP

There is a mark for each spindle. The mnemonics for each spindle are the following:

SYNCHRONP1 SYNCHRONP2 SYNCHRONP3 SYNCHRONP4

This mark is activated at the slave spindle and indicates that a position synchronization has begun. This mark may be used to distinguish between synchronization in position or in velocity and to know which mark, SYNSPEED or SYNCPOS, to attend to.

SYNSPEED

There is a mark for each spindle. The mnemonics for each spindle are the following:

SYNSPEED1 SYNSPEED2 SYNSPEED3 SYNSPEED4

This mark is activated at the slave spindle when it is synchronized in speed.

This mark is set to (=0) when exceeding the maximum speed error allowed, whose default value is set by machine parameter DSYNCVELW.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

SYNCPOSI

There is a mark for each spindle. The mnemonics for each spindle are the following:

SYNCPOSI1 SYNCPOSI2 SYNCPOSI3 SYNCPOSI4

This mark is activated at the slave spindle when it is synchronized in position.

This mark is set to (=0) when exceeding the maximum position error allowed, whose default value is set by machine parameter DSYNCPOSW.

GEAROK

There is a mark for each spindle. The mnemonics for each channel are the following.

GEAROK1 (can also be programmed as GEAROK)

GEAROK2 GEAROK3 GEAROK4

The spindle activates this mark when the parameter set selected at the CNC and at the PLC are the same. In order for both parameter sets to coincide, function M41 must be active at the CNC and the GEAR1 mark at the PLC, M42 with GEAR2 and so on.

If both parameter sets do not coincide, the CNC executes no action. Include this mark in the PLC maneuver to define the actions to carry out when both parameter sets do not coincide, such as stop the spindle or interrupt the execution of the part-program.

6.

LOGIC CNC INPUTS AND OUTPUTS.

Consulting signals for the spindle.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

6.4 Consultation signals of the independent interpolator.

The signal names are generic. Replace the text (axis) with the name or logic axis number.

IBUSY(axis)

For movements of the independent axis, the independent interpolator of the axis activates this mark when it has an instruction pending to execute.

IFREE(axis)

For movements of the independent axis, the independent interpolator of the axis activates this mark when it is ready to accept a new motion block. The axis interpolator can activate this mark even if there is block in execution, so it can join both blocks at the joining feedrate indicated in the first block.

IFHOUT(axis)

For movements of the independent axis, the independent interpolator of the axis activates this mark when the execution has been interrupted.

IEND(axis)

For movements of the independent axis, the independent interpolator of the axis activates this mark when it finishes generating the theoretical movement.

INSYNC(axis)

For synchronization movements of an independent axis and of an electronic cam, the interpolator of the axis activates this mark when synchronism has been reached. This mark stays active while maintaining synchronism.

PROBE1ACTIVE PROBE2ACTIVE

It is applied when latching the coordinate of an axis. There is a mark for each probe. Probe 1 will be the one assigned to parameter PRBDI1 and probe 2 will be the one assigned to parameter PRBDI2.

The independent interpolator activates this mark when a latching process is active with the indicated probe and it deactivates when there is no active latching process with the indicated probe.

LATCH1ACTIVE(axis) LATCH2ACTIVE(axis)

It is applied when latching the coordinate of an axis. There is a mark for each probe. Probe 1 will be the one assigned to parameter PRBDI1 and probe 2 will be the one assigned to parameter PRBDI2.

The independent interpolator activates this mark when activating a latching process on the axis with the indicated probe and it deactivates when the latching process ends or it is canceled.

The latching process on the axis is canceled when the PLC activates the IRESET(axis) mark. A reset of the channel cancels the latching processes on all axes of the channel.

Functions M02 and M30 will not be considered executed (done) until all the latching processes active on the axes of the channel are finished.

6.

LOGIC CNC INPUTS AND OUTPUTS.
Consultation signals of the independent interpolator.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

LATCH1DONE(axis)
LATCH2DONE(axis)

It is applied when latching the coordinate of an axis. There is a mark for each probe. Probe 1 will be the one assigned to parameter PRBDI1 and probe 2 will be the one assigned to parameter PRBDI2.

The independent interpolator activates this mark when the latch event on the axis with the indicated probe occurs; it deactivates it when activating a new latching process on the axis with the same probe.

6.

LOGIC CNC INPUTS AND OUTPUTS.

Consultation signals of the independent interpolator.

6.5 Consulting logic signals; laser.

ACTIVEMATERIALON

The CNC activates this mark when there is an active material file.

PIERCING

Piercing enabled. The OEM must manage this variable in the subroutine associated with piercing (by default, Piercing.fst), to indicate to the PLC when to start (value 1) and end (value 0) the piercing operation. The instruction #PLC makes it possible to manage a PLC mark from the part or subroutine program, without interrupting the block preparation.

This mark shows the state of the variable V.PLC.PIERCING. The OEM can also use this variable in the subroutine associated with the piercing (by default, Piercing.fst) to modify the status of this mark. Entering this variable interrupts the block preparation.

CUTTING

Active cutting. The OEM must manage this mark in the subroutines associated with cutting (by default, Cutting.fst / Cuttingoff.fst), to indicate to the PLC when to begin (value 1) and end (value 0) the cutting operation. The instruction #PLC makes it possible to manage a PLC mark from the part or subroutine program, without interrupting the block preparation.

This mark shows the state of the variable V.PLC.CUTTING. The OEM can also use this variable in the subroutines associated with cutting (by default, Cuttingon.fst / Cuttingoff.fst) to modify the status of this mark. Entering this variable interrupts the block preparation.

COMVARACT

The CNC activates this mark when the user validates the common parameters of the active piercing or cutting table. The PLC should disable this mark when management with these variables is complete.

CUTVARACT

The CNC activates this mark when the user validates the cutting parameters of the active table. The PLC should disable this mark when management with these variables is complete.

When the subroutine Cuttingon.fst changes only the cutting type (#CUTTING ON), the CNC does not activate this mark, as the synchronization with the PLC is implemented through M functions.

The subroutine Cuttingon.fst (or the one associated with the cutting process) performs a series of operations with the variables associated with the technological tables, operations that may also be required when the user modifies the table directly (which means modifying the variables). The PLC manages the effects of these variables on the machine devices (analog inputs, digital inputs, etc).

PIRVARACT

The CNC activates this mark when the user validates the piercing parameters of the active table. The PLC should disable this mark when management with these variables is complete.

When the subroutine Piercing.fst changes only the piercing type (#PIERCING), the CNC does not activate this mark, as the synchronization with the PLC is implemented through M functions.

The subroutine Piercing.fst (or the one associated with the piercing process) performs a series of operations with the variables associated with the technological tables, operations that may also be required when the user modifies the table directly (which means modifying the variables). The PLC manages the effects of these variables on the machine devices (analog inputs, digital inputs, etc).

6.

INPOSGAP

The CNC activates this mark if the gap is within the range defined by the parameters GAPMIN-GAPMAX. If the gap exceeds the range set by these parameters, the CNC deactivates the INPOSGAP mark. The GAPERRORCANCEL parameter establishes the CNC behavior when the gap exceeds the GAPMIN - GAPMAX range.

GAPERRORCANCEL	Meaning.
Yes.	The CNC cancels the out-of-range gap error and does not stop the movement of the axes.
No	The CNC returns an out-of-range gap error and stops the movement of the axes, according to the braking ramp and controlling the gap during the ramp.

INTOL

The CNC activates this mark if the gap is within the tolerance limit defined by the parameter GAPTOL regarding the gap value programmed. If the axis exceeds the tolerance set in the GAPTOL parameter, the CNC deactivates the INTOL mark. The GAPTOLCANCEL parameter establishes the CNC behavior when the gap exceeds the tolerance set in GAPTOL.

GAPTOLCANCEL	Meaning.
Yes.	The CNC cancels the out-of-range gap error and does not stop the movement of the axes.
No	The CNC returns an out-of-range gap error and stops the movement of the axes, according to the braking ramp and controlling the gap during the ramp.

INPOSLIMIT

The CNC activates this mark (changes from 0 to 1) when the leap programmed with a #LEAP reaches the highest point (POSLIMIT command). The CNC deactivates this mark (changes from 1 to 0) when the leap starts to lower from this point. If the leap does not reach the highest point due to lack of space, the CNC will not activate this mark. In the event of any error, the CNC will deactivate this mark.

6.

LOGIC CNC INPUTS AND OUTPUTS.

Consulting logic signals: laser.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

6.6 Tool manager consulting signals.

6.

LOGIC CNC INPUTS AND OUTPUTS.
Tool manager consulting signals.

TMOPERATION

There is a register for each channel. The mnemonics for each channel are the following.

TMOPERATIONC1 (same as TMOPERATION)
TMOPERATIONC2 TMOPERATIONC3 TMOPERATIONC4

This register indicates the type of operation to be carried out by the tool manager.

TMOPSTROBE

There is a mark for each channel. The mnemonics for each channel are the following.

TMOPSTROBEC1 (same as TMOPSTROBE)
TMOPSTROBEC2 TMOPSTROBEC3 TMOPSTROBEC4

The CNC sets this mark high (=1) to let the PLC know that it must execute the operation indicated by TMOPERATION.

LEAVEPOS

There is a register for each magazine. The mnemonics for each magazine are the following.

LEAVEPOSMZ1 (can also be programmed as LEAVEPOS)
LEAVEPOSMZ2 LEAVEPOSMZ3 LEAVEPOSMZ4

This register indicates the magazine position to leave the tool. While selecting a turret position (#ROTATEMZ instruction), this register takes the value of ·0· if it is a positive relative positioning and the value of ·1· if it is a negative relative positioning.

TAKEPOS

There is a register for each magazine. The mnemonics for each magazine are the following.

TAKEPOSMZ1 (can also be programmed as TAKEPOS)
TAKEPOSMZ2 TAKEPOSMZ3 TAKEPOSMZ4

This register indicates the magazine position of the tool to be taken. While selecting a turret position (#ROTATEMZ instruction), this register takes the value of ·0· if it is an absolute positioning and the value of ·1· if it is a relative positioning.

NEXTPOS

There is a register for each magazine. The mnemonics for each magazine are the following.

NEXTPOSMZ1 (can also be programmed as NEXTPOS)
NEXTPOSMZ2 NEXTPOSMZ3 NEXTPOSMZ4

This register indicates the magazine position occupied by the next tool. While selecting a turret position (#ROTATEMZ instruction); in an absolute positioning, this register indicates the position to reach and, in a relative positioning, the number of positions to rotate.

TWORNOUT

There is a mark for each channel. The mnemonics for each channel are the following.

TWORNOUTC1 (can also be programmed as TWORNOUT)
TWORNOUTC2 TWORNOUTC3 TWORNOUTC4

The CNC channel sets this mark high (=1) to "tell" the PLC that the tool has been rejected because it is worn out (real life > maximum life span).



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TMINEM

There is a register for each magazine. The mnemonics for each magazine are the following.

TMINEMZ1 (can also be programmed as TMINEM)

TMINEMZ2 TMINEMZ3 TMINEMZ4

The CNC sets this mark high (=1) to inform the PLC that an emergency has occurred at the tool manager.

MZID

There is a register for each channel. The mnemonics for each channel are the following.

MZIDC1 MZIDC2 MZIDC3 MZIDC4

This register indicates which magazine contains the tool requested by the channel.

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.

6.

LOGIC CNC INPUTS AND OUTPUTS.

Tool manager consulting signals.

6.7 Keystroke consulting signals.

KEYBD1 / KEYBD2

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). If there is only one keyboard, these registers coincide with KEYBD1_1 to KEYBD3_1. When there are several keyboards, the contents of these registers are not always the same as KEYBD1_1 to KEYBD3_1; therefore, they may be used indistinctively.

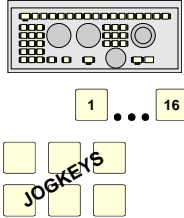
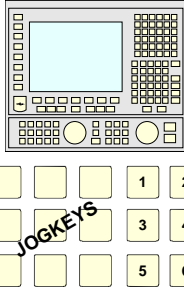
KEYBD1_1 / KEYBD2_1

..

KEYBD1_8 / KEYBD2_8

These registers indicate (bit=1) which key has been pressed on each operator panel. Registers KEYBD1_1 to KEYBD3_1 correspond to the first jog panel, KEYBD1_2 to KEYBD3_2 to the second one and so on.

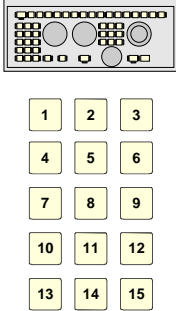
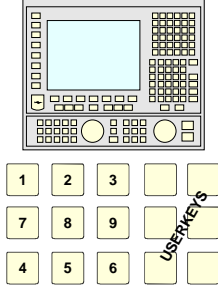
Registers KEYBD1 / KEYBD1_1 to KEYBD1_8. User keys.

Bit	OP-PANEL	LCD-10K
		
0	User key 1	User key 1
1	User key 2	User key 2
2	User key 3	User key 3
3	User key 4	User key 4
4	User key 5	User key 5
5	User key 6	User key 6
6	User key 7	---
7	User key 8	---
8	User key 9	---
9	User key 10	---
10	User key 11	---
11	User key 12	---
12	User key 13	---
13	User key 14	---
14	User key 15	---
15	User key 16	---

6.

LOGIC CNC INPUTS AND OUTPUTS.
Keystroke consulting signals.

Registers KEYBD1 / KEYBD1_1 to KEYBD1_8. Jog keys.

Bit	OP-PANEL	LCD-10K
		
16	Jog key 1	Jog key 1
17	Jog key 2	Jog key 2
18	Jog key 3	Jog key 3
19	Jog key 4	Jog key 4
20	Jog key 5	Jog key 5
21	Jog key 6	Jog key 6
22	Jog key 7	Jog key 7
23	Jog key 8	Jog key 8
24	Jog key 9	Jog key 9
25	Jog key 10	---
26	Jog key 11	---
27	Jog key 12	---
28	Jog key 13	---
29	Jog key 14	---
30	Jog key 15	---
31	----	---

Registers KEYBD2 / KEYBD2_1 to KEYBD2_8.

Bit	Key	Bit	Selector
0	Spindle override +	10	Generic key 3
1	Spindle clockwise	11	ZERO
2	Spindle positioning	12	---
3	Spindle stop	13	Single block
4	Spindle override -	14	CNC OFF
5	Spindle counterclockwise	15	RESET
6	START	16 - 20	Feed override
7	STOP	21 - 23	---
8	Generic key 1	24 - 27	Mode selector
9	Generic key 2	28 - 31	ZERO

When the operator panel has a spindle speed override switch, the keys associated with the speed override (bits 0 and 4) no longer have this functionality and may be configured from the PLC.

6.

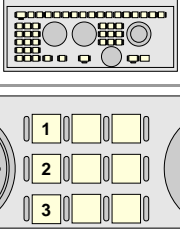
LOGIC CNC INPUTS AND OUTPUTS.
Keystroke consulting signals.



CNC 8070

(REF: 1709)

Registers KEYBD2 / KEYBD2_1 to KEYBD2_8. Generic keys.

Bit	OP-PANEL
	
8	Generic key 1
9	Generic key 2
10	Generic key 3

Registers KEYBD2 / KEYBD2_1 to KEYBD2_8. Feedrate override selector

KEYBD2					
20	19	18	17	16	
0	0	0	0	0	0 %
0	0	0	0	1	2 %
0	0	0	1	0	4 %
0	0	0	1	1	10 %
0	0	1	0	0	20 %
0	0	1	0	1	30 %
0	0	1	1	0	40 %
0	0	1	1	1	50 %
0	1	0	0	0	60 %
0	1	0	0	1	70 %
0	1	0	1	0	80 %
0	1	0	1	1	90 %
0	1	1	0	0	100 %
0	1	1	0	1	110 %
0	1	1	1	0	120 %
0	1	1	1	1	130 %
1	0	0	0	0	140 %
1	0	0	0	1	150 %
1	0	0	1	0	160 %
1	0	0	1	1	170 %
1	0	1	0	0	180 %
1	0	1	0	1	190 %
1	0	1	1	0	200 %

6.

LOGIC CNC INPUTS AND OUTPUTS.
Keystroke consulting signals.

Registers KEYBD2 / KEYBD2_1 to KEYBD2_8. Movement selector (handwheel, incremental jog or continuous jog).

KEYBD2				
27	26	25	24	
0	0	0	0	Handwheel x100
0	0	0	1	Handwheel x10
0	0	1	0	Handwheel x1
0	0	1	1	Jog 1
0	1	0	0	Jog 10
0	1	0	1	Jog 100
0	1	1	0	Jog 1000
0	1	1	1	Jog 10000
1	0	0	0	Continuous jog

Registers KEYBD3 / KEYBD3_1 to KEYBD3_8. Spindle speed override selector.

KEYBD3					
4	3	2	1	0	
0	0	0	0	0	0 %
0	0	0	0	1	2 %
0	0	0	1	0	4 %
0	0	0	1	1	10 %
0	0	1	0	0	20 %
0	0	1	0	1	30 %
0	0	1	1	0	40 %
0	0	1	1	1	50 %
0	1	0	0	0	60 %
0	1	0	0	1	70 %
0	1	0	1	0	80 %
0	1	0	1	1	90 %
0	1	1	0	0	100 %
0	1	1	0	1	110 %
0	1	1	1	0	120 %
0	1	1	1	1	130 %
1	0	0	0	0	140 %
1	0	0	0	1	150 %
1	0	0	1	0	160 %
1	0	0	1	1	170 %
1	0	1	0	0	180 %
1	0	1	0	1	190 %
1	0	1	1	0	200 %

6.

LOGIC CNC INPUTS AND OUTPUTS.

Keystroke consulting signals.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

6.8 General modifiable signals.



The `_EMERGEN`, `_STOP`, `_FEEDHOL`, `_XFERINH` and `CYSTART` signals must be defined in the PLC program.

`_EMERGEN`

There is a mark for each channel. The mnemonics for each channel are the following.

`_EMERGENC1` (can also be programmed as `_EMERGEN`)
`_EMERGENC2` `_EMERGENC3` `_EMERGENC4`

If the PLC sets this mark low (`=0`), the CNC stops the axes and the spindle and displays the corresponding error message. The CNC interrupts the movement of the axes leaving them without command; the axes do not stop using a set acceleration ramp.

While the `_EMERGEN` mark is low (`=0`), the CNC does not allow executing programs and aborts any axis or spindle movement.

`I-EMERG AND` (rest of conditions) = `_EMERGEN`

If the emergency-stop button is pressed (`I-EMERG=0`) or any other emergency situation occurs (`=0`), the `_EMERGEN` mark is set low (`=0`) causing an emergency at the CNC.

`_STOP`

There is a mark for each channel. The mnemonics for each channel are the following.

`_STOPC1` (can also be programmed as `_STOP`)
`_STOPC2` `_STOPC3` `_STOPC4`

If the PLC sets this mark at a low logic level, the CNC channel interrupts the execution of the part program. To resume the execution of the program, besides setting this mark at a high logical level, the PLC should activate the `CYSTART` mark. See "[CYSTART](#)" on page 419.

- This mark does not stop the spindle.
- This mark does not affect the independent movement of an axis (independent interpolator), which also does not affect the key [STOP].
- This mark does not turn off the laser.
- This mark does not disable gap control, except in approach movement. In this case, the axis will remain stationary and without gap control until the movement is resumed.

The treatment received by the signal `_STOP` is similar to that given to the [STOP] key in the CNC. With this signal at a low logic level, all keys remain enabled.

`_XFERINH`

There is a mark for each channel. The mnemonics for each channel are the following.

`_XFERINHC1` (can also be programmed as `_XFERINH`)
`_XFERINHC2` `_XFERINHC3` `_XFERINHC4`

If the PLC sets this mark low (`=0`), the CNC channel interrupts the execution of the program at the end of the block being executed and inhibits the execution of the next block. If the axis needs more braking distance than it has with the block being executed, the CNC can continue executing more blocks until the axis comes to a complete stop respecting the dynamics of the machine. When this mark is set back high (`=1`), the CNC resumes the execution of the program.

Having this mark (`=0`), the CNC does not allow jog movements for the axes of the channel; pressing a jog key is ignored.

6.

LOGIC CNC INPUTS AND OUTPUTS.
General modifiable signals.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

This mark always affects the movements of the independent axis programmed from the CNC; the ones programmed from the PLC depend on parameter XFITOIND. If the PLC deactivates the `_XFERINH` mark, the CNC channel interrupts the movements of the independent axis when they reach position and inhibits the execution of the next movement.

XFITOIND	Meaning.
Yes	The <code>_XFERINH</code> mark affects the movement of the independent axis programmed from the PLC and from the CNC.
No	The <code>_XFERINH</code> mark does not affect the movements of the independent axis programmed from the PLC; but it does affect the ones programmed from the CNC.

To manage the transfer inhibit in an independent movement, the PLC also has a particular mark per axis (`_IXFERINH(axis)` mark). See "`_IXFERINH(axis)`" on page 439.

`_FEEDHOL`

There is a mark for each channel. The mnemonics for each channel are the following.

`_FEEDHOLC1` (can also be programmed as `_FEEDHOL`)

`_FEEDHOLC2` `_FEEDHOLC3` `_FEEDHOLC4`

If the PLC sets this mark at a high logic level, the CNC channel momentarily interrupts the movement of the axes. When the mark returns to the high logic level, the movement of the axes continues. All the stops and starts of the axes are carried out with the corresponding acceleration and deceleration.

- This mark does not stop the spindle.
- This mark does not affect the independent movement of an axis (independent interpolator).
- On hirth axes, if the axis does not stop in a particular position, the CNC does not activate the `MATCH(axis)` mark.
- This mark does not turn off the laser. If the power control is active, the laser removes the minimum defined power.

If the PLC sets the `_FEEDHOL` at a low logic level in a motionless block, the CNC continues executing the program until it detects a motion block.

The "Freal" text of the screens of the automatic and jog modes shows the status of this mark. The text appears in red when the `_FEEDHOL` mark is active. The status of the mark is not displayed if the screen does not show this text.

`CYSTART`

There is a mark for each channel. The mnemonics for each channel are the following.

`CYSTARTC1` (can also be programmed as `CYSTART`)

`CYSTARTC2` `CYSTARTC3` `CYSTARTC4`

When the operator presses the [START] key, the CNC lets the PLC know by activating the START mark. If the rest of the conditions are met (hydraulic, safety, etc.), the PLC must activate the `CYSTART` mark in order for the program to start running.

START AND (rest of conditions) = `CYSTART`

`SBLOCK`

There is a mark for each channel. The mnemonics for each channel are the following.

`SBLOCKC1` (can also be programmed as `SBLOCK`)

`SBLOCKC2` `SBLOCKC3` `SBLOCKC4`

If the PLC sets this mark high (=1), the CNC switches to SINGLE BLOCK operation mode.

MANRAPID

There is a mark for each channel. The mnemonics for each channel are the following.

MANRAPIDC1 (can also be programmed as MANRAPID)

MANRAPIDC2 MANRAPIDC3 MANRAPIDC4

This mark is treated in a similar way to the Rapid traverse key. If the PLC turns this mark on, the CNC selects the rapid traverse for continuous jog movements. The incremental jog moves are carried out at the active feedrate in jog mode. When the PLC turns this mark off, all jog moves are carried out at the active feedrate in jog mode.

OVRCAN

There is a mark for each channel. The mnemonics for each channel are the following.

OVRCANC1 (can also be programmed as OVRCAN)

OVRCANC2 OVRCANC3 OVRCANC4

If the PLC sets this mark high (=1), the CNC applies 100 % override to the axis feedrate regardless of the value currently selected. While the OVRCAN mark is high (=1), the CNC channel will apply 100% of the feedrate corresponding to each work mode.

RESETIN

There is a mark for each channel. The mnemonics for each channel are the following.

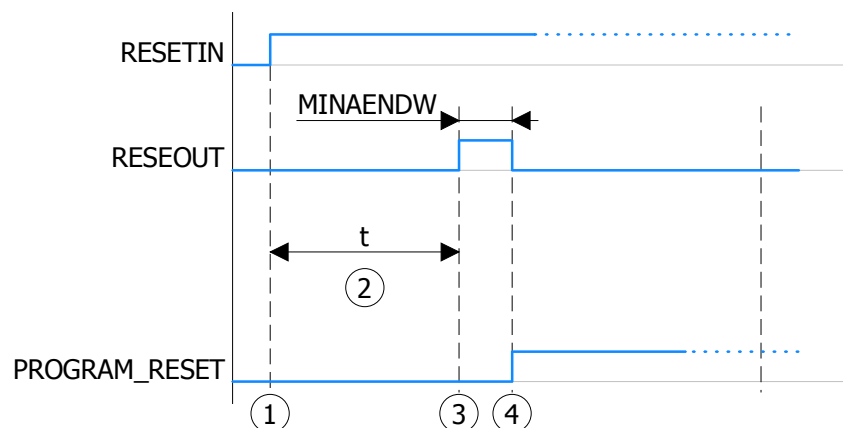
RESETINC1 (can also be programmed as RESETIN)

RESETINC2 RESETINC3 RESETINC4

Resetting the CNC. The treatment of this mark is similar to that of the [RESET] key. With an up-flank (change from 0 to 1) of this mark, the CNC channel is reset. After the reset has commenced, the PLC can deactivate this mark (RESETIN = 0). The reset process is as follows.

- 1 The user presses the [RESET] key on the operator panel or the PLC activates the RESETIN mark.
- 2 The channel assumes the initial conditions (those defined by the machine parameters).
- 3 The channel activates the RESEOUT (RESEOUT=1) mark to indicate that the reset has finalized. This mark stays active for the time period set in general parameter MINAENDW. See "[RESEOUT](#)" on page 390.
- 4 The channel executes the subroutine associated with the reset (PROGRAM_RESET), should there be one.

Reset process using RESETIN.



- (1) The PLC activates the RESETIN mark.
- (2) Time to assume the initial conditions.
- (3) The channel activates the RESEOUT mark.
This mark stays active for the time period set in general parameter MINAENDW.
- (4) End of reset.
The channel deactivates the RESEOUT mark.
The channel executes the subroutine associated with the reset (PROGRAM_RESET).
The RESETIN mark can be deactivated at any time.

6.

LOGIC CNC INPUTS AND OUTPUTS.
General modifiable signals.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

LATCHM

With this mark, it is possible to select how the JOG keys will work in JOG mode.

- If this mark is low (=0), the axes will move while the corresponding JOG key kept pressed.
- If it is high (=1), 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).

BLKSKIP1

There is a mark for each channel. The mnemonics for each channel are the following.

BLKSKIP1C1 (can also be programmed as BLKSKIP1)
 BLKSKIP1C2 BLKSKIP1C3 BLKSKIP1C4

The PLC sets this mark high (=1) to let the CNC channel know that the block skip condition "/" is met. Therefore, blocks with that condition will not be executed.

M01STOP

There is a mark for each channel. The mnemonics for each channel are the following.

M01STOPC1 (can also be programmed as M01STOP)
 M01STOPC2 M01STOPC3 M01STOPC4

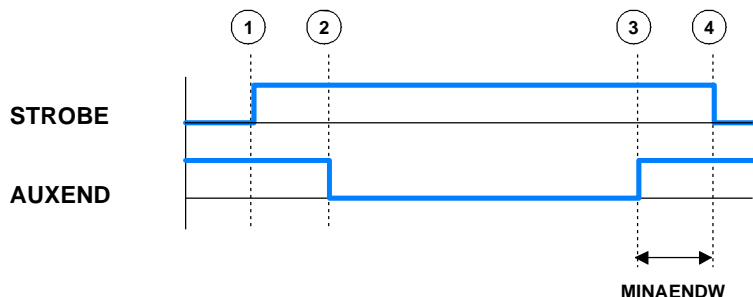
The PLC sets this mark high (=1) to "tell" the CNC channel not to ignore the conditional stops (M01).

AUXEND

There is a mark for each channel. The mnemonics for each channel are the following.

AUXENDC1 (can also be programmed as AUXEND)
 AUXENDC2 AUXENDC3 AUXENDC4

It is used when executing the auxiliary functions M and S. It works as follows:



- 1 The CNC channel indicates to the PLC, in registers MFUN and SFUN of the channel, the functions that it must execute and it activates the MSTROBE or SSTROBE mark to start the execution.
- 2 The PLC, when detecting that one of these marks has been activated, it must cancel the AUXEND mark to let the CNC know that the execution begins.
- 3 Once the required auxiliary functions have been executed, the PLC must activate the AUXEND mark to let the CNC know that the execution has ended. The AUXEND mark must be kept high (=1) longer than the time period established by (g.m.p.) MINAENDW.
- 4 After this time, the CNC deactivates the corresponding SSTROBE or MSTROBE mark thus ending the execution of the function.

TIMERON

The CNC has a timer to be used at will.

It times in seconds and it is enabled and disabled with this mark. It will be enabled (counting) with TIMERON set high (=1).

To reset and or know its accumulated timing, use the variable: (V.)PLC.TIMER. This variable is accessible from the PLC program, MDI or interface (any supported application).

Example of how to know the machining time.

```

CY1
() = MOV 0 R100
() = CNCWR (R100, PLC.TIMER, M11)
    Resets the timer to "0" on power-up.
END
PRG
AUTOMAT AND INCYCLE = TIMERON
    Timer active while machining.
() = CNCRD (PLC.TIMER, R300, M12)
    Register R300 shows the value of the timer.
END

```

PLCREADY

This mark indicates whether the PLC is running (=1) or stopped (=0).

- It must be high (=1) for the CNC to allow moving the axes and turning the spindle.
- If set low (=0), the PLC program stops and an error message is displayed.

NOWAIT

There is a mark for each channel. The mnemonics for each channel are the following.

NOWAITC1 NOWAITC2 NOWAITC3 NOWAITC4

It is applied to channel synchronization. The PLC sets this mark high (=1) to cancel all the synchronizations with the CNC channel.

For example, with the NOWAITC1 signal set to (=1), the waits programmed in any channel with the #WAIT, instruction and that refer to a mark of channel 1, they finish immediately and the program execution resumes.

DISCROSS1

..

DISCROSSn

It is applied to the cross compensation tables defined in the machine parameters. There is a mark for each table.

The PLC sets the mark to (=1) to disable the table. The DISCROSS1 mark is for table 1, DISCROSS2 for table 2 and so on.

EXRAPID

There is a mark for each channel. The mnemonics for each channel are the following.

EXRAPIDC1 (can also be programmed as EXRAPID)
EXRAPIDC2 EXRAPIDC3 EXRAPIDC4

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.

The treatment of this signal is similar to the rapid key of the operator panel.

6.

LOGIC CNC INPUTS AND OUTPUTS.
General modifiable signals.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

PLCABORT

There is a mark for each channel. The mnemonics for each channel are the following.

PLCABORTC1 (can also be programmed as PLCABORT)

PLCABORTC2 PLCABORTC3 PLCABORTC4

If the PLC sets this mark low (=0), the CNC channel aborts the CNCEX command launched from the PLC, but without initializing the conditions of the channel and keeping the history of the channel. This mark is only valid when the channel is executing a CNCEX; not in other program or MDI executions.

Once the execution has been canceled, the CNC deactivates this mark and sets the FREE mark of the channel high (=1). See "*FREE*" on page 397.

The PLCABORT mark is activated without having launched a CNCEX, the mark stays high (=1) until a CNCEX is executed (which will be aborted automatically) or until it is deactivated (RES PLCABORT).

PRGABORT

There is a mark for each channel. The mnemonics for each channel are the following.

PRGABORTC1 (can also be programmed as PRGABORT)

PRGABORTC2 PRGABORTC3 PRGABORTC4

If the PLC sets this mark low (=1), the CNC channel aborts the current execution of the part-program; but without affecting the spindle, the rest of the history is initialized. Then, the CNC resumes the execution of the program from the label indicated in the #ABORT instruction that is active in the part-program.

If no #ABORT instruction is active in the part-program, the PRGABORT mark has no effect.

NEXTMPGAXIS

This mark may be used to select an axis sequentially for jogging it with the handwheel. Only the axes being displayed in the active channel may be selected, regardless of the channel they belong to. The axes of another channel cannot be selected if they are not being displayed.

This mark is only taken into account when the CNC is in jog mode and the selector is in handwheel mode. With an up flank of this mark, transition from (=0) to (=1), the CNC behaves as follows.

- If no axis has been selected, it selects the first one that is displayed.
- If there is an axis selected, it selects the next one; if the last axis is selected, it selects the first one again.

An axis is de-selected when quitting the handwheel mode using the movement selector and after a reset.

This mark is meant for handwheels with push-button. On this type of handwheels, the push-button may be used to select, sequentially, the axis to be jogged. The usual thing in these cases is to connect the push-button of the handwheel to the digital input that will be in charge of managing the NEXTMPGAXIS mark.

**PANELOFF
PANELOFF1**

..

PANELOFF8

If the PLC activates one of these marks the CNC disables the corresponding jog panel. Each mark may be used to disable the jog panel integrated into the CAN bus; the rest of the elements of the bus are not affected. When the keyboard and the operator panel make up a single element, this mark only disables the jog panel.

There is one mark for each jog panel. The PANELOFF1 mark disables the first one of the bus, the PANELOFF2 mark the second one and so on. Both PANELOFF and PANELOFF1 are valid for the first jog panel.

6.

LOGIC CNC INPUTS AND OUTPUTS.
General modifiable signals.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

The "address" switch determines the order of the elements in the CAN bus. The first element will be the one with the lowest number and so on. The element with the lowest number corresponds to PANELOFF1.

Address	Element	PLC mark
0	CNC	
1	Remote (I/O) group	
2	Jog panel	PANELOFF1
3	Remote (I/O) group	
4	Remote (I/O) group	
5	Keyboard + Jog panel	PANELOFF2

CNCOFF

Setting this PLC mark high (=1) initiates the CNC shut-down sequence. Activating this mark is the same as pressing the key combination [ALT]+[F4].

SYNC

There is a register for each channel. The mnemonics for each channel are the following.

SYNC1 SYNC2 SYNC3 SYNC4

This register is used when using, from one channel, a particular spindle for synchronization even if the spindle is in another channel. For example, in the case of dual-turret lathe with a single spindle.

- With the G33 function, when threading with a particular spindle.
- With the G95 function, when programming the feedrate as a function of the turning speed of a particular spindle.

To do that, the PLC indicates in channel register SYNC the spindle to be used, only for synchronization. The SYNC register will take values 1 through 4; when assigning a 0 value, it will use the master spindle of the channel.

The CNC will check the contents of this register at the beginning of the block. If the PLC modifies this register during the execution of the block, the change will not be effective until the beginning of the next block.

RETRACE

There is a mark for each channel. The mnemonics for each channel are the following.

RETRACEC1 RETRACEC2 RETRACEC3 RETRACEC4

If the PLC sets this mark high (=1) while executing a program in automatic mode, the retrace function is activated in the selected channel. The retrace function stops the execution of the program and starts executing backwards the path traveled so far at the current block plus the last n blocks executed. The maximum number of blocks to execute in retrace mode is set by machine parameter NRETBK.

The retrace function may be activated during an interpolation, in the middle of a block and also at the end of the block, whether the execution was interrupted by M0 or by the single block mode.

If the PLC sets this mark high (=1), the CNC finishes the RETRACE function. The retrace function also ends after M30, with a reset or when the CNC channel sets mark RETRAEND high (=1) See "*RETRAEND*" on page 397.

The retrace function must be enabled in machine parameter RETRACAC.

6.

LOGIC CNC INPUTS AND OUTPUTS.
General modifiable signals.

INHIBITMPG1

..

INHIBITMPG12

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.

If the handwheel is disabled, the CNC ignores the pulses coming from the handwheel and, therefore, it does not move the axis. While the handwheel is disabled, the variable associated with the handwheel does not save the pulses sent by the handwheel.

(V.)G.HANDP[hw] Number of pulses sent by the handwheel since the system was started up.

If it is an individual handwheel, associated with an axis, the ENABLE(axis) mark of the axis will stay active. If the handwheel mode is selected in jog mode for that axis, that axis will appear highlighted even if the PLC has disabled the handwheels that could move it.

KEYBD1CH

..

KEYBD8CH

On power-up, the CNC always assumes the keyboard configuration set in the machine parameters. These registers may be used to change the default behavior of the keyboards with respect to the channels, set by machine parameters. These registers can associate a jog panel with a particular channel, always with the active channel or restore the configuration defined in the machine parameters.

KEYBD1CH .. KEYBD8CH	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.

VOLCOMP1

..

VOLCOMP4

If the PLC activates one of these marks (changing it from 0 to 1), the CNC activates the corresponding volumetric compensation (VOLCOMP parameter). If the PLC deactivates one of these marks (changing it from 1 to 0), the CNC deactivates the corresponding volumetric compensation.

Mark.	Meaning.
VOLCOMP1	Activate the first volumetric compensation (machine parameter VOLCOMP 1).
VOLCOMP2	Activate the second volumetric compensation (machine parameter VOLCOMP 2).
VOLCOMP3	Activate the third volumetric compensation (machine parameter VOLCOMP 3).
VOLCOMP4	Activate the fourth volumetric compensation (machine parameter VOLCOMP 4).

All volumetric compensations can be simultaneously active, as long as there are no common axes among them. The CNC applies the volumetric compensation after applying the leadscrew compensation and the cross compensation. Volumetric compensation remains active after a reset, error or end of program (M30).

QWERTYOFF1

..

QWERTYOFF8

If the PLC turns one of these marks on, it disables the corresponding alphanumeric keyboard. Each mark may be used to disable a USB keyboard (10K model) or one integrated into the CAN bus (15" model); the rest of the elements of the bus are not affected. The keyboards are not enabled or disabled immediately, especially when activating the USB keyboard, it may take a few seconds.

The way the QWERTYOFF1 works depends on the CNC model.

- On the 10K model, this mark affects the alphanumeric keyboard and the softkeys of the monitor (USB interface).
- On the 15" model, this mark affects the softkeys of the monitor (USB interface) and the first node of the CAN bus. In order for the mark to affect the keyboard associated with the central unit, the number of this keyboard must be the lowest one in the bus.

Marks QWERTYOFF2 to QWERTYOFF8 affect the rest of the CAN keyboards, depending on the logic numbers of the nodes, from lowest to highest.

FLIMITAC

If the PLC activates this mark, the CNC activates the feedrate safety limits (parameter FLIMIT) on all the axes of the system during the execution of the block. If the PLC deactivates this mark, the CNC restores the programmed feedrate.

The feedrate safety limit is applied to the movements in automatic (G0, G1, etc.) and in jog mode (jog, handwheels, etc.). This parameter affects neither the threading operations nor the independent axis movements, which are executed at the programmed feedrate.

FLIMITACCH

There is a mark for each channel. The mnemonics for each channel are the following.

FLIMITACCHC1 (can also be programmed as FLIMITACCH)

FLIMITACCHC2 FLIMITACCHC3 FLIMITACCHC4

If the PLC activates this mark, the CNC activates the feedrate safety limits (parameter FLIMIT) on all the axes of the channel during the execution of the block. If the PLC deactivates this mark, the CNC restores the programmed feedrate.

The feedrate safety limit is applied to the movements in automatic (G0, G1, etc.) and in jog mode (jog, handwheels, etc.). This parameter affects neither the threading operations nor the independent axis movements, which are executed at the programmed feedrate.

SLIMITAC

If the PLC activates this mark, the CNC activates the speed safety limits (parameter SLIMIT) on all the spindles of the system during the execution of the current block. If the PLC deactivates this mark, the CNC restores the programmed speed.

PT100OFF1

..

PT100OFF20

The PT100 inputs are activated via the machine parameters. The PLC has the following marks in order to be able to temporarily disable the sensor (for example, during a spindle change).

If the PLC activates one of these marks, the CNC disables the corresponding sensor (parameter PT100 n). The PT100OFF1 mark corresponds to the input 1 sensor, PT100OFF2 is for the input 2 sensor and so forth.

6.

LOGIC CNC INPUTS AND OUTPUTS.
General modifiable signals.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

INT1

..

INT4

There is a mark for each channel. The mnemonics for each channel are the following. Here is an example of the mnemonics for INT1; it is the same for the rest of the marks.

INT1C1 (can also be programmed as INT1)
 INT1C2 INT1C3 INT1C4

When the PLC activates one of these marks, the channel interrupts the execution of the program and executes the corresponding interruption subroutine associated with parameters INT1SUB to INT4SUB. 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.).

If the program is 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 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.

The CNC cancels the mark when it starts executing the subroutine or when it rejects the execution of the subroutine.

PROBE1ENA**PROBE2ENA**

Marks activated by default. These mark indicate that the probe has is enabled. When executing a G100 or G103 command, the CNC will issue an error message if the mark of the active probe (the one selected with #SELECT PROBE) is not enabled. These marks do not limit the monitoring of the safe mode.

These marks should be tested in the subroutines Sub_Probe_Tool_Begin.fst and Sub_Probe_Piece_Begin.fst so the subroutine waits until the mark is active.

PROBE1MONIT**PROBE2MONIT**

These marks are associated with the safe mode of the probe. If the mark is active, the probe is in safe mode monitoring collisions.

Loop level monitoring, controlling all the collision scenarios in any of the two probes. The CNC can monitor the probes connected to the local inputs and to the remote CAN inputs. The CNC monitors the two probe inputs at the same time.

These marks should be tested in the subroutines Sub_Probe_Tool_Begin.fst and Sub_Probe_Piece_Begin.fst to warn, if they are deactivated, that the probe is in non-safe mode.

LCOUNTALARMOFF1**LCOUNTALARMOFF2**

These marks refer to the local feedback inputs 1 (LCOUNTALARMOFF1) and 2 (LCOUNTALARMOFF2) for the 8070.

If the PLC activates one of these marks (by changing it from 0 to 1), it disables the corresponding local feedback input. If the PLC deactivates one of these marks (by changing it from 1 to 0), it enables the feedback alarms.

When the spindle and the C axis are parameterized in two sets for the same spindle, each with its respective feedback while sharing the input signal (although with different resolutions), the movement of the spindle can sound an alarm for the C feedback. These marks allow the C feedback alarms to be disabled (for a local feedback) when the spindle is in use, although assuming there may be a loss of feedback and that the C axis must be referenced when re-activating it.

6.**LOGIC CNC INPUTS AND OUTPUTS.**

General modifiable signals.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

The alarm report is conditioned to the parameter FBACKAL. No alarm is detected if the local feedback is set to non-differential TTL.

6.

LOGIC CNC INPUTS AND OUTPUTS.

General modifiable signals.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

6.9 Modifiable signals for axes and spindles.

When the spindle is working in closed loop (M19 or G63), it behaves like an axis.

The signal names are generic. Replace the word (axis) with the spindle name or with the name or logic number of the axis. For example, the name of the LIMITPOS(axis) mark for a machine with the X, Y, Z, Z2, B axes and spindle S.

LIMITPOSX, LIMITPOSZ2, LIMITPOSB, LIMITPOSS

LIMITPOS3 for the Z axis.

LIMITPOS5 for the B axis.

LIMITPOS(axis)

LIMITNEG(axis)

If the PLC sets this mark high (=1), the CNC interprets that the corresponding axis or spindle has exceeded the positive (POS) or negative (NEG) travel limits.

It stops the axes and the spindle and displays the relevant error message.

To take the axis to the work zone, access the JOG mode and move the axis or spindle that overran the travel limit. They can only be moved in the proper direction.

DECEL(axis)

This mark is used during home search. The PLC sets this mark high (=1) to indicate that the home switch is pressed. The CNC decelerates the axis, it switches from the fast homing feedrate indicated by (a.m.p.) REFFEED1, to the slow homing feedrate indicated by the (a.m.p.) REFFEED2.

When searching home on axes that use distance-coded reference marks, the PLC sets this mark to (=1) to indicate that the axis has reached the travel limit. In this case, it is necessary to reverse the moving direction to go on with the home search.

INHIBIT(axis)

If the PLC sets this mark high (=1) the CNC inhibits any movement of the corresponding axis or spindle. This movement will resume when the PLC sets this mark back low (=0). If the axis or spindle is moving with other axes, all the axes will stop.

For independent axes and electronic cam, if the PLC sets this signal to (=1), 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.

For independent axes, this signal also stops the monitoring of the synchronization.

The screens of the automatic and jog modes show the status of this mark.

- For the spindles, the text "Sreal" appears in red when this mark is active for the spindle. The status of the mark is not displayed if the screen does not show this text.
- For the axes, the name of the axis preceding the coordinate appears in red when this mark is active for the axis.

AXISPOS(axis)

AXISNEG(axis)

The CNC uses these marks when operating in JOG mode.

If the PLC sets one of these marks high (=1), the CNC will move the relevant axis in the indicated direction: positive (POS) or negative (NEG). The CNC will move the axis at the corresponding feedrate and selected override (%).

The treatment of these marks is similar to the JOG keys of the operator panel.

6.

LOGIC CNC INPUTS AND OUTPUTS.

Modifiable signals for axes and spindles.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

SERVO(axis)ON

This mark must be high (=1) in order to be able to move the corresponding axis. If the SERVO(axis)ON mark is set low (=0) while moving the axis or spindle, the CNC stops the axes and the spindle. It also displays the relevant error message.

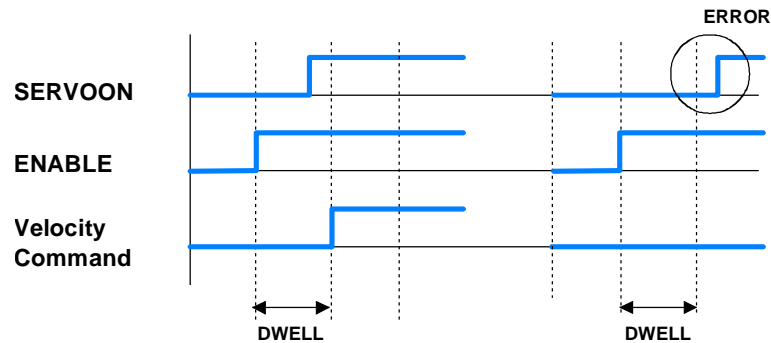
- In order to be able to continuously control the axis, the SERVOON mark must always be high (=1).

(there are no errors) AND (axis drive OK) = SERVOonON

- To control the axis only when they move, use the ENABLE mark. The CNC sets it high (=1) whenever it has to move the axis. See "**ENABLE(axis)**" on page 400.

(no errors) AND (drive OK) AND ENABLE = SERVOON

If the axis moves while locked (meaning SERVOON=0), the CNC stores that displacement as axis lag (following error). When controlling it back (SERVOON=1) it restores its position.



After activating the ENABLE mark, the CNC waits for a time period indicated by parameter DWELL only if SERVOonON is low (=0). If after this time period the SERVOonON signal is still deactivated, the CNC shows the error "axis locked up".

To control the axis only while moving, the (a.m.p.) DWELL must be assigned a value greater than 2 PLC cycle scans in order to avoid the error message "axis locked".

When the CNC works in any square-corner mode (G05, G50 or HSC), it can issue the "axis locked up" error in the block-to-block transition if the SERVOonON mark of any of the axes involved has not been activated before. To avoid this problem when the CNC works with dead axes, the PLC must activate the DEAD(axis) mark so the CNC does not apply block transition on these dead axes.

DRO(axis)

This mark, together with the relevant SERVOonON mark allows the axis or spindle as a dro axis. To do that, the DRO mark must be set high (=1) and SERVOonON low (=0).

When working as DRO axis or spindle, the CNC neither closes the position loop nor generates following error because the real and theoretical coordinates are the same.

The programmed movements are considered executed without making any movement or causing error messages. When programming an interpolation with other axes, the other ones will move at the corresponding feedrate; but the dro axis will not move.

When the DRO marks goes back low (=0), the axis is no longer a dro axis and it assumes the current position having a following error of "0".

LIM(axis)OFF

If the PLC activates this mark, the CNC ignores the software travel limits and the work zones set for the corresponding axis. If the PLC deactivates this mark, the CNC considers again the software travel limits and the work zones set for the indicated axis.

This mark makes it easier to take the tool back to the permitted zone if it had exceeded the software limits or it had invaded the forbidden zone.

The software travel limits may be set with machine parameters, using functions G198 and G199 or using variables. The CNC always applies the most restrictive travel limits. The work zones may be set using functions G120, G121, G122, G123.

6.

LOGIC CNC INPUTS AND OUTPUTS.
Modifiable signals for axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

SPENA(axis) DRENA(axis)

The PLC uses these marks to enable the drives when communicating via Sercor or to enable the servo drives and inverters when communicating via Mechatrolink. The SPENA(axis) mark corresponds to the "speed enable" signal and the DRENA(axis) to the "drive enable" signal of the device. The operation of these two signals is the following.

The operation of these two signals is the following.

- On PLC power-up, it must cancel both signals.
- For the device to run normally, both signals must be active. The motor will respond to any changes of velocity command.
- If the PLC cancels the DRENA signal (drive enable), the power circuit of the device shuts down and the motor loses its torque. In this situation, the motor is no longer governed and will turn freely until it stops by inertia.
- If the PLC cancels the SPENA signal (speed enable), the internal velocity command of the drive is switched to "0 rpm". In this situation, the motor keeps its torque while braking and, once stopped, the drive's power circuit shut down and the motor has no torque.

PARKED(axis)

The PLC sets this mark high (=1) to let the CNC know that the corresponding axis or spindle is parked.

For further information, see the consulting signals for axes and spindles: PARK and UNPARK described earlier in this chapter. See "*PARK(axis) UNPARK(axis)*" on page 402.

LUBRENA(axis) LUBROK(axis)

These marks together with the axis consulting signal LUBR(axis) must be used to lubricate the axes. See "*LUBR(axis)*" on page 401.

The LUBRENA mark indicates whether to use this feature (=1) or not (=0). With LUBRENA set to (=1), the CNC acts as follows:

- 1 When the axis has traveled the distance set by (a.m.p.) DISTLUBRI, it sets the LUBR mark high (=1) to let the PLC know that the axis must be lubricated.
- 2 After lubricating the axis, the PLC sets the LUBROK(axis) mark high (=1) to let the CNC know that the axis has been lubricated.
- 3 The CNC sets the LUBR mark low (=0) and resets its count to "0".

Remind to set the LUBROK mark low (=0) so the feature works properly. Otherwise, its count will always stay at "0".

DIFFCOMP(axis)

This mark is used on Gantry axes to correct the position difference between the master and the slave axes. The slave axis will move until reaching the position of the master axis at the feedrate set by parameter REFFEED2.

The correction must be enabled in machine parameter DIFFCOMP and it is applied in the following cases.

- With the up-flank of SERVO*ON if DIFFCOMP is set high.
- With the up-flank of DIFFCOMP if SERVO*ON is set high.

This process can only be interrupted with RESET.

FBACKSEL(axis)

The CNC uses this mark when the system has external+internal feedback. See parameter FBACKSRC.

An up flank of this mark switches to external feedback (direct feedback) and a down flank switches to internal feedback (motor feedback). If the combined feedback is active (parameter FBMIXTIME), it is used with the up flank of this mark.

6.

LOGIC CNC INPUTS AND OUTPUTS.

Modifiable signals for axes and spindles.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

When the type of feedback is internal+external, the CNC uses the internal feedback (motor feedback) on power-up, when resetting the drive and when initializing the Sercos ring.

The ACTFBACK(axis) mark indicates which is the active feedback. See "[ACTFBACK\(axis\)](#)" on page 405.

DEAD(axis)

When the system has a dead axis, the transition between blocks (blending) where the dead axis is involved, it issues the "axis locked up" error message. This is due to the delay between the activation of the ENABLE and SERVOON signals .

To avoid this error, the PLC offers this mark that indicates how the CNC must handle the transitions between blocks when a dead axis is involved.

- If the PLC sets this mark high (=1) whenever the axis is involved in the movement, the CNC does not apply the transition (blend) between blocks; in other words, the movements where the dead axis is involved, they wait a time period set in the DWELL parameter.
- If the PLC sets this mark low (=0), the CNC blends the blocks even when the axis is involved in the movement.

This mark must be set high (=1) when the axis is enabled as dead axis and low (=0) when it is enabled as a live axis.

SWITCH(axis)

When the system has multi-axis groups, this mark may be used to toggle between the various axes or spindles of the group. The PLC activates this mark to let the CNC know which axis or spindle of the group is active. If the PLC turns this mark off, the CNC assumes that the axis or spindle does not belong to the current configuration.

TANDEMOFF(axis)

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. For example, in a C axis tandem, there is no need to engage each motor which requires generating an oscillation movement on each motor without affecting the other.

This mark refers to the slave axis or spindle of the tandem. If the PLC activates this mark, the slave axis is decoupled from the master and both axes may be moved separately. The slave axis can only be moved from the PLC through the PLCOFFSET. The master axis may be moved as usual from the jog panel, from the MDI/MDA mode, etc. When moving the master axis, the CNC will not generate a velocity command for the slave axis nor will apply any compensation.

Even if the tandem is decoupled, it will remain active in terms of programming, display, etc. The CNC displays the coordinates of the master axis, does not allow programming the slave axis and none of the axes may be parked. The CNC can home the master axis and at the end of the home search, the CNC will also initialize the position value of the slave axis (with the position value of the master).

6.

LOGIC CNC INPUTS AND OUTPUTS.
Modifiable signals for axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

6.10 Spindle modifiable signals.

GEAR1
GEAR2
GEAR3
GEAR4

There is a mark for each spindle. The mnemonics for each spindle are the following: Here is an example of the mnemonics for GEAR1; it is the same for the rest of the marks.

GEAR1SP1 (can also be programmed as GEAR1)
GEAR1SP2 GEAR1SP3 GEAR1SP4

The PLC uses these marks to let the CNC know which spindle range (gear) is currently selected (=1). When requesting a gearchange, the CNC informs the PLC about it using auxiliary functions: M41, M42, M43 or M44. The gear change concludes when the PLC receives the confirmation signal AUXEND.

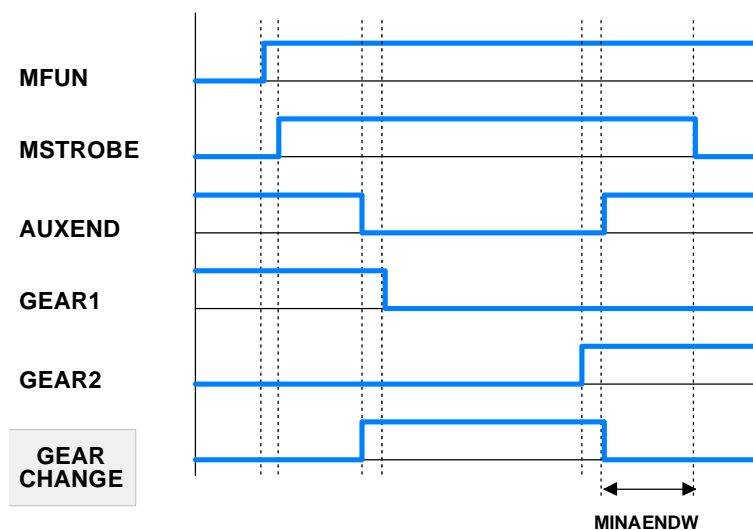
The CNC assumes the parameter set of the new gear when the spindle speed reaches the speed set in parameter SZERO and when the PLC receives the confirmation of one of marks GEAR1 through GEAR4.

On Sercos spindles, when the range change involves a gear change (NPULSES, INPUTREV, OUTPUTREV), functions M41 to M44 also change the gear at the drive.

When the parameter set selected at the CNC and at the PLC are the same, the spindle activates the GEAROK mark. In order for both parameter sets to coincide, function M41 must be active at the CNC and the GEAR1 mark at the PLC, M42 with GEAR2 and so on.

Example of a GEAR1 GEAR2 change.

If Gear 2 (M42) is requested while gear 1 is active.



- 1 The CNC indicates to the PLC the gear requested with MFUN1=42 and sets the MSTROBE mark to (=1).
- 2 When detecting the request, the PLC sets an internal indicator.
DFU MSTROBE AND CPS MFUN* EQ 42 = SET M1002
- 3 The change begins and lets the CNC know by setting AUXEND (=0).

```
NOT M1002 AND <rest of conditions> \
= AUXEND \
= (starts the gear change)
```

During the gear change, the CNC is informed that gear 1 is dropped and gear 2 is now selected. The active gear indicator, GEAR1 through GEAR4 must be set before activating the AUXEND signal.

```
I21 = GEAR1
I22 = GEAR2
```

6.

LOGIC CNC INPUTS AND OUTPUTS.

Spindle modifiable signals.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

- 4 Once the gear change is over, it cancels the indicator (M1002) and it lets the CNC know by setting AUXEND high (=1).

(GEAR change completed) = RES M1002

The AUXEND mark must be kept high (=1) longer than the time period set by g.m.p. MINAENDW so the CNC cancels the MSTROBE mark and concludes the gear change.

PLCCNTL SANALOG

There is one signal for each spindle. The mnemonics for each spindle are the following:

PLCCNTL1 (can also be programmed as PLCCNTL)

PLCCNTL2 PLCCNTL3 PLCCNTL4

SANALOG1 (can also be programmed as SANALOG)

SANALOG2 SANALOG3 SANALOG4

These signals are used with analog spindles, position-Sercos, velocity-Sercos and Mechatrolink.

When the PLC sets the PLCCNTL mark high (=1), the spindle decelerates with a ramp until it stops and it is then controlled by the PLC. The SANALOG register sets the spindle command voltage to be applied. The spindle control via PLC is used, for example, for oscillating the spindle during a gear change.

- Analog spindle.

10 V of velocity command correspond to SANALOG = 32767. In other words:

For 4V, program $SANALOG = (4 \times 32767) / 10 = 13107$

For -4V, program $SANALOG = (-4 \times 32767) / 10 = -13107$

- Velocity-Sercos spindle.

The command in SANALOG will be given in 0.0001 rpm.

- Sercos spindle in position.

The command in SANALOG will be given in 0.0001 degrees.

- Mechatrolink spindle.

The command in SANALOG will be given in hundredths of a Hertz.

The command indicated in SANALOG is not applied with a ramp; therefore, it is up to the PLC program to apply the command gradually when necessary.

The spindle reference point is not lost when the spindle is controlled via PLC. There is no need to home the spindle again when its control is switched back to the CNC.

The PLC has no priority over a spindle synchronization. When attempting to control a synchronized spindle (either master or slave) using PLCCNTL, a warning will be issued indicating that it is not possible. Also, if the gear change of a synchronized spindle involves a command from the PLC, this change will not be possible.

6.

LOGIC CNC INPUTS AND OUTPUTS.
Spindle modifiable signals.



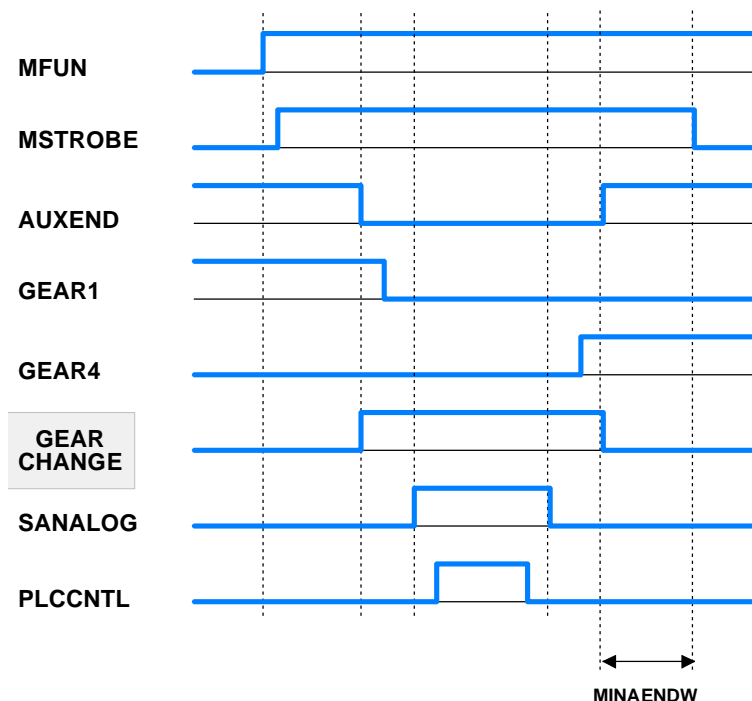
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Example similar to the one used for GEAR1 through GEAR4

The spindle oscillation during a gear change is controlled by the PLC. Gear 4 is requested while gear 1 is active.



The example for the GEAR1 through GEAR4 signals describes how to detect and carry out the gear change. This example shows how to control the spindle oscillation during a gear change.

The PLC sets SANALOG to the value corresponding to the residual velocity command and activates the PLCCNTL mark to indicate that the spindle is controlled by the PLC.

When done, the PLCCNTL mark must be set low (=0) and the SANALOG signal must be set to "0".

SPDLEREV

There is a mark for each spindle. The mnemonics for each spindle are the following:

- SPDLEREV1 (can also be programmed as SPDLEREV)
- SPDLEREV2 SPDLEREV3 SPDLEREV4

When the PLC sets this mark high (=1), the CNC reverses the spindle turning direction. To do this, it decelerates and accelerates applying the ramps set by machine parameters.

The PLC always applies this mark to the spindles in open loop(M3/M4); On spindles in closed loop (M19), it depends on parameter M19SPDLEREV. If an M2 or M4 function is executed while the SPDLEREV mark is high (=1), the spindle will turn in the opposite direction to the one assigned to the function.

If the PLC activates or cancels the SPDLEREV mark when the spindle is controlled by the PLCA (PLCCNTL mark active), the CNC does not generate ramps to invert the SANALOG command.

- PLCM3
- PLCM4
- PLCM5

The PLC offers a mark for each spindle. The mnemonics for each spindle are the following: Here is an example of the mnemonics for PLCM3; it is the same for the rest of the spindles.

- PLCM3SP1 (can also be programmed as PLCM3)
- PLCM3SP2 PLCM3SP3 PLCM3SP4

The PLC turns this mark on to let the CNC know that it must execute the corresponding M function at the indicated spindle. The spindle must belong to a channel, M functions cannot

6.

LOGIC CNC INPUTS AND OUTPUTS.
Spindle modifiable signals.



CNC 8070

(REF: 1709)

be sent to spindles that are not assigned to any channel. The PLC may change the spindle speed using the variable "(V.)PLC.S.sn", but without generating a gear change even if the gear change is automatic (parameter AUTOGEAR).

These M functions are treated the same way as those executed from the CNC. When the CNC activates one of these marks, the CNC activates the MSTROBE mark and writes the corresponding M function into the MFUN register. Once the M function has been synchronized at the PLC (AUXEND signal), the CNC starts sending the velocity command to that spindle, updates (refreshes) the M function history and deactivates the mark at the PLC.

The CNC will admit M functions even if the channel is in execution (executing a program, jogging an axis, etc.) as long as the channel status is other than "In error" and "Not ready" and the channel is not doing a reset or validating machine parameters. If during tool inspection, the PLC changes the turning direction of a spindle using these marks, the change will be identified when repositioning and will appear as pending to reposition.

The CNC ignores the PLC requests in the following cases, when the CNC ignores the mark set by the PLC and it erases it so the request does not remain pending.

- When the spindle is working as a C axis.
- When the spindle is threading (rigid tapping, normal tapping or electronic threading).
- When the CNC status is "In error" or "Not ready", is doing a reset or validating parameters.

If the PLC activates these marks while synchronizing another M function of the program being executed or while homing the spindle, the PLC keeps this mark activated until the CNC can attend to it.

SLIMITACSPDL

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 during the execution of the current block. If the PLC deactivates this mark, the CNC restores the programmed speed.

6.

LOGIC CNC INPUTS AND OUTPUTS.

Spindle modifiable signals.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

6.11 Modifiable logic signals; laser.

LASERON

The PLC activates this mark to indicate that the laser is on.

SHUTTERON

The PLC activates this mark to indicate that the laser source shutter is open.

LASERREADY

The PLC activates this mark to indicate that the laser is ready.

PWMON

This mark may be used to activate and deactivate the PWM via PLC. The PWM is configured using the following registers.

Registers.	Meaning.
PWMFREQ	Frequency of the PWM (between 2 and 5000 Hz; by default, 0).
PWMDUTY	Duty cycle of the PWM (between 1 and 1000; by default, 500).

Activating the PWM via PLC has priority over activating it from the CNC.

PWM status from the CNC.	PWM status from the PLC.	PWM status.
Off.	Off.	PWM off.
Active.	Off.	PWM active from the CNC.
Off.	Active.	PWM active from the PLC.
Active.	Active.	PWM active from the PLC.

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.

During tool inspection, the CNC does not deactivate the PWM. This mark may be used to deactivate the PWM via PLC during tool inspection and resume the PWM at the end of the inspection.

PWMFREQ

PWM frequency, when the PWM has been activated from the PLC (2-5000 Hz; by default, 0).

PWMDUTY

PWM duty cycle, when the PWM has been activated from the PLC (1-5000 tenths of %; by default, 500).

ENABLEPWROUT

This mark is active by default. This mark is associated with the power control via an analog output associated with the spindle (#PWRCTRL ON [OUT]). The PLC deactivates this mark to disable the active power control. The PLC activates this mark to enable the active power control in the CNC; if no power control is active, this mark does nothing.

If the PLC deactivates this mark, the CNC disable the power control, setting the maximum defined in the instruction #PWRCTRL ON.

ENABLEPWRDUTY.

This mark is active by default. This mark is associated with the PWM duty control (#PWRCTRL ON [DUTY]). The PLC deactivates this mark to disable the active duty control. The PLC activates this mark to enable the active duty control in the CNC; if no duty control is active, this mark does nothing.

If the PLC deactivates this mark, the CNC disables the duty control, setting the maximum specified in the instruction #PWRCTRL ON.

ENABLEGAP

This mark is active by default. The PLC deactivates this mark to disable the active gap control. The PLC activates this mark to enable the active gap control in the CNC; if no gap control is active, this mark does nothing.

GAPCOLLISION

The gap control sensor may have a collision signal connected to the CNC, which should be managed by the PLC via this mark. The PLC activates this mark to indicate that the sensor collision signal has been received. This mark is not kept in memory. When this mark is active, the CNC will display an error or not, depending on how the parameter GAPCOLLISION has been set.

ENABLELEAP

The PLC deactivates this mark to disable the active leapfrog. The PLC activates this mark to enable the active leapfrog in the CNC; if no gap control is active, this mark does nothing.

6.**LOGIC CNC INPUTS AND OUTPUTS.**

Modifiable logic signals; laser.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

6.12 Modifiable signals of the independent interpolator.

The signal names are generic. Replace the text (axis) with the name or logic axis number.

_IXFERINH(axis)

If the PLC sets this mark (=0), the movement of the independent axis ends and remains waiting for the PLC to activate this mark again. To manage the transfer inhibit in an independent movement, the PLC also has a general mark per channel (_XFERINH(axis) mark). The CNC evaluates both marks as follows.

- In independent movements programmed from the CNC, it first evaluates the transfer inhibit of the channel (_XFERINH mark) and then the particular one of the axis (_IXFERINH(axis) mark).
- In independent movements programmed from the PLC, the transfer inhibit of the channel is evaluated depending on parameter XFITOIND. The CNC always evaluates the particular transfer inhibit of the axis.

XFITOIND	Meaning.
Yes	The CNC firsts evaluates the transfer inhibit of the channel (_XFERINH mark) and then the particular one of the axis (_IXFERINH(axis) mark).
No	The CNC only evaluates the particular transfer inhibit of the axis (_IXFERINH(axis) mark).

I RESET(axis)

For movements of an independent axis, if the PLC sets this mark to (=1), it interrupts the instruction in execution and eliminates the instructions pending execution.

For electronic-cam movements, it interrupts the cam synchronization movement switching to zero speed.

It sets the initial conditions at the independent interpolator of the axis.

I ABORT(axis)

For movements of an independent axis, if the PLC sets this mark to (=1), the positioning block being executed (if any) stops, without eliminating also the rest of the positioning blocks pending execution.

It only affects positioning blocks; neither the pending instructions nor the synchronization movement are eliminated.

6.

LOGIC CNC INPUTS AND OUTPUTS.

Modifiable signals of the independent interpolator.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

6.13 Tool manager modifiable signals.

6.

LOGIC CNC INPUTS AND OUTPUTS.
Tool manager modifiable signals.

SETTMEM

There is a mark for each magazine. The mnemonics for each magazine are the following.

SETTMEMZ1 (can also be programmed as SETTMEM)

SETTMEMZ2 SETTMEMZ3 SETTMEMZ4

The PLC sets this mark high (=1) to activate the tool manager emergency.

RESTMEM

There is a mark for each magazine. The mnemonics for each magazine are the following.

RESTMEMZ1 (can also be programmed as RESTMEM)

RESTMEMZ2 RESTMEMZ3 RESTMEMZ4

The PLC sets this mark high (=1) to cancel the tool manager emergency.

CUTTINGON

There is a mark for each channel. The mnemonics for each channel are the following.

CUTTINGONC1 (same as CUTTINGON)

CUTTINGONC2 CUTTINGONC3 CUTTINGONC4

When associating a maximum life span to a tool (monitoring), the CNC checks this mark to find out whether the tool is machining (=1) or not (=0).

```
PRG
()= CNCRD (G.GS0, R300, M12)
  Register R300 shows the status of the G functions.
AUTOMAT AND INCYCLE AND NOT BOR300 = CUTTINGON
  If it is in automatic mode (AUTOMAT), a (INCYCLE) block is being executed and the
  GOO function is not active, then, the tool is considered to be machining.
END
```

TREJECT

There is a mark for each channel. The mnemonics for each channel are the following.

TREJECTC1 (can also be programmed as TREJECT)

TREJECTC2 TREJECTC3 TREJECTC4

If the PLC sets this mark high (=1), the CNC interprets that the tool must be rejected.

MZTOCH1

There is a mark for each magazine. The mnemonics for each magazine are the following.

MZTOCH1MZ1 (can also be programmed as MZTOCH1)

MZTOCH1MZ2 MZTOCH1MZ3 MZTOCH1MZ4

Use it with an asynchronous magazine or synchronous with arm. The PLC must set this mark high (=1) after taking the tool from the magazine to the changer arm 1.

CH1TOSPDL

There is a mark for each magazine. The mnemonics for each magazine are the following.

CH1TOSPDLMZ1 (same as CH1TOSPDL)

CH1TOSPDLMZ2 CH1TOSPDLMZ3 CH1TOSPDLMZ4

Use it with an asynchronous magazine or synchronous with arm. The PLC must set this mark high (=1) after taking the tool from the changer arm 1 to the spindle.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

SPDLTOCH1

There is a mark for each magazine. The mnemonics for each magazine are the following.

SPDLTOCH1MZ1 (same as SPDLTOCH1)
SPDLTOCH1MZ2 SPDLTOCH1MZ3 SPDLTOCH1MZ4

Use it with an asynchronous magazine with one changer arm. The PLC must set this mark high (=1) after taking the tool from the spindle to the changer arm 1.

SPDLTOCH2

There is a mark for each magazine. The mnemonics for each magazine are the following.

SPDLTOCH1MZ1 (same as SPDLTOCH1)
SPDLTOCH2MZ2 SPDLTOCH2MZ3 SPDLTOCH2MZ4

Use it with an asynchronous magazine or synchronous with arm. The PLC must set this mark high (=1) after taking the tool from the spindle to the changer arm 2.

CH1TOMZ

There is a mark for each magazine. The mnemonics for each magazine are the following.

CH1TOMZ1 (can also be programmed as CH1TOMZ)
CH1TOMZ2 CH1TOMZ3 CH1TOMZ4

Use it with an asynchronous magazine or synchronous with arm. The PLC must set this mark high (=1) after taking the tool from the tool changer arm 1 to the magazine.

CH2TOMZ

There is a mark for each magazine. The mnemonics for each magazine are the following.

CH2TOMZ1 (can also be programmed as CH2TOMZ)
CH2TOMZ2 CH2TOMZ3 CH2TOMZ4

Use it with an asynchronous magazine or synchronous with arm. The PLC must set this mark high (=1) after taking the tool from the tool changer arm 2 to the magazine.

SPDLTOGR

There is a mark for each channel. The mnemonics for each channel are the following.

SPDLTOGRC1 (can also be programmed as SPDLTOGR)
SPDLTOGRC2 SPDLTOGRC3 SPDLTOGRC4

Use it with a magazine that admits ground tools. The PLC must set this mark high (=1) after taking the tool from the spindle to ground.

GRTOSPDL

There is a mark for each channel. The mnemonics for each channel are the following.

GRTOSPDL1 (can also be programmed as GRTOSPDL)
GRTOSPDL2 GRTOSPDL3 GRTOSPDL4

Use it with a magazine that admits ground tools. The PLC must set this mark high (=1) after taking the tool from the ground to the spindle.

6.

LOGIC CNC INPUTS AND OUTPUTS.

Tool manager modifiable signals.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

MZTOSPDL

There is a mark for each magazine. The mnemonics for each magazine are the following.

MZTOSPDLMZ1 (can also be programmed as MZTOSPDL)

MZTOSPDLMZ2 MZTOSPDLMZ3 MZTOSPDLMZ4

Use it with a synchronous magazine (without arm). The PLC must set this mark high (=1) after taking the tool from the magazine to the spindle.

SPDLTOMZ

There is a mark for each magazine. The mnemonics for each magazine are the following.

SPDLTOMZ1 (can also be programmed as SPDLTOMZ)

SPDLTOMZ2 SPDLTOMZ3 SPDLTOMZ4

Use it with a synchronous magazine (without arm). The PLC must set this mark high (=1) after taking the tool from the spindle to the magazine.

MZROT

There is a mark for each magazine. The mnemonics for each magazine are the following.

MZROTMZ1 (can also be programmed as MZROT)

MZROTMZ2 MZROTMZ3 MZROTMZ4

Use it with a turret type magazine. The PLC must set this mark high (=1) when the turret has rotated.

TCHANGEOK

There is a mark for each magazine. The mnemonics for each magazine are the following.

TCHANGEOKMZ1 (same as TCHANGEOK)

TCHANGEOKMZ2 TCHANGEOKMZ3 TCHANGEOKMZ4

The PLC must set this mark high (=1) when the tool change has ended (M06).

MZPOS

There is a register for each magazine. The mnemonics for each magazine are the following.

MZPOSMZ1 (can also be programmed as MZPOS)

MZPOSMZ2 MZPOSMZ3 MZPOSMZ4

The PLC must indicate the current magazine position in this register.

6.

LOGIC CNC INPUTS AND OUTPUTS.
Tool manager modifiable signals.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

6.14 Keystroke modifiable signals.

KEYLED1 / KEYLED2

These registers control the LED's (lamps) of the keys of all the operator panels at the same time.

KEYLED1_1 / KEYLED2_1

..

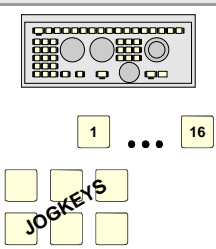
KEYLED1_8 / KEYLED2_8

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.

The following instruction changes the status of the led of the first user key (bit 0) every time the key is pressed.

```
DFU BOKEYBD1_2 = CPL BOKEYLED1_2
```

Registers KEYLED1 / KEYLED1_1 to KEYLED1_8. User keys.

Bit	OP-PANEL
	
0	User key 1
1	User key 2
2	User key 3
3	User key 4
4	User key 5
5	User key 6
6	User key 7
7	User key 8
8	User key 9
9	User key 10
10	User key 11
11	User key 12
12	User key 13
13	User key 14
14	User key 15
15	User key 16

6.

LOGIC CNC INPUTS AND OUTPUTS.

Keystroke modifiable signals.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Registers KEYLED2 / KEYLED2_1 to KEYLED2_8.

Bit	Key	Bit	Key
0	Spindle override +	16	---
1	Spindle clockwise	17	---
2	Spindle positioning	18	---
3	Spindle stop	19	---
4	Spindle override -	20	---
5	Spindle counterclockwise	21	---
6	START	22	---
7	STOP	23	---
8	---	24	---
9	---	25	---
10	---	26	---
11	ZERO	27	---
12	---	28	---
13	Single block	29	---
14	---	30	---
15	RESET	31	---

KEYDIS1 / KEYDIS2 / KEYDIS3 / KEYDIS4

These registers inhibit (bit=1) the keys and the switches on all operator panels at the same time.

KEYDIS1_1 / KEYDIS2_1 / KEYDIS3_1 / KEYDIS4_1

..

KEYDIS1_8 / KEYDIS2_8 / KEYDIS3_8 / KEYDIS4_8

These registers inhibit (bit=1) the keys and the switches on each operator panel. Registers KEYDIS1_1 to KEYDIS4_1 correspond to the first jog panel, KEYDIS1_2 to KEYDIS4_2 to the second one and so on.

When selecting one of the inhibited positions of the feedrate override, the CNC will take the value of the lowest position allowed. If all of them are inhibited, it will take the value of 0%. For example, being allowed only the 110% and 120% positions, if the 50% position is selected, the CNC will take the value of 0%.

When selecting one of the inhibited positions of the spindle speed override, the CNC will take the value of the lowest position allowed. If all the positions are inhibited, the CNC keeps the active value.

The following instruction inhibits the first jog key (bit 16) of the second keyboard.

```
( ) = B16KEYDIS1_2
```

6.

LOGIC CNC INPUTS AND OUTPUTS.
Keystroke modifiable signals.

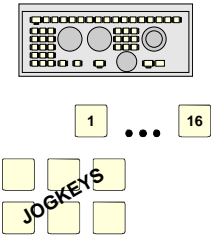
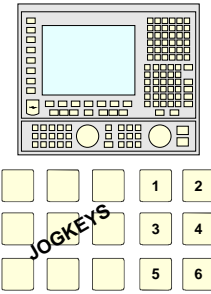


FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Registers KEYDIS1 / KEYDIS1_1 to KEYDIS1_8. User keys.

Bit	OP-PANEL	LCD-10K
		
0	User key 1	User key 1
1	User key 2	User key 2
2	User key 3	User key 3
3	User key 4	User key 4
4	User key 5	User key 5
5	User key 6	User key 6
6	User key 7	---
7	User key 8	---
8	User key 9	---
9	User key 10	---
10	User key 11	---
11	User key 12	---
12	User key 13	---
13	User key 14	---
14	User key 15	---
15	User key 16	---

6.

LOGIC CNC INPUTS AND OUTPUTS.

Keystroke modifiable signals.

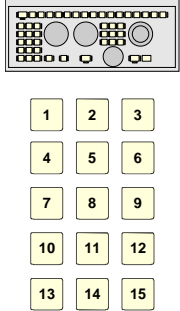
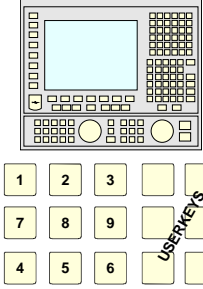


FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Registers KEYDIS1 / KEYDIS1_1 to KEYDIS1_8. Jog keys.

Bit	OP-PANEL	LCD-10K
		
16	Jog key 1	Jog key 1
17	Jog key 2	Jog key 2
18	Jog key 3	Jog key 3
19	Jog key 4	Jog key 4
20	Jog key 5	Jog key 5
21	Jog key 6	Jog key 6
22	Jog key 7	Jog key 7
23	Jog key 8	Jog key 8
24	Jog key 9	Jog key 9
25	Jog key 10	---
26	Jog key 11	---
27	Jog key 12	---
28	Jog key 13	---
29	Jog key 14	---
30	Jog key 15	---
31	----	---

Registers KEYDIS2 / KEYDIS2_1 to KEYDIS2_8.

Bit	Key	Bit	Key
0	Spindle override +	16	---
1	Spindle clockwise	17	---
2	Spindle positioning	18	---
3	Spindle stop	19	---
4	Spindle override -	20	---
5	Spindle counterclockwise	21	---
6	START	22	---
7	STOP	23	---
8	---	24	---
9	---	25	---
10	---	26	---
11	ZERO	27	---
12	---	28	---
13	Single block	29	---
14	---	30	---
15	RESET	31	---

6.

LOGIC CNC INPUTS AND OUTPUTS.
Keystroke modifiable signals.



CNC 8070

(REF: 1709)

Registers KEYDIS3 / KEYDIS3_1 to KEYDIS3_8. Feedrate override and movement selector (handwheel, incremental jog or continuous jog).

Bit	Selector
0	0 %
1	2 %
2	4 %
3	10 %
4	20 %
5	30 %
6	40 %
7	50 %
8	60 %
9	70 %
10	80 %
11	90 %
12	100 %
13	110 %
14	120 %
15	130 %

Bit	Selector
16	140 %
17	150 %
18	160 %
19	170 %
20	180 %
21	190 %
22	200 %
23	Handwheel x100
24	Handwheel x10
25	Handwheel x1
26	Jog 1
27	Jog 10
28	Jog 100
29	Jog 1000
30	Jog 10000
31	Continuous jog

Registers KEYDIS4 / KEYDIS4_1 to KEYDIS4_8. Spindle speed override selector.

Bit	Selector
0	0 %
1	2 %
2	4 %
3	10 %
4	20 %
5	30 %
6	40 %
7	50 %
8	60 %
9	70 %
10	80 %
11	90 %
12	100 %
13	110 %
14	120 %
15	130 %

Bit	Selector
16	140 %
17	150 %
18	160 %
19	170 %
20	180 %
21	190 %
22	200 %
23	---
24	---
25	---
26	---
27	---
28	---
29	---
30	---
31	---

6.

LOGIC CNC INPUTS AND OUTPUTS.

Keystroke modifiable signals.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

6.

LOGIC CNC INPUTS AND OUTPUTS.

Keystroke modifiable signals.



CNC 8070

(REF: 1709)

TOOL AND MAGAZINE MANAGEMENT.

7

To properly configure the tool magazines and the tool change, you must:

- Set the machine parameters.
- Set the tool table and the tool magazine table.
- Write the PLC program.
- Program the subroutine associated with the tool and with the M06 function.

The machine parameters define the number of tool magazines available and their characteristics. Up to four magazines may be used and each may be of a different type. See "[7.1 Types of tool magazine.](#)" on page 451.

When creating the PLC program and the subroutine associated with the tool and with the M06 function, you must bear in mind the number of magazines and channels available. For the communication between the tool manager and the PLC, each channel and each magazine has its own group of marks and registers.

The PLC maneuver will be different depending on the type of tool magazine. Later sections of this chapter show an example for each type of magazine.

About the magazines.

The CNC can have up to four magazines and each may be of a different type. Each magazine has its own configuration parameters.

The number of magazines is independent from the number of spindles and channels available. A magazine is not associated with any particular channel or spindle; i.e. a magazine may be shared by several channels and a channel can request tools from different magazines.



The access to the magazine depends on the mechanical configuration of the machine; in other words, to the physical possibility of the machine to access the magazines.

All the magazines can carry out tool changes simultaneously. However, one magazine can only be involved in a tool change process. If from one channel, one wishes to pick up or leave a tool in a magazine already involved in a tool change, the tool manager will wait for the tool change to be done before attending to the new request.

Two magazines may be involved in a tool change. The magazine receiving the tool and the magazine from where the new tool is picked up may be different.

About the tools.

Each tool is identified by its number, that is unique for the whole system; it cannot be repeated in different magazines nor in ground tools.

The list of available tools is saved in the tool table, the only one for the whole system. This table indicates the position and the magazine where each tool is located, whether it is a ground tool or the tool is active in a channel.

The tools are always stored in the same magazine. When doing a tool change, it is always stored in the magazine from which it was picked up.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Ground tools.

A ground tool is a tool that is not stored in any magazine and is loaded manually when requested. Ground tools are also defined in the tool table, but they are not associated with any magazine position.

Ground tool loading and unloading is global to the system; it is not associated with any particular magazine or channel.

Tool manager.

The CNC has a tool magazine management that knows at all times the location of each tool. When requesting a tool change or tool search, the tool manager "tells" the PLC the operations to be carried out.

- Take a tool from the magazine and insert it in the spindle.
- Leave the tool of the spindle in the magazine and take another one.
- Leave the spindle tool on the ground.
- Etc.

Depending on the type of magazine and the operation requested, several actions may be needed sometimes. For example, in some magazines, to take a tool from the spindle to the main magazine, the tool must be carried from the spindle to the changer arm and then from the changer arm to the magazine.

It is up to the PLC to control those movements. It must inform the manager about the actions carried out so it updates its information.

For the communication between the tool manager and the PLC, each channel and magazine has its own group of marks and registers. See "[7.3 Communication between manager and PLC.](#)" on page 454.

Subroutines associated with the tool change.

There are two subroutines associated with the tool change.

Subroutines associated with the tool.

The subroutine associated with the tool is executed automatically every time a T function (tool selection) is executed.

There is one subroutine in each channel.

Subroutines associated with the M06 function.

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

It is recommended to set this function in the "M" function table so it executes the subroutine for the tool changer installed on the machine.

This subroutine is common to the whole system.



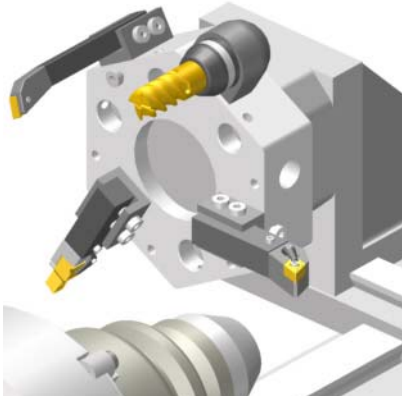
Within both subroutines, the "modal" condition of the canned cycles no longer has any influence. This characteristic is restored when the execution of the subroutine ends.

7.

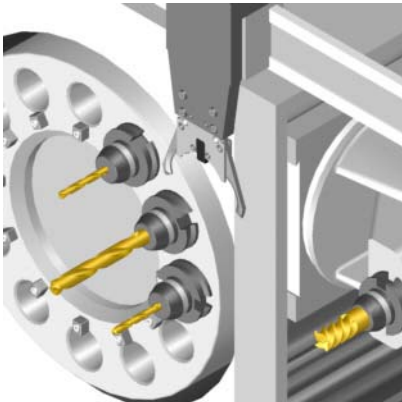
7.1 Types of tool magazine.

Tool magazines may be divided into 4 large groups:

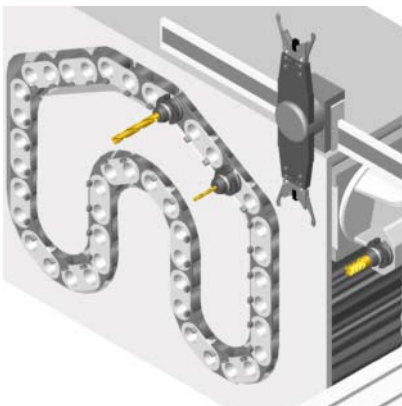
Turret type.



Synchronous magazine without changer arm.



Synchronous magazine with changer arm (1 or 2 holders). The arm cannot move while machining the part.



Asynchronous magazine. Tool changer arm of independent movements that may be moved while machining.

7.

TOOL AND MAGAZINE MANAGEMENT.
Types of tool magazine.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

"Random" or "Non-Random" magazine.

Depending on how the tools are stored in the magazine during a tool change, the magazine may be either random or non-random. In a random magazine, the tool may occupy any position whereas in a non-random magazine, the tools must always occupy the same position.

In any case and even when all the magazines are random, the tools are always stored in the same magazine from which they were picked up.

"Cyclic" or "Non-Cyclic" magazine.

A cyclic magazine requires a tool change command 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.

7.

TOOL AND MAGAZINE MANAGEMENT.
Types of tool magazine.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.2 Tool table, active tool table and tool magazine table

After setting the machine parameters for the tool magazines, define the tool table and then the magazine table.

Tool table.

The tool table must contain, among other things, the geometry, type of monitoring and tool size. All the tools must be defined, including the ground ones.

The tool manager inserts the special tools always in the same tool pocket regardless of the number of pockets they occupy.

Tool magazine table.

There is one table for each magazine. Each table shows which tool is in each pocket and on each holder of the changer arm (if any).

Although the magazine table may be initialized manually, it is up to the tool manager to dynamically update all its data.

Example:

Having 10 tools and a 10-position magazine. The tools are small, except T2 that is large and T4 that has a special size (0 to the left and 1 to the right).

Load the tools one by one in the magazine using the corresponding softkey.

- T1 in position 1.
- T2 in position 3. Verify that it cannot go in position 2 because the tool is too large.
- T3 in position 5. T2 occupies positions 2-3-4.
- T4 in position 6. T4 occupies positions 6-7.
- T5 in position 8.
- T6 in position 9.
- T7 in position 10.

T8, T9 and T10 do not fit in the magazine; therefore, they will be ground tools.

Active-tools table.

The active-tools table shows which tools are active in the spindles.

7.**TOOL AND MAGAZINE MANAGEMENT.**

Tool table, active tool table and tool magazine table

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.3 Communication between manager and PLC.

For the communication between the tool manager and the PLC, each channel and magazine has its own group of marks and registers.



The CNC maintains compatibility with the marks and registers of previous versions whose mnemonics do not refer to any particular channel or magazine. From software version V2.00 on, these signals refer to the first channel or magazine accordingly.

The communication between the manager and the PLC takes place in two stages; when executing the T function and when executing the M06 function.

- When executing the T function, the CNC lets the tool manager know about it.

The tool manager sends a command to the PLC to select the next tool in the magazine (if possible).

The CNC continues with the execution of the program without waiting for the tool manager to complete the operation.

- When executing the M06 function, its associated subroutine is called upon. The M06 function must also be programmed inside that subroutine so the CNC "tells" the tool manager to start making the tool change.

The tool manager sends a command to the PLC to make the change.

The CNC waits for the tool manager to complete the operation before continuing with the execution of the program.

Considerations and recommendations.

The management of the tool change should be included in the subroutine associated with the M06 and leave the control of the external devices up to the PLC.

Use the auxiliary functions to govern the various devices (magazine rotation, magazine movement, tool changer arm, etc.) from the M06 subroutine.

On asynchronous magazines (changer arm with independent movements) when the change implies leaving a tool in the magazine, the TCHANGEOK mark may be activated so the CNC goes on executing the program while the tool is being taken to the magazine.

7.

TOOL AND MAGAZINE MANAGEMENT.
Communication between manager and PLC.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.3.1 Manager --> PLC communication.

The manager uses the following registers and marks to inform the PLC about the operations it must carry out. Some signals are per channel whereas others are per magazine. The following table shows the mnemonics for each mark (M) or register (R) in each channel or magazine.

	Channel -1-	Channel -2-	Channel -3-	Channel -4-
M	TMOPSTROBE TMOPSTROBEC1	TMOPSTROBEC2	TMOPSTROBEC3	TMOPSTROBEC4
R	TMOPERATION TMOPERATIONC1	TMOPERATIONC2	TMOPERATIONC3	TMOPERATIONC4
R	MZIDC1	MZIDC2	MZIDC3	MZIDC4

	Magazine -1-	Magazine -2-	Magazine -3-	Magazine -4-
R	LEAVEPOS LEAVEPOSMZ1	LEAVEPOSMZ2	LEAVEPOSMZ3	LEAVEPOSMZ4
R	TAKEPOS TAKEPOSMZ1	TAKEPOSMZ2	TAKEPOSMZ3	TAKEPOSMZ4
R	NEXTPOS NEXTPOSMZ1	NEXTPOSMZ2	NEXTPOSMZ3	NEXTPOSMZ4

The meaning of the marks and registers is as follows.

Marks / Registers.	Meaning,
TMOPSTROBE	The channel tool manager sets this mark high (=1) to let the PLC know that it must execute the operation indicated by the TMOPERATION mark of the channel.
LEAVEPOS	This register indicates the magazine position to leave the tool. While selecting a turret position (#ROTATEMZ instruction), this register takes the value of -0- if it is a positive relative positioning and the value of -1- if it is a negative relative positioning.
TAKEPOS	This register indicates the magazine position of the tool to be taken. While selecting a turret position (#ROTATEMZ instruction), this register takes the value of -0- if it is an absolute positioning and the value of -1- if it is a relative positioning.
NEXTPOS	This register indicates the magazine position occupied by the next tool. While selecting a turret position (#ROTATEMZ instruction); in an absolute positioning, this register indicates the position to reach and, in a relative positioning, the number of positions to rotate.
MZID	This register indicates which magazine contains the tool requested by the channel. 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.
TMOPERATION	This register indicates the type of operation to be carried out by the tool manager.

7.

TOOL AND MAGAZINE MANAGEMENT.
Communication between manager and PLC.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.3.2 PLC --> Manager communication.

The PLC uses the following marks to inform the manager about the operations that it has carried out. There is one group of marks for each magazine.

The PLC, depending on the type of magazine, must take some actions to execute the operations requested by the manager. After ending each one of them, it must activate certain marks to inform the manager that the action has been completed. The manager sets them back to "0" once they are read.

The following table shows the mnemonics for each mark (M) or register (R) in each channel or magazine.

	Magazine -1-	Magazine -2-	Magazine -3-	Magazine -4-
M	MZTOCH1 MZTOCH1MZ1	MZTOCH1MZ2	MZTOCH1MZ3	MZTOCH1MZ4
M	CH1TOSPD CH1TOSPDLMZ1	CH1TOSPDLMZ2	CH1TOSPDLMZ3	CH1TOSPDLMZ4
M	SPDLTOCH1 SPDLTOCH1MZ1	SPDLTOCH1MZ2	SPDLTOCH1MZ3	SPDLTOCH1MZ4
M	SPDLTOCH2 SPDLTOCH2MZ1	SPDLTOCH2MZ2	SPDLTOCH2MZ3	SPDLTOCH2MZ4
M	CH1TOMZ CH1TOMZ1	CH1TOMZ2	CH1TOMZ3	CH1TOMZ4
M	CH2TOMZ CH2TOMZ1	CH2TOMZ2	CH2TOMZ3	CH2TOMZ4
M	SPDLTOGR SPDLTOGRC1	SPDLTOGRC2	SPDLTOGRC3	SPDLTOGRC4
M	GRTOSPD GRTOSPDLC1	GRTOSPDLC2	GRTOSPDLC3	GRTOSPDLC4
M	MZTOSPD MZTOSPDLMZ1	MZTOSPDLMZ2	MZTOSPDLMZ3	MZTOSPDLMZ4
M	SPDLTOMZ SPDLTOMZ1	SPDLTOMZ2	SPDLTOMZ3	SPDLTOMZ4
M	MZROT MZROTMZ1	MZROTMZ2	MZROTMZ3	MZROTMZ4
M	TCHANGEOK TCHANGEOKMZ1	TCHANGEOKMZ2	TCHANGEOKMZ3	TCHANGEOKMZ4
R	MZPOS MZPOSZ1	MZPOSZ2	MZPOSZ3	MZPOSZ4

The meaning of the marks and registers is as follows.

Marks / Registers.	Meaning,
MZTOCH1	Use it with an asynchronous magazine or synchronous with arm. The PLC must set this mark high (=1) after taking the tool from the magazine to the changer arm 1.
CH1TOSPD	Use it with an asynchronous magazine or synchronous with arm. The PLC must set this mark high (=1) after taking the tool from the changer arm 1 to the spindle.
SPDLTOCH1	Use it with an asynchronous magazine with one changer arm. The PLC must set this mark high (=1) after taking the tool from the spindle to the changer arm 1.
SPDLTOCH2	Use it with an asynchronous magazine or synchronous with arm. The PLC must set this mark high (=1) after taking the tool from the spindle to the changer arm 2.
CH1TOMZ	Use it with an asynchronous magazine or synchronous with arm. The PLC must set this mark high (=1) after taking the tool from the tool changer arm 1 to the magazine.

7.

TOOL AND MAGAZINE MANAGEMENT.
Communication between manager and PLC.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Marks / Registers.	Meaning.
CH2TOMZ	Use it with an asynchronous magazine or synchronous with arm. The PLC must set this mark high (=1) after taking the tool from the tool changer arm 2 to the magazine.
SPDLTOGR	Use it with a magazine that admits ground tools. The PLC must set this mark high (=1) after taking the tool from the spindle to ground.
GRTOSPDL	Use it with a magazine that admits ground tools. The PLC must set this mark high (=1) after taking the tool from the ground to the spindle.
MZTOSPDL	Use it with a synchronous magazine (without arm). The PLC must set this mark high (=1) after taking the tool from the magazine to the spindle.
SPDLTOMZ	Use it with a synchronous magazine (without arm). The PLC must set this mark high (=1) after taking the tool from the spindle to the magazine.
MZROT	Use it with a turret-type magazine and with a synchronous magazine. The PLC must set this mark high (=1) when the turret has rotated. In the synchronous magazine, it is used to optimize the change by orienting the magazine while machining. The PLC must set this mark to (=1) to indicate that the operation has been completed, whether it has been oriented or not.
TCHANGEOK	On asynchronous magazines (with changer arm of independent movements) the following mark may be activated so the CNC goes on executing the program while the tool is being taken to the magazine. The PLC must set this mark high (=1) to "tell" the manager to go on executing the program while the tool is being taken to the magazine.
MZPOS	On random magazines, magazine orientations may be optimized if the manager knows the position selected at all times. The PLC must indicate in this register the current position of the magazine; when not using this register, the PLC must set it to ·0·. On turret magazines, it is possible to select a position (#ROTATEMZ instruction). The PLC must indicate in this register the current position of the magazine; if the register has a value of ·0·, the PLC will issue the corresponding error.

7.

TOOL AND MAGAZINE MANAGEMENT.

Communication between manager and PLC.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.3.3 Manager Emergency.

The manager may be set in an Emergency state when a malfunction occurs (the PLC has executed the wrong action, incomplete tool change, etc.) or if the PLC activates the emergency.

The PLC marks related to manager emergency are the following: There is one group of marks for each magazine.

The following table shows the mnemonics for each mark (M) in each channel or magazine.

	Magazine -1-	Magazine -2-	Magazine -3-	Magazine -4-
M	SETTMEM SETTMEMZ1	SETTMEMZ2	SETTMEMZ3	SETTMEMZ4
M	RESTMEM RESTMEMZ1	RESTMEMZ2	RESTMEMZ3	RESTMEMZ4
M	TMINEM TMINEMZ1	TMINEMZ2	TMINEMZ3	TMINEMZ4

The meaning of the marks and registers is as follows.

Marks / Registers.	Meaning,
SETTMEM	The PLC sets this mark high (=1) to activate the tool manager emergency.
RESTMEM	The PLC sets this mark high (=1) to cancel the tool manager emergency.
TMINEM	The CNC sets this mark high (=1) to inform the PLC that an emergency has occurred at the tool manager. To generate an emergency at the manager from the PLC: (1) Activate the emergency. DFU (error condition) = SET SETTMEM (2) Make sure that the emergency has occurred before canceling it. TMINEM AND DFU (removal condition) = SET RESTMEM The manager sets the SETTMEM and RESTMEM signals low (=0).

7.

7.3.4 Tool monitoring.

The PLC marks related to tool monitoring are the following. There is one group of marks for each channel.

The following table shows the mnemonics for each mark (M) in each channel or magazine.

	Channel -1-	Channel -2-	Channel -3-	Channel -4-
M	CUTTINGON CUTTINGONC1	CUTTINGONC2	CUTTINGONC3	CUTTINGONC4
M	TREJECT TREJECTC1	TREJECTC2	TREJECTC3	TREJECTC4
M	TWORNOUT TWORNOUTC1	TWORNOUTC2	TWORNOUTC3	TWORNOUTC4

The meaning of the marks and registers is as follows.

Marks / Registers.	Meaning.
CUTTINGON	When a tool is assigned a maximum life span (monitoring), the CNC checks this mark in order to know whether the tool is machining (=1) or not (=0). It is usually considered to be machining when the following conditions are met: <ul style="list-style-type: none"> • The spindle is turning (M3 or M4) or when the tool is threadcutting and the 0% of feedrate is not selected. • The automatic operating mode is selected, there is a block in execution and the G00 function is not active. • The execution is not interrupted.
TREJECT	If the PLC sets this mark high (=1), the CNC interprets that the tool must be rejected.
TWORNOUT	The CNC sets this mark high (=1) to "tell" the PLC that the tool has been rejected because it is worn out (real life > maximum life span).

7.

TOOL AND MAGAZINE MANAGEMENT.
Communication between manager and PLC.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.4 Variables related to tool magazine management.

The variables associated with the magazine that are involved in the tool magazine management are the following. There is one group of variables for each channel. Replace the [ch] character with the channel number, maintaining the brackets.

V.[ch].TM.MZMODE

Operating mode of the tool magazine manager.

With a ·0· value	Normal mode (by default and after reset).
With a ·1· value	Magazine loading mode.
With a ·2· value	Magazine unloading mode.

V.[ch].TM.MZSTATUS

Tool manager status.

With a ·0· value	Normal.
With a ·1· value	An error has occurred.
With a ·2· value	An error has occurred; but it will wait for the current maneuver to be completed.
With a ·4· value	Emergency.

V.[ch].TM.MZRUN

Tool manager running.

With a ·0· value	There is no sequence in execution.
With a ·1· value	There is a sequence in execution.

V.[ch].TM.MZWAIT

Tool manager executing a maneuver. It indicates whether to wait or not for the maneuver to end.

With a ·0· value	No need to wait.
With a ·1· value	It has to wait.

There is no need to program it in the subroutine associated with M06. The subroutine itself waits for the manager's maneuvers to finish. This way, block preparation is not interrupted.

7.

TOOL AND MAGAZINE MANAGEMENT.
Variables related to tool magazine management.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.5 Tool loading and unloading from the magazines.

Tool loading and unloading from the magazine.

Each magazine table has softkeys for initializing, loading and unloading tools in the magazine either manually or automatically. Refer to the operating manual.

The tools may also be loaded and unloaded in the magazine by program or in MDI mode.

Tool loading and unloading from the spindle.

The tools must always be loaded and unloaded in the spindle and on the changer arm in manual mode from the magazine table. Refer to the operating manual.

Loading the magazine by program or in MDI mode.

The tools are loaded in the magazine by taking them one by one from ground to the magazine going through the spindle.

Set the variable: V.TM.MZMODE=1 to "tell" the tool manager that the magazine loading mode has been chosen.

If a T1 M6 is executed next, the tool manager interprets that T1 must be taken from ground to the magazine going through the spindle and it will let the PLC know by setting TMOOPERATION=9.

The subroutine associated with the M06 function and the PLC program must contain the maneuver needed to make the tool change.

After loading the tool, the tool manager updates the magazine table.

Unloading the magazine by program or MDI.

The tools are unloaded from the magazine taking them one by one from the magazine to the ground going through the spindle.

Set variable: V.TM.MZMODE=2 to "tell" the tool manager that the magazine unloading mode has been chosen.

If a T1 M6 is executed next, the tool manager interprets that T1 must be taken from magazine to the ground going through the spindle and it will let the PLC know by setting TMOOPERATION=10.

The subroutine associated with the M06 function and the PLC program must contain the maneuver needed to make the tool change.

After unloading the tool, the tool manager updates the magazine table.

Placing a tool in the spindle by program or in MDI mode

Set variable: V.TM.MZMODE=0 to "tell" the tool manager that the normal mode has been chosen.

If T1 M6 is executed next, the tool manager checks whether there is already a tool in the spindle (to remove it first) and whether the requested tool is in the magazine or it has to be picked up from the ground. In any case, it lets the PLC know which operation it must carry out by setting TMOOPERATION to the proper value.

The subroutine associated with the M06 function and the PLC program must contain the maneuver needed to make the tool change.

After placing the tool, the tool manager updates the magazine table.

7.

TOOL AND MAGAZINE MANAGEMENT.

Tool loading and unloading from the magazines.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.6 Magazine-less system.

When there is no tool magazine, changing the tool only requires programming the T function without the need for M6. In this circumstances, a programmed M6 function is not interpreted as a tool change, but as just another M function without any special meaning. This does not prevent it from having an associated subroutine like any other M function.

It will be synchronized with the tool manager after the tool change (MZWAIT) after managing the T block in execution.

Execution of the T function.

- 1 When the CNC executes the T function, it lets the tool manager know about it.
- 2 The tool manager sends a command to the PLC to make the change.
- 3 The CNC waits for the tool manager to complete the operation before continuing with the execution of the program.

Behavior of function M06.

When there is no tool magazine, the M6 function behaves as follows.

- The M6 function will not activate PLC marks (e.g. DM06) nor will it execute anything related to the tool change (like the tool change subroutine, etc.).
- The M6 function will not be displayed in the M function history.
- The subroutine associated with the M6 function (if it has one) will execute modal canned cycles or modal subroutines with motion.
- The subroutine associated with M6 will have no special treatment when called upon from hidden files, canned cycles, etc.

7.

TOOL AND MAGAZINE MANAGEMENT.
Magazine-less system.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.6.1 Valid operations and marks activated by the PLC with each one of them.

Valid operations of the tool manager for this magazine.

The possible operations in this type of magazine are:

TMOPERATION	Meaning.
0	Do nothing.
3	Insert a ground tool in the spindle.
4	Leave the spindle tool on the ground.
8	Leave the spindle tool on the ground and take a tool from ground.

Values of registers TAKEPOS and LEAVEPOS used by the manager.

The TAKEPOS and LEAVEPOS signals do not assume any value. Here is a summary table of the marks that must be activated by the PLC at the end of each operation.

TM => PLC			PLC => TM
TMOPERATION	TAKEPOS	LEAVEPOS	
3	0	0	GRTOSPDL
4	0	0	SPDLTOGR
8	0	0	SPDLTOGR + GRTOSPDL

Example of operation.

The following example shows, assuming that there is no tool in the spindle, it shows the functions executed by the CNC, the values sent by the tool manager to the PLC in each operation and the marks to be activated by the PLC in each case.

It is a non-random magazine, each tool occupies the position of its own number and T7 and T8 are ground tools.

CNC	TMOPERATION	TAKEPOS	LEAVEPOS	
T7	3	0	0	GRTOSPDL
T8	8	0	0	SPDLTOGR + GRTOSPDL
T0	4	0	0	SPDLTOGR



TOOL AND MAGAZINE MANAGEMENT.
 Magazine-less system.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.6.2 Detailed description of the operations of the magazine.

Here is a sample of a detailed description of the valid operations for this magazine. In a system without magazines, the TAKEPOS, LEAVEPOS, NEXTPOS and MZID signals are not involved.

TMOPERATION = 3

Insert a ground tool in the spindle.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark: GRTOSPDL.

TMOPERATION = 4

Leave the spindle tool on the ground.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark: SPDLTOGR.

TMOPERATION = 8

Leave the spindle tool on the ground and take a tool from ground.

The sequence of this operation is the following.

- 1 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.
- 2 Leave the ground tool in the spindle and activate the mark GRTOSPDL.

7.6.3 Basic PLC programming.

When executing the T function.

When executing the T function, the tool manager sends to the PLC, in the TMOPERATION register, the code for the operation to be carried out.

DFU TMOPSTROBE AND (CPS TMOPERATION EQ 3) = SET GRTOSPDL

A tool change has been requested and TMOPERATION=3. The PLC has taken the tool from ground to the spindle.

DFU TMOPSTROBE AND (CPS TMOPERATION EQ 4) = SET SPDLTOGR

A tool change has been requested and TMOPERATION=4. The PLC has taken the tool from the spindle to ground.

DFU TMOPSTROBE AND (CPS TMOPERATION EQ 8) = SET SPDLTOGR

A tool change has been requested and TMOPERATION=4. The PLC has taken the tool from the spindle to ground.

DFD SPDLTOGR AND (CPS TMOPERATION EQ 8) = SET GRTOSPDL

The manager has canceled the SPDLTOGR mark and TMOPERATION=8. The PLC has taken the tool from ground to the spindle.

7.

TOOL AND MAGAZINE MANAGEMENT.
Magazine-less system.



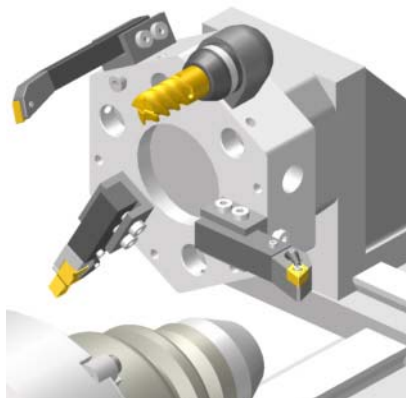
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.7 Turret type magazine.

It is a typical magazine for lathes. To make the change, the magazine rotates until it places the new tool in the work position. The tool cannot be changed while the part is being machined.



Usually, in this type of magazine, the communication between the manager and the PLC is configured so it is carried out in a single stage, with the execution of the T function.

It is recommended that the routine associated with T include the M06 function. When the CNC selects a tool, the routine associated with the T executes the M06 function (tool change), the machining stops and the tool manager sends the PLC the code for the operation to be carried out. If the subroutine associated with T does not include the function M06, when the CNC selects a tool, it internally executes a T# M06 block to make the change.

Subroutine associated with T.	Behavior when the T function is executed.
Exists.	The CNC executes the T function and the subroutine executes the M06 block.
It does not exist.	The CNC internally generates a T# M06 block.

- 1 When the CNC executes the T function, it lets the tool manager know about it and calls the associated subroutine. The M06 function must be programmed inside that subroutine so the CNC "tells" the tool manager to start making the tool change.
- 2 The tool manager sends a command to the PLC to select the next tool in the magazine.
- 3 The subroutine associated with the T function executes the M06 function. The M06 subroutine can also have an associated subroutine. If a subroutine associated with the M06 function is defined, the M06 function must also be programmed inside that subroutine so the CNC "tells" the tool manager to start making the tool change.
- 4 The tool manager sends a command to the PLC to make the change.
- 5 The CNC waits for the tool manager to complete the operation before continuing with the execution of the program.

The management of the tool change should be included in the subroutine associated with the M06 and leave the control of the external devices up to the PLC. Use the auxiliary functions to govern the various devices (magazine rotation, magazine movement, tool changer arm, etc.) from the M06 subroutine.

Ground tools in a turret type magazine.

This magazine offers the possibility of working with ground tools. If there is a tool in the work position and another one is requested that is not in the turret, the CNC considers it to be a ground tool.

The maneuver for loading/unloading a tool in the spindle (in the work position) either directly or going through the spindle is the same. In the latter case, the variable V.TM.MZMODE must have the proper value to load or unload.

7.

TOOL AND MAGAZINE MANAGEMENT.

Turret type magazine.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.7.1 Valid operations and marks activated by the PLC with each one of them.

Valid operations of the tool manager for this magazine.

The possible operations in this type of magazine are:

TMOPERATION	Meaning.
0	Do nothing.
1	Assume the current tool as active tool.
3	Insert a ground tool in the spindle.
4	Leave the spindle tool on the 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 turret.
15	Select a turret position.

Values of registers TAKEPOS and LEAVEPOS used by the manager.

The TAKEPOS and LEAVEPOS signals can assume the following values.

Value	Meaning
0	Do nothing.
#	Magazine position.
-4	Ground tool.

If TMOPERATION = 15, it is a special case where the TAKEPOS, LEAVEPOS and NEXTPOS signals can assume the following values.

Signal.	Meaning.
TAKEPOS	This register takes the value of ·0· if it is an absolute positioning and the value of ·1· if it is a relative positioning.
LEAVEPOS	This register takes the value of ·0· if it is a positive relative positioning and the value of ·1· if it is a negative relative positioning.
NEXTPOS	In an absolute positioning, this register indicates the position to reach and, in a relative positioning, the number of positions to rotate.

Here is a summary table of the TAKEPOS and LEAVEPOS values used by the manager in each operation as well as the marks that must be activated by the PLC at the end of each operation.

TM => PLC			PLC => TM
TMOPERATION	TAKEPOS	LEAVEPOS	
1	#	0	MZTOSPD
3	0	0	GRTOSPD
4	0	0	SPDLTOGR
9	-4	#	GRTOSPD
10	#	-4	SPDLTOGR
11	0	0	MZROT
15	- - -	- - -	MZROT

Example of operation.

The following example shows, assuming that there is no tool in the spindle, it shows the functions executed by the CNC, the values sent by the tool manager to the PLC in each operation and the marks to be activated by the PLC in each case.

It is a non-random magazine, each tool occupies the position of its own number and T7 is a ground tool.

CNC	TMOPERATION	TAKEPOS	LEAVEPOS	
T1 M6	11 1	0 1	0 0	MZROT MZTOSPD
T2 M6	11 1	0 2	0 0	MZROT MZTOSPD
T7 M6	11 3	0 0	0 0	GRTOSPD

7.

TOOL AND MAGAZINE MANAGEMENT.

Turret type magazine.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.7.2 Detailed description of the operations of the magazine.

Here is a sample of a detailed description of the valid operations for this magazine. For each operation, it indicates which of the TAKEPOS, LEAVEPOS, NEXTPOS and MZID signals are involved and their meanings. It also shows the sequence to follow to complete the operation.

The TAKEPOS, LEAVEPOS and NEXTPOS signals can assume the following values.

Value	Meaning
0	Do nothing.
#	Magazine position.
-4	Ground tool.

TMOPERATION = 1

Assume the current tool as active tool.

TAKEPOS Magazine position to pick up the tool.

The sequence of this operation is the following.

- 1 After ending this operation, activate the MZTOSPD L mark.

TMOPERATION = 3

Insert a ground tool in the spindle.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark: GRTOSPD L.

TMOPERATION = 4

Leave the spindle tool on the ground.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark: SPDLTOGR.

TMOPERATION = 9

Take a ground tool to the magazine going through the spindle.

TAKEPOS Ground tool.

LEAVEPOS Magazine position to leave the tool.

The sequence of this operation is the following.

- 1 Leave the ground tool in the spindle and activate the mark GRTOSPD L.

TMOPERATION = 10

Take a tool from the magazine and leave on the ground going through the spindle.

TAKEPOS Magazine position to pick up the tool.

LEAVEPOS Ground tool.

The sequence of this operation is the following.

- 1 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.

TMOPERATION = 11

Orient the magazine.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark MZROT.

7.

TOOL AND MAGAZINE MANAGEMENT.
Turret type magazine.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TMOPERATION = 15

Select a magazine position.

TAKEPOS This register takes the value of ·0· if it is an absolute positioning and the value of ·1· if it is a relative positioning.

LEAVEPOS This register takes the value of ·0· if it is a positive relative positioning and the value of ·1· if it is a negative relative positioning.

NEXTPOS In an absolute positioning, this register indicates the position to reach and, in a relative positioning, the number of positions to rotate.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark MZROT.

7.

TOOL AND MAGAZINE MANAGEMENT.

Turret type magazine.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.7.3 Communication between the PLC and the M06 subroutine.

The communication between the PLC and the M06 subroutine takes place using a series of generic marks and registers. The program of the M06 subroutine offered as an example uses the following marks and registers.

Communication between the PLC and the M06 subroutine

Marks activated by the M06 subroutine so the PLC activates the relevant mark of the tool manager.

Mark.	Meaning.
M1107	SPDLTOGR The tool has been taken from the spindle to ground.
M1108	GRTOSPDL The tool has been taken from ground to the spindle.
M1109	MZTOSPDL Assume the current tool as active tool.
M1110	SPDLTOGR Assume the current tool as active tool.
M1111	MZROT Mark that indicates that the magazine has rotated.

M functions at the PLC

M functions that imply movements controlled by the PLC:

Function.	Meaning.
M109	Select in the magazine the position indicated by TAKEPOS and insert the tool in the spindle.
M110	Select in the magazine the position indicated by LEAVEPOS and leave the spindle tool.

Set all the M functions with "before-before" synchronization to the program continues after the M function is completed.

7.

TOOL AND MAGAZINE MANAGEMENT.
Turret type magazine.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.7.4 Program of the M06 subroutine.

%L SUB_MZ_ROT

```
V.PLC.M[1111]=1
#WAIT FOR [V.PLC.M[1111]=0]; MZROT mark to the tool manager.
#RET
```

%L SUB_SPD_TO_GR ; Remove the tool from the spindle (take it to ground).

```
#MSG ["Extract tool T%D and press START", V.TM.TOOL]
; Message for the operator to extract the tool.
M0 ; Wait for the operation to be completed.
#MSG [""] ; Remove message.
V.PLC.M[1107]=1 ; SPDLTOGR mark to the tool manager.
#WAIT FOR [V.PLC.M[1107]=0]
#RET
```

%L SUB_GR_TO_SPD ; Insert the ground tool in the spindle.

```
#MSG ["Insert tool T%D and press START", V.TM.NXTOOL]
; Message for the operator to insert the tool.
M0 ; Wait for the operation to be completed.
#MSG [""] ; Remove message.
V.PLC.M[1108]=1 ; GRTOSPDL mark to the tool manager.
#WAIT FOR [V.PLC.M[1108]=0]
#RET
```

%L SUB_MZ_TO_SPD ; Take the tool from the magazine and insert it in the spindle.

```
M109 ; Auxiliary function to execute an action.
V.PLC.M[1109]=1 ; MZTOSPDL mark to the tool manager.
#WAIT FOR [V.PLC.M[1109]=0]
#RET
```

%L SUB_SPD_TO_MZ ; Leave the spindle tool in the magazine.

```
M110 ; Auxiliary function to execute an action.
V.PLC.M[1110]=1 ; SPDLTOMZ mark to the tool manager.
#WAIT FOR [V.PLC.M[1110]=0]
#RET
```

%L SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.

```
G1 Z_ F_ ; Move the spindle.
#RET
```

%L SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point

```
G1 Z_ F_ ; Move the spindle.
#RET
```

%SUB_M6.nc

```
#ESBLK ; Begin of single block activation
#DSTOP ; STOP key disable
M6 ; Order the tool manager to start the tool change.
```

```
$IF [[V.G.FULLSTATUS & 255]<9] | [[V.G.FULLSTATUS & 255]>13]
$IF [[V.G.CNCAUTSTATUS & 4096] | [V.G.CNCAUTSTATUS & 8192]]
V.S.EXECUTION = 0 ; Simulated Theoretical or G
$ELSE
V.S.EXECUTION = 1 ; Execution
$ENDIF
$ELSE
V.S.EXECUTION = 0 ; Simulation
$ENDIF
```

```
$IF [V.S.EXECUTION == 1] ; CNC in execution
```

```
$SWITCH V.PLC.TMOPERATION ; It analyzes the type of operation.
```

7.

TOOL AND MAGAZINE MANAGEMENT.

Turret type magazine.


FAGOR

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

```

$CASE 1 ; Assume the current tool as active tool.
  LL SUB_MZ_TO_SPD
$BREAK

$CASE 3 ; Insert the ground tool in the spindle.
  LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
  LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
$BREAK

$CASE 4 ; Leave the spindle tool on the ground.
  LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
  LL SUB_SPD_TO_GR ; Remove the tool from the spindle.
$BREAK

$CASE 9 ; Take a ground tool to the magazine going through the
          spindle.
  LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
  LL SUB_SPD_TO_MZ ; Leave the spindle tool in the magazine.
$BREAK

$CASE 10 ; Take a tool from the magazine and leave on the
          ground going through the spindle.
  LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
  LL SUB_MZ_TO_SPD ; Take the tool from the magazine to the spindle.
$BREAK

$CASE 11 ; Orient the turret.
  LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
  LL SUB_MZ_ROT
$BREAK

$ENDSWITCH ; End of analysis of type of operation.

$WHILE V.TM.MZWAIT == 1 ; Wait for the tool manager.
$ENDWHILE

$ENDIF
#DSBLK ; End of single block activation.
#ESTOP ; STOP key enable
#RET

```

7.

TOOL AND MAGAZINE MANAGEMENT.

Turret type magazine.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.7.5 Basic PLC programming.

When executing the -T- function

When executing a T function, the tool manager sends the code TMOOPERATION=11 to the PLC. In general, it is an optimization of the change that permits orienting the magazine while machining.

```
DFU TMOPSTROBE AND CPS TMOOPERATION EQ 11 = SET MZROT
```

Activate the MZROT mark to "tell" the tool manager that the operation has finished.

When executing the M06 function

Every time the M06 subroutine ends an action, it lets the PLC know so it activates the relevant mark of the tool manager.

```
DFU M1107 = SET SPDLTOGR
DFD SPDLTOGR = RES M1107
```

The tool has been taken from the spindle to ground.

```
DFU M1108 = SET GRTOSPDL
DFD GRTOSPDL = RES M1108
```

The tool has been taken from ground to the spindle.

```
DFU M1109 = SET MZTOSPDL
DFD MZTOSPDL = RES M1109
```

The tool has been taken from the magazine to the spindle.

```
DFU M1110 = SET SPDLTOGR
DFD SPDLTOGR = RES M1110
```

The tool has been taken from the spindle to the magazine.

```
DFU M1111 = SET MZROT
DFD MZROT = RES M1111
```

The magazine has rotated already.

The M06 subroutine uses the following M functions to "tell" the PLC which movements it must carry out.

- | | |
|------|--|
| M109 | Select in the magazine the position indicated by TAKEPOS and insert the tool in the spindle. |
| M110 | Select in the magazine the position indicated by LEAVEPOS and leave the spindle tool. |

Programming it depends on the type of machine. The auxiliary function will conclude after executing the requested movement.

Certain operations require using the information transferred by the tool manager in the following registers:

LEAVEPOS This register indicates the magazine position to leave the tool.

TAKEPOS This register indicates the magazine position of the tool to be taken.

Manager emergency signal.

Treatment of the tool manager emergency signal.

```
DFU B11KEYBD1 AND NOT TMINEM = SET SETTMEM
DFU TMINEM = RES SETTMEM
```

Pressing the USER12 key activates the emergency.

```
TMINEM = B11KEYLED1
```

The lamp of the USER12 key turns on when there is an emergency.

```
TMINEM AND DFU B12KEYBD1 = SET RESTMEM
```

Pressing the USER13 key removes the emergency.

7.

TOOL AND MAGAZINE MANAGEMENT.

Turret type magazine.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.8 Synchronous magazine without changer arm.

In a synchronous tool changer without arm, the magazine must move up to the spindle to change the tool. The tool cannot be changed while the part is being machined.



The communication between the manager and the PLC takes place in two stages; the first one when executing the T function and the second one when executing the M06 function.

Execution of the T function.

- 1 When the CNC executes the T function, it lets the tool manager know about it.
- 2 The tool manager sends a command to the PLC to select the next tool in the magazine (if possible).
- 3 The CNC continues with the execution of the program without waiting for the tool manager to complete the operation.

Execution of the M06 function.

- 1 When the CNC executes the M06 function, it calls the associated subroutine. The M06 function must also be programmed inside that subroutine so the CNC "tells" the tool manager to start making the tool change.
- 2 The tool manager sends a command to the PLC to make the change.
- 3 The CNC waits for the tool manager to complete the operation before continuing with the execution of the program.

The management of the tool change should be included in the subroutine associated with the M06 and leave the control of the external devices up to the PLC. Use the auxiliary functions to govern the various devices (magazine rotation, magazine movement, etc.) from the M06 subroutine.

7.

TOOL AND MAGAZINE MANAGEMENT.
Synchronous magazine without changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.8.1 Valid operations and marks activated by the PLC with each one of them.

Valid operations of the tool manager for this magazine.

The possible operations in this type of magazine are:

TMOPERATION	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 a tool from the same magazine.
6	Leave the spindle tool in the magazine and take a tool from ground.
7	Leave the spindle tool on the ground and take a tool from the magazine.
8	Leave the spindle tool on the ground and take a tool 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 a tool from the same magazine (as TMOPERATION=5). It is an optimized operation, only valid when the magazine is random and the tool is special.
13	Orienting two magazines.
14	Leave the spindle tool in the magazine and take a tool from another magazine.

Values of registers TAKEPOS and LEAVEPOS used by the manager.

The TAKEPOS and LEAVEPOS signals can assume the following values.

Value	Meaning
0	Do nothing.
#	Magazine position.
-4	Ground tool.

Here is a summary table of the TAKEPOS and LEAVEPOS values used by the manager in each operation as well as the marks that must be activated by the PLC at the end of each operation.

TM => PLC			PLC => TM
TMOPERATION	TAKEPOS	LEAVEPOS	
1	#	0	MZTOSPDL
2	0	#	SPDLTOMZ
3	0	0	GRTOSPDL
4	0	0	SPDLTOGR
5	#	#	SPDLTOMZ + MZTOSPDL
6	-4	#	SPDLTOMZ + GRTOSPDL
7	#	-4	SPDLTOGR + MZTOSPDL
8	0	0	SPDLTOGR + GRTOSPDL
9	-4	#	GRTOSPDL + SPDLTOMZ

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine without changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TM => PLC			PLC => TM
TMOPERATION	TAKEPOS	LEAVEPOS	
10	#	-4	MZTOSPDL + SPDLTOGR
11	0	0	MZROT
12	#	#	SPDLTOMZ + MZTOSPDL
13	#	0	MZROT + MZROT
14	#	#	SPDLTOMZ + MZTOSPDL

Example of operation.

The following example shows, assuming that there is no tool in the spindle, it shows the functions executed by the CNC, the values sent by the tool manager to the PLC in each operation and the marks to be activated by the PLC in each case.

It is a non-random magazine, each tool occupies the position of its own number and T7, T8 and T9 are ground tools.

CNC	TMOPERATION	TAKEPOS	LEAVEPOS	
T1 M6	11 1	0 1	0 0	MZROT MZTOSPDL
T2 M6	11 5	0 2	0 1	MZROT SPDLTOMZ + MZTOSPDL
T7 M6	11 6	0 -4	0 2	MZROT SPDLTOMZ + GRTOSPDL
T8 M6	11 8	0 0	0 0	MZROT SPDLTOGR + GRTOSPDL
T3 T4 M6	11 11 7	0 0 4	0 0 -4	MZROT MZROT SDPLTOGR + MZTOSPDL
T0 M6	11 2	0 0	0 4	MZROT SPDLTOMZ
T9 M6	11 3	0 0	0 0	MZROT GRTOSPDL
T0 M6	11 4	0 0	0 0	MZROT SPDLTOGR

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine without changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.8.2 Detailed description of the operations of the magazine.

Here is a sample of a detailed description of the valid operations for this magazine. For each operation, it indicates which of the TAKEPOS, LEAVEPOS, NEXTPOS and MZID signals are involved and their meanings. It also shows the sequence to follow to complete the operation.

The TAKEPOS, LEAVEPOS and NEXTPOS signals can assume the following values.

Value	Meaning
0	Do nothing.
#	Magazine position.
-4	Ground tool.

TMOPERATION = 1

Take a tool from the magazine and insert it in the spindle.

TAKEPOS=# Magazine position to pick up the tool.

The sequence of this operation is the following.

- 1 After ending this operation, activate the MZTOSPD L mark.

TMOPERATION = 2

Leave the spindle tool in the magazine.

LEAVEPOS=# Magazine position to leave the tool.

The sequence of this operation is the following.

- 1 After ending this operation, activate the SPDLTOMZ mark.

TMOPERATION = 3

Insert a ground tool in the spindle.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark: GRTOSPD L.

TMOPERATION = 4

Leave the spindle tool on the ground.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark: SPDLTOGR.

TMOPERATION = 5

Leave the spindle tool in the magazine and take a tool from the same magazine.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=# Magazine position to leave the tool.

The sequence of this operation is the following.

- 1 Leave the tool of the spindle in the magazine and activate the mark SPDLTOMZ.
- 2 Leave the tool of the magazine in the spindle and activate the mark MZTOSPD L.

TMOPERATION = 6

Leave the spindle tool in the magazine and take a tool from ground.

TAKEPOS=-4 Ground tool.

LEAVEPOS=# Magazine position to leave the tool.

The sequence of this operation is the following.

- 1 Leave the tool of the spindle in the magazine and activate the mark SPDLTOMZ.
- 2 Leave the ground tool in the spindle and activate the mark GRTOSPD L.

7.

TOOL AND MAGAZINE MANAGEMENT.
Synchronous magazine without changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TMOPERATION = 7

Leave the spindle tool on the ground and take a tool from the magazine.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=-4 Ground tool.

The sequence of this operation is the following.

- 1 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.
- 2 Leave the tool of the magazine in the spindle and activate the mark MZTOSPD.

TMOPERATION = 8

Leave the spindle tool on the ground and take a tool from ground.

The sequence of this operation is the following.

- 1 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.
- 2 Leave the ground tool in the spindle and activate the mark GRTOSPD.

TMOPERATION = 9

Take a ground tool to the magazine going through the spindle.

TAKEPOS=-4 Ground tool.

LEAVEPOS=# Magazine position to leave the tool.

The sequence of this operation is the following.

- 1 Leave the ground tool in the spindle and activate the mark GRTOSPD.
- 2 Leave the tool of the spindle in the magazine and activate the mark SPDLTOMZ.

TMOPERATION = 10

Take a tool from the magazine and leave on the ground going through the spindle.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=-4 Ground tool.

The sequence of this operation is the following.

- 1 Leave the tool of the magazine in the spindle and activate the mark MZTOSPD.
- 2 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.

TMOPERATION = 11

Orient the magazine.

This operation is an optimization of the tool change that permits orienting the magazine while machining. Activate the MZROT mark to indicate that the operation has been completed, whether it has been oriented or not.

TMOPERATION = 12

Leave the spindle tool in the magazine and take a tool from the same magazine (as TMOPERATION=5). It is an optimized operation, only valid when the magazine is random and the tool is special.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=# Magazine position to leave the tool.

7.

TOOL AND MAGAZINE MANAGEMENT.
Synchronous magazine without changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TMOPERATION = 13

Orienting two magazines.

This operation is an optimization of the tool change that permits orienting two magazines while machining. Activate the MZROT mark in both magazines to indicate that the operation has been completed, whether the magazines have been oriented or not.

NEXTPOS=# Magazine position to leave the tool.
TAKEPOS=# Magazine position to pick up the tool.
MZID The lower portion of this register indicates the destination magazine for the tool and the higher portion the source magazine for the tool.

TMOPERATION = 14

Leave the spindle tool in the magazine and take a tool from another magazine.

TAKEPOS=# Magazine position to pick up the tool.
LEAVEPOS=# Magazine position to leave the tool.
MZID The lower portion of this register indicates the destination magazine for the tool and the higher portion the source magazine for the tool.

The sequence of this operation is the following.

- 1 Leave the tool of the spindle in the magazine and activate the mark SPDLTOMZ.
- 2 Leave the tool of the magazine in the spindle and activate the mark MZTOSPD.

7.

TOOL AND MAGAZINE MANAGEMENT.
Synchronous magazine without changer arm.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.8.3 Communication between the PLC and the M06 subroutine.

The communication between the PLC and the M06 subroutine takes place using a series of generic marks and registers. The program of the M06 subroutine offered as an example uses the following marks and registers.

Communication between the PLC and the M06 subroutine

Marks activated by the M06 subroutine so the PLC activates the relevant mark of the tool manager.

Mark.	Meaning.
M1107	SPDLTOGR The tool has been taken from the spindle to ground.
M1108	GRTOSPDL The tool has been taken from ground to the spindle.
M1109	MZTOSPDL The tool has been taken from the magazine to the spindle.
M1110	SPDLTOMZ The tool has been taken from the spindle to the magazine.

M functions at the PLC

M functions that imply movements controlled by the PLC:

Function.	Meaning.
M109	Select in the magazine the position indicated by TAKEPOS and insert the tool in the spindle.
M110	Select in the magazine the position indicated by LEAVEPOS and leave the spindle tool.

Set all the M functions with "before-before" synchronization to the program continues after the M function is completed.

7.

TOOL AND MAGAZINE MANAGEMENT.
Synchronous magazine without changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.8.4 Program of the M06 subroutine.

```

%L SUB_SPD_TO_GR ; Remove the tool from the spindle (take it to ground).
    #MSG ["Extract tool T%D and press START", V.TM.TOOL]
                                     ; Message for the operator to extract the tool.
    M0                               ; Wait for the operation to be completed.
    #MSG ["" ]                       ; Remove message.
    V.PLC.M[1107]=1                  ; SPDLTOGR mark to the tool manager.
    #WAIT FOR [V.PLC.M[1107]=0]
#RET

%L SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
    #MSG ["Insert tool T%D and press START", V.TM.NXTOOL]
                                     ; Message for the operator to insert the tool.
    M0                               ; Wait for the operation to be completed.
    #MSG ["" ]                       ; Remove message.
    V.PLC.M[1108]=1                  ; GRTOSPDL mark to the tool manager.
    #WAIT FOR [V.PLC.M[1108]=0]
#RET

%L SUB_MZ_TO_SPD ; Take the tool from the magazine and insert it in the spindle.
    M109                             ; Auxiliary function to execute an action.
    V.PLC.M[1109]=1                  ; MZTOSPDL mark to the tool manager.
    #WAIT FOR [V.PLC.M[1109]=0]
#RET

%L SUB_SPD_TO_MZ ; Leave the spindle tool in the magazine.
    M110                             ; Auxiliary function to execute an action.
    V.PLC.M[1110]=1                  ; SPDLTOMZ mark to the tool manager.
    #WAIT FOR [V.PLC.M[1110]=0]
#RET

%L SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    G1 Z_ F_                          ; Move the spindle.
#RET

%L SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    G1 Z_ F_                          ; Move the spindle.
#RET

%SUB_M6.nc
    #ESBLK                            ; Begin of single block activation
    #DSTOP                             ; STOP key disable
    M6                                 ; Order the tool manager to start the tool change.

    $IF [[ [V.G.FULLSTATUS & 255] < 9 ] | [ [V.G.FULLSTATUS & 255] > 13 ] ]
        $IF [[ [V.G.CNCAUTSTATUS & 4096] | [V.G.CNCAUTSTATUS & 8192] ] ]
            V.S.EXECUTION = 0          ; Simulated Theoretical or G
        $ELSE
            V.S.EXECUTION = 1          ; Execution
        $ENDIF
    $ELSE
        V.S.EXECUTION = 0             ; Simulation
    $ENDIF

    $IF [V.S.EXECUTION == 1]          ; CNC in execution

    $SWITCH V.PLC.TMOPERATION ; It analyzes the type of operation.

    $CASE 1                           ; Take a tool from the magazine and insert it in the
                                     ; spindle.
        LL SUB_SPD_AUTCHG             ; Move the spindle to the automatic tool change point
        LL SUB_MZ_TO_SPD              ; Take the tool from the magazine to the spindle.
    $BREAK

```

7.

TOOL AND MAGAZINE MANAGEMENT.
Synchronous magazine without changer arm.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

```

$CASE 2 ; Leave the spindle tool in the magazine.
  LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
  LL SUB_SPD_TO_MZ ; Leave the spindle tool in the magazine.
$BREAK

$CASE 3 ; Insert the ground tool in the spindle.
  LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
  LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
$BREAK

$CASE 4 ; Leave the spindle tool on the ground.
  LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
  LL SUB_SPD_TO_GR ; Remove the tool from the spindle.
$BREAK

$CASE 5 ; Leave the spindle tool in the magazine and take
          another one from the magazine.
  LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
  LL SUB_SPD_TO_MZ ; Leave the spindle tool in the magazine.
  LL SUB_MZ_TO_SPD ; Take the tool from the magazine to the spindle.
$BREAK

$CASE 6 ; Leave the spindle tool in the magazine and take
          another one from ground.
  LL SUB_SPD_AUTCHG ; the spindle to the automatic tool change point
  LL SUB_SPD_TO_MZ ; Leave the spindle tool in the magazine.
  LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
  LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
$BREAK

$CASE 7 ; Leave the spindle tool on the ground and take another
          one from the magazine.
  LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
  LL SUB_SPD_TO_GR ; Remove the tool from the spindle.
  LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
  LL SUB_MZ_TO_SPD ; Take the tool from the magazine to the spindle.
$BREAK

$CASE 8 ; Leave the spindle tool on the ground and take another
          one from ground.
  LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
  LL SUB_SPD_TO_GR ; Remove the tool from the spindle.
  LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
$BREAK

$CASE 9 ; Take a ground tool to the magazine going through the
          spindle.
  LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
  LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
  LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
  LL SUB_SPD_TO_MZ ; Leave the spindle tool in the magazine.
$BREAK

$CASE 10 ; Take a tool from the magazine and leave on the
          ground going through the spindle.
  LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
  LL SUB_MZ_TO_SPD ; Take the tool from the magazine to the spindle.
  LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
  LL SUB_SPD_TO_GR ; Remove the tool from the spindle.
$BREAK

$ENDSWITCH ; End of analysis of type of operation.

$WHILE V.TM.MZWAIT == 1
$ENDWHILE ; Wait for the tool manager.
$ENDIF
#DSBLK ; End of single block activation.
#ESTOP ; STOP key enable
#RET

```

7.

TOOL AND MAGAZINE MANAGEMENT.
Synchronous magazine without changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.8.5 Basic PLC programming.

When executing the -T- function

When executing a T function, the tool manager sends the code TMOPERATION=11 to the PLC. In general, it is an optimization of the change that permits orienting the magazine while machining.

In this case, the magazine is not oriented and the MZROT mark is activated to indicate that the operation has been completed.

```
DFU TMOPSTROBE AND CPS TMOPERATION EQ 11 = SET MZROT
```

Activate the MZROT mark to "tell" the tool manager that the operation has finished.

When executing the M06 function

Every time the M06 subroutine ends an action, it lets the PLC know so it activates the relevant mark of the tool manager.

```
DFU M1107 = SET SPDLTOGR
```

```
DFD SPDLTOGR = RES M1107
```

The tool has been taken from the spindle to ground.

```
DFU M1108 = SET GRTOSPDL
```

```
DFD GRTOSPDL = RES M1108
```

The tool has been taken from ground to the spindle.

```
DFU M1109 = SET MZTOSPDL
```

```
DFD MZTOSPDL = RES M1109
```

The tool has been taken from the magazine to the spindle.

```
DFU M1110 = SET SPDLTOMZ
```

```
DFD SPDLTOMZ = RES M1110
```

The tool has been taken from the spindle to the magazine.

```
DFU M1111 = SET MZROT
```

```
DFD MZROT = RES M1111
```

The magazine has rotated already.

The M06 subroutine uses the following M functions to "tell" the PLC which movements it must carry out.

M109 Select in the magazine the position indicated by TAKEPOS and insert the tool in the spindle.

M110 Select in the magazine the position indicated by LEAVEPOS and leave the spindle tool.

Programming it depends on the type of machine. The auxiliary function will conclude after executing the requested movement.

Certain operations require using the information transferred by the tool manager in the following registers:

LEAVEPOS This register indicates the magazine position to leave the tool.

TAKEPOS This register indicates the magazine position of the tool to be taken.

Manager emergency signal.

Treatment of the tool manager emergency signal.

```
DFU B11KEYBD1 AND NOT TMINEM = SET SETTMEM
```

```
DFU TMINEM = RES SETTMEM
```

Pressing the USER12 key activates the emergency.

```
TMINEM = B11KEYLED1
```

The lamp of the USER12 key turns on when there is an emergency.

```
TMINEM AND DFU B12KEYBD1 = SET RESTMEM
```

Pressing the USER13 key removes the emergency.

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine without changer arm.

FAGOR 

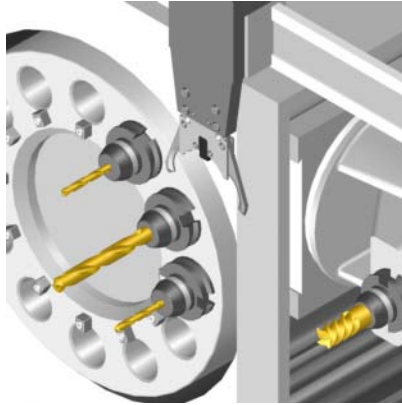
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.9 Synchronous magazine with changer arm and 1 claw

Synchronous magazines with tool changer arm (1 or 2 claws) have the magazine close to the spindle and the tool is changed using a tool changer arm. The tool cannot be changed while the part is being machined.



The communication between the manager and the PLC takes place in two stages; the first one when executing the T function and the second one when executing the M06 function.

Execution of the T function.

- 1 When the CNC executes the T function, it lets the tool manager know about it.
- 2 The tool manager sends a command to the PLC to select the next tool in the magazine (if possible).
- 3 The CNC continues with the execution of the program without waiting for the tool manager to complete the operation.

Execution of the M06 function.

- 1 When the CNC executes the M06 function, it calls the associated subroutine. The M06 function must also be programmed inside that subroutine so the CNC "tells" the tool manager to start making the tool change.
- 2 The tool manager sends a command to the PLC to make the change.
- 3 The CNC waits for the tool manager to complete the operation before continuing with the execution of the program.

The management of the tool change should be included in the subroutine associated with the M06 and leave the control of the external devices up to the PLC. Use the auxiliary functions to govern the various devices (magazine rotation, magazine movement, tool changer arm, etc.) from the M06 subroutine.

7.

TOOL AND MAGAZINE MANAGEMENT.
Synchronous magazine with changer arm and 1 claw



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.9.1 Valid operations and marks activated by the PLC with each one of them.

Valid operations of the tool manager for this magazine.

The possible operations in this type of magazine are:

TMOPERATION	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 a tool from the same magazine.
6	Leave the spindle tool in the magazine and take a tool from ground.
7	Leave the spindle tool on the ground and take a tool from the magazine.
8	Leave the spindle tool on the ground and take a tool 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 a tool from the same magazine (as TMOPERATION=5). It is an optimized operation, only valid when the magazine is random and the tool is special.
13	Orienting two magazines.
14	Leave the spindle tool in the magazine and take a tool from another magazine.

Values of registers TAKEPOS and LEAVEPOS used by the manager.

The TAKEPOS and LEAVEPOS signals can assume the following values.

Value	Meaning
0	Do nothing.
#	Magazine position.
-4	Ground tool.

Here is a summary table of the TAKEPOS and LEAVEPOS values used by the manager in each operation as well as the marks that must be activated by the PLC at the end of each operation. In the case of the marks shown between parenthesis, the manager can execute the operations in any order, but it must always execute both.

TM => PLC			PLC => TM
TMOPERATION	TAKEPOS	LEAVEPOS	
1	#	0	MZTOCH1 + CH1TOSPDL
2	0	#	SPDLTOCH1 + CH1TOMZ
3	0	0	GRTOSPDL
4	0	0	SPDLTOGR
5	#	#	SPDLTOCH1 + CH1TOMZ + MZTOCH1 + CH1TOSPDL
6	-4	#	SPDLTOCH1 + (CH1TOMZ & GRTOSPDL)
7	#	-4	(SPDLTOGR & MZTOCH1) + CH1TOSPDL

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 1 claw



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TM => PLC			PLC => TM
TMOPERATION	TAKEPOS	LEAVEPOS	
8	0	0	SPDLTOGR + GRTOSPDL
9	-4	#	GRTOSPDL + SPDLTOCH1 + CH1TOMZ
10	#	-4	MZTOCH1 + CH1TOSPDL + SPDLTOGR
11	0	0	MZROT
12	#	#	SPDLTOCH1 + CH1TOMZ + MZTOCH1 + CH1TOSPDL
13	#	0	MZROT + MZROT
14	#	#	SPDLTOCH1 + CH1TOMZ + MZTOCH1 + CH1TOSPDL

Example of operation.

The following example shows, assuming that there is no tool in the spindle, it shows the functions executed by the CNC, the values sent by the tool manager to the PLC in each operation and the marks to be activated by the PLC in each case.

It is a non-random magazine, each tool occupies the position of its own number and T7, T8 and T9 are ground tools.

CNC	TMOPERATION	TAKEPOS	LEAVEPOS	
T1 M6	11 1	0 1	0 0	MZROT MZTOCH1 + CH1TOSPDL
T2 M6	11 5	0 2	0 1	MZROT SPDLTOCH1 + CH1TOMZ + MZTOCH1 + CH1TOSPDL
T7 M6	11 6	0 -4	0 2	MZROT SPDLTOCH1 + CH1TOMZ + GRTOSPDL
T8 M6	11 8	0 0	0 0	MZROT SPDLTOGR + GRTOSPDL
T3 T4 M6	11 11 7	0 0 4	0 0 -4	MZROT MZROT SDPLTOGR + MZTOCH1 + CH1TOSPDL
T0 M6	11 2	0 0	0 4	MZROT SPDLTOCH1 + CH1TOMZ
T9 M6	11 3	0 0	0 0	MZROT GRTOSPDL
T0 M6	11 4	0 0	0 0	MZROT SPDLTOGR

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 1 claw



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.9.2 Detailed description of the operations of the magazine.

Here is a sample of a detailed description of the valid operations for this magazine. For each operation, it indicates which of the TAKEPOS, LEAVEPOS, NEXTPOS and MZID signals are involved and their meanings. It also shows the sequence to follow to complete the operation.

The TAKEPOS, LEAVEPOS and NEXTPOS signals can assume the following values.

Value	Meaning
0	Do nothing.
#	Magazine position.
-4	Ground tool.

TMOPERATION = 1

Take a tool from the magazine and insert it in the spindle.

TAKEPOS=# Magazine position to pick up the tool.

The sequence of this operation is the following.

- 1 Take the tool from the magazine with holder 1 and activate the mark MZTOCH1.
- 2 Insert the tool of holder 1 in the spindle and activate CH1TOSPD.

TMOPERATION = 2

Leave the spindle tool in the magazine.

LEAVEPOS=# Magazine position to leave the tool.

The sequence of this operation is the following.

- 1 Take the tool from the spindle with holder 1 and activate the mark SPDLTOCH1.
- 2 Leave the tool of holder 1 in the magazine and activate the mark CH1TOMZ.

TMOPERATION = 3

Insert a ground tool in the spindle.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark: GRTOSPD.

TMOPERATION = 4

Leave the spindle tool on the ground.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark: SPDLTOGR.

TMOPERATION = 5

Leave the spindle tool in the magazine and take a tool from the same magazine.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=# Magazine position to leave the tool.

The sequence of this operation is the following.

- 1 Take the tool from the spindle with holder 1 and activate the mark SPDLTOCH1.
- 2 Leave the tool of holder 1 in the magazine and activate the mark CH1TOMZ.
- 3 Take the tool from the magazine with holder 1 and activate the mark MZTOCH1.
- 4 Insert the tool of holder 1 in the spindle and activate CH1TOSPD.

TMOPERATION = 6

Leave the spindle tool in the magazine and take a tool from ground.

TAKEPOS=-4 Ground tool.

LEAVEPOS=# Magazine position to leave the tool.

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 1 claw


FAGOR

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

The sequence of this operation is the following. The tool manager admits the sequence 1-3-2.

- 1 Take the tool from the spindle with holder 1 and activate the mark SPDLTOCH1.
- 2 Leave the tool of holder 1 in the magazine and activate the mark CH1TOMZ.
- 3 Leave the ground tool in the spindle and activate the mark GRTOSPDL.

TMOPERATION = 7

Leave the spindle tool on the ground and take a tool from the magazine.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=-4 Ground tool.

The sequence of this operation is the following. The tool manager admits the sequence 2-1-3.

- 1 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.
- 2 Take the tool from the magazine with holder 1 and activate the mark MZTOCH1.
- 3 Insert the tool of holder 1 in the spindle and activate CH1TOSPDL.

TMOPERATION = 8

Leave the spindle tool on the ground and take a tool from ground.

The sequence of this operation is the following.

- 1 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.
- 2 Leave the ground tool in the spindle and activate the mark GRTOSPDL.

TMOPERATION = 9

Take a ground tool to the magazine going through the spindle.

TAKEPOS=-4 Ground tool.

LEAVEPOS=# Magazine position to leave the tool.

The sequence of this operation is the following.

- 1 Leave the ground tool in the spindle and activate the mark GRTOSPDL.
- 2 Take the tool from the spindle with holder 1 and activate the mark SPDLTOCH1.
- 3 Leave the tool of holder 1 in the magazine and activate the mark CH1TOMZ.

TMOPERATION = 10

Take a tool from the magazine and leave on the ground going through the spindle.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=-4 Ground tool.

The sequence of this operation is the following.

- 1 Take the tool from the magazine with holder 1 and activate the mark MZTOCH1.
- 2 Insert the tool of holder 1 in the spindle and activate CH1TOSPDL.
- 3 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.

TMOPERATION = 11

Orient the magazine.

This operation is an optimization of the tool change that permits orienting the magazine while machining. Activate the MZROT mark to indicate that the operation has been completed, whether it has been oriented or not.

TMOPERATION = 12

Leave the spindle tool in the magazine and take a tool from the same magazine (as TMOPERATION=5). It is an optimized operation, only valid when the magazine is random and the tool is special.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=# Magazine position to leave the tool.

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 1 claw



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TMOPERATION = 13

Orienting two magazines.

This operation is an optimization of the tool change that permits orienting two magazines while machining. Activate the MZROT mark in both magazines to indicate that the operation has been completed, whether the magazines have been oriented or not.

NEXTPOS=# Magazine position to leave the tool.
TAKEPOS=# Magazine position to pick up the tool.
MZID The lower portion of this register indicates the destination magazine for the tool and the higher portion the source magazine for the tool.

TMOPERATION = 14

Leave the spindle tool in the magazine and take a tool from another magazine.

TAKEPOS=# Magazine position to pick up the tool.
LEAVEPOS=# Magazine position to leave the tool.
MZID The lower portion of this register indicates the destination magazine for the tool and the higher portion the source magazine for the tool.

The sequence of this operation is the following.

- 1 Take the tool from the spindle with holder 1 and activate the mark SPDLTOCH1.
- 2 Leave the tool of holder 1 in the magazine and activate the mark CH1TOMZ.
- 3 Take the tool from the other magazine with holder ·1· and activate the mark MZTOCH1.
- 4 Insert the tool of holder 1 in the spindle and activate CH1TOSPDL.

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 1 claw

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.9.3 Communication between the PLC and the M06 subroutine.

The communication between the PLC and the M06 subroutine takes place using a series of generic marks and registers. The program of the M06 subroutine offered as an example uses the following marks and registers.

Communication between the PLC and the M06 subroutine

Marks activated by the M06 subroutine so the PLC activates the relevant mark of the tool manager.

Mark.	Meaning.
M1101	MZTOCH1 The tool has been taken from the magazine to holder 1.
M1102	CH1TOSPDL The tool has been taken from holder 1 to the spindle.
M1103	SPDLTOCH1 The tool has been taken from the spindle to holder 1.
M1105	CH1TOMZ The tool has been taken from holder 1 to the magazine.
M1107	SPDLTOGR The tool has been taken from the spindle to ground.
M1108	GRTOSPDL The tool has been taken from ground to the spindle.

M functions at the PLC

M functions that imply movements controlled by the PLC:

Function.	Meaning.
M101	Select in the magazine the position indicated by TAKEPOS and take the tool with holder 1.
M102	Take the tool of holder 1 to the spindle.
M103	Take the spindle tool with holder 1.
M105	Select in the magazine the position indicated by LEAVEPOS and leave the tool of holder 1.

Set all the M functions with "before-before" synchronization to the program continues after the M function is completed.

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 1 claw



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.9.4 Program of the M06 subroutine.

```

%L SUB_MZ_TO_CH1 ; Take the tool from the magazine with holder 1.
    M101 ; Auxiliary function to execute an action.
    V.PLC.M[1101]=1 ; MZTOCH1 mark to the tool manager.
    #WAIT FOR [V.PLC.M[1101]=0]
#RET

%L SUB_CH1_TO_SPD ; Take the tool of holder 1 to the spindle.
    M102 ; Auxiliary function to execute an action.
    V.PLC.M[1102]=1 ; CH1TOSPD mark to the tool manager.
    #WAIT FOR [V.PLC.M[1102]=0]
#RET

%L SUB_SPD_TO_CH1 ; Take the spindle tool with holder 1.
    M103 ; Auxiliary function to execute an action.
    V.PLC.M[1103]=1 ; SPDLTOCH1 mark to the tool manager.
    #WAIT FOR [V.PLC.M[1103]=0]
#RET

%L SUB_CH1_TO_MZ ; Take the tool of holder 1 to the magazine.
    M105 ; Auxiliary function to execute an action.
    V.PLC.M[1105]=1 ; CH1TOMZ mark to the tool manager.
    #WAIT FOR [V.PLC.M[1105]=0]
#RET

%L SUB_SPD_TO_GR ; Remove the tool from the spindle (take it to ground).
    #MSG ["Extract tool T%D and press START", V.TM.TOOL]
    M0 ; Message for the operator to extract the tool.
    #MSG ["" ] ; Wait for the operation to be completed.
    #MSG ["" ] ; Remove message.
    V.PLC.M[1107]=1 ; SPDLTOGR mark to the tool manager.
    #WAIT FOR [V.PLC.M[1107]=0]
#RET

%L SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
    #MSG ["Insert tool T%D and press START", V.TM.NXTOOL]
    M0 ; Message for the operator to insert the tool.
    #MSG ["" ] ; Wait for the operation to be completed.
    #MSG ["" ] ; Remove message.
    V.PLC.M[1108]=1 ; GRTOSPD mark to the tool manager.
    #WAIT FOR [V.PLC.M[1108]=0]
#RET

%L SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    G1 Z_ F_ ; Move the spindle.
#RET

%L SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    G1 Z_ F_ ; Move the spindle.
#RET

%SUB_M6.nc
    #ESBLK ; Begin of single block activation
    #DSTOP ; STOP key disable
    M6 ; Order the tool manager to start the tool change.

    $IF [[ [V.G.FULLSTATUS & 255]<9 ] | [[V.G.FULLSTATUS & 255]>13]]
        $IF [[ [V.G.CNCAUTSTATUS & 4096 ] | [V.G.CNCAUTSTATUS & 8192]]
            V.S.EXECUTION = 0 ; Simulated Theoretical or G
        $ELSE
            V.S.EXECUTION = 1 ; Execution
        $ENDIF
    $ELSE
        V.S.EXECUTION = 0 ; Simulation
    $ENDIF

```

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 1 claw


FAGOR

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

```

$IF [V.S.EXECUTION == 1] ; CNC in execution

$SWITCH V.PLC.TMOPERATION ; It analyzes the type of operation.

$CASE 1 ; Take a tool from the magazine and insert it in the
        ; spindle.
    LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    LL SUB_MZ_TO_CH1 ; Take the tool from the magazine with holder 1.
    LL SUB_CH1_TO_SPD ; Take the tool of holder 1 to the spindle.
$BREAK

$CASE 2 ; Leave the spindle tool in the magazine.
    LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    LL SUB_SPD_TO_CH1 ; Take the spindle tool with holder 1.
    LL SUB_CH1_TO_MZ ; Leave the tool of holder 1 in the magazine.
$BREAK

$CASE 3 ; Insert the ground tool in the spindle.
    LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
$BREAK

$CASE 4 ; Leave the spindle tool on the ground.
    LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    LL SUB_SPD_TO_GR ; Remove the tool from the spindle.
$BREAK

$CASE 5 ; Leave the spindle tool in the magazine and take
        ; another one from the magazine.
    LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    LL SUB_SPD_TO_CH1 ; Take the spindle tool with holder 1.
    LL SUB_CH1_TO_MZ ; Leave the tool of holder 1 in the magazine.
    LL SUB_MZ_TO_CH1 ; Take the tool from the magazine with holder 1.
    LL SUB_CH1_TO_SPD ; Take the tool of holder 1 to the spindle.
$BREAK

$CASE 6 ; Leave the spindle tool in the magazine and take
        ; another one from ground.
    LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    LL SUB_SPD_TO_CH1 ; Take the spindle tool with holder 1.
    LL SUB_CH1_TO_MZ ; Leave the tool of holder 1 in the magazine.
    LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
$BREAK

$CASE 7 ; Leave the spindle tool on the ground and take another
        ; one from the magazine.
    LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    LL SUB_SPD_TO_GR ; Remove the tool from the spindle.
    LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    LL SUB_MZ_TO_CH1 ; Take the tool from the magazine with holder 1.
    LL SUB_CH1_TO_SPD ; Take the tool of holder 1 to the spindle.
$BREAK

$CASE 8 ; Leave the spindle tool on the ground and take another
        ; one from ground.
    LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    LL SUB_SPD_TO_GR ; Remove the tool from the spindle.
    LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
$BREAK

$CASE 9 ; Take a ground tool to the magazine going through the
        ; spindle.
    LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
    LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    LL SUB_SPD_TO_CH1 ; Take the spindle tool with holder 1.
    LL SUB_CH1_TO_MZ ; Leave the tool of holder 1 in the magazine.
$BREAK

```

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 1 claw



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

```
$CASE 10 ; Take a tool from the magazine and leave on the  
          ; ground going through the spindle.  
  LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point  
  LL SUB_MZ_TO_CH1  ; Take the tool from the magazine with holder 1.  
  LL SUB_CH1_TO_SPD ; Take the tool of holder 1 to the spindle.  
  LL SUB_SPD_GMCHG  ; Move the spindle to the manual tool change point.  
  LL SUB_SPD_TO_GR   ; Remove the tool from the spindle.  
$BREAK  
  
$ENDSWITCH ; End of analysis of type of operation.  
  
$WHILE V.TM.MZWAIT == 1 ; Wait for the tool manager.  
$ENDWHILE  
  
$ENDIF  
#DSBLK ; End of single block activation.  
#ESTOP ; STOP key enable  
#RET
```

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 1 claw



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.9.5 Basic PLC programming.

When executing the -T- function

When executing a T function, the tool manager sends the code TMOOPERATION=11 to the PLC. In general, it is an optimization of the change that permits orienting the magazine while machining.

In this case, the magazine is not oriented and the MZROT mark is activated to indicate that the operation has been completed.

```
DFU TMOPSTROBE AND CPS TMOOPERATION EQ 11 = SET MZROT
```

Activate the MZROT mark to "tell" the tool manager that the operation has finished.

When executing the M06 function

Every time the M06 subroutine ends an action, it lets the PLC know so it activates the relevant mark of the tool manager.

```
DFU M1101 = SET MZTOCH1
```

```
DFD MZTOCH1 = RES M1101
```

The tool has been taken from the magazine to holder 1.

```
DFU M1102 = SET CH1TOSPDL
```

```
DFD CH1TOSPDL = RES M1102
```

The tool has been taken from holder 1 to the spindle.

```
DFU M1103 = SET SPDLTOCH1
```

```
DFD SPDLTOCH1 = RES M1103
```

The tool has been taken from the spindle to holder 1.

```
DFU M1105 = SET CH1TOMZ
```

```
DFD CH1TOMZ = RES M1105
```

The tool has been taken from holder 1 to the magazine.

```
DFU M1107 = SET SPDLTOGR
```

```
DFD SPDLTOGR = RES M1107
```

The tool has been taken from the spindle to ground.

```
DFU M1108 = SET GRTOSPDL
```

```
DFD GRTOSPDL = RES M1108
```

The tool has been taken from ground to the spindle.

The M06 subroutine uses the following M functions to "tell" the PLC which movements it must carry out.

M101	Select in the magazine the position indicated by TAKEPOS and take the tool with holder 1.
M102	Take the tool of holder 1 to the spindle.
M103	Take the spindle tool with holder 1.
M105	Select in the magazine the position indicated by LEAVEPOS and leave the tool of holder 1.

Programming it depends on the type of machine. The auxiliary function will conclude after executing the requested movement.

Certain operations require using the information transferred by the tool manager in the following registers:

LEAVEPOS	This register indicates the magazine position to leave the tool.
TAKEPOS	This register indicates the magazine position of the tool to be taken.

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 1 claw



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Manager emergency signal.

Treatment of the tool manager emergency signal.

DFU B11KEYBD1 AND NOT TMINEM = SET SETTMEM
DFU TMINEM = RES SETTMEM

Pressing the USER12 key activates the emergency.

TMINEM = B11KEYLED1

The lamp of the USER12 key turns on when there is an emergency.

TMINEM AND DFU B12KEYBD1 = SET RESTMEM

Pressing the USER13 key removes the emergency.

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 1 claw

FAGOR 

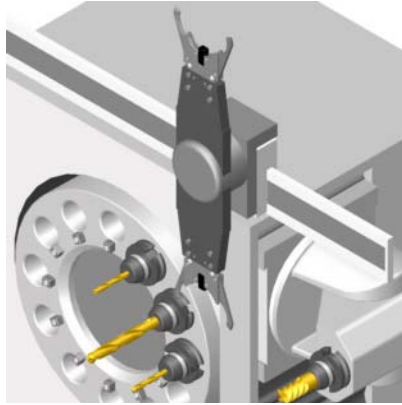
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.10 Synchronous magazine with changer arm and 2 claws.

Synchronous magazines with tool changer arm (1 or 2 claws) have the magazine close to the spindle and the tool is changed using a tool changer arm. The tool cannot be changed while the part is being machined.



The communication between the manager and the PLC takes place in two stages; the first one when executing the T function and the second one when executing the M06 function.

Execution of the T function.

- 1 When the CNC executes the T function, it lets the tool manager know about it.
- 2 The tool manager sends a command to the PLC to select the next tool in the magazine (if possible).
- 3 The CNC continues with the execution of the program without waiting for the tool manager to complete the operation.

Execution of the M06 function.

- 1 When the CNC executes the M06 function, it calls the associated subroutine. The M06 function must also be programmed inside that subroutine so the CNC "tells" the tool manager to start making the tool change.
- 2 The tool manager sends a command to the PLC to make the change.
- 3 The CNC waits for the tool manager to complete the operation before continuing with the execution of the program.

The management of the tool change should be included in the subroutine associated with the M06 and leave the control of the external devices up to the PLC. Use the auxiliary functions to govern the various devices (magazine rotation, magazine movement, tool changer arm, etc.) from the M06 subroutine.

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 2 claws.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.10.1 Valid operations and marks activated by the PLC with each one of them.

Valid operations of the tool manager for this magazine.

The possible operations in this type of magazine are:

TMOPERATION	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 a tool from the same magazine.
6	Leave the spindle tool in the magazine and take a tool from ground.
7	Leave the spindle tool on the ground and take a tool from the magazine.
8	Leave the spindle tool on the ground and take a tool 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 a tool from the same magazine (as TMOPERATION=5). It is an optimized operation only valid in the following types of synchronous magazine. <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 the magazine and take a tool from another magazine.

Values of registers TAKEPOS and LEAVEPOS used by the manager.

The TAKEPOS and LEAVEPOS signals can assume the following values.

Value	Meaning
0	Do nothing.
#	Magazine position.
-4	Ground tool.

Here is a summary table of the TAKEPOS and LEAVEPOS values used by the manager in each operation as well as the marks that must be activated by the PLC at the end of each operation. In the case of the marks shown between parenthesis, the manager can execute the operations in any order, but it must always execute both.

TM => PLC			PLC => TM
TMOPERATION	TAKEPOS	LEAVEPOS	
1	#	0	MZTOCH1 + CH1TOSPDL
2	0	#	SPDLTOCH2 + CH2TOMZ
3	0	0	GRTOSPDL
4	0	0	SPDLTOGR
5	#	#	(SPDLTOCH2 & MZTOCH1) + (CH1TOSPDL & CH2TOMZ)
6	-4	#	SPDLTOCH2 + (CH2TOMZ & GRTOSPDL)

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 2 claws.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TM => PLC			PLC => TM
TMOPERATION	TAKEPOS	LEAVEPOS	
7	#	-4	(SPDLTOGR & MZTOCH1) + CH1TOSPD
8	0	0	SPDLTOGR + GRTOSPD
9	-4	#	GRTOSPD + SPDLTOCH2 + CH2TOMZ
10	#	-4	MZTOCH1 + CH1TOSPD + SPDLTOGR
11	0	0	MZROT
12	#	#	SPDLTOCH2 + CH2TOMZ + MZTOCH1 + CH1TOSPD
13	#	0	MZROT + MZROT
14	#	#	SPDLTOCH2 + CH2TOMZ + MZTOCH1 + CH1TOSPD

Example of operation.

The following example shows, assuming that there is no tool in the spindle, it shows the functions executed by the CNC, the values sent by the tool manager to the PLC in each operation and the marks to be activated by the PLC in each case.

It is a non-random magazine, each tool occupies the position of its own number and T7, T8 and T9 are ground tools.

CNC	TMOPERATION	TAKEPOS	LEAVEPOS	
T1 M6	11 1	0 1	0 0	MZROT MZTOCH1 + CH1TOSPD
T2 M6	11 5	0 2	0 1	MZROT MZTOCH1 + SPDLTOCH2 + CH1TOSPD + CH2TOMZ
T7 M6	11 6	0 -4	0 2	MZROT SPDLTOCH2 + CH2TOMZ + GRTOSPD
T8 M6	11 8	0 0	0 0	MZROT SPDLTOGR + GRTOSPD
T3 T4 M6	11 11 7	0 0 4	0 0 -4	MZROT MZROT SDPLTOGR + MZTOCH1 + CH1TOSPD
T0 M6	11 2	0 0	0 4	MZROT SPDLTOCH2 + CH2TOMZ
T9 M6	11 3	0 0	0 0	MZROT GRTOSPD
T0 M6	11 4	0 0	0 0	MZROT SPDLTOGR

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 2 claws.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.10.2 Detailed description of the operations of the magazine.

Here is a sample of a detailed description of the valid operations for this magazine. For each operation, it indicates which of the TAKEPOS, LEAVEPOS, NEXTPOS and MZID signals are involved and their meanings. It also shows the sequence to follow to complete the operation.

The TAKEPOS, LEAVEPOS and NEXTPOS signals can assume the following values.

Value	Meaning
0	Do nothing.
#	Magazine position.
-4	Ground tool.

TMOPERATION = 1

Take a tool from the magazine and insert it in the spindle.

TAKEPOS=# Magazine position to pick up the tool.

The sequence of this operation is the following.

- 1 Take the tool from the magazine with holder 1 and activate the mark MZTOCH1.
- 2 Insert the tool of holder 1 in the spindle and activate CH1TOSPD.

TMOPERATION = 2

Leave the spindle tool in the magazine.

LEAVEPOS=# Magazine position to leave the tool.

The sequence of this operation is the following.

- 1 Take the tool from the spindle with holder ·2· and activate the mark SPDLTOCH2.
- 2 Leave the tool of holder ·2· in the magazine and activate the mark CH2TOMZ.

TMOPERATION = 3

Insert a ground tool in the spindle.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark: GRTOSPD.

TMOPERATION = 4

Leave the spindle tool on the ground.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark: SPDLTOGR.

TMOPERATION = 5

Leave the spindle tool in the magazine and take a tool from the same magazine.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=# Magazine position to leave the tool.

The sequence of this operation is the following. The tool manager also admits the sequences 1-2-4-3, 2-1-3-4, 2-1-4-3.

- 1 Take the tool from the spindle with holder ·2· and activate the mark SPDLTOCH2.
- 2 Take the tool from the magazine with holder 1 and activate the mark MZTOCH1.
- 3 Leave the tool of holder ·2· in the magazine and activate the mark CH2TOMZ.
- 4 Insert the tool of holder 1 in the spindle and activate CH1TOSPD.

TMOPERATION = 6

Leave the spindle tool in the magazine and take a tool from ground.

TAKEPOS=-4 Ground tool.

LEAVEPOS=# Magazine position to leave the tool.

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 2 claws.


FAGOR

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

The sequence of this operation is the following. The tool manager admits the sequence 1-3-2.

- 1 Take the tool from the spindle with holder ·2· and activate the mark SPDLTOCH2.
- 2 Leave the tool of holder ·2· in the magazine and activate the mark CH2TOMZ.
- 3 Leave the ground tool in the spindle and activate the mark GRTOSPDL.

TMOPERATION = 7

Leave the spindle tool on the ground and take a tool from the magazine.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=-4 Ground tool.

The sequence of this operation is the following. The tool manager admits the sequence 2-1-3.

- 1 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.
- 2 Take the tool from the magazine with holder 1 and activate the mark MZTOCH1.
- 3 Insert the tool of holder 1 in the spindle and activate CH1TOSPDL.

TMOPERATION = 8

Leave the spindle tool on the ground and take a tool from ground.

The sequence of this operation is the following.

- 1 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.
- 2 Leave the ground tool in the spindle and activate the mark GRTOSPDL.

TMOPERATION = 9

Take a ground tool to the magazine going through the spindle.

TAKEPOS=-4 Ground tool.

LEAVEPOS=# Magazine position to leave the tool.

The sequence of this operation is the following.

- 1 Leave the ground tool in the spindle and activate the mark GRTOSPDL.
- 2 Take the tool from the spindle with holder ·2· and activate the mark SPDLTOCH2.
- 3 Leave the tool of holder ·2· in the magazine and activate the mark CH2TOMZ.

TMOPERATION = 10

Take a tool from the magazine and leave on the ground going through the spindle.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=-4 Ground tool.

The sequence of this operation is the following.

- 1 Take the tool from the magazine with holder 1 and activate the mark MZTOCH1.
- 2 Insert the tool of holder 1 in the spindle and activate CH1TOSPDL.
- 3 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.

TMOPERATION = 11

Orient the magazine.

This operation is an optimization of the tool change that permits orienting the magazine while machining. Activate the MZROT mark to indicate that the operation has been completed, whether it has been oriented or not.

7.

TOOL AND MAGAZINE MANAGEMENT.
Synchronous magazine with changer arm and 2 claws.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TMOPERATION = 12

Leave the spindle tool in the magazine and take a tool from the same magazine (as TMOPERATION=5). It is an optimized operation only valid in the following types of synchronous magazine.

- Non-random having a tool changer arm with two claws.
- Random when having special tools.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=# Magazine position to leave the tool.

TMOPERATION = 13

Orienting two magazines.

This operation is an optimization of the tool change that permits orienting two magazines while machining. Activate the MZROT mark in both magazines to indicate that the operation has been completed, whether the magazines have been oriented or not.

NEXTPOS=# Magazine position to leave the tool.

TAKEPOS=# Magazine position to pick up the tool.

MZID The lower portion of this register indicates the destination magazine for the tool and the higher portion the source magazine for the tool.

TMOPERATION = 14

Leave the spindle tool in the magazine and take a tool from another magazine.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=# Magazine position to leave the tool.

MZID The lower portion of this register indicates the destination magazine for the tool and the higher portion the source magazine for the tool.

The sequence of this operation is the following.

- 1 Take the tool from the spindle with holder ·2· and activate the mark SPDLTOCH2.
- 2 Leave the tool of holder ·2· in the magazine and activate the mark CH2TOMZ.
- 3 Take the tool from the other magazine with holder ·1· and activate the mark MZTOCH1.
- 4 Insert the tool of holder 1 in the spindle and activate CH1TOSPD.

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 2 claws.

7.10.3 Communication between the PLC and the M06 subroutine.

The communication between the PLC and the M06 subroutine takes place using a series of generic marks and registers. The program of the M06 subroutine offered as an example uses the following marks and registers.

Communication between the PLC and the M06 subroutine

Marks activated by the M06 subroutine so the PLC activates the relevant mark of the tool manager.

Mark.	Meaning.
M1101	MZTOCH1 The tool has been taken from the magazine to holder 1.
M1102	CH1TOSPDL The tool has been taken from holder 1 to the spindle.
M1104	SPDLTOCH2 The tool has been taken from the spindle to holder 2.
M1106	CH2TOMZ The tool has been taken from holder 2 to the magazine.
M1107	SPDLTOGR The tool has been taken from the spindle to ground.
M1108	GRTOSPDL The tool has been taken from ground to the spindle.

M functions at the PLC

M functions that imply movements controlled by the PLC:

Function.	Meaning.
M101	Select in the magazine the position indicated by TAKEPOS and take the tool with holder 1.
M102	Take the tool of holder 1 to the spindle.
M104	Take the spindle tool with holder 2.
M106	Select in the magazine the position indicated by LEAVEPOS and leave the tool of holder 2.

Set all the M functions with "before-before" synchronization to the program continues after the M function is completed.

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 2 claws.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.10.4 Program of the M06 subroutine.

```

%L SUB_MZ_TO_CH1 ; Take the tool from the magazine with holder 1.
    M101 ; Auxiliary function to execute an action.
    V.PLC.M[1101]=1 ; MZTOCH1 mark to the tool manager.
    #WAIT FOR [V.PLC.M[1101]=0]
#RET

%L SUB_CH1_TO_SPD ; Take the tool of holder 1 to the spindle.
    M102 ; Auxiliary function to execute an action.
    V.PLC.M[1102]=1 ; CH1TOSPD mark to the tool manager.
    #WAIT FOR [V.PLC.M[1102]=0]
#RET

%L SUB_SPD_TO_CH2 ; Take the spindle tool with holder 2.
    M104 ; Auxiliary function to execute an action.
    V.PLC.M[1104]=1 ; SPDLTOCH2 mark to the tool manager.
    #WAIT FOR [V.PLC.M[1104]=0]
#RET

%L SUB_CH2_TO_MZ ; Take the tool of holder 2 to the magazine.
    M106 ; Auxiliary function to execute an action.
    V.PLC.M[1106]=1 ; CH2TOMZ mark to the tool manager.
    #WAIT FOR [V.PLC.M[1106]=0]
#RET

%L SUB_SPD_TO_GR ; Remove the tool from the spindle (take it to ground).
    #MSG ["Extract tool T%D and press START", V.TM.TOOL]
    M0 ; Message for the operator to extract the tool.
    #MSG [""] ; Wait for the operation to be completed.
    #MSG [""] ; Remove message.
    V.PLC.M[1107]=1 ; SPDLTOGR mark to the tool manager.
    #WAIT FOR [V.PLC.M[1107]=0]
#RET

%L SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
    #MSG ["Insert tool T%D and press START", V.TM.NXTOOL]
    M0 ; Message for the operator to insert the tool.
    #MSG [""] ; Wait for the operation to be completed.
    #MSG [""] ; Remove message.
    V.PLC.M[1108]=1 ; GRTOSPD mark to the tool manager.
    #WAIT FOR [V.PLC.M[1108]=0]
#RET

%L SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    G1 Z_ F_ ; Move the spindle.
#RET

%L SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    G1 Z_ F_ ; Move the spindle.
#RET

%SUB_M6.nc
    #ESBLK ; Begin of single block activation
    #DSTOP ; STOP key disable
    M6 ; Order the tool manager to start the tool change.

    $IF [[V.G.FULLSTATUS & 255]<9] | [[V.G.FULLSTATUS & 255]>13]]
        $IF [[V.G.CNCAUTSTATUS & 4096] | [V.G.CNCAUTSTATUS & 8192]]
            V.S.EXECUTION = 0 ; Simulated Theoretical or G
        $ELSE
            V.S.EXECUTION = 1 ; Execution
        $ENDIF
    $ELSE
        V.S.EXECUTION = 0 ; Simulation
    $ENDIF

```

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 2 claws.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

```

$IF [V.S.EXECUTION == 1] ; CNC in execution

$SWITCH V.PLC.TMOPERATION ; It analyzes the type of operation.

$CASE 1 ; Take a tool from the magazine and insert it in the
        ; spindle.
    LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    LL SUB_MZ_TO_CH1 ; Take the tool from the magazine with holder 1.
    LL SUB_CH1_TO_SPD ; Take the tool of holder 1 to the spindle.
$BREAK

$CASE 2 ; Leave the spindle tool in the magazine.
    LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    LL SUB_SPD_TO_CH2 ; Take the spindle tool with holder 2.
    LL SUB_CH2_TO_MZ ; Leave the tool of holder 2 in the magazine.
$BREAK

$CASE 3 ; Insert the ground tool in the spindle.
    LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
$BREAK

$CASE 4 ; Leave the spindle tool on the ground.
    LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    LL SUB_SPD_TO_GR ; Remove the tool from the spindle.
$BREAK

$CASE 5 ; Leave the spindle tool in the magazine and take
        ; another one from the magazine.
    LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    LL SUB_SPD_TO_CH2 ; Take the spindle tool with holder 2.
    LL SUB_MZ_TO_CH1 ; Take the tool from the magazine with holder 1.
    LL SUB_CH2_TO_MZ ; Leave the tool of holder 2 in the magazine.
    LL SUB_CH1_TO_SPD ; Take the tool of holder 1 to the spindle.
$BREAK

$CASE 6 ; Leave the spindle tool in the magazine and take
        ; another one from ground.
    LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    LL SUB_SPD_TO_CH2 ; Take the spindle tool with holder 2.
    LL SUB_CH2_TO_MZ ; Leave the tool of holder 2 in the magazine.
    LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
$BREAK

$CASE 7 ; Leave the spindle tool on the ground and take another
        ; one from the magazine.
    LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    LL SUB_SPD_TO_GR ; Remove the tool from the spindle.
    LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    LL SUB_MZ_TO_CH1 ; Take the tool from the magazine with holder 1.
    LL SUB_CH1_TO_SPD ; Take the tool of holder 1 to the spindle.
$BREAK

$CASE 8 ; Leave the spindle tool on the ground and take another
        ; one from ground.
    LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    LL SUB_SPD_TO_GR ; Remove the tool from the spindle.
    LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
$BREAK

$CASE 9 ; Take a ground tool to the magazine going through the
        ; spindle.
    LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
    LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
    LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
    LL SUB_SPD_TO_CH2 ; Take the spindle tool with holder 2.
    LL SUB_CH2_TO_MZ ; Leave the tool of holder 2 in the magazine.
$BREAK

```

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 2 claws.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

```
$CASE 10 ; Take a tool from the magazine and leave on the  
          ; ground going through the spindle.  
  LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point  
  LL SUB_MZ_TO_CH1  ; Take the tool from the magazine with holder 1.  
  LL SUB_CH1_TO_SPD ; Take the tool of holder 1 to the spindle.  
  LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.  
  LL SUB_SPD_TO_GR  ; Remove the tool from the spindle.  
$BREAK  
  
$ENDSWITCH ; End of analysis of type of operation.  
  
$WHILE V.TM.MZWAIT == 1 ; Wait for the tool manager.  
$ENDWHILE  
  
$ENDIF  
#DSBLK ; End of single block activation.  
#ESTOP ; STOP key enable  
#RET
```

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 2 claws.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.10.5 Basic PLC programming.

When executing the -T- function

When executing a T function, the tool manager sends the code TMOOPERATION=11 to the PLC. In general, it is an optimization of the change that permits orienting the magazine while machining.

In this case, the magazine is not oriented and the MZROT mark is activated to indicate that the operation has been completed.

DFU TMOPTROBE AND CPS TMOOPERATION EQ 11 = SET MZROT

Activate the MZROT mark to "tell" the tool manager that the operation has finished.

When executing the M06 function

Every time the M06 subroutine ends an action, it lets the PLC know so it activates the relevant mark of the tool manager.

DFU M1101 = SET MZTOCH1

DFD MZTOCH1 = RES M1101

The tool has been taken from the magazine to holder 1.

DFU M1102 = SET CH1TOSPDL

DFD CH1TOSPDL = RES M1102

The tool has been taken from holder 1 to the spindle.

DFU M1104 = SET SPDLTOCH2

DFD SPDLTOCH2 = RES M1104

The tool has been taken from the spindle to holder 2.

DFU M1106 = SET CH2TOMZ

DFD CH2TOMZ = RES M1106

The tool has been taken from holder 2 to the magazine.

DFU M1107 = SET SPDLTOGR

DFD SPDLTOGR = RES M1107

The tool has been taken from the spindle to ground.

DFU M1108 = SET GRTOSPDL

DFD GRTOSPDL = RES M1108

The tool has been taken from ground to the spindle.

The M06 subroutine uses the following M functions to "tell" the PLC which movements it must carry out.

M101	Select in the magazine the position indicated by TAKEPOS and take the tool with holder 1.
M102	Take the tool of holder 1 to the spindle.
M104	Take the spindle tool with holder 2.
M106	Select in the magazine the position indicated by LEAVEPOS and leave the tool of holder 2.

Programming it depends on the type of machine. The auxiliary function will conclude after executing the requested movement.

Certain operations require using the information transferred by the tool manager in the following registers:

LEAVEPOS	This register indicates the magazine position to leave the tool.
TAKEPOS	This register indicates the magazine position of the tool to be taken.

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 2 claws.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Manager emergency signal.

Treatment of the tool manager emergency signal.

DFU B11KEYBD1 AND NOT TMINEM = SET SETTMEM
DFU TMINEM = RES SETTMEM

Pressing the USER12 key activates the emergency.

TMINEM = B11KEYLED1

The lamp of the USER12 key turns on when there is an emergency.

TMINEM AND DFU B12KEYBD1 = SET RESTMEM

Pressing the USER13 key removes the emergency.

7.

TOOL AND MAGAZINE MANAGEMENT.

Synchronous magazine with changer arm and 2 claws.



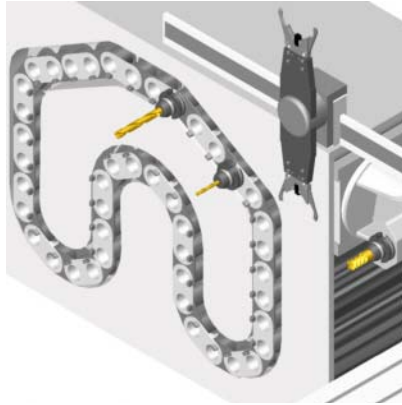
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.11 Asynchronous magazine with changer arm.

Asynchronous magazines have the magazine far from the spindle and the tool is changed using a tool changer arm. Most of the movements may be carried out while machining the part, thus minimizing machining time.



The communication between the manager and the PLC takes place in two stages; the first one when executing the T function and the second one when executing the M06 function.

Execution of the T function.

- 1 When the CNC executes the T function, it lets the tool manager know about it.
- 2 The tool manager sends a command to the PLC to select the next tool in the magazine.
- 3 The CNC continues with the execution of the program without waiting for the tool manager to complete the operation.

Execution of the M06 function.

- 1 When the CNC executes the M06 function, it calls the associated subroutine. The M06 function must also be programmed inside that subroutine so the CNC "tells" the tool manager to start making the tool change.
- 2 The tool manager sends a command to the PLC to make the change.
- 3 The CNC waits for the tool manager to complete the operation before continuing with the execution of the program.

The management of the tool change should be included in the subroutine associated with the M06 and leave the control of the external devices up to the PLC. Use the auxiliary functions to govern the various devices (magazine rotation, magazine movement, tool changer arm, etc.) from the M06 subroutine.

If the tool change implies leaving a tool in the magazine, once executed the change and with the tool in the arm, it is possible to activate the TCHANGEOK mark for the CNC to continue executing the program while the tool is being left in the magazine.

7.

TOOL AND MAGAZINE MANAGEMENT.
Asynchronous magazine with changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.11.1 Valid operations and marks activated by the PLC with each one of them.

Valid operations of the tool manager for this magazine.

The possible operations in this type of magazine are:

TMOPERATION	Meaning.
0	Do nothing.
1	Take a tool from the arm 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 a tool from the arm. The tools of the spindle and of the arm are from the same magazine.
6	Leave the spindle tool in the magazine and take a tool from ground.
7	Leave the spindle tool on the ground and take a tool from the arm.
8	Leave the spindle tool on the ground and take a tool 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	Take a tool from the magazine and insert it in the arm.
14	Leave the spindle tool in the magazine and take a tool from the arm. The tools of the spindle and of the arm are from the different magazines.
16	Leave the tool of the arm (holder 2) in the magazine. Take a tool from the magazine and insert it in the arm (holder 1).

Values of registers TAKEPOS and LEAVEPOS used by the manager.

The TAKEPOS and LEAVEPOS signals can assume the following values.

Value	Meaning
0	Do nothing.
#	Magazine position.
-1	Claw ·1· of the tool changer arm.
-4	Ground tool.

Here is a summary table of the TAKEPOS and LEAVEPOS values used by the manager in each operation as well as the marks that must be activated by the PLC at the end of each operation.

The TCHANGEOK mark is optional and must only be used when possible. Once the tool change is done and the tool is in the arm, activate the TCHANGEOK mark to resume the execution of the program while leaving the tool in the magazine.

TM => PLC			PLC => TM
TMOPERATION	TAKEPOS	LEAVEPOS	
1	-1	0	CH1TOSPD
2	0	#	SPDLTOCH2 + TCHANGEOK + CH2TOMZ
3	0	0	GRTOSPD
4	0	0	SPDLTOGR
5	-1	#	(a) SPDLTOCH2 + CH2TOMZ + CH1TOSPD (b) SPDLTOCH2 + CH1TOSPD + TCHANGEOK + CH2TOMZ

7.

TOOL AND MAGAZINE MANAGEMENT.
Asynchronous magazine with changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TM => PLC			PLC => TM
TMOPERATION	TAKEPOS	LEAVEPOS	
6	-4	#	(a) SPDLTOCH2 + CH2TOMZ + GRTOSPDL (b) SPDLTOCH2 + GRTOSPDL + TCHANGEOK + CH2TOMZ
7	-1	-4	SPDLTOGR + CH1TOSPDL
8	0	0	SPDLTOGR + GRTOSPDL
9	-4	#	GRTOSPDL + SPDLTOCH2 + TCHANGEOK + CH2TOMZ
10	#	-4	MZTOCH1 + CH1TOSPDL + SPDLTOGR
11	#	0	MZTOCH1
	#	#	CH1TOMZ + MZTOCH1
14	-1	#	SPDLTOCH2 + CH2TOMZ + CH1TOSPDL
16	#	#	CH2TOMZ + MZTOCH1

Example of operation.

The following example shows, assuming that there is no tool in the spindle, it shows the functions executed by the CNC, the values sent by the tool manager to the PLC in each operation and the marks to be activated by the PLC in each case.

It is a non-random magazine, each tool occupies the position of its own number and T7, T8 and T9 are ground tools.

CNC	TMOPERATION	TAKEPOS	LEAVEPOS	
T1 M6	11 1	1 -1	0 0	MZTOCH1 CH1TOSPDL
T2 M6	11 5	2 -1	0 1	MZTOCH1 SPDLTOCH2 + CH1TOSPDL + CH2TOMZ
T7 M6	11 6	0 -4	0 2	MZROT SPDLTOCH2 + CH2TOMZ + GRTOSPDL
T8 M6	11 8	0 0	0 0	MZROT SPDLTOGR + GRTOSPDL
T3 T4 M6	11 11 7	0 0 4	0 0 -4	MZTOCH1 CH1TOMZ + MZTOCH1 SDPLTOGR + CH1TOSPDL
T0 M6	11 2	0 0	0 4	MZROT SPDLTOCH2 + CH2TOMZ
T9 M6	11 3	0 0	0 0	MZROT GRTOSPDL
T0 M6	11 4	0 0	0 0	MZROT SPDLTOGR

7.

TOOL AND MAGAZINE MANAGEMENT.

Asynchronous magazine with changer arm.

7.11.2 Detailed description of the operations of the magazine.

Here is a sample of a detailed description of the valid operations for this magazine. For each operation, it indicates which of the TAKEPOS, LEAVEPOS, NEXTPOS and MZID signals are involved and their meanings. It also shows the sequence to follow to complete the operation.

The TAKEPOS, LEAVEPOS and NEXTPOS signals can assume the following values.

Value	Meaning
0	Do nothing.
#	Magazine position.
-1	Claw ·1· of the tool changer arm.
-4	Ground tool.

TMOPERATION = 1

Take a tool from the arm and insert it in the spindle.

TAKEPOS=-1 Tool in the changer arm.

The sequence of this operation is the following. Previously, while machining (when executing the T) the manager sends the code: TMOPERATION=11 to take the tool from the magazine with claw 1.

- 1 Insert the tool of holder 1 in the spindle and activate CH1TOSPD.

TMOPERATION = 2

Leave the spindle tool in the magazine.

LEAVEPOS=# Magazine position to leave the tool.

The sequence of this operation is the following.

- 1 Take the tool from the spindle with holder ·2· and activate the mark SPDLTOCH2.
- 2 Start sending the changer arm to the magazine to leave the tool in holder 2.
- 3 When the arm leaves the collision zone, activate TCHANGEOK, if so desired, to continue executing the program.
- 4 Leave the tool of holder ·2· in the magazine and activate the mark CH2TOMZ.

TMOPERATION = 3

Insert a ground tool in the spindle.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark: GRTOSPD.

TMOPERATION = 4

Leave the spindle tool on the ground.

The sequence of this operation is the following.

- 1 When the operation is completed, activate the mark: SPDLTOGR.

TMOPERATION = 5

Leave the spindle tool in the magazine and take a tool from the arm. The tools of the spindle and of the arm are from the same magazine.

TAKEPOS=-1 Tool in the changer arm.

LEAVEPOS=# Magazine position to leave the tool.

In this operation, the tool manager admits 2 sequences. Previously, while machining (when executing the T) the manager sends the code: TMOPERATION=11 to take the tool from the magazine with claw 1.

7.

TOOL AND MAGAZINE MANAGEMENT.
Asynchronous magazine with changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

First sequence:

- 1 Take the tool from the spindle with holder ·2· and activate the mark SPDLTOCH2.
- 2 Leave the tool of holder ·2· in the magazine and activate the mark CH2TOMZ.
- 3 Insert the tool of holder 1 in the spindle and activate CH1TOSPDL.

Second sequence:

- 1 Take the tool from the spindle with holder ·2· and activate the mark SPDLTOCH2.
- 2 Insert the tool of holder 1 in the spindle and activate CH1TOSPDL.
- 3 Start sending the changer arm to the magazine to leave the tool in holder 2.
- 4 When the arm leaves the collision zone, activate TCHANGEOK, if so desired, to continue executing the program.
- 5 Leave the tool of holder ·2· in the magazine and activate the mark CH2TOMZ.

TMOPERATION = 6

Leave the spindle tool in the magazine and take a tool from ground.

TAKEPOS=-4 Ground tool.

LEAVEPOS=# Magazine position to leave the tool.

In this operation, the tool manager admits 2 sequences.

First sequence:

- 1 Take the tool from the spindle with holder ·2· and activate the mark SPDLTOCH2.
- 2 Leave the tool of holder ·2· in the magazine and activate the mark CH2TOMZ.
- 3 Leave the ground tool in the spindle and activate the mark GRTOSPDL.

Second sequence:

- 1 Take the tool from the spindle with holder ·2· and activate the mark SPDLTOCH2.
- 2 Leave the ground tool in the spindle and activate the mark GRTOSPDL.
- 3 Start sending the changer arm to the magazine to leave the tool in holder 2.
- 4 When the arm leaves the collision zone, activate TCHANGEOK, if so desired, to continue executing the program.
- 5 Leave the tool of holder ·2· in the magazine and activate the mark CH2TOMZ.

TMOPERATION = 7

Leave the spindle tool on the ground and take a tool from the arm.

TAKEPOS=-1 Tool in the changer arm.

LEAVEPOS=-4 Ground tool.

The sequence of this operation is the following. Previously, while machining (when executing the T) the manager sends the code: TMOPERATION=11 to take the tool from the magazine with claw 1.

- 1 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.
- 2 Insert the tool of holder 1 in the spindle and activate CH1TOSPDL.

TMOPERATION = 8

Leave the spindle tool on the ground and take a tool from ground.

The sequence of this operation is the following.

- 1 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.
- 2 Leave the ground tool in the spindle and activate the mark GRTOSPDL.

7.

TOOL AND MAGAZINE MANAGEMENT.
Asynchronous magazine with changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TMOPERATION = 9

Take a ground tool to the magazine going through the spindle.

TAKEPOS=-4 Ground tool.

LEAVEPOS=# Magazine position to leave the tool.

The sequence of this operation is the following.

- 1 Leave the ground tool in the spindle and activate the mark GRTOSPDL.
- 2 Take the tool from the spindle with holder ·2· and activate the mark SPDLTOCH2.
- 3 Start sending the changer arm to the magazine to leave the tool in holder 2.
- 4 When the arm leaves the collision zone, activate TCHANGEOK, if so desired, to continue executing the program.
- 5 Leave the tool of holder ·2· in the magazine and activate the mark CH2TOMZ.

TMOPERATION = 10

Take a tool from the magazine and leave on the ground going through the spindle.

TAKEPOS=# Magazine position to pick up the tool.

LEAVEPOS=-4 Ground tool.

The sequence of this operation is the following.

- 1 Insert the tool of holder 1 in the spindle and activate CH1TOSPDL.
- 2 Leave the tool of the spindle on the ground and activate the mark SPDLTOGR.

TMOPERATION = 11

Take a tool from the magazine and insert it in the arm.

This operation is an optimization of the tool change that permits placing a tool in claw ·1· of the tool changer arm while machining. This operation may be used in the following cases.

First case. When executing a T function while machining.

TAKEPOS=# Position occupied by the tool.

- 1 Take the tool from the magazine with holder 1 and activate the mark MZTOCH1.

Second case. When requesting a new tool and there is another one in holder 1 of the arm.

TAKEPOS=# Position occupied by the tool.

LEAVEPOS=# Position to leave the tool.

- 1 Leave the tool of holder 1 in the magazine and activate the mark CH1TOMZ.
- 2 Take the tool from the magazine with holder 1 and activate the mark MZTOCH1.

TMOPERATION = 14

Leave the spindle tool in the magazine and take a tool from the arm. The tools of the spindle and of the arm are from the different magazines.

TAKEPOS=-1 Tool in the changer arm.

LEAVEPOS=# Magazine position to leave the tool.

MZID The lower portion of this register indicates the destination magazine for the tool and the higher portion the source magazine for the tool.

The sequence of this operation is the following. In this operation, the tool manager admits 2 sequences. Previously, while machining (when executing the T) the manager sends the code: TMOPERATION=11 to take the tool from the magazine with claw 1.

- 1 Take the tool from the spindle with holder 2 and activate the mark SPDLTOCH2.
- 2 Leave the tool of holder 2 in the magazine and activate the mark CH2TOMZ.
- 3 Insert the tool of holder 1 in the spindle and activate CH1TOSPDL.

7.

TOOL AND MAGAZINE MANAGEMENT.
Asynchronous magazine with changer arm.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

TMOPERATION = 16

Leave the tool of the arm (holder -2-) in the magazine. Take a tool from the magazine and insert it in the arm (holder -1-). This operation is an optimization of the tool change that permits placing the tool of claw -2- of the tool changer arm in the magazine.

The sequence of this operation is the following.

TAKEPOS=# Position occupied by the tool.

LEAVEPOS=# Position to leave the tool.

- 1 Leave the tool of holder -2- in the magazine and activate the mark CH2TOMZ.
- 2 Take the tool from the magazine with holder 1 and activate the mark MZTOCH1.

7.**TOOL AND MAGAZINE MANAGEMENT.**

Asynchronous magazine with changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.11.3 Communication between the PLC and the M06 subroutine.

The communication between the PLC and the M06 subroutine takes place using a series of generic marks and registers. The program of the M06 subroutine offered as an example uses the following marks and registers.

Communication between the PLC and the M06 subroutine

Marks activated by the M06 subroutine so the PLC activates the relevant mark of the tool manager.

Mark.	Meaning.
M1100	TCHANGEOK Continue executing the program.
M1101	MZTOCH1 The tool has been taken from the magazine to holder 1.
M1102	CH1TOSPDL The tool has been taken from holder 1 to the spindle.
M1104	SPDLTOCH2 The tool has been taken from the spindle to holder 2.
M1107	SPDLTOGR The tool has been taken from the spindle to ground.
M1108	GRTOSPDL The tool has been taken from ground to the spindle.

The PLC sets the CH2TOMZ mark when the tool has been left.

M functions at the PLC

M functions that imply movements controlled by the PLC:

Function.	Meaning.
M101	Select in the magazine the position indicated by TAKEPOS and take the tool with holder 1.
M102	Take the tool of holder 1 to the spindle.
M104	Take the spindle tool with holder 2.
M106	Start sending the changer arm to the magazine to leave the tool in holder 2.
M121	Take the changer arm to the change point.
M122	Take the changer arm to the magazine.
M123	Retract the changer arm.

Set all the M functions with "before-before" synchronization to the program continues after the M function is completed.

The PLC must consider the M106 completed when the arm exits the collision zone and machining is possible.

7.

TOOL AND MAGAZINE MANAGEMENT.
Asynchronous magazine with changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.11.4 Program of the M06 subroutine.

%L SUB_MZ_TO_CH1 ; Take the tool from the magazine with holder 1.

M101 ; Auxiliary function to execute an action.
 V.PLC.M[1101]=1 ; MZTOCH1 mark to the tool manager.
 #WAIT FOR [V.PLC.M[1101]==0]

#RET

%L SUB_CH1_TO_SPD ; Take the tool of holder 1 to the spindle.

M102 ; Auxiliary function to execute an action.
 V.PLC.M[1102]=1 ; CH1TOSPDL mark to the tool manager.
 #WAIT FOR [V.PLC.M[1102]==0]

#RET

%L SUB_SPD_TO_CH2 ; Take the spindle tool with holder 2.

M104 ; Auxiliary function to execute an action.
 V.PLC.M[1104]=1 ; SPDLTOCH2 mark to the tool manager.
 #WAIT FOR [V.PLC.M[1104]==0]

#RET

%L SUB_CH2_TO_MZ ; Start sending the changer arm to the magazine to leave the tool in holder 2.

M106 ; Auxiliary function to execute an action.
 ; The PLC must consider the M106 completed when the arm exits the collision zone and machining is possible.
 ; The PLC sets the CH2TOMZ mark when the tool has been left.

#RET

%L SUB_SPD_TO_GR ; Remove the tool from the spindle (take it to ground).

#MSG ["Extract tool T%D and press START", V.TM.TOOL]
 ; Message for the operator to extract the tool.
 M0 ; Wait for the operation to be completed.
 #MSG [""]
 ; Remove message.
 V.PLC.M[1107]=1 ; SPDLTOGR mark to the tool manager.
 #WAIT FOR [V.PLC.M[1107]==0]

#RET

%L SUB_GR_TO_SPD ; Insert the ground tool in the spindle.

#MSG ["Insert tool T%D and press START", V.TM.NXTOOL]
 ; Message for the operator to insert the tool.
 M0 ; Wait for the operation to be completed.
 #MSG [""]
 ; Remove message.
 V.PLC.M[1108]=1 ; GRTOSPDL mark to the tool manager.
 #WAIT FOR [V.PLC.M[1108]==0]

#RET

%L SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.

G1 Z_ F_ ; Move the spindle.
 #RET

%L SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point

G1 Z_ F_ ; Move the spindle.
 #RET

%L SUB_ARM_TO_CHG ; Take the changer arm to the change point.

M121 ; Auxiliary function to execute an action.
 #RET

%L SUB_ARM_TO_MZ ; Take the changer arm to the magazine.

M122 ; Auxiliary function to execute an action.
 #RET

%L SUB_ARM_BACK ; Retract the changer arm.

M123 ; Auxiliary function to execute an action.
 #RET

7.

TOOL AND MAGAZINE MANAGEMENT.
 Asynchronous magazine with changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

%SUB_M6.nc

```

#ESBLK ; Begin of single block activation
#DSTOP ; STOP key disable
M6 ; Order the tool manager to start the tool change.

$IF [[V.G.FULLSTATUS & 255]<9] | [[V.G.FULLSTATUS & 255]>13]]
  $IF [[V.G.CNCAUTSTATUS & 4096] | [V.G.CNCAUTSTATUS & 8192]]
    V.S.EXECUTION = 0 ; Simulated Theoretical or G
  $ELSE
    V.S.EXECUTION = 1 ; Execution
  $ENDIF
$ELSE
  V.S.EXECUTION = 0 ; Simulation
$ENDIF

$IF [V.S.EXECUTION == 1] ; CNC in execution

$SWITCH V.PLC.TMOPERATION ; It analyzes the type of operation.

$CASE 1 ; Take a tool from the magazine and insert it in the
; spindle.
; Previously, while machining (when executing the T),
; the tool is taken from the magazine to holder 1.
  LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
  LL SUB_ARM_TO_CHG ; Take the changer arm to the change point.
  LL SUB_CH1_TO_SPD ; Take the tool of holder 1 to the spindle.
  LL SUB_ARM_BACK ; Retract the changer arm.
$BREAK

$CASE 2 ; Leave the spindle tool in the magazine.
  LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
  LL SUB_ARM_TO_CHG ; Take the changer arm to the change point.
  LL SUB_SPD_TO_CH2 ; Take the spindle tool with holder 2.
  LL SUB_ARM_BACK ; Retract the changer arm.
  LL SUB_CH2_TO_MZ ; Start sending the changer arm to the magazine to
; leave the tool in holder 2.
V.PLC.M[1100]=1 ; Order the PLC to activate the TCHANGEOK mark
; "telling" the tool manager that it can continue
; executing.

$BREAK

$CASE 3 ; Insert the ground tool in the spindle.
  LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
  LL SUB_GR_TO_SPD ; Insert the ground tool in the spindle.
$BREAK

$CASE 4 ; Leave the spindle tool on the ground.
  LL SUB_SPD_GMCHG ; Move the spindle to the manual tool change point.
  LL SUB_SPD_TO_GR ; Remove the tool from the spindle.
$BREAK

$CASE 5 ; Leave the spindle tool in the magazine and take
; another one from the magazine.
; Previously, while machining (when executing the T),
; the tool is taken from the magazine to holder 1.
  LL SUB_SPD_AUTCHG ; Move the spindle to the automatic tool change point
  LL SUB_ARM_TO_CHG ; Take the changer arm to the change point.
  LL SUB_SPD_TO_CH2 ; Take the spindle tool with holder 2.
  LL SUB_CH1_TO_SPD ; Take the tool of holder 1 to the spindle.
  LL SUB_ARM_BACK ; Retract the changer arm.
  LL SUB_CH2_TO_MZ ; Start sending the changer arm to the magazine to
; leave the tool.
V.PLC.M[1100]=1 ; Order the PLC to activate the TCHANGEOK mark
; "telling" the tool manager that it can continue
; executing.

$BREAK

```

7.

TOOL AND MAGAZINE MANAGEMENT.
Asynchronous magazine with changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)


```

$CASE 6
    LL SUB_SPD_AUTCHG
    LL SUB_ARM_TO_CHG
    LL SUB_SPD_TO_CH2
    LL SUB_ARM_BACK
    LL SUB_SPD_GMCHG
    LL SUB_GR_TO_SPD
    LL SUB_CH2_TO_MZ
    V.PLC.M[1100]=1
; Leave the spindle tool in the magazine and take
; another one from ground.
; Move the spindle to the automatic tool change point
; Take the changer arm to the change point.
; Take the spindle tool with holder 2.
; Retract the changer arm.
; Move the spindle to the manual tool change point.
; Insert the ground tool in the spindle.
; Start sending the changer arm to the magazine to
; leave the tool.
; Order the PLC to activate the TCHANGEOK mark
; "telling" the tool manager that it can continue
; executing.

$BREAK

$CASE 7
    LL SUB_SPD_GMCHG
    LL SUB_SPD_TO_GR
    LL SUB_SPD_AUTCHG
    LL SUB_ARM_TO_CHG
    LL SUB_CH1_TO_SPD
    LL SUB_ARM_BACK
$BREAK
; Leave the spindle tool on the ground and take another
; one from the magazine.
; Previously, while machining (when executing the T),
; the tool is taken from the magazine to holder 1.
; Move the spindle to the manual tool change point.
; Remove the tool from the spindle.
; Move the spindle to the automatic tool change point
; Take the changer arm to the change point.
; Take the tool of holder 1 to the spindle.
; Retract the changer arm.

$CASE 8
    LL SUB_SPD_GMCHG
    LL SUB_SPD_TO_GR
    LL SUB_GR_TO_SPD
$BREAK
; Leave the spindle tool on the ground and take another
; one from ground.
; Move the spindle to the manual tool change point.
; Remove the tool from the spindle.
; Insert the ground tool in the spindle.

$CASE 9
    LL SUB_SPD_GMCHG
    LL SUB_GR_TO_SPD
    LL SUB_SPD_AUTCHG
    LL SUB_ARM_TO_CHG
    LL SUB_SPD_TO_CH2
    LL SUB_ARM_BACK
    LL SUB_CH2_TO_MZ
    V.PLC.M[1100]=1
; Take a ground tool to the magazine going through the
; spindle.
; Move the spindle to the manual tool change point.
; Insert the ground tool in the spindle.
; Move the spindle to the automatic tool change point
; Take the changer arm to the change point.
; Take the spindle tool with holder 2.
; Retract the changer arm.
; Start sending the changer arm to the magazine to
; leave the tool.
; Order the PLC to activate the TCHANGEOK mark
; "telling" the tool manager that it can continue
; executing.

$BREAK

$CASE 10
    LL SUB_MZ_TO_CH1
    LL SUB_SPD_AUTCHG
    LL SUB_ARM_TO_CHG
    LL SUB_CH1_TO_SPD
    LL SUB_ARM_BACK
    LL SUB_SPD_GMCHG
    LL SUB_SPD_TO_GR
$BREAK
; Take a tool from the magazine and leave on the
; ground going through the spindle.
; Take the tool from the magazine with holder 1.
; Move the spindle to the automatic tool change point
; Take the changer arm to the change point.
; Take the tool of holder 1 to the spindle.
; Retract the changer arm.
; Move the spindle to the manual tool change point.
; Remove the tool from the spindle.

$ENDSWITCH
; End of analysis of type of operation.

V.PLC.TMOPERATION = 0
$WHILE V.TM.MZWAIT == 1
$ENDWHILE
; Wait for the tool manager.

$ENDIF
#DSBLK
#ESTOP
#RET
; End of single block activation.
; STOP key enable

```

7.

TOOL AND MAGAZINE MANAGEMENT.
Asynchronous magazine with changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

7.11.5 Basic PLC programming.

When executing the -T- function

When executing the T function, the tool manager sends to the PLC the code: TMOOPERATION=11 to take the next tool in the arm and approach it to the spindle while machining.

DFU TMOPSTROBE AND CPS TMOOPERATION EQ 11 = ...

The following operations must be carried out:

Send the changer arm to the magazine.

If LEAVEPOS indicates a magazine position, leave the tool of holder 1 in that position and activate the CH1TOMZ mark.

Take the tool of the TAKEPOS position of the magazine with holder 1. Activate the MZTOCH1 mark to "tell" the tool manager that the tool has been taken.

When executing the M06 function

Every time the M06 subroutine ends an action, it lets the PLC know so it activates the relevant mark of the tool manager.

DFU M1100 = SET TCHANGEOK
DFD TCHANGEOK = RES M1100

Continue executing the program.

DFU M1101 = SET MZTOCH1
DFD MZTOCH1 = RES M1101

The tool has been taken from the magazine to holder 1.

DFU M1102 = SET CH1TOSPDL
DFD CH1TOSPDL = RES M1102

The tool has been taken from holder 1 to the spindle.

DFU M1104 = SET SPDLTOCH2
DFD SPDLTOCH2 = RES M1104

The tool has been taken from the spindle to holder 2.

DFU M1107 = SET SPDLTOGR
DFD SPDLTOGR = RES M1107

The tool has been taken from the spindle to ground.

DFU M1108 = SET GRTOSPDL
DFD GRTOSPDL = RES M1108

The tool has been taken from ground to the spindle.

The M06 subroutine uses the following M functions to "tell" the PLC which movements it must carry out.

M101	Select in the magazine the position indicated by TAKEPOS and take the tool with holder 1.
M102	Take the tool of holder 1 to the spindle.
M104	Take the spindle tool with holder 2.
M106	Start sending the changer arm to the magazine to leave the tool in holder 2.
M121	Take the changer arm to the change point.
M122	Take the changer arm to the magazine.
M123	Retract the changer arm.

Programming it depends on the type of machine. The auxiliary functions will be completed after executing the requested movement.

7.

TOOL AND MAGAZINE MANAGEMENT.
Asynchronous magazine with changer arm.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Treatment of the M106 function:

- Completed when the changer arm exits the collision zone and machining is possible.
- Activate the CH2TOMZ mark when the tool has been left to let the tool manager know that the tool has been taken from holder 2 to the magazine".

Certain operations require using the information transferred by the tool manager in the following registers:

LEAVEPOS This register indicates the magazine position to leave the tool.

TAKEPOS This register indicates the magazine position of the tool to be taken.

Manager emergency signal.

Treatment of the tool manager emergency signal.

DFU B11KEYBD1 AND NOT TMINEM = SET SETTMEM

DFU TMINEM = RES SETTMEM

Pressing the USER12 key activates the emergency.

TMINEM = B11KEYLED1

The lamp of the USER12 key turns on when there is an emergency.

TMINEM AND DFU B12KEYBD1 = SET RESTMEM

Pressing the USER13 key removes the emergency.

7.

TOOL AND MAGAZINE MANAGEMENT.
Asynchronous magazine with changer arm.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

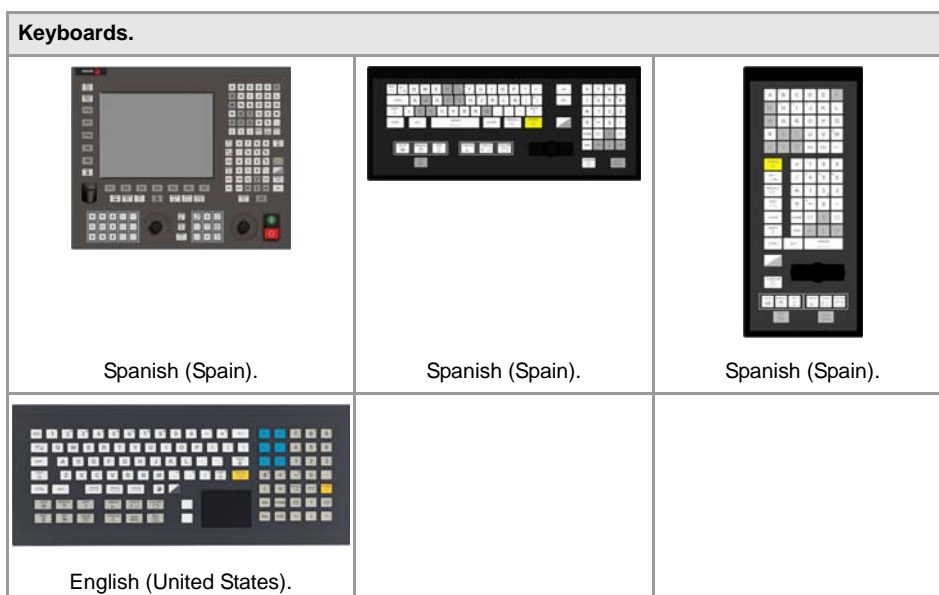
KEYBOARD SIMULATION VIA THE PLC. KEY CODES.

8

8.1 Selecting the language and the keyboard distribution.

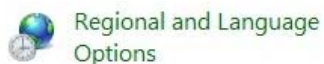
The keyboard distribution controls the characters that appear on screen when the keys are pressed. So that the characters on the screen to match the keys, the appropriate keyboard layout must be selected in the operating system. The key codes (scan codes) also depend on the selected keyboard layout.

In order to change the keyboard distribution, the desired input language and keyboard distribution must be added to Windows.



Adding an input language and a keyboard distribution.

- 1 Click on the Start menu > Control panel > Regional and language configuration. Depending on how Windows is configured, it may be necessary to first select "Clock, language and region".

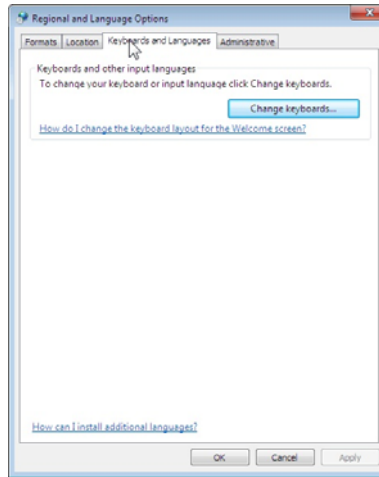


FAGOR AUTOMATION

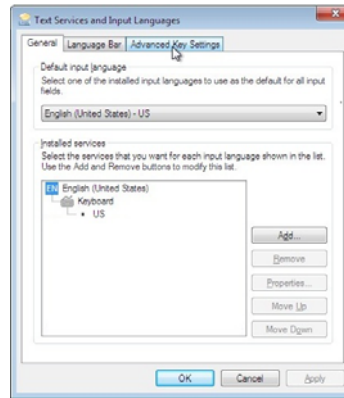
CNC 8070

(REF: 1709)

- Click on the "Keyboards and languages" tab and then click on "Change keyboards".



- In the "Installed services" section, click on the button "Add". Add the desired input language and the keyboard distribution (in this case, English (United States)). Click on "OK" to finish.



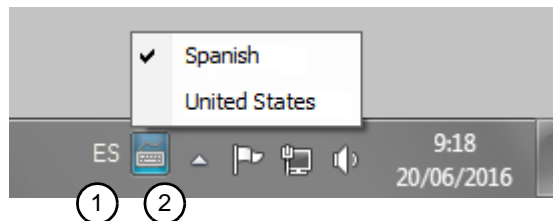
Changing the input language.



On the login screen, click on the language button (top left corner of the screen) and select the language "English (United States)".

Change the input language (only for the active window).

- In the language bar, click on the button "Input language" and select the language "English (United States)".
- Next, click on the button "Keyboard distribution" and select the distribution "United States".



8.

KEYBOARD SIMULATION VIA THE PLC. KEY CODES.
Selecting the language and the keyboard distribution.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

8.2 Key codes.

The key codes (scan codes) may be used to determine which is the last key accepted by the CNC and also to simulate the keyboard from the PLC. Each key is assigned two codes; one for pressing it and another one for releasing it. The code does not depend on the key (printed character), but rather its position on the keyboard (keyboard layout). For example, the code \$27 on a "Spanish (Spain)" keyboard layout corresponds to the character "Ñ" and on an "English (United States)" keyboard layout, it corresponds to the character ";". See ["8.5.1 Scan codes. "Spanish \(Spain\)" keyboard layout."](#) on page 527. See ["8.5.2 Scan codes. Keyboard distribution "English \(United States\)"."](#) on page 528.

CNC's own keys.

The keys of the CNC have no associated key code.

- The [START], [STOP] and [RESET] keys may be actuated from the PLC using the CYSTART, _STOP and RESETIN keys respectively.
- The keys that have an associated hotkey, such as those for accessing work modes, may be simulated through their hotkeys.
- Keys that do not have an associated hotkey cannot be simulated.
- The Fagor logo key cannot be simulated. This key is not found on all keyboards.

The task window ([CTRL] + [A]) shows the list of available hotkeys for the CNC.

8.

KEYBOARD SIMULATION VIA THE PLC. KEY CODES.
Key codes.

FAGOR 

FAGOR AUTOMATION

















CNC 8070

(REF: 1709)

8.2.1 Keyboard shortcuts. CNC's own keys.

8.

KEYBOARD SIMULATION VIA THE PLC. KEY CODES.
Key codes.

Work mode.	Key.	Hotkey.
Task window.		[CTRL]+[A]
Main menu.		[CTRL]+[SHIFT]+[F1]
Automatic mode.		[CTRL]+[F6]
Manual mode.		[CTRL]+[F7]
MDI/MDA mode.		[CTRL]+[F8]
EDISIMU mode.		[CTRL]+[F9]
User tables.		[CTRL]+[F10]
Tool and magazine table.		[CTRL]+[F11]
Utilities mode.		[CTRL]+[F12]
User mode.		[SHIFT]+[F1]
Calculator.		[CTRL]+[K]
Help.		[CTRL]+[F4]
Toolbar to modify the machine dynamics.		[CTRL]+[H]
Operations at the CNC.		Hotkey.
Previous horizontal menu.		[CTRL]+[F1]
Window change.		[CTRL]+[F2]
Changing the screen.		[CTRL]+[F3]
Two-color key.		[ALT]+[B]
Restore a value.		[CTRL]+[F5]
"Single block" mode.		[CTRL]+[B]
Show/Hide virtual operator panel.		[CTRL]+[J]
Show/Hide PLC messages.		[CTRL]+[M]
Show/Hide CNC messages.		[CTRL]+[O]
Minimize/Restore the CNC.		[CTRL]+[W]
Turn the CNC off.		[ALT]+[F4]
Channel synchronization window.		[ALT]+[S]
Show / hide the window for errors and warnings.		[ALT]+[W]

8.3 Consulting the last key accepted by the CNC.

The (V.)G.KEY variable keeps the code of the last key accepted by the CNC (press key code). This variable can be read and written from the PLC and read from the program and interface. To read the variable from the PLC, use the command CNCRD.

Reading variables from the PLC.

The command CNCRD saves the code of the key in the register R100. The PLC activates the M100 mark when the operation begins and keeps it active until the operation ends.
{ condition } = CNCRD(G.KEY,R100,M100)

8.4 Keyboard simulation via the PLC.

To simulate the keyboard from the PLC, write the code of the key in the variable (V.)G.KEY. Each key is assigned two codes; one for pressing it and another one for releasing it. For each key that the PLC sends, both codes must be written; it is recommended to use a delay of at least 200 ms between both codes (for safety). To write the variable from the PLC, use the command CNCWR.

Writing variables from the PLC.

The command CNCWR writes the value of the R101 register in the variable. The PLC activates the M101 mark when the operation begins and keeps it active until the operation ends.
{ condition } = CNCWR(R101,G.KEY,M101)

Uppercase and lowercase letters.

One key can have more than one character. To send the first character (lowercase), use the indicated codes; for the second character (uppercase), add the code corresponding to the [SHIFT] key, in the following way. When sending the two codes, it is recommended to use a delay of at least 200 ms between each (for safety).

- 1 Send the code associated for pressing the [SHIFT] key.
- 2 Send the code associated with pressing the letter key.
- 3 Send the code associated for releasing the [SHIFT] key.
- 4 Send the code associated with releasing the letter key.

Initialize the records.

```
() = MOV $1E R200 ; Press the [A] key
() = MOV $9E R201 ; Release the [A] key
() = MOV $2A R202 ; Press the [SHIFT] key
() = MOV $AA R203 ; Release the [SHIFT] key
```

The first user key (B0KEYBD1) sends the character "a".

```
DFU B0KEYBD1 = CNCWR(R200,G.KEY,M200) = TG1 200 200
T200 = M300
DFD M300 = CNCWR(R201,G.KEY,M201) = TG1 201 200
T201 = M301
```

The second user key (B1KEYBD1) sends the character "A".

```
DFU B1KEYBD1 = CNCWR(R202,G.KEY,M202)
              = CNCWR(R200,G.KEY,M203)
              = TG1 202 200
T202 = M302
DFD M302 = CNCWR(R201,G.KEY,M204)
          = CNCWR(R203,G.KEY,M205)
          = TG1 203 200
T203 = M303
```

8.

KEYBOARD SIMULATION VIA THE PLC. KEY CODES.

Consulting the last key accepted by the CNC.


FAGOR

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Keystroke combination.

To simulate from a keystroke combination from the PLC (for example [CTRL]+[F1]), proceed as follows. When sending the two codes, it is recommended to use a delay of at least 200 ms between each (for safety).

- 1 Send the code associated for pressing the first key (for example, the [CTRL] key).
- 2 Send the code associated for pressing the second key (for example, the [F1] key).
- 3 Send the code associated for releasing the first key (for example, the [CTRL] key).
- 4 Send the code associated for releasing the second key (for example, the [F1] key).

Initialize the records.

```
() = MOV $1D R204 ; Press the [CTRL] key
() = MOV $9D R205 ; Release the [CTRL] key
() = MOV $3B R206 ; Press the [F1] key
() = MOV $BB R207 ; Release the [F1] key
```

The first user key (B0KEYBD1) sends the combination [CTRL]+[F1].

```
DFU B0KEYBD1 = CNCWR(R204,G.KEY,M204)
              = CNCWR(R206,G.KEY,M206)
              = TG1 202 200
T202 = M302
DFD M302 = CNCWR(R207,G.KEY,M207)
          = CNCWR(R205,G.KEY,M205)
          = TG1 203 200
T203 =M303
```

8.

KEYBOARD SIMULATION VIA THE PLC. KEY CODES.
Keyboard simulation via the PLC.

8.5 Key codes depending on keyboard layout.

8.5.1 Scan codes. "Spanish (Spain)" keyboard layout.

On keys with several characters, to send the first character (lowercase), use the indicated codes; for the second character (uppercase), add the code corresponding to the [SHIFT] key, as follows; for the third character, use the code corresponding to the [ALT GR] key.

Character.	Códigos de tecla.		Character.	Códigos de tecla.	
	Press.	Release.		Press.	Release.
a A	\$1E	\$9E	x X	\$2D	\$AD
b B	\$30	\$B0	y Y	\$15	\$95
c C	\$2E	\$AE	z Z	\$2C	\$AC
d D	\$20	\$A0	ç Ç }	\$2B	\$AB
e E	\$12	\$92	° ª \	\$29	\$A9
f F	\$21	\$A1	' ?	\$0C	\$8C
g G	\$22	\$A2	ı ĭ	\$0D	\$8D
h H	\$23	\$A3	' ^ [\$1A	\$9A
i I	\$17	\$97	+ *]	\$1B	\$9B
j J	\$24	\$A4	' " {	\$28	\$A8
k K	\$25	\$A5	, ;	\$33	\$B3
l L	\$26	\$A6	. :	\$34	\$B4
m M	\$32	\$B2	- _	\$35	\$B5
n N	\$31	\$B1	< >	\$56	\$D6
ñ Ñ	\$27	\$A7	0 =	\$0B	\$8B
o O	\$18	\$98	1 ! @	\$02	\$82
p P	\$19	\$99	2 " #	\$03	\$83
q Q	\$10	\$90	3 .	\$04	\$84
r R	\$13	\$93	4 \$ ~	\$05	\$85
s S	\$1F	\$9F	5 %	\$06	\$86
t T	\$14	\$94	6 &	\$07	\$87
u U	\$16	\$96	7 /	\$08	\$88
v V	\$2F	\$AF	8 (\$09	\$89
w W	\$11	\$91	9)	\$0A	\$8A

Key.	Códigos de tecla.	
	Press.	Release.
F1	\$3B	\$BB
F2	\$3C	\$BC
F3	\$3D	\$BD
F4	\$3E	\$BE
F5	\$3F	\$BF
F6	\$40	\$C0
F7	\$41	\$C1
F8	\$42	\$C2
F9	\$43	\$C3
F10	\$44	\$C4
F11	\$57	\$D7
F12	\$58	\$D8
Escape.	\$01	\$81
Tab.	\$0F	\$8F
Enter.	\$1C	\$9C
Insert.	\$E0\$52	\$E0\$D2
Delete.	\$E0\$53	\$E0\$D3

Key.	Códigos de tecla.	
	Press.	Release.
Shift lock.	\$3A	\$BA
Uppercase (SHIFT).	\$2A/ \$36	\$AA/ \$B6
Control (CTRL).	\$1D	\$9D
Alternative (ALT).	\$38	\$B8
Alternative graphic (ALT GR).	\$E0\$38	\$E0\$B8
Space bar.	\$39	\$B9
Back (backspace).	\$0E	\$8E
Print screen.	\$E0\$37	\$E0\$B7
Page up.	\$E0\$51	\$E0\$D1
Page down.	\$E0\$49	\$E0\$C9
Begin.	\$E0\$47	\$E0\$C7
End.	\$E0\$4F	\$E0\$CF
Up arrow.	\$E0\$48	\$E0\$C8
Down arrow.	\$E0\$50	\$E0\$D0
Right arrow.	\$E0\$4D	\$E0\$CD
Left arrow.	\$E0\$4B	\$E0\$CB



KEYBOARD SIMULATION VIA THE PLC. KEY CODES.
Key codes depending on keyboard layout.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

8.5.2 Scan codes. Keyboard distribution "English (United States)".

On keys with several characters, to send the first character (lowercase), use the indicated codes; for the second character (uppercase), add the code corresponding to the [SHIFT] key.

Character.	Códigos de tecla.		Character.	Códigos de tecla.	
	Press.	Release.		Press.	Release.
a A	\$1E	\$9E	x X	\$2D	\$AD
b B	\$30	\$B0	y Y	\$15	\$95
c C	\$2E	\$AE	z Z	\$2C	\$AC
d D	\$20	\$A0	' ~	\$29	\$A9
e E	\$12	\$92	- _	\$0C	\$8C
f F	\$21	\$A1	= +	\$0D	\$8D
g G	\$22	\$A2	[{	\$1A	\$9A
h H	\$23	\$A3] }	\$1B	\$9B
i I	\$17	\$97	; :	\$27	\$A7
j J	\$24	\$A4	' "	\$28	\$A8
k K	\$25	\$A5	\	\$2B	\$AB
l L	\$26	\$A6	, <	\$33	\$B3
m M	\$32	\$B2	. >	\$34	\$B4
n N	\$31	\$B1	/ ?	\$35	\$B5
o O	\$18	\$98	1 !	\$02	\$82
p P	\$19	\$99	2 @	\$03	\$83
q Q	\$10	\$90	3 #	\$04	\$84
r R	\$13	\$93	4 \$	\$05	\$85
s S	\$1F	\$9F	5 %	\$06	\$86
t T	\$14	\$94	6 ^	\$07	\$87
u U	\$16	\$96	7 &	\$08	\$88
v V	\$2F	\$AF	8 *	\$09	\$89
w W	\$11	\$91	9 (\$0A	\$8A
			0)	\$0B	\$8B

Key.	Códigos de tecla.	
	Press.	Release.
F1	\$3B	\$BB
F2	\$3C	\$BC
F3	\$3D	\$BD
F4	\$3E	\$BE
F5	\$3F	\$BF
F6	\$40	\$C0
F7	\$41	\$C1
F8	\$42	\$C2
F9	\$43	\$C3
F10	\$44	\$C4
F11	\$57	\$D7
F12	\$58	\$D8
Escape.	\$01	\$81
Tab.	\$0F	\$8F
Enter.	\$1C	\$9C
Insert.	\$E0\$52	\$E0\$D2
Delete.	\$E0\$53	\$E0\$D3

Key.	Códigos de tecla.	
	Press.	Release.
Shift lock.	\$3A	\$BA
Uppercase (SHIFT).	\$2A/ \$36	\$AA/ \$B6
Control (CTRL).	\$1D	\$9D
Alternative (ALT).	\$38	\$B8
Alternative graphic (ALT GR).	\$E0\$38	\$E0\$B8
Space bar.	\$39	\$B9
Back (Backspace).	\$0E	\$8E
Print screen.	\$E0\$37	\$E0\$B7
Page up.	\$E0\$51	\$E0\$D1
Page down.	\$E0\$49	\$E0\$C9
Begin.	\$E0\$47	\$E0\$C7
End.	\$E0\$4F	\$E0\$CF
Up arrow.	\$E0\$48	\$E0\$C8
Down arrow.	\$E0\$50	\$E0\$D0
Right arrow.	\$E0\$4D	\$E0\$CD
Left arrow.	\$E0\$4B	\$E0\$CB

8.

KEYBOARD SIMULATION VIA THE PLC. KEY CODES.
Key codes depending on keyboard layout.

8.6 Example. Keyboard simulation via the PLC.

The first user key (B0KEYBD1) performs the following operations.

- 1 The CNC goes into jog mode.
- 2 The CNC goes into MDI mode.
- 3 The CNC performs a machine home search on the X axis.
- 4 After the home search, the CNC exits the MDI mode.

For each key to be sent from the PLC, the corresponding codes for "press key" and for the "release key" must be written. When sending both codes, the example uses a 200 ms delay between them (for safety).

; Initialize the records.

```
START OR DFU M313 = CYSTART
() = MOV $1D R200 = MOV $9D R201 ; [CTRL]
() = MOV $41 R202 = MOV $C1 R203 ; [F7]
() = MOV $42 R204 = MOV $C2 R205 ; [F8]
() = MOV $22 R206 = MOV $A2 R207 ; [G]
() = MOV $08 R208 = MOV $88 R209 ; [7]
() = MOV $05 R210 = MOV $85 R211 ; [4]
() = MOV $2D R212 = MOV $AD R213 ; [X]
() = MOV $02 R214 = MOV $82 R215 ; [1]
() = MOV $01 R216 = MOV $81 R217 ; [ESC]
```

; The CNC goes into jog mode. [CTRL]+[F7]

```
DFU B0KEYBD1 = CNCWR(R200,G.KEY,M200) = CNCWR(R202,G.KEY,M201)
= TG1 200 200
T200 = M300
DFD M300 = CNCWR(R201,G.KEY,M202) = CNCWR(R203,G.KEY,M203) = TG1 201 200
T201 = M301
```

; The CNC goes into MDI mode. [CTRL]+[F8]

```
DFD M301 = CNCWR(R200,G.KEY,M200) = CNCWR(R204,G.KEY,M204) = TG1 202 200
T202 = M302
DFD M302 = CNCWR(R201,G.KEY,M202) = CNCWR(R205,G.KEY,M205) = TG1 203 200
T203 = M303
```

; Home search. G74

```
DFD M303 = CNCWR(R206,G.KEY,M206) = TG1 204 200 ; G
T204 = M304
DFD M304 = CNCWR(R207,G.KEY,M207) = TG1 205 200
T205 = M305
DFD M305 = CNCWR(R208,G.KEY,M208) = TG1 206 200 ; 7
T206 = M306
DFD M306 = CNCWR(R209,G.KEY,M209) = TG1 207 200
T207 = M307
DFD M307 = CNCWR(R210,G.KEY,M210) = TG1 208 200 ; 4
T208 = M308
DFD M308 = CNCWR(R211,G.KEY,M211) = TG1 209 200
T209 = M309
DFD M309 = CNCWR(R212,G.KEY,M212) = TG1 210 200 ; X
T210 = M310
DFD M310 = CNCWR(R213,G.KEY,M213) = TG1 211 200
T211 = M311
DFD M311 = CNCWR(R214,G.KEY,M214) = TG1 212 200 ; 1
T212 = M312
DFD M312 = CNCWR(R215,G.KEY,M215) = TG1 213 200
T213 = M313
DFD M313 = SET M500 ; [START] (CYSTART=1)
DFD ZERO = SET M501
```

; The CNC comes out of MDI mode.

```
() = CNCRD(G.STATUS,R220,M220) ; CNC status ("1"=READY)
M500 AND M501 AND (CPS R220 EQ 1) = CNCWR(R216,G.KEY,M216) = TG1 214 200
T214 = M314
DFD M314 = CNCWR(R217,G.KEY,M217) = RES M500=RES M501
```

8.

KEYBOARD SIMULATION VIA THE PLC. KEY CODES.

Example. Keyboard simulation via the PLC.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

8.

KEYBOARD SIMULATION VIA THE PLC. KEY CODES.

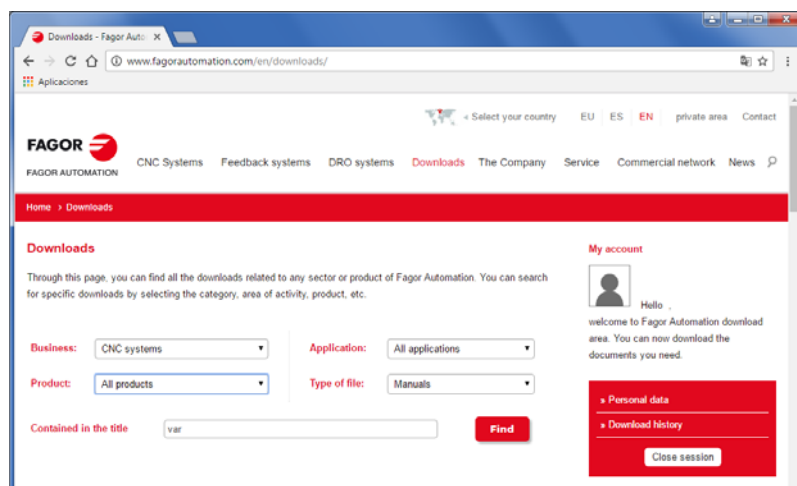
Example. Keyboard simulation via the PLC.

CNC VARIABLES.

9

All information about the CNC variables is in the "CNC Variables" manual, available on Fagor Automation's corporate website. The name of the electronic document is man_8070_var.pdf.

<http://www.fagorautomation.com/en/downloads/>



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

9.

CNC VARIABLES.



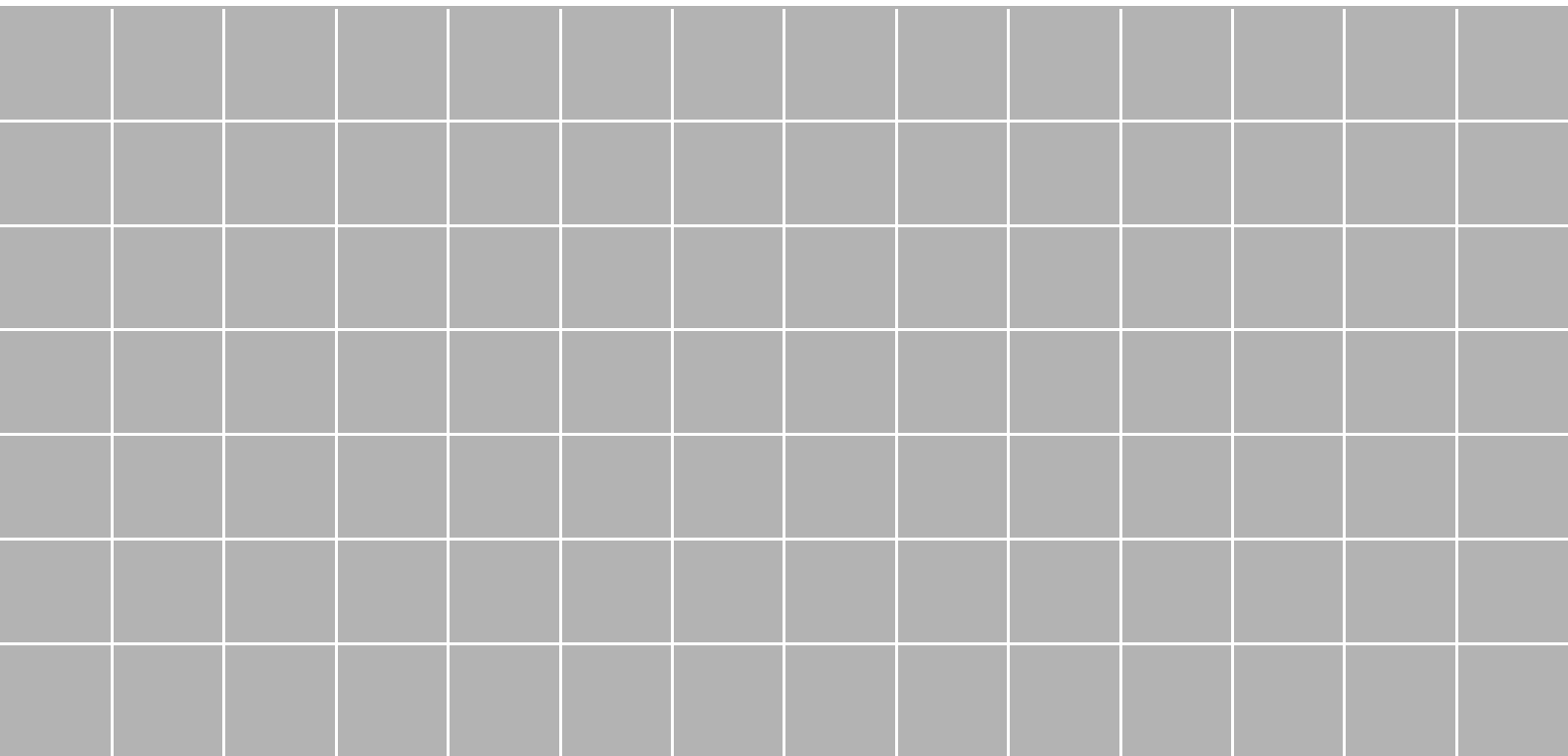
CNC 8070

(REF: 1709)

PART 2. CONCEPTS.



FAGOR AUTOMATION



BLANK PAGE

10.1 Configure the name and number of axes and spindles.

Valid names for axes and spindles.

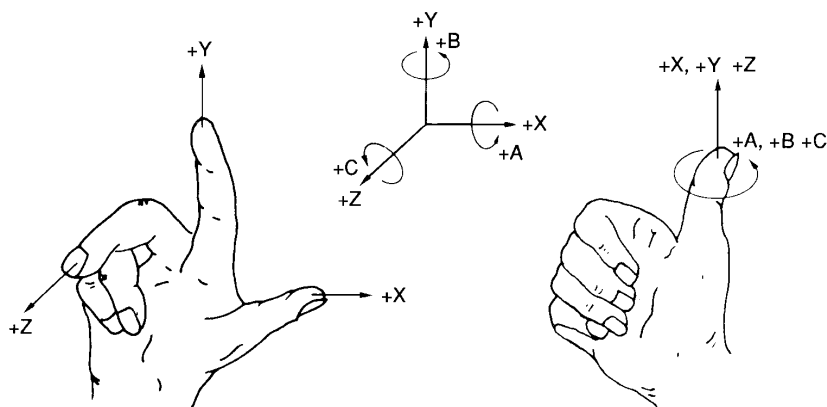
The axis name is defined by 1 or 2 characters. The first character must be one of the letters X - Y - Z - U - V - W - A - B - C. The second character is optional and will be a numerical suffix between 1 and 9. This way, the name of the spindles may be within the range X, X1...X9,...C, C1...C9. For example X, X1, Y3, Z9, W, W7, C...

The spindle name is defined by 1 or 2 characters. The first character must be the letter -S-. 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 S, S1 ... S9.

Name of the axes according to their type. DIN66217 standard.

Any of the names mentioned earlier may be assigned to any type of axis (rotary, auxiliary, etc.). However, if possible, we recommend to apply the DIN 66217 standard when naming the axes of the machine. The DIN 66217 standard names the various types of axes as follows.

Name.	Type of axis according to the DIN 66217 standard.
Xn Yn Zn	Main axes. Two axes make up the work plane and the third axis corresponds to the axis perpendicular to the plane.
Un Vn Wn	Auxiliary axes, parallel to X-Y-Z respectively.
An Bn Cn	Rotary axes, on X-Y-Z respectively.



10.1.1 Configure the number of axes and spindles of the system.

The axes and spindles of the system are configured using machine parameters NAXIS, AXISNAME, NSPDL and SPDLNAME. The order the axes and spindles are defined in parameters AXISNAME and SPDLNAME determines their logic number.

Parameter.	Meaning.
NAXIS	Number of axes of the system.
AXISNAME	List of axes of the system.
NSPDL	Number of spindles of the system
SPDLNAME	List of spindles of the system.

Logic number of the axes and spindles.

As with the name, the logic number permits identifying the axis or spindle in PLC variables, marks, etc.

The logic number of the axes and spindles is determined by the order in which they have been defined in machine parameters AXISNAME and SPDLNAME. The first axis of the AXISNAME table will be logic axis ·1· and so on. The logic numbering of the spindles continues from the last logic axis; hence, in a 3-axis system, the first spindle of the SPDLNAME table will be logic spindle ·4· and so on.

AXISNAME	SPDLNAME	Logic order.
AXISNAME 1		Logic number ·1·.
AXISNAME 2		Logic number ·2·.
AXISNAME 3		Logic number ·3·.
	SPDLNAME 1	Logic number ·4·.
	SPDLNAME 2	Logic number ·5·.

Index of the spindles in the system.

As with the name, the index may be used to identify the spindle in the variables.

The index of a spindle in the system is determined by the order in which it has been defined in machine parameter SPDLNAME. The index of the first spindle of the SPDLNAME table will be ·1· and so on.

AXISNAME SPDLNAME	Logic order.	Index in the system.
AXISNAME 1	Logic number ·1·.	
AXISNAME 2	Logic number ·2·.	
AXISNAME 3	Logic number ·3·.	
SPDLNAME 1	Logic number ·4·.	Spindle with index ·1·.
SPDLNAME 2	Logic number ·5·.	Spindle with index ·2·.

10.

GENERAL SETUP.

Configure the name and number of axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

10.1.2 Configure the number of axes and spindles of the channels.

In a single-channel or multi-channel system, the axes and spindles defined in the system must be distributed among the different channels. The axes and spindles of a channel are configured using parameters CHNAXIS, CHAXISNAME, CHNSPDL and CHSPDLNAME. The order the axes and spindles are defined in parameters CHAXISNAME and CHSPDLNAME determines their index in the channel.

Parameter.	Meaning.
CHNAXIS	Number of axes of the channel.
CHAXISNAME	List of axes of the channel.
CHNSPDL	Number of spindles of the channel.
CHSPDLNAME	List of spindles of the channel.

A channel may have initially associated with it one, several or no of the axes defined in the system. In any case, the number of axes assigned to the channel cannot be higher than the number of axes of the system, defined by parameter NAXIS. The sum of the axes assigned to the channels cannot exceed the number of axes of the system either. This is also valid for the spindles.

Index of an axis or spindle in the channel.

As with the name, the index in the channel permits identifying the axis or spindle in PLC variables, marks, etc.

The logic number of the axes and spindles is determined by the order in which they have been defined in machine parameters CHAXISNAME and CHSPDLNAME. The index of the first axis of the CHAXISNAME table will be ·1· and so on. The index of the first spindle of the CHSPDLNAME table will be ·1· and so on.

CHAXISNAME	CHSPDLNAME	Index in the channel.
CHAXISNAME 1		Axis with index ·1·.
CHAXISNAME 2		Axis with index ·2·.
CHAXISNAME 3		Axis with index ·3·.
	CHSPDLNAME 1	Spindle with index ·1·.
	CHSPDLNAME 2	Spindle with index ·2·.

10.

GENERAL SETUP.

Configure the name and number of axes and spindles.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

10.2 Configuration examples.

Here are various machine configurations. Each one indicates the logic number and index of each axis and spindle in the channel. Each example also shows the work plane associated with functions G17, G18 and G19.

The lathe examples indicate the value of machine parameter GEOCONFIG because the behavior of the functions related to the work planes changes depending on this parameter.

Example list.	Page.
Milling machine with 1 channel, 3 axes and 1 spindle.	539
Milling machine with 1 channel, 5 axes (2 free) and 1 spindle.	540
Milling machine with 3 channels, 9 axes and 2 spindles. Channel 1: 3 axes and 1 spindle. Channel 2: 4 axes and 1 spindle. Channel 3: 2 axes and no spindle.	541
Lathe with 1 channel, 2 axes and 1 spindle. Configuration of "plane" type axes.	542
Lathe with 1 channel, 3 axes and 1 spindle. Configuration of "plane" type axes.	543
Lathe with 1 channel, 3 axes and 1 spindle. Configuration of trihedron type axes.	544
Lathe with 1 channel, 3 axes (1 free) and 1 spindle. Configuration of "plane" type axes.	545
Lathe with 2 channels, 4 axes and 2 spindles. Channel 1: 2 axes and 1 spindle. Configuration of "plane" type axes. Channel 2: 2 axes and 1 spindle. Configuration of "plane" type axes.	546
Lathe with 3 channels, 6 axes and 2 spindles. Channel 1: 2 axes and 1 spindle. Configuration of "plane" type axes. Channel 2: 2 axes and 1 spindle. Configuration of "plane" type axes. Channel 3: 2 axes and no spindle.	547

10.

GENERAL SETUP.
Configuration examples.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

10.2.1 Milling machine with 1 channel, 3 axes and 1 spindle.

The axes and spindles are distributed as follows.

Channel ·1·. 3 axes (X Y Z) 1 spindle (S).

General configuration of the channels, axes and spindles.

System.	Channel ·1·.
NCHANNEL = 1	
NAXIS = 3	CHNAXIS = 3
NSPDL = 1	CHNSPDL = 1

Configuration of the axes and spindles of the system.

AXISNAME	Value.	Logic order.	Index in the system.
AXISNAME 1	X	Logic number ·1·.	
AXISNAME 2	Y	Logic number ·2·.	
AXISNAME 3	Z	Logic number ·3·.	
SPDLNAME 1	S	Logic number ·4·.	Spindle with index ·1·.

Configuration of the axes and spindles of channel ·1·.

CHAXISNAME	Value.	Index in the channel.	Logic order.
CHAXISNAME 1	X	Axis with index ·1·.	Logic number ·1·.
CHAXISNAME 2	Y	Axis with index ·2·.	Logic number ·2·.
CHAXISNAME 3	Z	Axis with index ·3·.	Logic number ·3·.

CHSPDLNAME	Value.	Index in the channel.	Logic order.
CHSPDLNAME 1	S	Spindle with index ·1·.	Logic number ·4·.

Plane selection in channel ·1·.

Function.	Selected plane.
G17	X-Y plane. Perpendicular axis Z.
G18	Z-X plane. Perpendicular Y axis.
G19	Y-Z plane. Perpendicular axis X.
G20	Any plane and/or longitudinal axis.

10.

GENERAL SETUP.
Configuration examples.

10.2.2 Milling machine with 1 channel, 5 axes (2 free) and 1 spindle.

Let us suppose a single-channel machine with three axes and one spindle and two axes that have not been assigned initially. The axes and spindles are distributed as follows.

Channel ·1·. 3 axes (X Y Z) 1 spindle (S).

Not assigned. 2 axes (A B)

General configuration of the channels, axes and spindles.

System	Channel ·1·
NCHANNEL = 1	
NAXIS = 5	CHNAXIS = 3
NSPDL = 1	CHNSPDL = 1

Configuration of the axes and spindles of the system.

AXISNAME	Value.	Logic order.	Index in the system.
AXISNAME 1	X	Logic number ·1·.	
AXISNAME 2	Y	Logic number ·2·.	
AXISNAME 3	Z	Logic number ·3·.	
AXISNAME 4	A	Logic number ·4·.	
AXISNAME 5	B	Logic number ·5·.	
SPDLNAME 1	S1	Logic number ·6·.	Spindle with index ·1·.

Configuration of the axes and spindles of channel ·1·.

CHAXISNAME	Value.	Index in the channel.	Logic order.
CHAXISNAME 1	X	Axis with index ·1·.	Logic number ·1·.
CHAXISNAME 2	Y	Axis with index ·2·.	Logic number ·2·.
CHAXISNAME 3	Z	Axis with index ·3·.	Logic number ·3·.

CHSPDLNAME	Value.	Index in the channel.	Logic order.
CHSPDLNAME 1	S	Spindle with index ·1·.	Logic number ·10·.

Plane selection in channel ·1·.

Function.	Selected plane.
G17	X-Y plane. Perpendicular axis Z.
G18	Z-X plane. Perpendicular Y axis.
G19	Y-Z plane. Perpendicular axis X.
G20	Any plane and/or longitudinal axis.

10.

GENERAL SETUP.
Configuration examples.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

10.2.3 Milling machine with 3 channels, 9 axes and 2 spindles.

Let us suppose tree-channel machine where the first two channels have axes and spindles for machining and the third channel is a loading and unloading system controlled by two axes. The axes and spindles are distributed as follows.

- Channel -1-. 3 axes (X Y Z) 1 spindle (S1).
- Channel -2-. 4 axes (X1 Y1 Z1 W) 1 spindle (S2).
- Channel -3-. 2 axes (U U1) 0 spindles.

General configuration of the channels, axes and spindles.

System	Channel -1-	Channel -2-	Channel -3-
NCHANNEL = 3			
NAXIS = 9	CHNAXIS = 3	CHNAXIS = 4	CHNAXIS = 2
NSPDL = 2	CHNSPDL = 1	CHNSPDL = 1	CHNSPDL = 0

Configuration of the axes and spindles of the system.

AXISNAME	Value.	Logic order.	Index in the system.
AXISNAME 1	X	Logic number -1-.	
AXISNAME 2	Y	Logic number -2-.	
AXISNAME 3	Z	Logic number -3-.	
AXISNAME 4	X1	Logic number -4-.	
AXISNAME 5	Y1	Logic number -5-.	
AXISNAME 6	Z1	Logic number -6-.	
AXISNAME 7	W	Logic number -7-.	
AXISNAME 8	U	Logic number -8-.	
AXISNAME 9	U1	Logic number -9-.	
SPDLNAME 1	S1	Logic number -10-.	Spindle with index -1-.
SPDLNAME 2	S2	Logic number -11-.	Spindle with index -2-.

Configuration of the axes and spindles of channel -1-.

CHAXISNAME	Value.	Index in the channel.	Logic order.
CHAXISNAME 1	X	Axis with index -1-.	Logic number -1-.
CHAXISNAME 2	Y	Axis with index -2-.	Logic number -2-.
CHAXISNAME 3	Z	Axis with index -3-.	Logic number -3-.

CHSPDLNAME	Value.	Index in the channel.	Logic order.
CHSPDLNAME 1	S1	Spindle with index -1-.	Logic number -10-.

Configuration of the axes and spindles of channel -2-.

CHAXISNAME	Value.	Index in the channel.	Logic order.
CHAXISNAME 1	X1	Axis with index -1-.	Logic number -4-.
CHAXISNAME 2	Y1	Axis with index -2-.	Logic number -5-.
CHAXISNAME 3	Z1	Axis with index -3-.	Logic number -6-.
CHAXISNAME 3	W	Axis with index -4-.	Logic number -7-.

CHSPDLNAME	Value.	Index in the channel.	Logic order.
CHSPDLNAME 1	S2	Spindle with index -1-.	Logic number -11-.

10.

GENERAL SETUP.
Configuration examples.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Configuration of the axes and spindles of channel -3-.

CHAXISNAME	Value.	Index in the channel.	Logic order.
CHAXISNAME 1	U1	Axis with index -1-.	Logic number -9-.
CHAXISNAME 2	U	Axis with index -2-.	Logic number -8-.

Plane selection in channels -1- and -2-.

Function.	Channel -1- Selected plane.	Channel -2- Selected plane.
G17	X-Y plane. Perpendicular axis Z.	X1-Y1 plane. Perpendicular axis Z1.
G18	Z-X plane. Perpendicular axis Y.	Z1-X1 plane. Perpendicular axis Y1.
G19	Y-Z plane. Perpendicular axis X.	Y1-Z1 plane. Perpendicular axis X1.
G20	Any plane and/or longitudinal axis.	Any plane and/or longitudinal axis.

LATHE WITH 1 CHANNEL, 2 AXES AND 1 SPINDLE. CONFIGURATION OF "PLANE" TYPE AXES.

The axes and spindles are distributed as follows.

Channel -1-. 2 axes (X Z) 1 spindle (S).

General configuration of the channels, axes and spindles.

System	Channel -1-
NCHANNEL = 1	GEOCONFIG = Plane
NAXIS = 2	CHNAXIS = 2
NSPDL = 1	CHNSPDL = 1
	IPLANE = G18

Configuration of the axes and spindles of the system.

AXISNAME	Value	Logic order.	Index in the system.
AXISNAME 1	X	Logic number -1-.	
AXISNAME 2	Z	Logic number -2-.	
SPDLNAME 1	S	Logic number -3-.	Spindle with index -1-.

Configuration of the axes and spindles of channel -1-.

CHAXISNAME	Value	Index in the channel.	Logic order.
CHAXISNAME 1	X	Axis with index -1-.	Logic number -1-.
CHAXISNAME 2	Z	Axis with index -2-.	Logic number -2-.

CHSPDLNAME	Value	Index in the channel.	Logic order.
CHSPDLNAME 1	S	Spindle with index -1-.	Logic number -3-.

Plane selection in channel -1-.

With this configuration, the work plane is always G18 and will be formed by the first two axes defined in the channel. If the X (first axis of the channel) and Z (second axis of the channel) have been defined, the work plane will be the ZX (Z as abscissa and X as ordinate). Functions G17 and G19 are not valid. Since there are only two axes, function G20 makes no sense.

(REF: 1709)

10.

GENERAL SETUP.
Configuration examples.



FAGOR AUTOMATION

CNC 8070

10.2.4 Lathe with 1 channel, 3 axes and 1 spindle. Configuration of "plane" type axes.

The axes and spindles are distributed as follows.

Channel ·1·. 3 axes (X Z Y) 1 spindle (S).

General configuration of the channels, axes and spindles.

System	Channel ·1·
NCHANNEL = 1	GEOCONFIG = Plane
NAXIS = 3	CHNAXIS = 3
NSPDL = 1	CHNSPDL = 1
	IPLANE = G18

Configuration of the axes and spindles of the system.

AXISNAME	Value	Logic order.	Index in the system.
AXISNAME 1	X	Logic number ·1·.	
AXISNAME 2	Z	Logic number ·2·.	
AXISNAME 3	Y	Logic number ·3·.	
SPDLNAME 1	S	Logic number ·4·.	Spindle with index ·1·.

Configuration of the axes and spindles of channel ·1·.

CHAXISNAME	Value	Index in the channel.	Logic order.
CHAXISNAME 1	X	Axis with index ·1·.	Logic number ·1·.
CHAXISNAME 2	Z	Axis with index ·2·.	Logic number ·2·.
CHAXISNAME 3	Y	Axis with index ·3·.	Logic number ·3·.

CHSPDLNAME	Value	Index in the channel.	Logic order.
CHSPDLNAME 1	S	Spindle with index ·1·.	Logic number ·4·.

Plane selection in channel ·1·.

Function	Selected plane.
G18	ZX plane Z axis for longitudinal compensation.

With a plane-type-axis configuration, the G18 plane is always active; in this case the ZX plane. Functions G17 and G19 are not valid. Function G20 permits selecting the rest of the axes (Y axis) as the axis for longitudinal compensation.

10.

GENERAL SETUP.

Configuration examples.

10.2.5 Lathe with 1 channel, 3 axes and 1 spindle. Configuration of trihedron type axes.

The axes and spindles are distributed as follows.

Channel ·1·. 3 axes (X Y Z) 1 spindle (S).

General configuration of the channels, axes and spindles.

System	Channel ·1·
NCHANNEL = 1	GEOCONFIG = Trihedron
NAXIS = 3	CHNAXIS = 3
NSPDL = 1	CHNSPDL = 1

Configuration of the axes and spindles of the system.

AXISNAME	Value	Logic order.	Index in the system.
AXISNAME 1	X	Logic number ·1·.	
AXISNAME 2	Y	Logic number ·2·.	
AXISNAME 3	Z	Logic number ·3·.	
SPDLNAME 1	S	Logic number ·4·.	Spindle with index ·1·.

Configuration of the axes and spindles of channel ·1·.

CHAXISNAME	Value	Index in the channel.	Logic order.
CHAXISNAME 1	X	Axis with index ·1·.	Logic number ·1·.
CHAXISNAME 2	Y	Axis with index ·2·.	Logic number ·2·.
CHAXISNAME 3	Z	Axis with index ·3·.	Logic number ·3·.

CHSPDLNAME	Value	Index in the channel.	Logic order.
CHSPDLNAME 1	S	Spindle with index ·1·.	Logic number ·4·.

Plane selection in channel ·1·.

Function.	Selected plane.
G17	X-Y plane. Perpendicular axis Z.
G18	Z-X plane. Perpendicular Y axis.
G19	Y-Z plane. Perpendicular axis X.
G20	Any plane and/or longitudinal axis.

With a trihedron-type-axis 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 in parameter IPLANE).

10.

GENERAL SETUP.
Configuration examples.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

10.2.6 Lathe with 1 channel, 3 axes (1 free) and 1 spindle. Configuration of "plane" type axes.

The axes and spindles are distributed as follows.

Channel ·1·. 3 axes (X Z U) 1 spindle (S).

General configuration of the channels, axes and spindles.

System	Channel ·1·
NCHANNEL = 1	GEOCONFIG = Plane
NAXIS = 3	CHNAXIS = 2
NSPDL = 1	CHNSPDL = 1
	IPLANE = G18

Configuration of the axes and spindles of the system.

AXISNAME	Value	Logic order.	Index in the system.
AXISNAME 1	X	Logic number ·1·.	
AXISNAME 2	Z	Logic number ·2·.	
AXISNAME 3	U	Logic number ·3·.	
SPDLNAME 1	S	Logic number ·4·.	Spindle with index ·1·.

Configuration of the axes and spindles of channel ·1·.

CHAXISNAME	Value	Index in the channel.	Logic order.
CHAXISNAME 1	X	Axis with index ·1·.	Logic number ·1·.
CHAXISNAME 2	Z	Axis with index ·2·.	Logic number ·2·.

CHSPDLNAME	Value	Index in the channel.	Logic order.
CHSPDLNAME 1	S	Spindle with index ·1·.	Logic number ·4·.

Plane selection in channel ·1·.

Function	Selected plane.
G18	ZX plane Z axis for longitudinal compensation.

With a plane-type-axis configuration, the G18 plane is always active; in this case the ZX plane. Functions G17 and G19 are not valid.

10.

GENERAL SETUP.
Configuration examples.

10.2.7 Lathe with 2 channels, 4 axes and 2 spindles. Configuration of "plane" type axes.

The axes and spindles are distributed as follows.

Channel ·1·. 2 axes (X Z) 1 spindle (S).

Channel ·2·. 2 axes (X1 Z1) 1 spindle (S1).

General configuration of the channels, axes and spindles.

System	Channel ·1·	Channel ·2·
NCHANNEL = 2	GEOCONFIG = Plane	GEOCONFIG = Plane
NAXIS = 4	CHNAXIS = 2	CHNAXIS = 2
NSPDL = 2	CHNSPDL = 1	CHNSPDL = 1
	IPLANE = G18	IPLANE = G18

Configuration of the axes and spindles of the system.

AXISNAME	Value	Logic order.	Index in the system.
AXISNAME 1	X	Logic number ·1·.	
AXISNAME 2	Z	Logic number ·2·.	
AXISNAME 3	X1	Logic number ·3·.	
AXISNAME 4	Z1	Logic number ·4·.	
SPDLNAME 1	S	Logic number ·5·.	Spindle with index ·1·.
SPDLNAME 2	S1	Logic number ·6·.	Spindle with index ·2·.

Configuration of the axes and spindles of channel ·1·.

CHAXISNAME	Value	Index in the channel.	Logic order.
CHAXISNAME 1	X	Axis with index ·1·.	Logic number ·1·.
CHAXISNAME 2	Z	Axis with index ·2·.	Logic number ·2·.

CHSPDLNAME	Value	Index in the channel.	Logic order.
CHSPDLNAME 1	S	Spindle with index ·1·.	Logic number ·5·.

Configuration of the axes and spindles of channel ·2·.

CHAXISNAME	Value	Index in the channel.	Logic order.
CHAXISNAME 1	X1	Axis with index ·1·.	Logic number ·3·.
CHAXISNAME 2	Z1	Axis with index ·2·.	Logic number ·4·.

CHSPDLNAME	Value	Index in the channel.	Logic order.
CHSPDLNAME 1	S1	Spindle with index ·1·.	Logic number ·6·.

Plane selection in channels ·1· and ·2·.

With this configuration, the work plane is always G18 and will be formed by the first two axes defined in the channel. If the X (first axis of the channel) and Z (second axis of the channel) have been defined, the work plane will be the ZX (Z as abscissa and X as ordinate). Functions G17 and G19 are not valid. Since there are only two axes, function G20 makes no sense.

10.

GENERAL SETUP.
Configuration examples.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

10.2.8 Lathe with 3 channels, 6 axes and 2 spindles. Configuration of "plane" type axes.

The axes and spindles are distributed as follows.

- Channel -1-. 2 axes (X Z) 1 spindle (S).
- Channel -2-. 2 axes (X1 Z1) 1 spindle (S1).
- Channel -3-. 2 axes (U V)

General configuration of the channels, axes and spindles.

System	Channel -1-	Channel -2-	Channel -3-
NCHANNEL = 3	GEOCONFIG = Plane	GEOCONFIG = Plane	
NAXIS = 6	CHNAXIS = 2	CHNAXIS = 2	CHNAXIS = 2
NSPDL = 2	CHNSPDL = 1	CHNSPDL = 1	CHNSPDL = 0
	IPLANE = G18	IPLANE = G18	

Configuration of the axes and spindles of the system.

AXISNAME	Value	Logic order.	Index in the system.
AXISNAME 1	X	Logic number -1-.	
AXISNAME 2	Z	Logic number -2-.	
AXISNAME 3	X1	Logic number -3-.	
AXISNAME 4	Z1	Logic number -4-.	
AXISNAME 5	U	Logic number -5-.	
AXISNAME 6	V	Logic number -6-.	
SPDLNAME 1	S	Logic number -7-.	Spindle with index -1-.
SPDLNAME 2	S1	Logic number -8-.	Spindle with index -2-.

Configuration of the axes and spindles of channel -1-.

CHAXISNAME	Value	Index in the channel.	Logic order.
CHAXISNAME 1	X	Axis with index -1-.	Logic number -1-.
CHAXISNAME 2	Z	Axis with index -2-.	Logic number -2-.

CHSPDLNAME	Value	Index in the channel.	Logic order.
CHSPDLNAME 1	S	Spindle with index -1-.	Logic number -7-.

Configuration of the axes and spindles of channel -2-.

CHAXISNAME	Value	Index in the channel.	Logic order.
CHAXISNAME 1	X1	Axis with index -1-.	Logic number -3-.
CHAXISNAME 2	Z1	Axis with index -2-.	Logic number -4-.

CHSPDLNAME	Value	Index in the channel.	Logic order.
CHSPDLNAME 1	S1	Spindle with index -1-.	Logic number -8-.

Configuration of the axes and spindles of channel -3-.

CHAXISNAME	Value	Index in the channel.	Logic order.
CHAXISNAME 1	U	Axis with index -1-.	Logic number -5-.
CHAXISNAME 2	V	Axis with index -2-.	Logic number -6-.

Plane selection in channels -1- and -2-.

With this configuration, the work plane is always G18 and will be formed by the first two axes defined in the channel. If the X (first axis of the channel) and Z (second axis of the channel) have been defined, the work plane will be the ZX (Z as abscissa and X as ordinate). Functions G17 and G19 are not valid. Since there are only two axes, function G20 makes no sense.

10.

GENERAL SETUP.
Configuration examples.

10.

GENERAL SETUP.

Configuration examples.



CNC 8070

(REF: 1709)

11.1 Configure an axis as rotary axis.

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.

Machine parameter configuration.

Machine parameters LIMIT+ and LIMIT- set the travel limits for the axis; there are no module limits.

Parameter.	Value.
AXISTYPE	Rotary.
AXISMODE	Linearlike.
UNIDIR	Not used.
SHORTESTWAY	Not used.
LIMIT+ LIMIT-	Travel limits.
MODUPLIM MODLOWLIM	Not used.

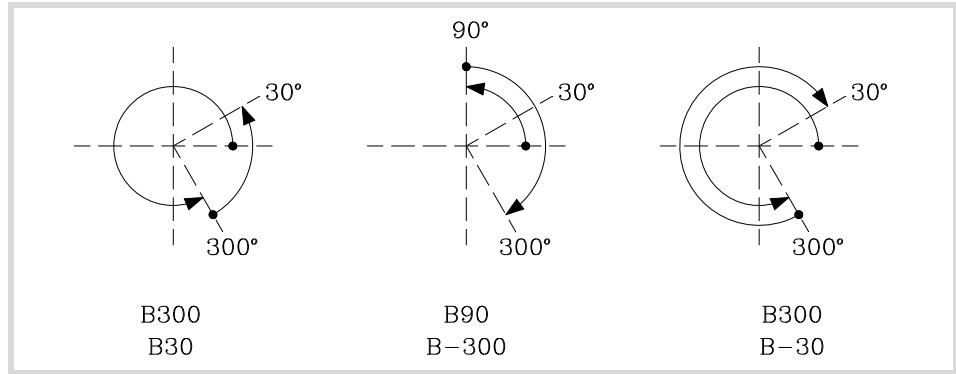
Normal rotary axis.

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.

11.

CONFIGURING AN AXIS.
Configure an axis as rotary axis.



Machine parameter configuration.

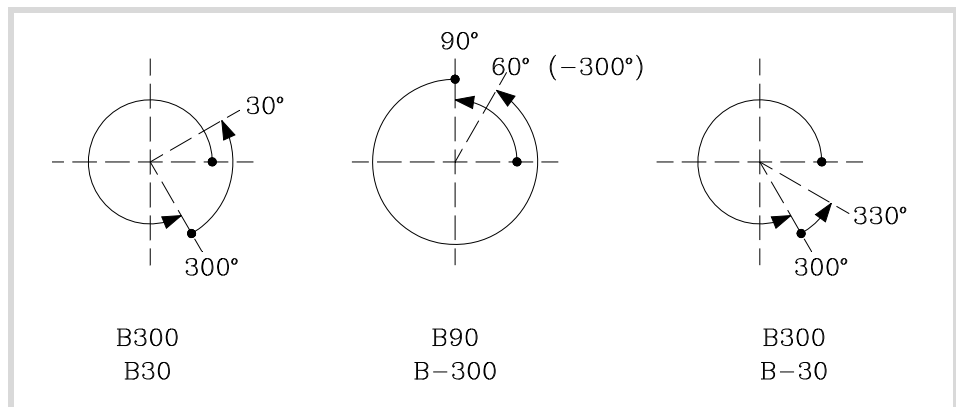
Machine parameters MODUPLIM and MODLOWLIM set the module limits for the axis; there are no travel limits.

Parameter.	Value.
AXISTYPE	Rotary.
AXISMODE	Module.
UNIDIR	No.
SHORTESTWAY	No.
LIMIT+ LIMIT-	Not used. There are no travel limits.
MODUPLIM MODLOWLIM	Limits of the module. Limits to display the position values.

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.



Machine parameter configuration.

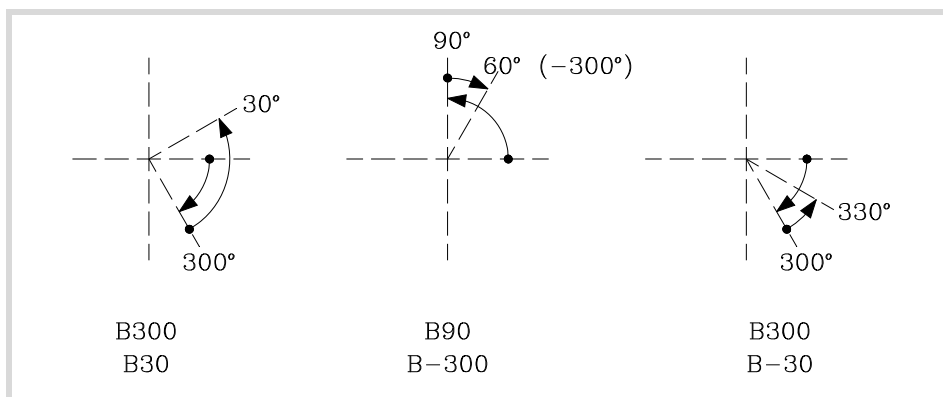
Machine parameters MODUPLIM and MODLOWLIM set the module limits for the axis; there are no travel limits.

Parameter.	Value.
AXISTYPE	Rotary.
AXISMODE	Module.
UNIDIR	Positive / Negative.
SHORTESTWAY	No.
LIMIT+ LIMIT-	Not used. There are no travel limits.
MODUPLIM MODLOWLIM	Limits of the module. Limits to display the position values.

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.



Machine parameter configuration.

Machine parameters MODUPLIM and MODLOWLIM set the module limits for the axis; there are no travel limits.

Parameter.	Value.
AXISTYPE	Rotary.
AXISMODE	Module.
UNIDIR	No.
SHORTESTWAY	Yes.
LIMIT+ LIMIT-	Not used. There are no travel limits.
MODUPLIM MODLOWLIM	Limits of the module. Limits to display the position values.

11.

CONFIGURING AN AXIS.
Configure an axis as rotary axis.



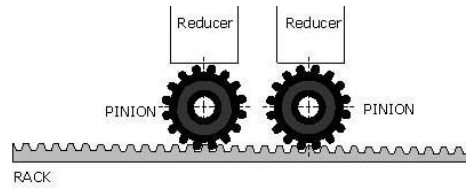
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

11.2 Configure two axes as a tandem axis.

A tandem axis consists in two motors mechanically coupled (slaved) and making up a single transmission system (axis or spindle). Only one axis (master tandem axis) can be displayed and programmed at the CNC. The other axis (slave tandem axis) is not displayed and is not programmable; it is controlled by the CNC.

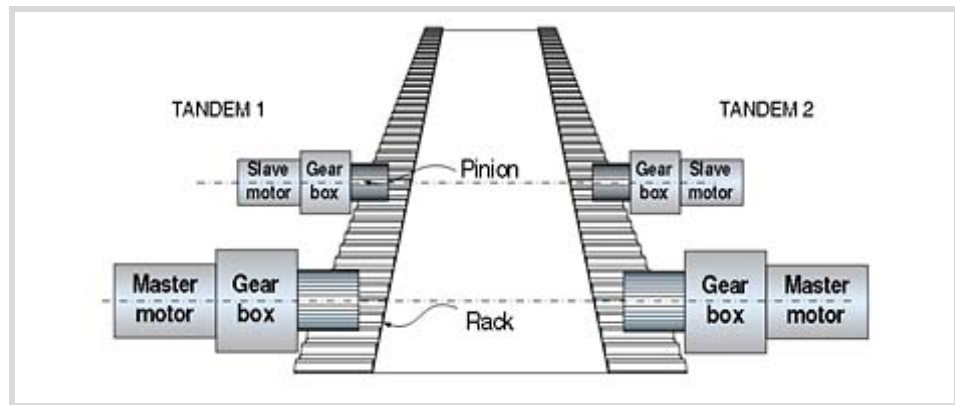


Example of a tandem axis made up of two pinions and a rack.

The following aspects may be pointed out in this configuration:

- A tandem axis helps provide the necessary torque to move an axis when a single motor is not capable of supplying enough torque to do it.
- Applying a pre-load torque between the main motor and the slave motor reduces the backlash on the rack and pinion.
- The rigidity of the rack-and-pinion system is greater than that of long leadscrews.

One of the many applications for the control of a tandem axis has to do with Gantry machines. The following example shows a gantry axis with two tandem axes.



Requirements and limitations of tandem axes.

Each tandem pair (master and slave) must meet the following requirements:

- Each master tandem axis (spindle) admits one single slave tandem axis (spindle).
- A preload may be applied between the two motors.
- Each motor may have a different rated torque.
- The turning direction of each motor may be different from the other's.
- The torque distribution between both motors may be different from 1:1 ratio. For example, on motors whose rated torque is different.
- Both axes and drives must be of the same type (same AXISTYPE and DRIVETYPE parameters for both axes).
- The channel changing permission (parameter AXISEXCH) must be the same in both axes.
- Both axes and drives must have the same software limits (same LIMIT+ and LIMIT- parameters for both axes).
- When the axes are rotary, both axes must be of the same type (same AXISMODE and SHORTESTWAY parameters for both axes).
- The fast and slow home search speeds (parameters REFFEED1 and REFFEED2) must be the same for both axes.

11.

CONFIGURING AN AXIS.
Configure two axes as a tandem axis.

11.2.1 Tandem axis configuration. Machine parameters.

The tandem pairs of the system are configured by machine parameters. Each tandem pair has the following general machine parameters to configure it.

Parameter.	Meaning.
TANDEM	Table of tandem axes or spindles.
MASTERAXIS	Master or main axis/spindle.
SLAVEAXIS	Slave axis/spindle.
TORQDIST	Torque distribution.
PRELOAD	Preload between both motors.
PRELFITI	Filter time to apply the preload.
TPROGAIN	Proportional gain (Kp) for the tandem.
TINTIME	Integral gain (Ki) for the tandem.
TCOMPLIM	Compensation limit.

Axes or spindles of the tandem system.

A tandem system can consist of a pair of linear axes, rotary axes or spindles. The pair of axes or spindles making up the tandem system is set by machine parameters MASTERAXIS and SLAVEAXIS inside the TANDEM table. The axes must be previously defined in parameter AXISNAME and the spindles in parameter SPDLNAME.

The master motor of the tandem, besides generating torque, it is in charge of positioning. The slave motor only provides torque.

The CNC closes the position loop only with the position of the master axis of the tandem. The velocity command of the master axis/spindle of the tandem is also sent to the slave axis/spindle of the tandem closing the velocity loop. The tandem control changes the velocity command of the master axis/spindle and that of the slave axis/spindle according to the torque distribution and the selected preload.

Type of servo system and feedback.

Tandem axes or spindles must be velocity-Sercos; if not, both axes will move, but it will not be a tandem system. All the axes that will be interpolated with the tandem axis must also operate in velocity-Sercos mode.

In a tandem system, the master axis or spindle can carry either external or internal feedback. The slave axis or spindle normally uses internal feedback. The tandem does not admit combined feedback (external+internal).

Considerations for tandem spindles.

- The PLC must enable and disable the SPENA(axis) and DRENA(axis) signals of the master and slave spindles at the same time.
- Both spindles must be in the same gear (range).
- To control the tandem spindle from the PLC, for example in a gear change, it is only necessary to activate the marks PLCCNTL, SPDLEREV and SANALOG of the master spindle.

Torque load distribution in the tandem.

The torque distribution sets the torque percentage supplied by each motor to obtain the total necessary torque in the tandem. The torque distribution in the tandem is defined with parameter TORQDIST that indicates the percentage to be provided by the master motor. The difference between the value of this parameter and 100% is the percentage applied to the slave motor.

The torque distribution between both motors may be different from 1:1 ratio; for example, on motors whose rated torque is different. If the motors are identical and they're both supposed to output the same torque, this parameter should be set to 50%.

11.

CONFIGURING AN AXIS.

Configure two axes as a tandem axis.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Apply a preload value in the tandem.

The preload is the amount of torque applied before, in both directions, to both motors of the tandem to set a traction between them in order to eliminate the backlash when the tandem is in rest position. In order for the two motors to supply opposite torques, the preload value must be greater than the maximum torque needed at all times, including accelerations.



Applying the preload necessarily implies mechanically joining the master and slave axes that make up the tandem; otherwise, the motors will move even without control command.

The preload is defined with parameter PRELOAD that indicates the percentage of the rated torque of the master motor that will be applied as preload.

Time to apply the preload.

Parameter PRELFITI may be set in order to apply the preload gradually avoiding torque steps at the tandem compensator. This parameter enables a first order filter that sets the time it takes the CNC to apply the preload gradually.

Proportional gain (Kp) for the tandem.

The proportional controller generates an output proportional to the torque error between the two motors. The proportional gain may be set with parameter TPROGAIN.

Integral gain (Ki) for the tandem.

The integral controller generates an output proportional to the integral of the torque error between the two motors. The integral gain may be set with parameter TINTTIME.

11.

CONFIGURING AN AXIS.

Configure two axes as a tandem axis.



FAGOR AUTOMATION

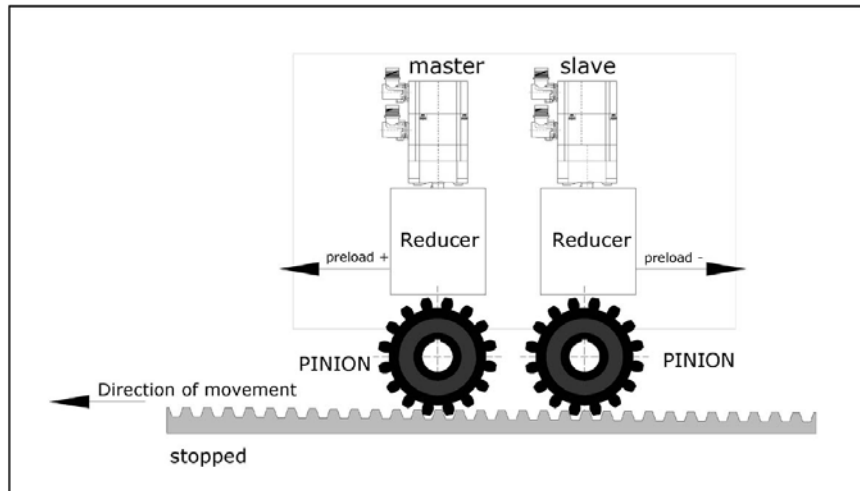
CNC 8070

(REF: 1709)

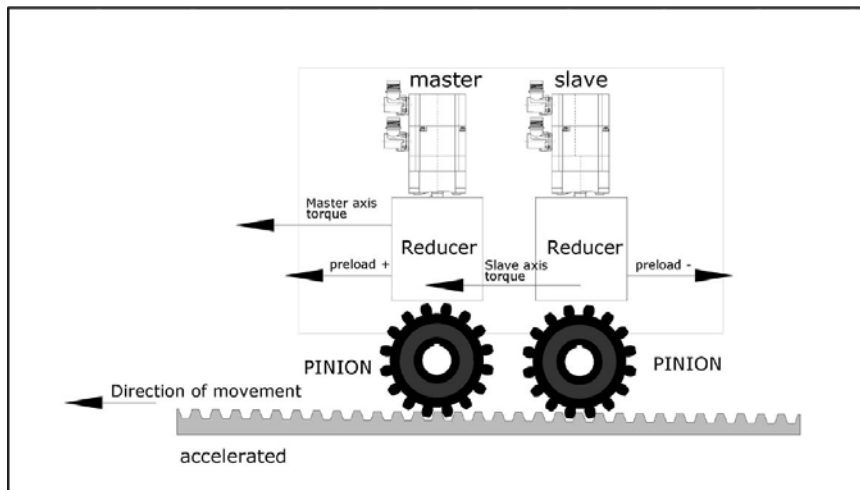
11.2.2 Effect of the preload.

The following diagrams show the effect of pre-load in different situations.

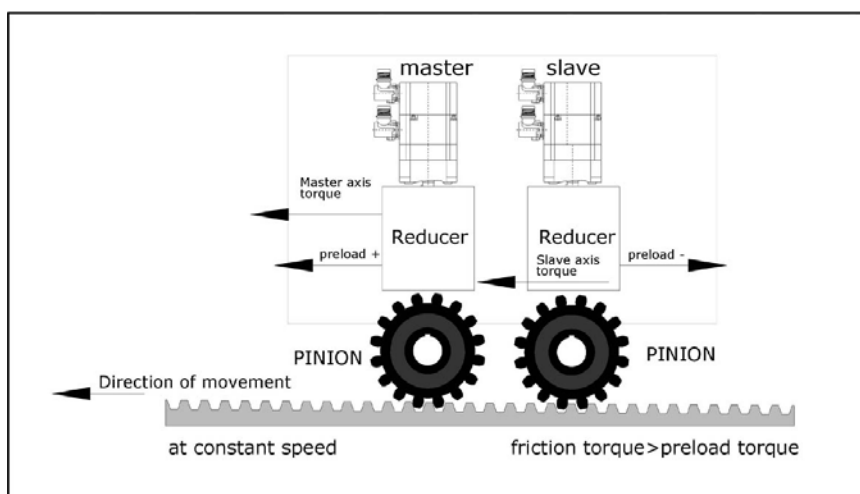
Preload at rest.



Preload with acceleration.



Preload at constant speed. Friction torque > Preload.



11.

CONFIGURING AN AXIS.
Configure two axes as a tandem axis.

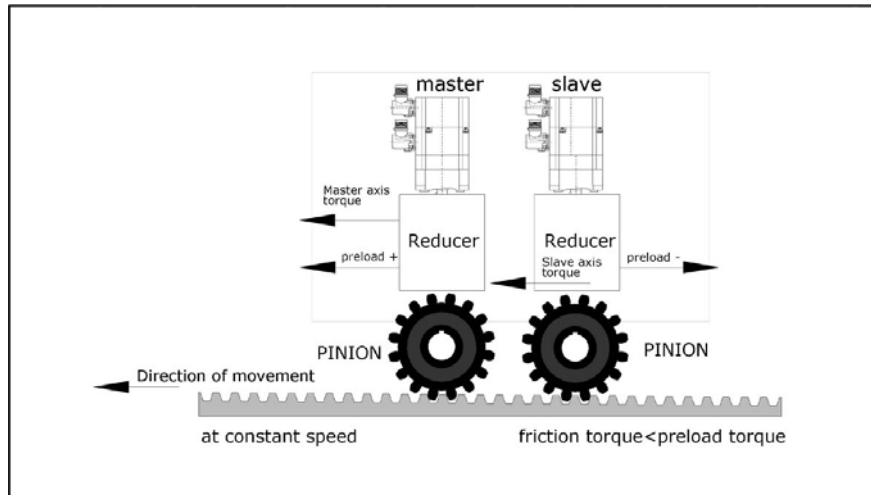


FAGOR AUTOMATION

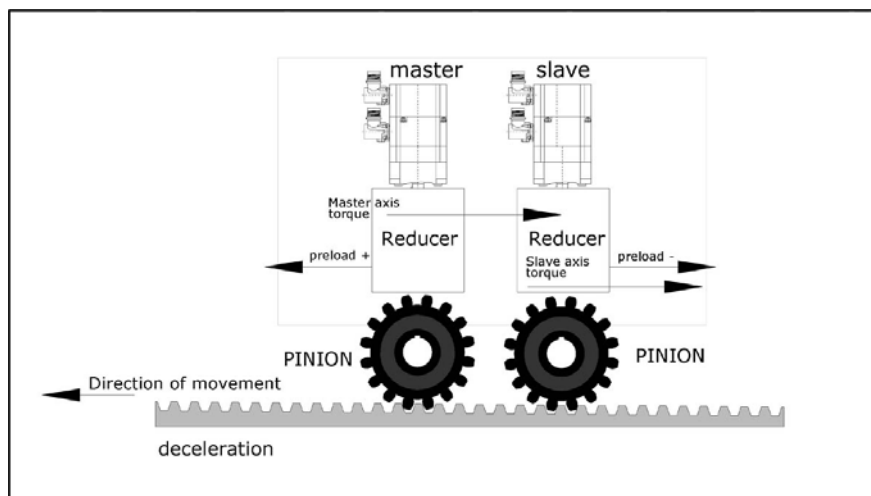
CNC 8070

(REF: 1709)

Preload at constant speed. Friction torque < Preload.



Preload with deceleration.



11.

CONFIGURING AN AXIS.

Configure two axes as a tandem axis.



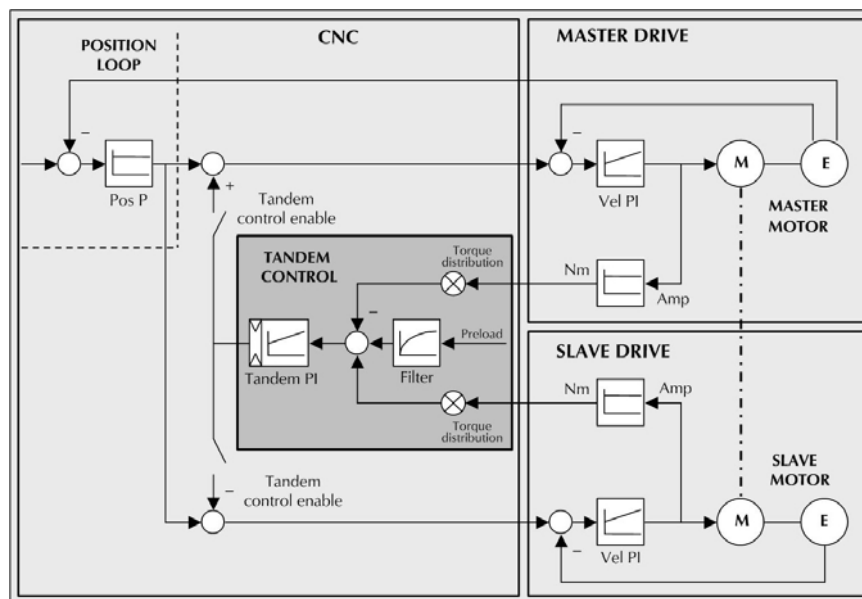
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

11.2.3 Tandem axis configuration. Block diagram.

The block diagram of the tandem control system shows the master tandem axis with its slave tandem axis. The block diagram for a gantry machine consists of two identical diagrams to the one shown in the figure.

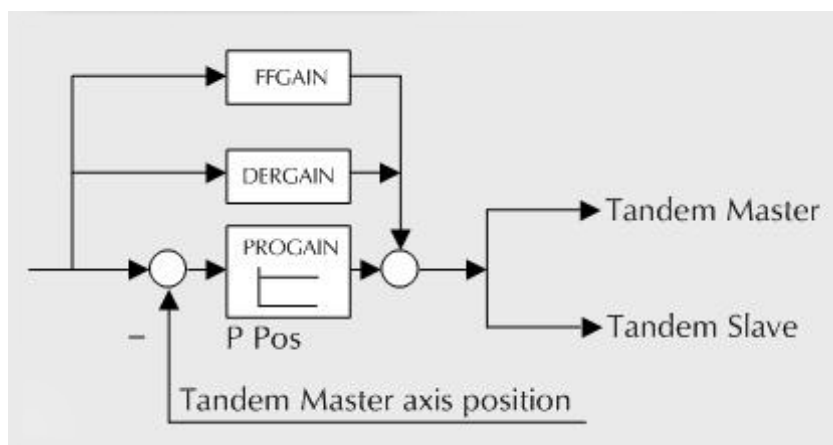


The block diagram has an area for the drive and another one for the CNC that comprises the position loop and the control of the tandem.

Position and velocity loop.

The position loop is closed only with the position of the master axis of the tandem. The velocity command of the tandem master axis is also sent to the tandem slave axis closing the velocity loop.

The tandem control changes the velocity command of the master axis and that of the slave axis according to the torque distribution and the selected preload.



The feed-forward and AC-forward values of the master axis are applied to the slave axis; consequently, they must have the same gear ratios.

11.

CONFIGURING AN AXIS.

Configure two axes as a tandem axis.



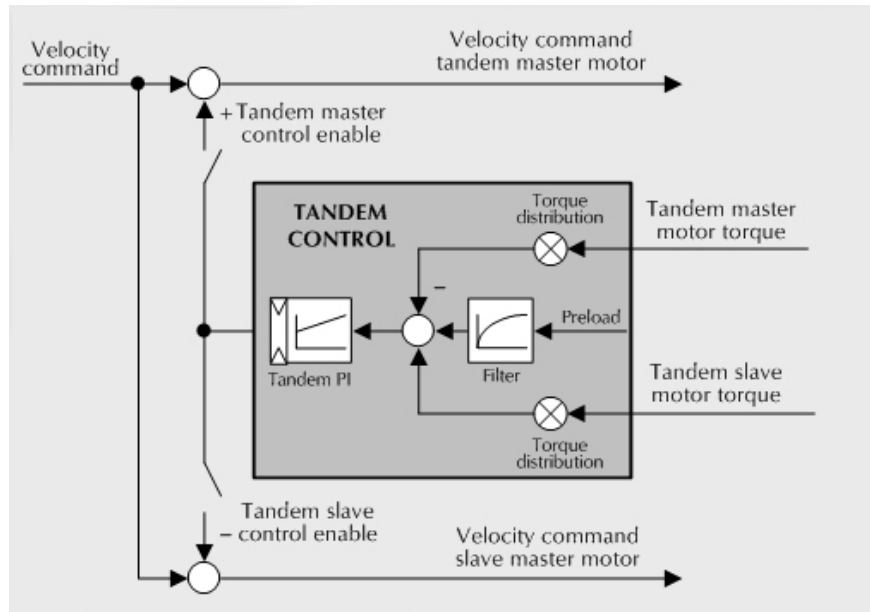
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Tandem axis control.

The block diagram showing the application of the tandem axis control is the following. The meaning of the nomenclature being used is:



Torque of the master motor of the tandem.

It is the percentage of the rated torque shown by the sercos variable TV2 of the drive that governs the master axis of the tandem. It is read through the sercos fast channel in each loop.

Torque of the slave motor of the tandem.

It is the percentage of the rated torque shown by the sercos variable TV2 of the drive that governs the slave axis of the tandem. It is read through the sercos fast channel in each loop.

Torque distribution.

Normalizing gain of the torque generated by the motors in order to distribute the torque in a ratio other than 1:1.

Preload.

Previous torque applied to both tandem axes in opposite direction. This sets a traction between them in order to eliminate the rack-and-pinion backlash when it is in rest position. It is determined as the torque difference supplied by each axis.



Applying the preload necessarily implies mechanically joining the master and slave axes that make up the tandem axis. Otherwise, the motors will move even without the control velocity command.

Preload filter.

First-order filter to prevent torque steps from coming in when configuring the preload.

PI of the tandem.

PI for making each motor provide its corresponding torque. It increases its velocity command if the torque being supplied is too low and it decreases it if the torque being supplied is too high.

When defining a tandem axis, at each loop, the CNC reads via Sercos the torque supplied by the master and the slave axes. This may result in a smaller number of read and write variables available in the Sercos fast channel.

(REF: 1709)

11.

CONFIGURING AN AXIS.

Configure two axes as a tandem axis.



FAGOR AUTOMATION

CNC 8070

11.2.4 Tandem related variables.

Variables related to general machine parameters.

These variables are read-only synchronous and are evaluated during block preparation. They have generic names. Replace the letter "nb" with numbers, keeping the brackets.

Variable	PRG	PLC	INT
(V.)MPG.TMASTERAXIS[nb] Tandem [nb]. Logic number of the master axis	R	R	R
(V.)MPG.TSLAVEAXIS[nb] Tandem [nb]. Logic number of the slave axis	R	R	R
(V.)MPG.TORQDIST[nb] Tandem [nb]. Torque distribution	R	R	R
(V.)MPG.PRELOAD[nb] Tandem [nb]. Preload	R	R	R
(V.)MPG.PRELFITI[nb] Tandem [nb]. Time to apply the preload	R	R	R
(V.)MPG.TPROGAIN[nb] Tandem [nb]. Proportional gain	R	R	R
(V.)MPG.TINTIME[nb] Tandem [nb]. Integral gain	R	R	R
(V.)MPG.TCOMPLIM[nb] Tandem [nb]. Compensation limit	R	R	R

Tandem related variables.

These variables are read/write (R/W) synchronous and are evaluated during execution. They correspond to linear and rotary axes and to the spindle. The mnemonics of the variables have generic names.

- Replace "Xn" with the name or logic number of the axis.
- Replace the "ch" character with the channel number, maintaining the brackets.

	PRG	PLC	INT
(V.)[ch].A.TPIIN.xn Input of the PI of the master axis of the tandem (in rpm)	R	R	R
(V.)[ch].A.TPIOUT.xn Output of the PI of the master axis of the tandem (in rpm)	R	R	R
(V.)[ch].A.TFILTOUT.xn Output of the pre-load filter	R	R	R
(V.)[ch].A.PRELOAD.xn Preload	R/W	R/W	R/W
(V.)[ch].A.FTEO.xn Velocity command for Sercos	R	R	R
(V.)[ch].A.TORQUE.xn Current torque in Sercos	R	R	R

(V.)A.TPIOUT.[Xn] (V.)A.TPIIN.[Xn]

The axis must be a valid tandem master, otherwise, it returns a value of zero.

(V.)A.TORQUE

The PLC reading of this variable comes in tenths (x10).

11.

CONFIGURING AN AXIS.

Configure two axes as a tandem axis.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

11.2.5 Tandem adjustment procedure.

This procedure must bear in mind the type of machine. In general, the resonance frequency of a tandem machine is low; therefore, the CNC must generate position commands without frequency components higher than the resonance frequency.

It is recommended to begin the process with low jerk values (lower than 10 m/sec³) and low Kv. They can always be increased in a later readjustment.

11.

CONFIGURING AN AXIS.

Configure two axes as a tandem axis.

Steps for the adjustment.

- 1 Move both axes independently.

The first step is to ensure the perfect operation of both the master and the slave axes separately. Also verify that both axes move in the same direction with similar dynamics.

- 2 Move one of the axes at a slow and constant speed.

- Do not make jerky movements because the second motor is being dragged by the first one. In this situation, any acceleration or deceleration forces it to go from one side of the backlash to the other thus making it jerky.
- Verify that the turning direction of both motors are coherent once the movement has been carried out.



Observe that reversing the turning direction of a motor reverses the direction of the torque and, consequently, it will be necessary to change the direction of its values monitored with drive parameters SP43 and TP85.

- Verify that the gear ratio in both motors is the same (same feed for same turning speed).
- Carry out a basic adjustment of the velocity loop so the machine can move. It will be readjusted later with both motors together.
- Do not set the friction parameters (there is already enough torque to move the machine).

- 3 Repeat the procedure with the second axis.

When adjusting the loops, use the same parameters if the motors are identical and the torque distribution is 50%. If the motors are different, the axes must be adjusted so their dynamic response is the same or very similar. When using AC-forward ("ACFGAIN" = YES), remember that each motor has half the inertia for a 50% torque distribution.

- 4 Enable the tandem with both motors.

First, disable the PI of the tandem, apply power and verify that the system is at rest.

Then, enter a low proportional value and eliminate the integral value of the PI of the tandem. Without preload, verify that the machine moves and that each motor supplies its corresponding torque according to the parameter "TORQDIST" (e.g. half the torque for 50% distribution).

To define the pre-load, monitor the torque of each motor (Sercos variable TV2). While stopped, increase the preload gradually until the motors supply torque in opposite directions.

Having enabled the tandem, move slowly in both directions and verify that it works properly. Make sure that it is not jerky and that each motor supplies its corresponding torque according to parameters "TORQDIST" and "PRELOAD".

Finally, readjust the velocity loop in both motors with the method used normally.



While changing the parameters of the velocity loop, the best thing would be to change them on both drives at the same time. But, since this is not possible, it is recommended to make small changes in the values or make them while the motor is stopped.

11.3 Analog axes.

11.3.1 Configure the number of the analog output and of the feedback input.

Both the feedback input and the velocity command output are configured by machine parameters. Each axis has the following machine parameters to configure it.

Parameter.	Meaning.
ANAOUTID	Number of the analog output associated with the axis.
COUNTERTYPE	Type of feedback input for the axis.
COUNTERID	Number of the feedback input for the axis.

Configure the analog output of the axis.

The velocity command for an analog axis may be taken from an analog output of the remote modules or of a Sercos drive. It is selected with parameter ANAOUTID. In either case, this parameter indicates the analog output used for the command.

Managing the analog command from CANfagor remote modules.

Parameter.	Value.
ANAOUTID	1 - 16

In this case, this parameter indicates the number of the analog output used for the command. The analog output modules are numbered following the logic order of the remote groups (rotary switch of the power supply). If there are several analog-output modules in each group, the order is from top to bottom and from left to right. The analog outputs of the first module will be 1 through 4, those of the second module 5 through 8 and so on.

Managing the analog command from Sercos drives.

Parameter.	Value.
ANAOUTID	101 -132 201 - 232

In this case, this parameter indicates the number of the analog output and the drive to be used. The parameter is set with a three-digit number; the first digit indicates the number of the analog output to be used (1 or 2) and the other two digits indicate the logic address of the drive (1 through 32). The "address" rotary selector switch sets the logic address.

For example, if the parameter is with a value of ·107·, it indicates that the CNC uses analog input ·1· of the drive whose logic address is ·7·.

Configure the feedback input for the axis.

The feedback inputs for an analog axis may be located in remote modules, in a Sercos drive or locally in the central unit itself. It is selected with parameter COUNTERTYPE.

Once it has been determined where the feedback input is, Parameter COUNTERID must be set indicating which feedback input will be used (either local or remote) or the drive number.

When using the feedback input of a Sercos drive, home search is not possible; therefore, the axis must have absolute feedback.



When using the feedback input of a Sercos drive, set drive parameter PP5=-0.0001.

11.

Feedback input in CANfagor remote modules.

Parameter.	Value.
COUNTERTYPE	Remote.
COUNTERID	Number of the feedback input.

A counter module is used as feedback input in the remote modules. Parameter COUNTERID indicates the number of the feedback input to be used.

The counter modules are numbered following the logic order of the remote groups (rotary switch of the power supply). If there are several Counter modules in each group, the order is from top to bottom and from left to right. The counters of the first module will be 1 through 4, those of the second module 5 through 8 and so on.

Feedback input in Sercos drives.

Parameter.	Value.
COUNTERTYPE	Drive.
COUNTERID	Number of the feedback input.

In this case, the second feedback input of the drive will always be used. Parameter COUNTERID indicates the logic address of the drive (from 1 to 32). The "address" rotary selector switch sets the logic address.

Local feedback input.

Parameter.	Value.
COUNTERTYPE	Local.
COUNTERID	Number of the feedback input.

The central unit has two feedback inputs. Parameter COUNTERID indicates the feedback input being used.

11.

CONFIGURING AN AXIS.
Analog axes.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

11.3.2 Configure 2 axes with the same feedback input and analog output.

There are machines having 2 axes that are never in the same configuration at the same time, because one of them is always parked; for example when changing spindles with an internal motor used to move its articulation.

In these cases, both motors are controlled by the same servo drive and there is a single feedback input and a single analog output for the CNC. This way, although the motors are identical, the axes may be totally different, compensations, dynamics, etc.

Configuring the axes.

To use the same feedback input, parameters COUNTERTYPE and COUNTERID of both axes must be the same. To use the same analog output, parameter ANAOUTID of both axes must be identical.

Besides, the motors must necessarily be identical because the following machine parameters must have the same values for both axes.

Parameter	Meaning
ABSFEEDBACK	Type of feedback.
SINMAGNI	Multiplying factor for the sinusoidal signal.
I0TYPE	Reference mark (I0) type.
REFPULSE	Flank of the reference signal (I0).
ABSOFF	Offset of the feedback device with respect to I0.
EXTMULT	External multiplying factor.
I0CODDI1	Pitch between 2 fixed I0 signals
I0CODDI2	Pitch between 2 variable I0 signals

The PLC routine must include the detection of the active axis and of the parked axis.

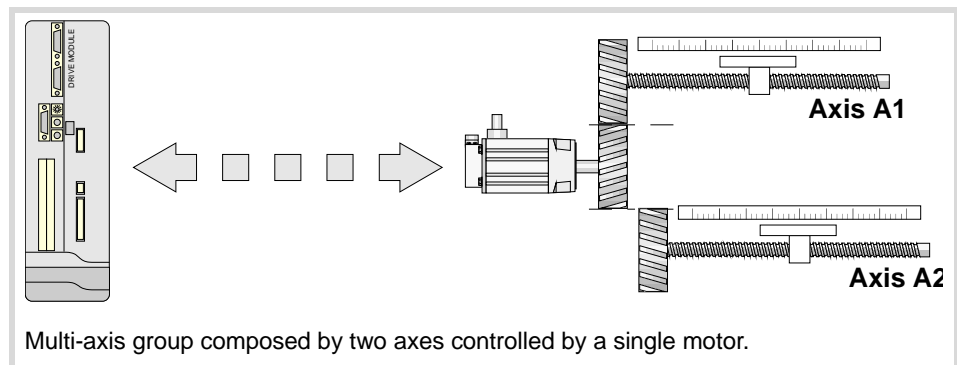
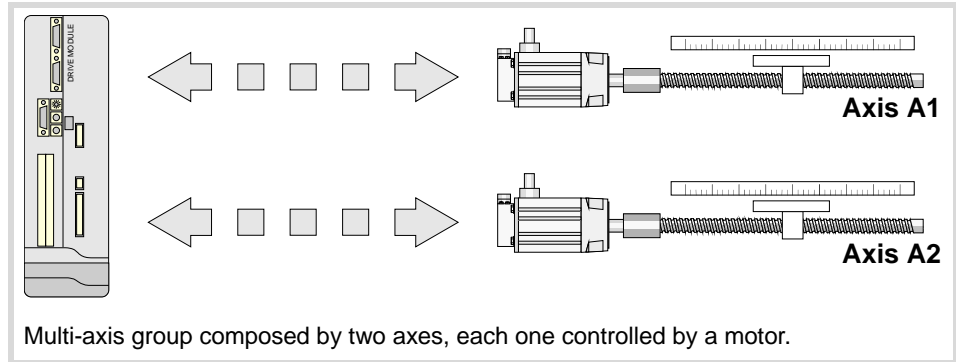
11.

CONFIGURING AN AXIS.

Analog axes.

11.4 Multi-axis management.

A multi-axis group consists of several Sercos axes or spindles controlled by a single drive, although not simultaneously. The axes and spindles may have different dynamics and may be controlled either by the same motor or by different motors.



The internal (motor) feedback of both axes will be located at the drive whereas the external (direct) feedback of each axis may be located at the remote modules, at a Sercos drive or locally in the central unit itself. When using a single drive, it is possible to use the free feedback input of that drive or of another one. Each axis can have its own feedback system or both axes can share the same feedback system.

Requirements and limitations of a multi-axis group.

The axes associated with the same drive must meet the following requirements.

- Any Sercos axis of the system may be part of a multi-axis group, except those that belong to gantry or tandem axes.
- Since each axis and spindle defined at the CNC can have several parameter sets, the sum of sets of all the axes and/or spindles included in a group must not exceed the 8 sets available at the drive. For example, a group may consist of 8 axes with 1 parameter set each, a spindle with 4 sets plus 2 axes with 2 sets each, etc.
- Since the axes are controlled by the same drive, all the axes of a group must have the same Sercos address (parameter DRIVEID).
- All the axes of a group must work in the same Sercos operating mode, position or speed, (parameter OPMODEP). If the external feedback (direct feedback) of the axis is connected to a drive, the axis may be operated in either position-Sercos mode or velocity-Sercos mode (velocity-Sercos mode is recommended). If the external feedback (direct feedback) of the axis is connected to local or remote feedback input, the axis can only work in velocity-Sercos mode.
- If the external feedback is local or remote, the CNC homes the axis as if it were analog; the drive does not know that the axis has been homed and does not update its position value, it only receives the velocity command. If the external feedback is connected to another drive, it will not be possible to home the axis.
- The axis being governed by the drive is selected via PLC (mark SWITCH(axis)). The PLC must either park the axes not governed by the drive or set them as DRO axes.
- The CNC keeps monitoring (following error, tendency test, etc.) all the axes sharing the drive even if the axis is not selected.

11.

CONFIGURING AN AXIS.
Multi-axis management.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Behavior of the CNC and the drive on power-up and when switching axes.

Behavior of the CNC and the drive on power-up.

On power-up, the drive controls the master axis of the group, the first axis defined in the group (parameter MULAXISNAME_1). The CNC assumes default set of the axis (parameter DEFAULTSET). The PLC program must define the set and the gear assumed by the drive.

Behavior of the CNC and the drive when switching axes.

When the PLC switches an axis and/or spindle at the drive, the PLC routine must define the set and the gear assumed by the drive. The CNC assumes default set (parameter DEFAULTSET).

Parameter matching between the CNC and the Sercos drive

In a multi-axis group, the CNC matches the drive with the general parameters and the parameters of the first set of the master axis. During parameter matching, the CNC only uses the parameters of the first set of the drive, the rest of the sets of the drive must be set via DDSSetup or at the drive itself. The CNC uses the following criterion for parameter matching.

- The CNC sends the general parameters of the master axis to the general parameters of the drive.
- The CNC sends the parameters of the default set of the master axis (set defined in parameter DEFAULTSET) to the general parameters or of the first set of the drive accordingly.
- The CNC sends the gear parameters of the master axis to the gear parameters of the drive.

The gears that correspond to the slave axes must be set at the drive itself.

11.

CONFIGURING AN AXIS.
Multi-axis management.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

11.4.1 Configuration of a multi-axis group. Machine parameters.

Configuring a multi-axis group.

Multi-axis groups are configured via machine parameters. Each multi-axis group has the following general machine parameters to configure it.

Parameter.	Meaning.
MULTIAXIS	Table of multi-axis groups.
MULNGROUP	Number of multi-axis groups in the system.
GROUP n	Multi-axis groups.
MULNAXIS	Number of axes and/or spindles making up the multi-axis group.
MULAXISNAME n	Name of the axes and/or spindles of the group.

Defining the multi-axis groups of the system.

Parameter MULTIAXIS shows the multi-axis group definition table for defining the number of groups of the system and the composition of each group.

Parameter MULNGROUP sets the number of multi-axis groups of the system. The table shows a GROUP_n parameter for each multi-axis group.

Defining the axes or spindles that make up the multi-axis group.

Parameter MULNAXIS sets the number of axes and/or spindles making up the multi-axis group. Since each axis or spindle of the CNC can have several parameter sets, the number of axes of the group is limited by the 8 parameter sets available at the drive. The parameter sets of the drive are distributed among all the axes and/or spindles that make up the multi-axis group, so a group may be formed by 8 axes with 1 parameter set for each, 1 spindle with 4 sets plus 2 axes with 2 sets each, etc.

The table shows a MUMLAXISNAME_n parameter for each axis or spindle that belong to the multi-axis group. Any Sercos axis of the system may be part of a group, except those that belong to gantry or tandem axes. The first axis of the group will be the master axis, the one the drive will use by default; the order of the rest of the axes is irrelevant. On system power-up, the CNC will use the parameters of the master axis to run the parameter matching with the drive.

Setting the drive to be used.

Parameter.	Meaning.
DRIVESET	Set of the drive to be used.

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). The CNC sends the changes of Kv, feed-forward (parameter FFGAIN) and AC-forward (parameter ACFGAIN) to the set indicated in this parameter.

Being parameter matching active, the CNC sends the parameters of set ·1· of the master axis to the set ·0· and to the gear ·1· (first) of the drive. For the rest of the sets of the master axis, the CNC only sends the gear data to the drive; set ·2· of the CNC to the gear 2 of the drive and so on. At the master axis of multi-axis group, in order for parameter matching to work properly, this parameter must coincide with the set number where it is; i.e. DRIVESET=1 in set ·1·, DRIVESET=2 in ·2· and so on. At the slave axis of a multi-axis group, this parameter indicates the gear to be used at the drive. If the axis belongs to a multi-axis group, the parameter does not admit a ·0· (zero) value.

11.

CONFIGURING AN AXIS.
 Multi-axis management.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Configuring the external feedback (direct feedback).

Parameter.	Meaning.
COUNTERTYPE	Type of feedback input for the axis; drive, local or remote.
COUNTERID	Number of the feedback input for the axis.

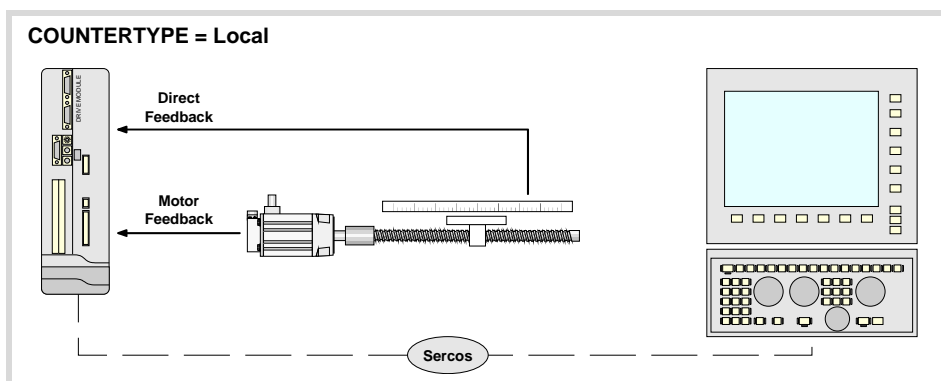
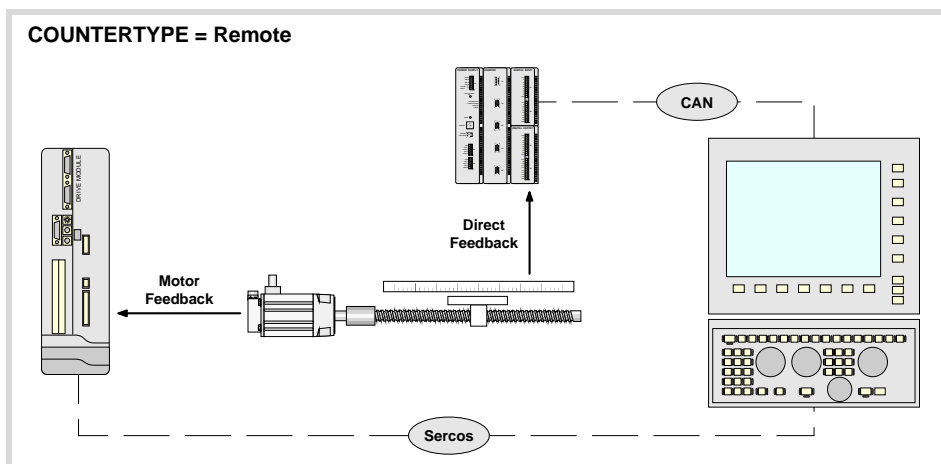
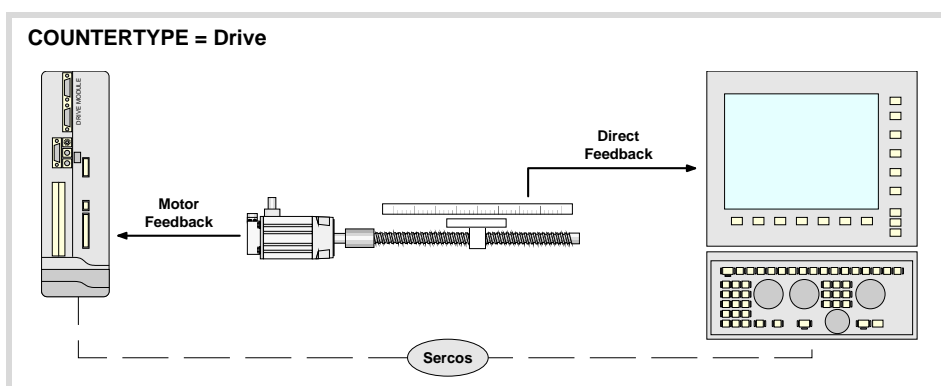
Type of feedback input for the axis.

The internal feedback (motor) of both axes is located at the drive. Parameter COUNTERTYPE indicates the location of the external (direct) feedback of the axis; at the remote modules (4 inputs per module, differential TTL, 1 Vpp sinusoidal), at the free input of a Sercos drive or at the local feedback inputs of the central unit.

When using a Sercos drive, the axis can use the second free feedback input of that drive or of another one. When the axis uses the second feedback input of another drive, the CNC does not allow homing the axis; therefore, the axis must have absolute feedback.



When the internal (motor) feedback and the external (direct) feedback of an axis are at different drives, set parameter PP5=-0,0001 at the drive that receives the external feedback.



Number of the feedback input for the axis.

Depending on the type of feedback, this parameter indicates which feedback input will be used (either local or remote) or the drive number. When two axes will not be in the configuration at the same time, the CNC allows using the same feedback input for both axes.

Type of feedback.	Number of the feedback input for the axis.
Feedback input in CANfagor remote modules.	A counter module is used as feedback input in the remote modules. This parameter indicates the number of the feedback input being used. The counter modules are numbered following the logic order of the remote groups (rotary switch of the power supply). If there are several Counter modules in each group, the order is from top to bottom and from left to right. The counters of the first module will be 1 through 4, those of the second module 5 through 8 and so on.
Feedback input in Sercos drives.	In this case, the second feedback input of the drive will always be used. The parameter indicates the logic address of the drive (from 1 to 32). The "address" rotary selector switch sets the logic address.
Local feedback input.	The central unit has two feedback inputs. This parameter indicates the feedback input being used.

11.**CONFIGURING AN AXIS.**
Multi-axis management.

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

11.4.2 Configuration of a multi-axis group. The PLC routine generates an error.

Multi-axis groups are managed from the PLC using one of its marks.

PLC signal.	Meaning.
SWITCH(axis)	The PLC activates this mark to let the CNC know which axis or spindle that it must govern.

The PLC routine must define the set and the gear assumed by the drive on power-up, toggle between the various axes or spindle of the group and define the set and gear assumed by the drive for each change of axis or spindle. The axis or spindle changing process must consider the following aspects (assuming that the X axis is active and the X1 axis is parked)

- 1 Park the X axis.
- 2 Verify that the X axis is parked.
- 3 Switch to the X1 axis (SWITCHX1=1).
- 4 Otherwise, write the parameters at the drive. Verify that the parameters have been written at the drive and, if necessary, validate the parameters of the drive
- 5 Change the set and/or gear at the drive.
- 6 Verify that the set and/or gear has been changed (mark SERPLCAC) and that the set and/gear of the drive is correct (parameters GV21 and GV25 of the drive).
- 7 Enable the X1 axis.

11.

CONFIGURING AN AXIS.
Multi-axis management.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

11.4.3 Changing the set and the gear at the CNC and at the drive.

A set is a bunch of parameters that may be activated and gear is a bunch of parameters specifically related to gear ratios. The parameters that define the gear are included in the set, one gear per set. A set change activates all its parameters, including those of the gear, whereas a gear change only activates the specific gear-related parameters.

Axes and spindles included in a multi-axis group.

Function.	CNC	Drive.
G112 (axes only)	The CNC activates the programmed set.	The drive activates the gear defined in parameter DRIVESET of the program set.
M41-M44 (spindles only)	The CNC activates the programmed set.	The drive activates the gear defined in parameter DRIVESET of the program set.
(V.)A.SETGE.xn	Not used.	The drive activates the programmed set and/or gear.

If the axis or spindle does not belong to a multi-axis group, functions G112 and M41 to M44 act in the same way at the CNC; but at the drive they activate the programmed gear, not the one defined in parameter DRIVESET.

Spindle and C axis.

Function.	CNC	Drive.
#CAX ON	The CNC activates the set defined in parameter CAXSET.	The drive activates the gear defined in parameter CAXSET.
#CAX OFF	The CNC restores the set that was active before enabling the spindle as C axis.	The drive restores the gear that was active before enabling the spindle as C axis.

11.

CONFIGURING AN AXIS.
Multi-axis management.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

11.4.4 Configuration examples.

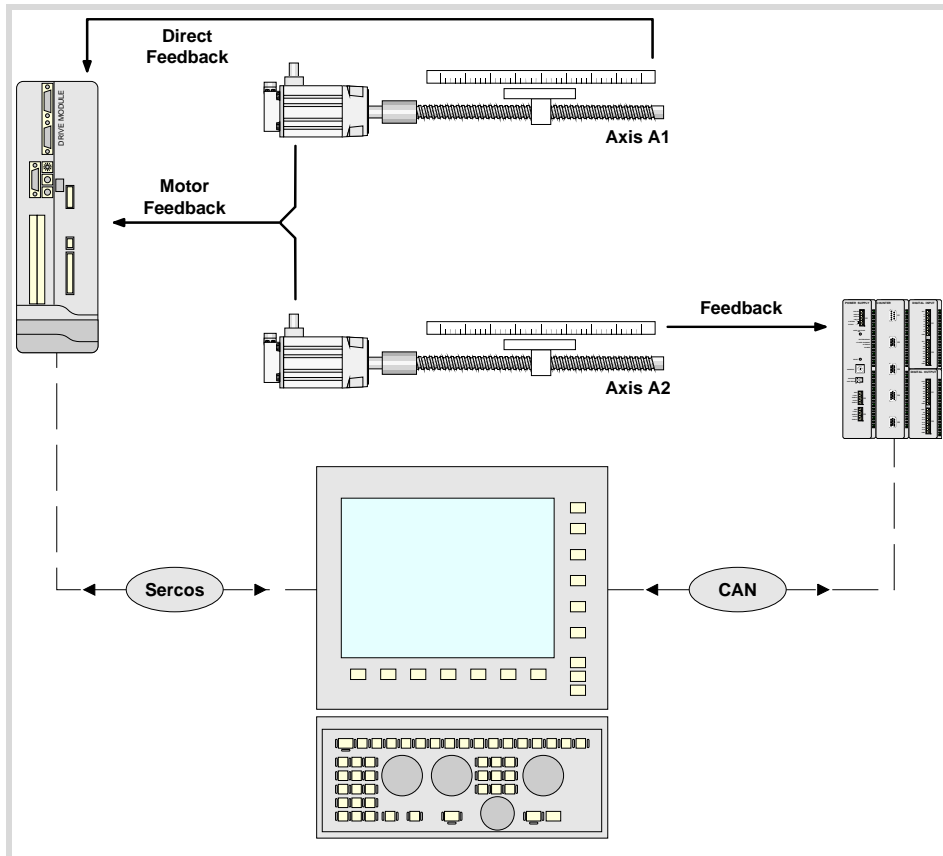
Example 1. 2-axis system, each with its own motor.

The system has the following characteristics.

- A single drive controls both axes, each with its own motor.
- Each axis uses a different type of feedback.

A1 axis The external (direct) feedback is connected to the drive itself.

A2 axis The external (direct) feedback is connected to the remote modules.



Configuration of the multi-axis group.

MULTIAXIS	GROUP_1
MULNGROUP = 1	MULNAXIS = 2 MULAXISNAME_1 = A1 MULAXISNAME_2 = A2

Configuring the axes.

	DRIVEID	OPMODEP	FBACKSRC	COUNTERTYPE	COUNTERID
A1 axis	1	Speed	External	Drive	1
A2 axis	1	Speed	External	Remote	1

Example 2. System with two axes and a single motor.

The system has the following characteristics.

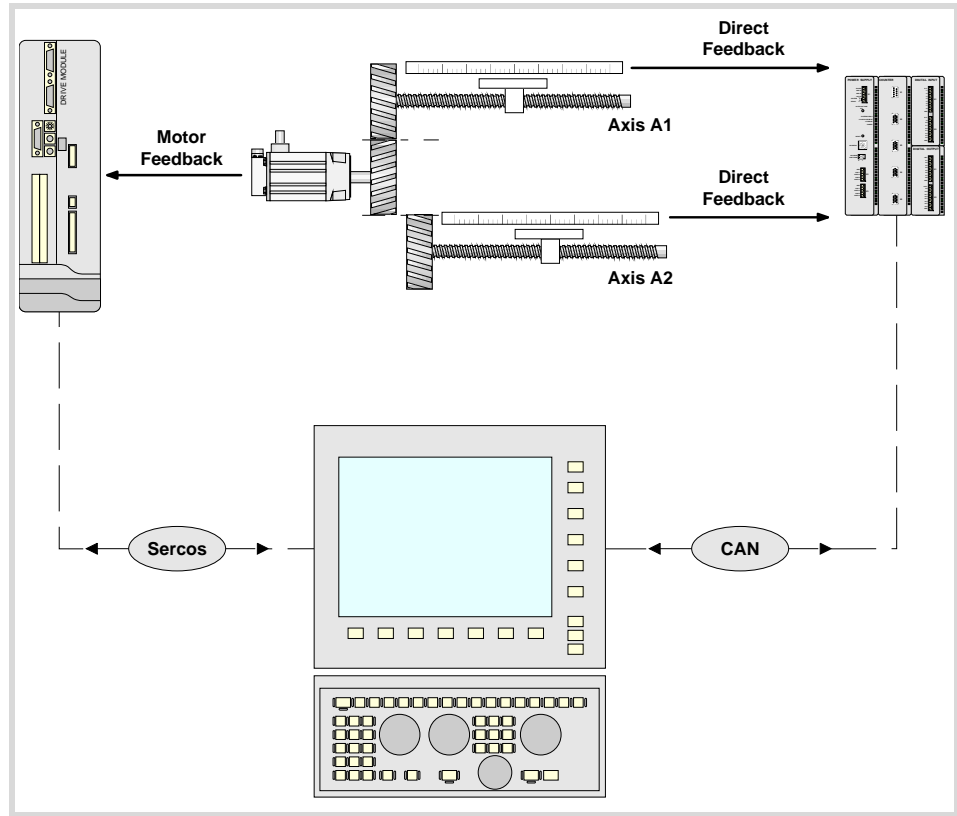
- A single drive controls both axes with the same motor.
- Each axis uses a different type of feedback.

A1 axis The external (direct) feedback is connected to the remote modules.

A2 axis The external (direct) feedback is connected to the remote modules.

11.

CONFIGURING AN AXIS.
 Multi-axis management.



Configuration of the multi-axis group.

MULTIAXIS	GROUP_1
MULNGROUP = 1	MULNAXIS = 2 MULAXISNAME_1 = A1 MULAXISNAME_2 = A2

Configuring the axes.

	DRIVEID	OPMODEP	FBACKSRC	COUNTERTYPE	COUNTERID
A1 axis	1	Speed	External	Remote	1
A2 axis	1	Speed	External	Remote	2

11.5 Home search.

What is home search?.

Home search 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). Home search may be done in three ways.

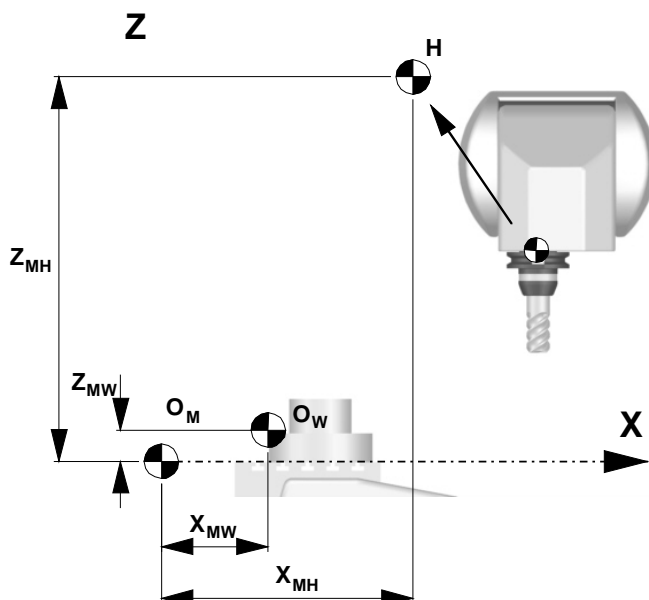
- Manual home search, in JOG mode. The axes are homed one by one. The CNC does not keep the part zero and the coordinates are displayed referred to machine reference zero.
- Automatic home search, in JOG mode. This kind is only available when the home search subroutine associated with function G74 (parameter REFPSUB) has been defined. All the axes are homed at the same time. The zero offsets are not canceled. The position values are displayed in the active reference system.
- Home search by program or MDI using function G74. The zero offsets are not canceled. The position values are displayed in the active reference system.

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

Machine reference system and machine reference point.

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.

- The machine reference zero is the origin point of the machine reference system, set by the machine manufacturer.
- The machine reference point is a point set by the manufacturer and referred to the machine reference zero. This point may be located anywhere on the machine. The position of the reference point, for each axis, is set at parameter REFVALUE.



OM	Machine zero
OW	Part zero
H	Machine reference point
X _{MW} , Y _{MW} , Z _{MW} ,	Coordinates of the part zero
X _{MH} , Y _{MH} , Z _{MH} ,	Coordinates of the reference point.

11.

CONFIGURING AN AXIS.
Home search.

FAGOR 

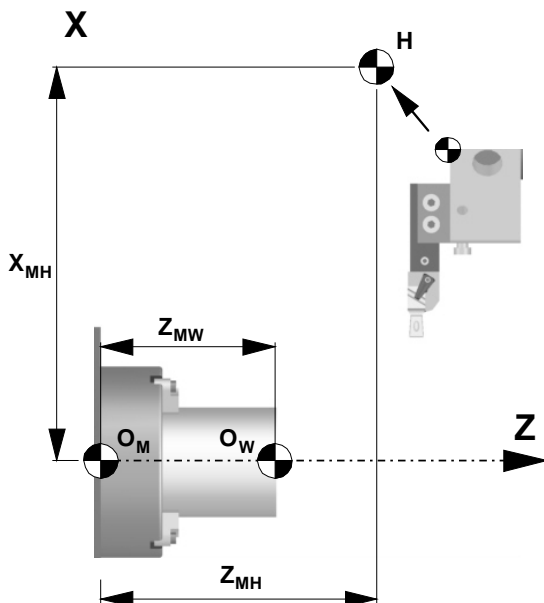
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

11.

CONFIGURING AN AXIS. Home search.



11.5.1 Home search (axes and spindles).

Home search with a feedback system that does not have distance-coded reference marks.

Parameter.	Meaning.
I0TYPE	Reference mark (I0) type.
DECINPUT	The axis has a home switch.
REFDIREC	Home searching direction.
REFFEED1	Fast home searching feedrate.
REFFEED2	Slow home searching feedrate.

When the feedback system does not have distance-coded reference marks, the axis must always be homed at a specific point of the machine which is referred to as Machine Reference Point. The installer must place a home switch for each axis at the machine reference point.

When starting the search, the axis moves in the direction set by parameter REFDIREC and at the feedrate indicated by parameter REFFFEED1 until reaching the home switch. When the axis presses the home switch, it starts moving in the opposite direction at the feedrate set by parameter REFFFEED2. It keeps on moving after releasing the home switch until the CNC detects a marker pulse (I0) from the feedback device.

Home search with a feedback system that has distance-coded reference marks.

Parameter.	Value.
I0TYPE	Increasing distance-coded Decreasing distance-coded
REFDIREC	Home searching direction.
REFFEED2	Slow home searching feedrate.

When the feedback system has distance-coded reference marks (coded I0) may be referenced (homed) at any point of the machine; there is no need for home switches. However, it will be necessary to set the machine reference point if the axis uses leadscrew error compensation because the amount of error at the machine reference point must be -0-.

The axis moves the minimum distance possible, less than 200 mm, in the direction set by parameter REFDIREC and at the feedrate indicated in "REFFEED2" until the CNC receives a reference marker pulse from the feedback system.. If before receiving the reference signal, the axis presses the home switch, it starts moving in the opposite direction and goes on with the process getting away from the switch until the CNC receives the reference marker pulse of the feedback system.

Redefine the machine reference point after having dismantled or replaced the feedback system.

Parameter.	Value.
REFVALUE	Position of the reference point.
REFSHIFT	Offset of the reference point.

Sometimes, to readjust the machine, it is necessary to take down the feedback device, thus when putting back up, the new home point might no coincide with the previous one.

Since the home point must still be the same, the difference between the new point and the old point must be assigned to parameter REFSHIFT . This way, when the axis finds the I0, it moves the distance indicated in REFSHIFT and at that point updates its coordinate to the value of REFVALUE..

11.

CONFIGURING AN AXIS.

Home search.



FAGOR AUTOMATION

CNC 8070

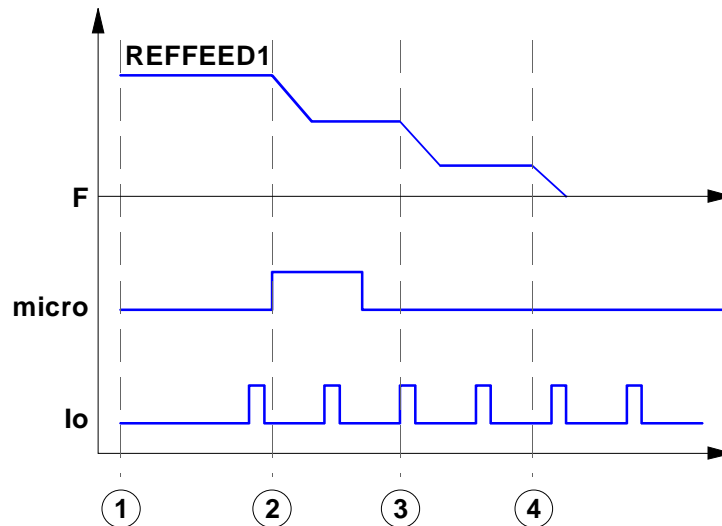
(REF: 1709)

When the axes are position-Sercos; during parameter matching, the CNC sends the value of parameter REFSHIFT to the drive so it takes it into account; this way the CNC coordinate and that of the drive will be the same.

Peculiarities of home search for Mechatrolink axes.

Parameter.	Value.
REFFEED1	Fast home searching feedrate.

The servo carries out the home search and the home switch must be wired to the servo. The homing procedure follows this sequence. Refer to the documentation of the device for further detail.



- 1 When homing starts, the axis moves in the homing direction (set by the parameters of the servo) at the feedrate set by parameter REFFFEED1.
- 2 When the axis presses the home switch, it switches to approach feedrate 1 (set by the parameters of the servo).
- 3 When the axis releases the home switch and receives the home signal, it switches to approach feedrate 2 (set by the parameters of the servo).
- 4 Once the axis has traveled the final positioning distance (set by the parameters of the servo), the servo concludes the home search.

Peculiarities of home search for spindles, tandem system and analog axes.

Peculiarities of home search for spindles.

Parameter.	Value.
REFINI	Home search in the first movement.
NPULSES	Number of encoder pulses.
NPULSES2	Number of pulses of the encoder (external feedback).

Parameter REFINI determines whether the CNC homes the spindle in its first movement or not. The CNC takes this parameter into account only when parameters NPULSES y NPULSES2 have been set with a value other than 0.

If the spindle has been homed, the spindle orientation in M19 does not force a new home search.

11.

CONFIGURING AN AXIS.
Home search.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

The CNC will home the spindle again in the following situations.

- When programming a new home search with function G74, via program or MDI/MDA mode.
- When the spindle exceeds the feedback pulses reading limit.
- After a failure at the Sercos ring.
- When replacing the encoder.

Peculiarities of home search for a tandem system.

In a tandem system, only the master axis is homed; the home search is transparent for the slave that just moves along with the master.

Peculiarities of home search for analog axes homed in parallel.

When programming a home search on several analog axes in parallel; i.e. with the same index (for example "G74 X1 Y1 Z1"), the execution sequence is the following.

- 1 All the axes move at the same time, each one at its feedrate REFFFEED1 until each one detects its home switch. The axes wait on the switch until all the axes involved in the home search detect the home switch.
- 2 Once all the axes have reached their switches, the home search begins sequentially on each axis at feedrate REFFFEED2 starting with the last axis reaching its home switch and following their order according to the logic number of the axes.
- 3 If any of the axes of the programmed group does not have a home switch, the axis waits for the rest of the axes to reach their switches and then starts looking for its reference mark at REFFFEED2 feedrate in its corresponding order (sequence).
- 4 If none of the axes of the programmed group has a home switch, the home search starts at REFFFEED2 with the axis having the lowest logic number and when that axis has finished, it goes on sequentially with the rest.

11.

CONFIGURING AN AXIS.
Home search.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

11.5.2 Home search (gantry axes).

Gantry axes may be homed in the same three ways described for the rest of the axes and spindles. The following requirements must be met when setting the parameters for the Gantry axes.

- The type of reference mark (parameter I0TYPE) must be the same for both axes.
- When not using distance-coded reference marks (I0), either both axes or just the master axis may have a home switch (parameter DECINPUT).

Analog axes and velocity-Sercos axes.

Home search with a feedback system that does not have distance-coded reference marks. The master and slave axes have a home switch.

Parameter.	Meaning.
I0TYPE	Reference mark (I0) type. Normal (non-distance-coded) on both axes. To be defined for the master and slave axes.
DECINPUT	The axis has a home switch. To be defined for the master and slave axes.
REFDIREC	Home searching direction. To be defined for the master axis.
REFFEED1	Fast home searching feedrate. To be defined for the master axis.
REFFEED2	Slow home searching feedrate. To be defined for the master axis.
REFVALUE	Position of the reference point. To be defined for the master and slave axes.
REFSHIFT	Offset of the reference point. To be defined for the master and slave axes.

The CNC starts moving both axes in the direction indicated by parameter REF D I R E C of the master axis. This movement is carried out at the feedrate indicated by parameter REF F E E D 1 of the master axis until any of the axis presses its home switch. After pressing the home switch, both axes move at REF F E E D 2 feedrate of the master axis until the axis that pressed the home switch detects its reference mark. After detecting the reference mark, the CNC resets the position value of this axis to the value set by parameter REF V A L U E and starts homing the second axis.

To home the second axis, both axes move at the feedrate indicated by parameter REF F E E D 1 of the master axis until the second axis presses its home switch. After pressing the home switch, both axes move at REF F E E D 2 feedrate of the master axis until the axis that pressed the home switch detects its reference mark. After detecting the reference mark, the CNC resets the position value of this axis to the value set by parameter REF V A L U E.

If the first axis pressing its home switch was the master and its parameter REF S H I F T is other than zero, the slave axis does not start homing until the master axis ends the movement corresponding to parameter REF S H I F T. The REF S H I F T parameter of the slave axis is applied when it resets the position value after having detected the reference mark without making the movement.

Home search with a feedback system that does not have distance-coded reference marks. Only the master axis has a home switch.

The CNC starts moving both axes in the direction indicated by parameter REF D I R E C of the master axis. This movement is carried out at the feedrate indicated by parameter REF F E E D 1 of the master axis until this axis presses its home switch. After pressing its home switch, both axes move at REF F E E D 2 feedrate of the master axis until the slave axis detects its reference mark. After detecting the reference mark, the CNC resets the position value of the slave axis to the value set by parameter REF V A L U E and considering parameter REF S H I F T. To update the position value, the axis does not move.

Then, the master axis searches the reference mark. Once the reference mark has been detected, the CNC resets its position value to the one set by parameter REF V A L U E and

11.

CONFIGURING AN AXIS.
Home search.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

considering parameter REFSHIFT. In this case, the axis will move to apply the REFSHIFT value.

Home search with a feedback system that has distance-coded reference marks or where none of the axes has a home switch.

The CNC starts moving both axes in the direction indicated by parameter REFDIR of the master axis. This movement is carried out at the feedrate indicated by parameter REFFFEED2 of the master axis until the slave axis detects its reference mark. After detecting the reference mark, the CNC resets the position value of this axis to the value set by parameter REFVALUE and considering parameter REFSHIFT if there is one.

Then, the master axis searches the reference mark. After detecting the reference mark, the CNC resets the position value of the master axis to the value set by parameter REFVALUE and considering parameter REFSHIFT if there is one.

Position-Sercos axes

To homing these axes, both must either have distance-coded reference marks or a home switch. If both axes have a home switch but do not have distance-coded reference marks, the home switches must be placed in such way that the master axis is not pressing its home switch while homing the slave axis (first to one to search its reference mark).

The CNC sends to the drive the command to home the slave axis. In the process, the CNC reads the increment in the real (actual) slave position value and sends it to the master axis as an increment of the theoretical position value, hence making the master axis follow the slave axis only by the lag between the theoretical and real position values.

When the slave axis has been homed, the CNC sends to the drive the command to home the master axis. In this case, the slave axis follows the master axis.

In order to prevent the axis following the one being homed from jerking due to the change in position value, the CNC makes sure that the increment of the real position value corresponds at the most with the homing feedrate (parameter REFFFEED1) and that it does not exceed the maximum acceleration allowed for that axis.

When both axes have been homed, the CNC takes into account the difference between the position values to correct it, if so permitted by parameter DIFFCOMP.

Coordinate (position) difference compensation after a home search.

Parameter.	Meaning.
DIFFCOMP	Compensate for the coordinate (position) difference between the two axes after G74.

The CNC corrects the position difference between the master and the slave axes after they are homed. The position value correction (compensation) may be applied to any type of axis; analog, position-Sercos and velocity-Sercos.

Position value compensation is enabled by parameter DIFFCOMP and it is applied with marks DIFFCOMP(axis) and SERVO*ON(axis) as follows.

- With the up-flank of SERVO*ON if DIFFCOMP is active.
- With the up-flank of DIFFCOMP if SERVO*ON is active.

To compensate the position value, the slave axis will move until reaching the position of the master axis at the feedrate set by parameter REFFFEED2. This process can only be interrupted with RESET. The REFPOIN(axis) mark of both axes is activated when the position value compensation is completed.

11.

CONFIGURING AN AXIS.
Home search.



FAGOR AUTOMATION

CNC 8070

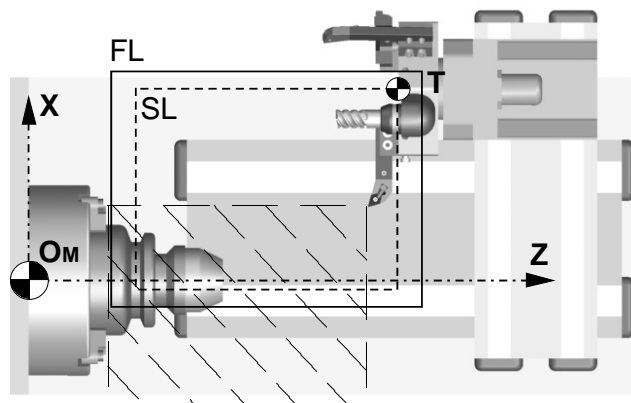
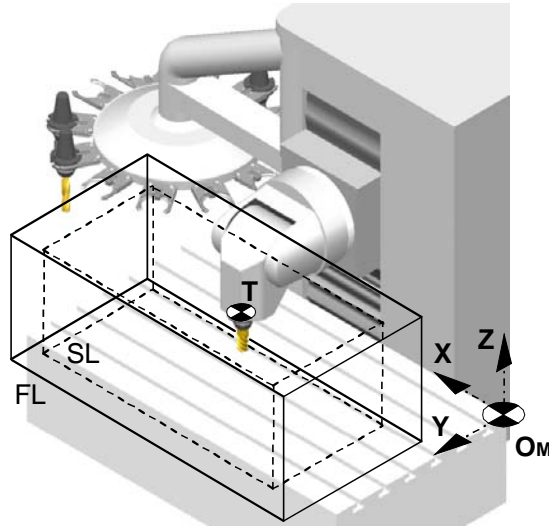
(REF: 1709)


11.6 Software limits of the axes.

The software limits set the travel limits for the axes to prevent the carriages from reaching the mechanical hard stops. The carriages reach the hard stops when the reference point of the tool holder is placed at the physical limits. The CNC lets set the software limits on linear axes and on linearlike rotary axes.

11.

CONFIGURING AN AXIS.
Software limits of the axes.



OM	Machine zero.
T	Reference point of the tool holder.
FL	Physical limits.
SL	Software limits applied by the CNC.
	Programmable positions of the axes (they depend on the active tool).

The programmable positions of the axes depend on the dimensions of each tool. When programming a position where the reference point of the tool holder is beyond the software limits, the CNC interrupts the execution and shows the pertinent error message.



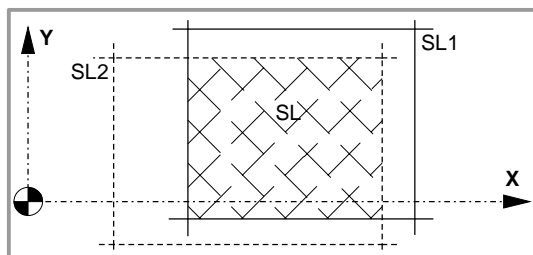
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Software limits applied by the CNC.

The CNC can have two groups of software limits active, where each group may consist of an upper limit and a lower limit for each axis; i.e. each axis may have two upper limits and two lower limits. Of the four possible software limits for each axis, the CNC will apply the most restrictive upper and lower limit even if they do not belong to the same group.



CNC behavior when an axis reaches the software limits.

When an axis reaches the software limits, the CNC interrupts the execution and shows the pertinent error message. To take the axis to the work zone, access the JOG mode and move the axis or spindle that overran the travel limit. The axis or spindle can only move in the direction that places it within the limits.

The CNC offers the following variable to indicate that an axis has reached the software limits.

Variable.	Meaning.
(V.)[ch].G.SOFTLIMIT	Software limits reached.

The PLC activates one of these marks to indicate to the CNC that the corresponding axis or spindle has reached the positive or negative travel limit.

PLC mark.	Meaning.
LIMITPOS(axis)	The axis has reached the upper software travel limit.
LIMITNEG(axis)	The axis has reached the lower software travel limit.

11.

CONFIGURING AN AXIS.
Software limits of the axes.

11.6.1 How to set the software travel limits.

Considerations for setting the software travel limits.

The software travel limits may be positive or negative, but the lower limits must be smaller than the upper ones; otherwise, the axis might not move in any direction.

If when changing the limits, an axis is positioned beyond them, that axis can only be moved in the direction that places it within those limits.

If for an axis, both upper and lower limits of a group are set to -0 , the CNC cancels those limits on that axis and applies the ones of the other group.

The software limits are always applied in radius, regardless of the setting of parameter DIAMPROG, of the active function G151/G152 and of the method chosen to set them.

Default software travel limits (first software travel limits).

The default software travel limits, those assumed by the CNC on power-up, are set by machine parameters; if both parameters are set to -0 , these limits are canceled.

Parameter.	Meaning.
LIMIT+	Upper software travel limits.
LIMIT-	Lower software travel limits.

These software travel limits may be changed by program or MDI using the following functions and variables. When modifying the limits with these functions or variables, the CNC assumes those values as the new limits in this group.

Function.	Variable.	Meaning.
G198	(V.)[ch].A.NEGLIMIT.xn	Lower software travel limits.
G199	(V.)[ch].A.POSLIMIT.xn	Upper software travel limits.

Setting both upper and lower limits of an axis to -0 cancels the limits, regardless of their values in the machine parameters. In this case, the CNC will apply the second software travel limits to the axis.

The new limits will be maintained after executing an M02 or M30, and after an emergency or a reset. On power-up or after validating the axis machine parameters the CNC assumes the software limits set by the machine parameters.

Modifying the software travel limits with functions G198/G199.

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)
G91 G198 X-700
      (New incremental lower limit X=-1500)
```

11.

CONFIGURING AN AXIS.
Software limits of the axes.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Second software travel limits.

These software travel limits are set with the following variables that can be written via part-program, MDI, PLC or interface. These variables are initialized on power-up assuming the maximum value possible. When modifying the limits with these variables, the CNC assumes those values as the new limits in this group.

Variable.	Meaning.
(V.)[ch].A.RTNEGLIMIT.xn	Lower software travel limits (second limits).
(V.)[ch].A.RTPOSLIMIT.xn	Upper software travel limits (second limits).

Setting both upper and lower limits of an axis to ·0· cancels the limits and the CNC applies the first software travel limits to the axis.

11.

CONFIGURING AN AXIS.
Software limits of the axes.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

11.6.2 Set the tolerance for an axis located at the software travel limits.

The tolerance for the software travel limits is set with machine parameter SWLIMITTOL. The CNC applies this tolerance to the active software limits.

Parameter.	Meaning.
SWLIMITTOL	Software limit tolerance.

This tolerance indicates the maximum variation or oscillation allowed to a real coordinate of an axis referred to the software limits, before issuing an error indicating travel limit overrun. The programmed theoretical movement of the axis is only possible up to the exact limit, but the real axis coordinate is allowed this margin before the error is issued. When they are DRO axes, the error is also issued when the real coordinate exceeds the limit over the tolerance.

When no theoretical movement has been programmed, the limit overrun error will only be issued when exceeding the tolerance in a sampling period (cycle time); for example, when hitting the axis causing it to overrun the limits abruptly. In any other case, if no theoretical movement has been programmed for the axis, the error will not be issued even if it overruns the limits.

11.

CONFIGURING AN AXIS.
Software limits of the axes.

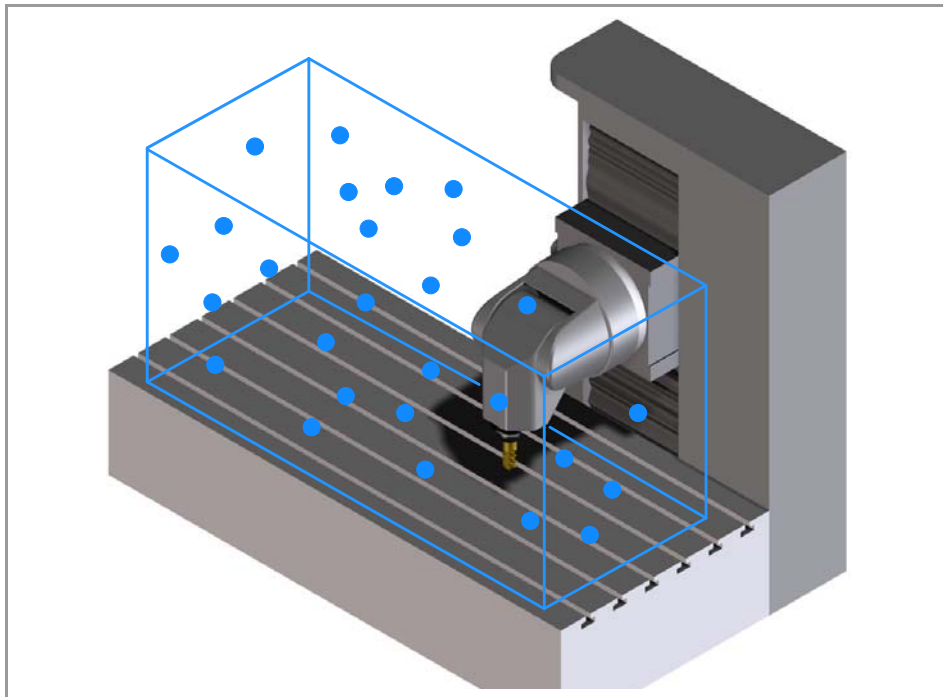


FAGOR AUTOMATION

CNC 8070

(REF: 1709)

One of the causes of the lack of precision in the positioning stems from geometric machine errors that arise during manufacturing and assembly, wear, elastic deformations, etc. Volumetric compensation corrects these geometric errors to a large extent, thus improving the precision of the positioning. The volume to be compensated is defined by a cloud of points, in each of which the error to be corrected is measured. This error is recorded in a file that is then uploaded to the CNC.



The CNC can have three types of volumetric compensation: basic, medium or large. The "Volumetric compensation" software option determines the type of compensation to use. The CNC can have four volumetric compensations configured, and each may be active simultaneously. Volumetric compensations are configured in the machine parameters and are activated from the PLC (marks VOLCOMP1 to VOLCOMP4). The tables that define the compensation are defined in an external file.

Type.	Description.
Basic.	<ul style="list-style-type: none"> • The volume to be compensated is defined by up to 25 points on each axis. • This compensation corrects translation errors. • The compensation tables are generated by the calibration application; they are not editable from the CNC. • The OEM must define all the compensation data (axes that move, axes to be compensated, position and size of the volume to be compensated, etc.) in the machine parameters and select the file containing the compensation data. • The basic compensation is the fastest to calibrate, but it can be less precise than the others (it corrects fewer error components).
Medium. Large.	<ul style="list-style-type: none"> • Volume compensation of up to 10 m³ (medium) or more than 10 m³ (large). • This compensation corrects the 21 geometric error components (translation, rotation and squaring). • The compensation tables are generated by the calibration application; they are not editable from the CNC. • The OEM need only define the axes to be compensated in the machine parameters and select the file containing the calibration data. The rest of the data (axes that move, error to be compensated, etc.) are implicit in the file. • Medium and large compensations are more precise than basic compensation, but they are slower to calibrate.

The volumetric compensation includes cross compensation and ballscrew compensation, and therefore it is not necessary to define the latter two; however, if they are defined, the CNC takes them into account. The CNC applies the cross compensation and ballscrew compensation first, followed by the volumetric compensation.

On the gantry axes, compensation is defined for the master axis and the CNC also applies it to the slave axis.

Basic volumetric compensation.

For basic volumetric compensation, it is necessary to define both axes: the one that moves and the one that is compensated. This makes it possible to define another type of compensation that does not involve the same three axes; for example, a common cross compensation or 2D compensation for the falling ram.

If the CNC has activated either a medium or large volumetric compensation (software option), it can also allow for a basic volumetric compensation to be defined if the file has an csv extension. In this case, the CNC will provide the necessary parameters for its definition.

12.

VOLUMETRIC COMPENSATION.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

12.1 Setup.

12.1.1 General; machine parameters.

VOLCOMP

Volumetric compensations tables.

This parameter shows the volumetric compensation table. Volumetric compensations are configured in the machine parameters and are activated from the PLC (marks VOLCOMP1 to VOLCOMP4). See "*VOLCOMP1 .. VOLCOMP4*" on page 589.

VOLCOMP n

Table of parameters for each volumetric compensation.

This parameter shows the parameter table to define the volumetric compensations of the system. Each table has the following machine parameters to configure it.

VOLCOMP	
VMOVAXIS1 VMOVAXIS2 VMOVAXIS3	In the basic volumetric compensation, the name of the axis that generates variations when moved. No function in the medium and large volumetric compensations.
NPOINTSAX1 NPOINTSAX2 NPOINTSAX3	Number of points of basic volumetric compensation on each axis. No function in the medium and large volumetric compensations.
INIPOSAX1 INIPOSAX2 INIPOSAX3	Initial position of the basic volumetric compensation on each axis. No function in the medium and large volumetric compensations.
INCREAX1 INCREAX2 INCREAX3	Interval between basic volumetric compensation points on each axis. No function in the medium and large volumetric compensations.
VCOMPAXIS1 VCOMPAXIS2 VCOMPAXIS3	Name of the first, second and third axis to be compensated.
VCOMPFILE	File containing volumetric compensation data.

VMOVAXIS1

VMOVAXIS2

VMOVAXIS3

Name of the axis that generates variations when moved.

Possible values: Any axes defined in AXISNAME.

Associated variable: (V.)MPG.VMOVAXIS1[tbl]

Associated variable: (V.)MPG.VMOVAXIS2[tbl]

Associated variable: (V.)MPG.VMOVAXIS3[tbl]

Parameter included in the VOLCOMP table.

Axes that define the volume to be compensated. First, second and third axis that, when moved, generates variations in the axes to be compensated. If it is a gantry axis, it is only necessary to define the master axis; the CNC also applies the compensation to the slave axis.

These parameters are only available for basic volumetric compensation; they have no function in medium and large volumetric compensations. In the medium and large volumetric compensations, the order of the axes is implicit in the file.

12.

**NPOINTSAX1
NPOINTSAX2
NPOINTSAX3****Number of points of the basic volumetric compensation for the MOVAXIS axis.***Possible values: From 2 to 25 points.**Default value: 10 points.**Associated variable: (V.)MPG.NPOINTSAX1[tl]**Associated variable: (V.)MPG.NPOINTSAX2[tl]**Associated variable: (V.)MPG.NPOINTSAX3[tl]**Parameter included in the VOLCOMP table.*

Number of points of basic volumetric compensation on each axis. Each volumetric compensation axis can have 25 points.

These parameters are only available for basic volumetric compensation; they have no function in medium and large volumetric compensations.

**INIPOSAX1
INIPOSAX2
INIPOSAX3****Initial position of the basic volumetric compensation for the MOVAXIS axis.***Possible values: Within ±99999.9999 mm / Within ±3937.00787 inch.**Default value: 0.**Associated variable: (V.)MPG.INIPOSAX1[tl]**Associated variable: (V.)MPG.INIPOSAX2[tl]**Associated variable: (V.)MPG.INIPOSAX3[tl]**Parameter included in the VOLCOMP table.*

Initial position of the basic volumetric compensation on each axis.

These parameters are only available for basic volumetric compensation; they have no function in medium and large volumetric compensations.

**INCREAX1
INCREAX2
INCREAX3****Interval between points for the MOVAXIS axis in basic volumetric compensation.***Possible values: Within ±99999.9999 mm / Within ±3937.00787 inch.**Default value: 10 mm / 0.3937 inch.**Associated variable: (V.)MPG.INCREAX1[tl]**Associated variable: (V.)MPG.INCREAX2[tl]**Associated variable: (V.)MPG.INCREAX3[tl]**Parameter included in the VOLCOMP table.*

Interval between points on each axis.

These parameters are only available for basic volumetric compensation; they have no function in medium and large volumetric compensations.

**VCOMPAXIS1
VCOMPAXIS2
VCOMPAXIS3****Name of the axis to be compensated.***Possible values: Any axes defined in AXISNAME.**Associated variable: (V.)MPG.VCOMPAXIS1[tl]**Associated variable: (V.)MPG.VCOMPAXIS2[tl]**Associated variable: (V.)MPG.VCOMPAXIS3[tl]**Parameter included in the VOLCOMP table.*

These parameters set the axes to be compensated with volumetric compensation. The axes associated with the same compensation may belong to different channels and they may be interchanged from one channel to another while the compensation is active. An axis can be included in several different compensations, but compensations that share axes cannot be active at the same time.

VCOMPFILE

File containing volumetric compensation data.

Associated variable: (V.)MPG.VCOMPFILE[tbl]

Parameter included in the VOLCOMP table.

File containing volumetric compensation data. The compensation tables are generated by the calibration application; they are not editable from the CNC. The units in the file data (millimeters or inches) must be those defined by the CNC (parameter INCHES).

12.1.2 PLC. General modifiable signals.

VOLCOMP1

..

VOLCOMP4

If the PLC activates one of these marks (changing it from 0 to 1), the CNC activates the corresponding volumetric compensation (parameter VOLCOMP). If the PLC deactivates one of these marks (changing it from 1 to 0), the CNC deactivates the corresponding volumetric compensation.

Mark.	Meaning.
VOLCOMP1	Activate the first volumetric compensation (machine parameter VOLCOMP 1).
VOLCOMP2	Activate the second volumetric compensation (machine parameter VOLCOMP 2).
VOLCOMP3	Activate the third volumetric compensation (machine parameter VOLCOMP 3).
VOLCOMP4	Activate the fourth volumetric compensation (machine parameter VOLCOMP 4).

All volumetric compensations can be simultaneously active, as long as there are no common axes among them. The CNC applies the volumetric compensation after applying the leadscrew compensation and the cross compensation. Volumetric compensation remains active after a reset, error or end of program (M30).

12.1.3 File containing basic volumetric compensation data.

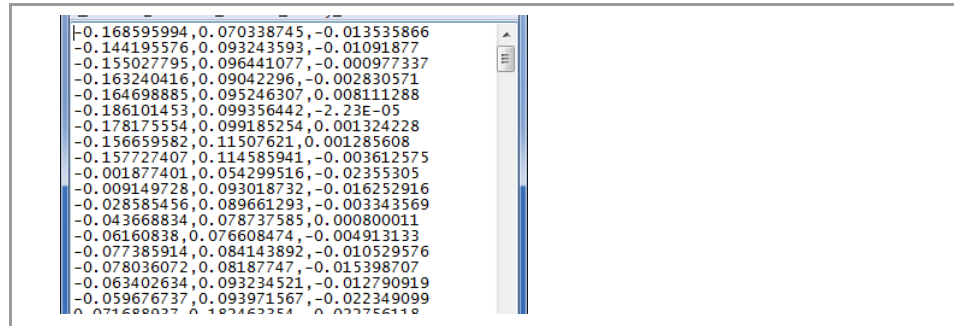
The data to be compensated must be a text file (in csv format). The OEM or the company responsible for the calibration must generate this file and define it in the machine parameter VCOMPFILE. The units in the file data (millimeters or inches) must be those defined by the CNC (parameter INCHES).

This file defines the error to be compensated at each volume point. At the intermediate points, the CNC interpolates the error based on the eight adjacent known points. For the points outside the defined volume, the error to be compensated will be zero.

Format of the data file.

The text file consists of three data columns and as many rows as there are points to be compensated, in which the data for each row are separated by the character "," (a comma).

$$\text{Number of rows} = \text{NPOINTSAX1} \times \text{NPOINTSAX2} \times \text{NPOINTSAX3}$$



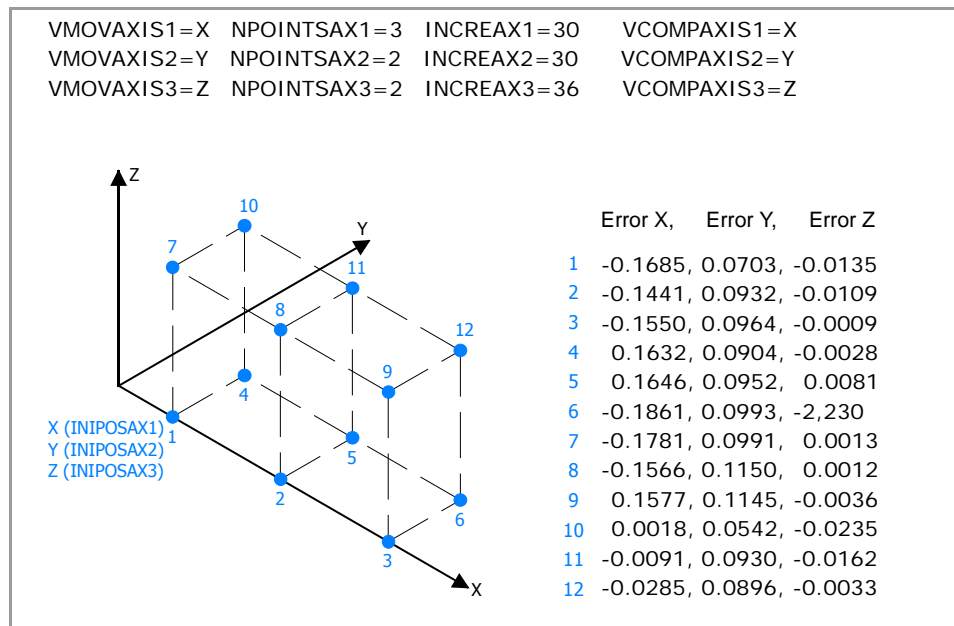
Example.

NPOINTSAX1=3
 NPOINTSAX2=2
 NPOINTSAX3=2

Number of rows on the table or points to be compensated.

$$\text{NPOINTSAX1} \times \text{NPOINTSAX2} \times \text{NPOINTSAX3} = 3 \times 2 \times 2 = 12$$

- The columns correspond to the axes defined in the parameters VCOMPAXIS1 (first column), VCOMPAXIS2 (second column) and VCOMPAXIS3 (third column).
- The rows are created with the consecutive error measurements, according to the order of the axes defined in MOVAXIS; first all the measurements on the first axis; then for each position on the second axis, all the measurements on the first axis once again; and for each position on the third axis, all the measurements of the first axis on each position of the second axis.



12.

VOLUMETRIC COMPENSATION. Setup.



FAGOR AUTOMATION

CNC 8070

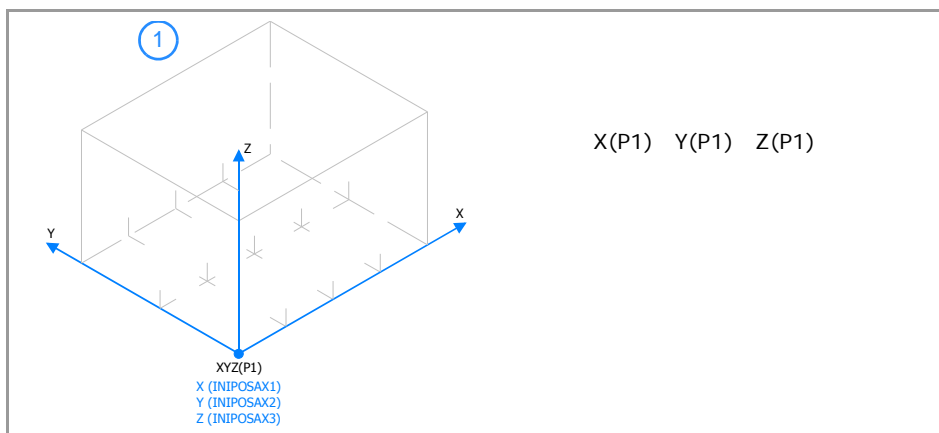
(REF: 1709)

12.1.4 Sequence to define the values in the file.

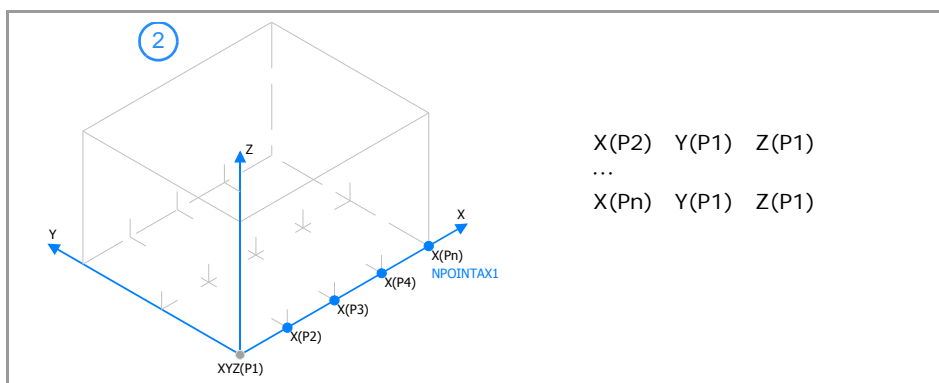
Assuming a trihedron on the following axes.

VMOVAXIS1=X VMOVAXIS2=Y VMOVAXIS3=Z.

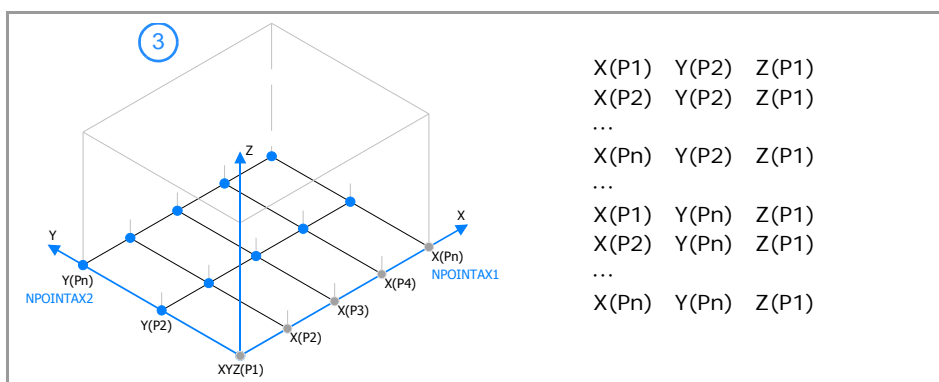
- 1 The first row corresponds to the point "X(P1), Y(P1), Z(P1)". This point corresponds to the initial point of the volume to be compensated (parameters INIPOSAX1, INIPOSAX2 and INIPOSAX3).



- 2 The second row corresponds to the point "X(P2), Y(P1), Z(P1)", the third to point "X(P3), Y(P1), Z(P1)", and so on until completing all the points of X (parameter NPOINTSAX1). The distance between the points on the X axis is defined in the parameter INCREAX1.



- 3 Repeat the same operation to define all the points on the X axis, from the point Y(P2) to the last point on the Y axis (parameter NPOINTSAX2). The distance between points on the Y axis is defined in the parameter INCREAX2.



4 Once all the points on the plane have been defined (coordinate Z(P1)), repeat the same operation from point "X(P1), Y(P1), Z(P2)" until the last point on the Z axis (parameter NPOINTSAX3).

12.

VOLUMETRIC COMPENSATION. Setup.

X(P1) Y(P1) Z(P2)
 X(P2) Y(P1) Z(P2)
 ...
 X(Pn) Y(P1) Z(P2)

X(P1) Y(P2) Z(P2)
 X(P2) Y(P2) Z(P2)
 ...
 X(Pn) Y(P2) Z(P2)
 ...
 X(P1) Y(Pn) Z(P2)
 X(P2) Y(Pn) Z(P2)
 ...
 X(Pn) Y(Pn) Z(P2)

X(P1) Y(P1) Z(Pn)
 X(P2) Y(P1) Z(Pn)
 ...
 X(Pn) Y(P1) Z(Pn)
 ...
 X(P1) Y(Pn) Z(Pn)
 X(P2) Y(Pn) Z(Pn)
 ...
 X(Pn) Y(Pn) Z(Pn)
 ...
 X(P1) Y(Pn) Z(Pn)
 X(P2) Y(Pn) Z(Pn)
 ...
 X(Pn) Y(Pn) Z(Pn)



CNC 8070

(REF: 1709)

12.2 Variables.

12.2.1 Variables. General; machine parameters.

(V.)MPG.VMOVAXIS1[tbl]

(V.)MPG.VMOVAXIS2[tbl]

(V.)MPG.VMOVAXIS3[tbl]

Variable that can only be read from the program, PLC and interface.

The variable returns the execution value; reading it interrupts block preparation.

Basic volumetric compensation table [tbl]. Name of the axis that generates changes when moved.

Syntax.

·tbl· Table number.

V.MPG.VMOVAXIS2[1]

Name of the second axis on the first volumetric compensation table.

(V.)MPG.NPOINTSAX1[tbl]

(V.)MPG.NPOINTSAX2[tbl]

(V.)MPG.NPOINTSAX3[tbl]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Basic volumetric compensation table [tbl]. Number of points of basic volumetric compensation on each axis.

Syntax.

·tbl· Table number.

V.MPG.NPOINTAX2[1]

Number of points from the first basic volumetric compensation on the second axis.

(V.)MPG.INIPOSAX1[tbl]

(V.)MPG.INIPOSAX2[tbl]

(V.)MPG.INIPOSAX3[tbl]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Basic volumetric compensation table [tbl]. Initial position of the basic volumetric compensation on each axis.

Syntax.

·tbl· Table number.

V.MPG.INIPOSAX2[1]

Initial position of the first basic volumetric compensation on the second axis.

(V.)MPG.INCREAX1[tbl]

(V.)MPG.INCREAX2[tbl]

(V.)MPG.INCREAX3[tbl]

Variable that can only be read from the program, PLC and interface.

The variable returns the value of block preparation.

Interval between basic volumetric compensation points on each axis.

Syntax.

·tbl· Table number.

V.MPG.INCREAX2[1]

Interval between points from the first basic volumetric compensation on the second axis.

12.

VOLUMETRIC COMPENSATION.
Variables.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

12.

(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 execution value; reading it interrupts block preparation.*

Volumetric compensation table [tbl]. Axis to be compensated.

Syntax.

·tbl· Table number.

V.MPG.VCOMPAXIS2[1]

Second axis to be compensated in the first volumetric compensation table.

(V.)MPG.VCOMPFILE[tbl]*Variable to be read via 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.

12.2.2 Variables. General modifiable signals.**(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 (changing it from 0 to 1), the CNC activates the corresponding volumetric compensation (parameter VOLCOMP).

V.PLC.VOLCOMP1

Volumetric compensation -1-.

V.PLC.VOLCOMP2

Volumetric compensation -2-.

V.PLC.VOLCOMP3

Volumetric compensation -3-.

V.PLC.VOLCOMP4

Volumetric compensation -4-.

12.2.3 Variables. Volumetric compensation.

(V.)[ch].A.VOLCOMP.xn

Variable that can only be read from the program, PLC and interface.

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. If the volumetric compensation is not active or the axis is outside the volume to be compensated, the variable returns the value to 0 (zero).

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.

Parameter valid for linear and rotary axes and spindles.

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.

This variable is only valid for the medium and large volumetric compensations; it has no function on the basic volumetric compensation.

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·.

12.

VOLUMETRIC COMPENSATION.
Variables.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

12.3 Error messages (cause and solution).

12.

VOLUMETRIC COMPENSATION.
Error messages (cause and solution).

0046 'Nonexistent axis'

DETECTION	During the validation of the machine parameters.
CAUSE	The possible causes are: <ul style="list-style-type: none"> • The user has assigned the name of an axis that does not exist to a machine parameter. • Either the VMOVAXIS or VCOMPAXIS axis is undefined in the basic volumetric compensation.
SOLUTION	Correct the machine parameters. The valid axis names are the ones defined in parameter AXISNAME.

1445 'Wrong parameter value'

DETECTION	During execution.
CAUSE	A parameter has an invalid value in an instruction or a fixed cycle.
SOLUTION	Check the program.

1447 'Software option not allowed'

DETECTION	During execution.
CAUSE	The CNC does not have the software option required to execute the programmed command.
SOLUTION	In diagnosis mode, it is possible to check the software options offered by the CNC.

9347 'Axis repeated in VOLCOMP'

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	The same axis is defined twice in the VMOVAXIS or VCOMPAXIS volumetric compensation parameters.
SOLUTION	Correct the machine parameters.

23101 'Error when analyzing the volumetric compensation definition file'

DETECTION	On CNC power-up or when validating the machine parameters.
CAUSE	The volumetric definition file does not contain the proper data. Basic volumetric compensation: <ul style="list-style-type: none"> • The file is not in csv format. • The format of the file defined in VCOMPFILE is incorrect. • The file defined in VCOMPFILE does not have all the points defined. The number of points is defined by the expression: NPOINTSAX1 x NPOINTSAX2 x NPOINTSAX3 Medium and large volumetric compensation: <ul style="list-style-type: none"> • The format of the file defined in VCOMPFILE is incorrect.
SOLUTION	Select a proper file. The file must be in csv format and have all the points defined in the basic volumetric compensation.

23102 'Volumetric compensation system error; Compensation off'

DETECTION	During execution.
CAUSE	The CNC has detected an internal error when calculating the volumetric compensation and has canceled the compensation values.
SOLUTION	Contact Fagor Automation.

23103 'The VOLCOMP mark cannot be activated if a table is not validated'

DETECTION	During execution.
CAUSE	The PLC has tried to activate a volumetric compensation that is not defined properly and therefore the machine cannot be compensated.
SOLUTION	Define a proper volumetric compensation in the machine parameters (VOLCOMP parameters).



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

23104 'The VOLCOMP mark cannot be activated while there is another table active on common axes'

DETECTION	During execution.
CAUSE	The PLC has tried to activate a volumetric compensation while another one is active and one of the axes used is common to both compensations.
SOLUTION	There cannot be two active volumetric compensations having an axis in common. Before activating a compensation, cancel the active one using the corresponding PLC mark.

12.

VOLUMETRIC COMPENSATION.
Error messages (cause and solution).

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

12.

VOLUMETRIC COMPENSATION.

Error messages (cause and solution).



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).

13

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.

Default HSC mode.

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 part surface finish (SURFACE mode) optimizing the contour error (CONTERROR mode) or the machining speed (FAST mode).

The default machining mode is defined by parameter HSCDEFAULTMODE, where Fagor offers the SURFACE mode as default. The more sophisticated algorithms of the SURFACE mode obtain more accurate machining. Tests run at Fagor have obtained an average accuracy improvement of 25% to 30%. In parallel, the CNC controls machine movements more smoothly reducing considerably the vibrations due to part geometry or machine dynamics. Reducing machine vibrations results in higher surface quality of the machined parts.

13.1 Configuration of the HSC mode.

The HSC mode is configured with the following axis and general machine parameters.

Parameter.	Meaning.
HSCDEFAULTMODE	Default mode when programming #HSC ON.
FEEDAVRG	Calculating feedrate average.
SMOOTHFREQ	Smoothing frequency in the interpolation.
CORNER	Maximum angle of the corner to machine it in square corner mode.
HSCFILTFREQ	Filter frequency (CONTERROR mode).
FASTFACTOR	Default feedrate percentage (FAST mode).
FTIMELIM	Time difference permitted in feedrate interpolation (FAST mode).
MINCORFEED	Minimum feedrate at the corners.
FSMOOTHFREQ	Smoothing frequency in the interpolation (FAST mode).
FASTFILTFREQ	Filter frequency (FAST mode).
FREQRES	First resonance frequency of the machine.
SOFTFREQ	Path filter frequency for profiles of linear acceleration.
HSCROUND	Default value of the maximum path error in HSC.
SURFFILFREQ	Axis filter frequency (SURFACE mode).

Parameter.	Meaning.
CORNERACC	Maximum acceleration permitted at the corners.
CURVACC	Maximum contouring acceleration permitted.
CORNERJERK	Maximum Jerk permitted at the corners.
CURVJERK	Maximum contouring Jerk permitted.
FASTACC	Maximum acceleration permitted (FAST mode).
MAXERROR	Position error.
CONTERROR	Tolerance per axis for smoothing the generated n-dimensional path.

General machine parameters.

FEEDAVRG Calculating feedrate average.

This parameter enables the adjustment of the feedrate according to the block reading speed and their size. This adjustment eliminates the need to slow down due to a poor supply of small blocks; although as a result, the feedrate reached is lower, the overall machining time will improve. The block reading speed depends on machine parameter PREPFREQ.

This parameter is only valid when the acceleration profile is trapezoidal or square sine (parameter SLOPETYPE), that is the default acceleration profile for the HSC CONTERROR mode.

SMOOTHFREQ Smoothing frequency in the interpolation..

This parameter sets the smoothing frequency in path interpolation. This parameter avoids accelerating and decelerating throughout a path, beyond a particular frequency when generating an averaged speed.

This parameter is only valid when the acceleration profile is trapezoidal or square sine (parameter SLOPETYPE), that is the default acceleration profile for the HSC CONTERROR mode.

13.

CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).
Configuration of the HSC mode.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

CORNER

Maximum angle of the corner to machine it in square corner mode.

This parameter indicates the maximum angle between two paths, under which the machining operation is carried out in square corner mode.

HSCFILTFREQ

Filter frequency (CONTERROR mode).

This parameter activates an automatic FIR filter for all the axes of the channel during the execution in CONTERROR mode. This filter may be used to smooth the response of the axes generating a smoother tool path and, if necessary, reduce the feedrate so the path error is adjusted to the programmed path.

This parameter inserts a variable (non constant) phase shift depending on the frequencies. This phase shift can cause a change on the path if it is not executed at the same feedrate as, for example, when changing the feedrate override percentage or when moving back and forth on the same path.

FASTFACTOR

Default feedrate percentage (FAST mode).

This parameter indicates the feedrate at the corners and indicates the feedrate percentage to be reached, of the maximum that the CNC can reach in FAST mode. This parameter indicates the default value and it may be changed from the part program.

FTIMELIM

Time difference permitted in feedrate interpolation (FAST mode).

The feedrate interpolation in HSC FAST mode improves the machining quality. Parameter FTIMELIM allows limiting the time it takes the CNC to interpolate the feedrate in each segment. When programmed with a 0 value, the CNC does not interpolate the feedrate and, therefore, the execution is faster. When increasing the value of this parameter, it increases the maximum total execution time allowed; e.g. a value of 200% or 300% will allow an execution time twice or three times as long in the interpolated segment.

In rather large blocks where the feedrate adaptation represents a longer additional time than the one indicated in FTIMELIM, the CNC will adapt the feedrate using the maximum dynamics to avoid unnecessary waste of time.

MINCORFEED

Minimum feedrate at the corners.

This value should not be modified because it is possible to exceed the dynamics of the axes.

FSMOOTHFREQ

Smoothing frequency in the interpolation (FAST mode).

This parameter sets the smoothing frequency in path interpolation for the HSC FAST mode. This parameter avoids accelerating and decelerating throughout a path, beyond a particular frequency when generating an averaged speed. It is only applied when the HSC FAST mode is active.

FASTFILTFREQ

Filter frequency (FAST mode).

This parameter activates an automatic "low pass" filter for all the axes of the channel while executing in HSC FAST mode that permits smoothing the response of the axes by generating a smoother path. This filter has the drawback of rounding the corners slightly.

This parameter inserts a constant phase shift regardless of the frequencies.

13.

CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).

Configuration of the HSC mode.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

13.

CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).
 Configuration of the HSC mode.

FREQRES

First resonance frequency of the machine.

Resonance frequency that the CNC must eliminate when generating the velocity command.

This parameter is only valid when the acceleration profile is trapezoidal or square sine (parameter SLOPETYPE), that is the default acceleration profile for the HSC CONTERROR mode.

SOFTFREQ

Path filter frequency for profiles of linear acceleration.

This parameter may be used to smooth the velocity profile in HSC FAST mode, which improves machining time and surface quality. This parameter must be set using the FineTune application.

HSCROUND

Default value of the maximum path error in HSC.

Maximum path error in HSC.

SURFFILFREQ

Axis filter frequency (SURFACE mode).

This parameter activates an automatic filter for all the axes of the channel while executing in HSC SURFACE mode that permits smoothing the response of the axes by generating a smoother path.

HSCDEFAULTMODE

Default mode when programming #HSC ON.

Default mode to activate HSC machining.

Axis machine parameters.**CORNERACC**

Maximum acceleration permitted at the corners.

This parameter sets the maximum acceleration allowed for the axis during block transition. If this parameter is set with a -0- value, the maximum acceleration of the axis is respected.

CURVACC

Maximum contouring acceleration permitted.

If this parameter is set with a -0- value, the maximum acceleration of the axis is respected.

CORNERJERK

Maximum Jerk permitted at the corners.

If this parameter is set with a -0- value, the maximum jerk of the axis is respected.

CURVJERK

Maximum contouring Jerk permitted.

If this parameter is set with a -0- value, the maximum jerk of the axis is respected.

FASTACC

Maximum acceleration permitted (FAST mode).

If this parameter is set with a -0- value, the maximum acceleration of the axis is respected.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

MAXERROR
Position error.

Maximum axis position error in HSC when working outside the plane/trihedron.

CONTERROR
Tolerance per axis for smoothing the generated n-dimensional path.

For 5-axis machining, this parameter may be used to set the tolerance per axis for smoothing the generated n-dimensional path. This parameter does not affect the 3D error when working with RTCP. We recommend setting this parameter on rotary axes.

13.

CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).

Configuration of the HSC mode.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

13.2 Influence of the type of acceleration and of the filters in HSC mode.

13.

CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).

Influence of the type of acceleration and of the filters in HSC mode.

HSC optimizing the chordal error (CONTERROR mode).

In this mode, the CNC modifies the geometry through intelligent algorithms for eliminating unnecessary points and generating splines or polynomial transitions between blocks always respecting the error limits set for this mode. This way, the contour is traveled at a variable feedrate according to the curvature and the programmed parameters (acceleration and feedrate) but without going beyond the set error limits.

This mode uses this type of acceleration is defined with parameter SLOPETYPE. The following situations can occur when the type of acceleration has jerk.

- Jerk control permits pretty smooth velocity and acceleration profiles.

In this case, if there is no need to eliminate resonance frequencies, there is no need to activate filters to make the path smoother. Since there are no filters, the part will be more accurate because the path goes through the programmed points. The exception is on sections where the CNC disconnects the spline because it exceeds the error limit and, in that case, it executes a rounding polynomial that passes at a distance from the programmed point lower than the error limit.

- Since the HSC mode tries to respect the jerk throughout the path, some irregularities may result on the velocity profile, especially at the corners, and, consequently, affect the quality of the surface finish.

In this case, the amount of jerk can be increased at the corners (parameter CORNERJERK) or apply the smoothing frequency in the interpolation of the path (parameter SMOOTHFREQ). Activating the smoothing frequency generates smoother velocity and acceleration profiles, but increasing the execution time.

The effects of the filters set by the FILTER parameter are added to the effects of the filters active in HSC; parameter HSCFILTFREQ for the CONTERROR mode.

HSC optimizing the feedrate (FASTmode).

It is the recommended way (mode) to work in the following circumstances.

- When feedrate is more important than accuracy.
- When the CONTERROR mode does not provide the desired results. When the error with which the part has been generated is not significantly lower than the error demanded to the CONTERROR, the execution is slower and the machined surfaces show ridges.
- When the dynamics of the machine does not respond in wide range of frequencies; i.e. the machine shows resonance or a limited bandwidth.

The FAST mode provides a smoother surface finish at a more constant feedrate. From the program, it is possible to indicate the percentage of feedrate to apply, over the maximum that the CNC can reach depending on the setting of parameters LACC1 and LACC2. Programming it is optional; if not defined, it assumes the percentage set in parameter FASTFACTOR. This parameter affect directly the feedrate when going through the corners and it must be borne in mind when reaching a compromise between the feedrate at the corner and its finish quality.

This mode uses linear acceleration, regardless of the setting of parameter SLOPETYPE. For best results, it is essential to set parameters LACC1 and LACC2 properly. Since there is no jerk control, a filter is required for all the axes, inserted automatically, to make the machine movement smoother. The frequency of this filter for all the axis is set in parameter FASTFILTFREQ.

Using axis filters results in smoother velocity and acceleration profiles, but lower precision at the corners depending on geometry and type of filter. The higher the linear acceleration or the lower the range of frequencies of the machine, the lower the frequency of the filter needed to generate a greater loss of accuracy to narrow the bandwidth of the system.

The effects of the filters set by the FILTER parameter are added to the effects of the filters active in HSC; parameter FASTFILTFREQ for the FAST mode.

(REF: 1709)

13.3 Procedure for analysis and adjustment of the HSC.

To adjust the HSC mode, the CNC axes must have parameters G00FEED, ACCEL, LACC1, LACC2 and ACCJERK already tested and adjusted to the maximum value allowed by the dynamics of the machine. The following procedure is meant for analyzing the behavior of the CNC while machining and evaluate whether the result may be improved or not by adjusting a parameter. The steps to follow are.

- 1 Evaluate the time the CNC has for its calculations and if the blocks are supplied properly.
- 2 Evaluate the feedrate on the path in case there are irregularities from one pass to the next.
- 3 Analyze the problems occurring at specific points.

The CNC has a set of variables that, when analyzed while machining, help find aspect of the HSC that may be improved in terms of execution time and machining quality. The oscilloscope will serve as a tool to analyze the evolution HSC while machining.

Evaluate the time the CNC has for its calculations and if the blocks are supplied properly.

An improper supply of blocks forces to lower the feedrate unnecessarily. The CNC offers the following variables to check whether the HSC has a proper supply of blocks or not.

Variable.	Meaning.
V.[ch].G.PERFRATE	Percentage of blocks managed by the CNC in block preparation, with respect to the best possible.
V.[ch].G.DROPRATE	Percentage of reduction of the maximum feedrate possible.
V.G.NCTIMERATE	Percentage of loop time (cycle time) used by the real time portion of the CNC.
V.G.LOOPTIMERATE	Percentage of loop time (cycle time) used by the position loop.
V.[ch].G.CHTIMERATE	Percentage of loop time (cycle time) used by the channel.
V.[ch].G.PREPTIMERATE	Percentage of loop time (cycle time) used by the channel for block preparation.
V.[ch].G.IPOTIMERATE	Percentage of loop time (cycle time) used by the interpolator of the channel.

In order to ensure a proper supply of blocks to the HSC, the following variables must return a value near 100%.

V.[ch].G.PERFRATE A value lower than 100 means that the CNC can process more blocks.

V.[ch].G.DROPRATE A value lower than 100 means that the CNC could increase the feedrate if it had more blocks.

In either case, 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 with the following variable.

V.[ch].G.NCTIMERATE With a value lower than 50, parameter PREPFREQ may be increased.

To properly analyze this aspect, parameter FEEDAVRG must be set to ·No· to prevent the CNC from limiting the feedrate depending on the supply of blocks.

Evaluate the feedrate on the path in case there are irregularities from one pass to the next.

For similar machining passes, the velocity graph should look the same. However, in some passes, the feedrate could decrease even more and the machining could be less

13.

symmetrical than what appears on the graph. The CNC has the following variables to analyze the cause of this feedrate reduction.

Variable.	Meaning.
V.[ch].G.PARLIMF	Cause that limits the feedrate at the block being executed.
V.[ch].G.AXLIMF	Logic number of the axis that limits the feedrate at the block being executed.
V.[ch].G.PARLIMC	Cause that limits the feedrate at the corner in the current block.
V.[ch].G.AXLIMC	Logic number of the axis that limits the feedrate at the corner, in the block being executed.
V.[ch].G.LIMERROR	Value of the error that cancels the splines (CONTERROR mode).
V.[ch].G.PATHFEED	Theoretical feedrate on the tool path.
V.[ch].G.FREAL	Real feedrate on the tool path.

To analyze the cause of feedrate reduction, use a trace of the oscilloscope that shows an undesired feedrate reduction. Move the cursor to the value where the theoretical feedrate (V.[ch].G.PATHFEED) decreases and check whether the cause that limits the feedrate may be adjusted or the generated error is greater than the one programmed.

Causes that limit the feedrate on the tool path.

Check the V.[ch].G.AXLIMF variable to know which axis is limiting the feedrate and the V.[ch].G.PARLIMF variable to know the reason why this axis is limiting the feedrate. This last 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.

The error incurred by the spline may be checked with the V.G.LIMERROR variable. Depending on the cause, some HSC adjusting parameter may be tweaked.

- Parameter CURVACC may be changed if the feedrate is limited by the acceleration due to curvature).
- Parameter CURVJERK may be changed if the feedrate is limited by the jerk due to curvature).

Causes that limit the feedrate at the corners.

Check the V.[ch].G.AXLIMC variable to know which axis is limiting the feedrate and the V.[ch].G.PARLIMC variable to know the reason why this axis is limiting the feedrate. This last 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 (CONTERROR).
9	Geometry at the corner (FAST mode).

13.

CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).
Procedure for analysis and adjustment of the HSC.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Check whether the generated error is greater than the one programmed or not.

Check the vV.[ch].G.LIMERROR variable to know the value of the error that cancels the splines. This last 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	Value of the error that cancels the splines.

If the value returned by this variable is greater than the one programmed, it would be a good idea to execute the HSC with more error or generate the program with less error in the CAM.

13.

CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).
Procedure for analysis and adjustment of the HSC.

Analyze the problems occurring at specific points.

If a problem comes up at a specific point, use a trace of the oscilloscope to analyze the following variables and locate the velocity profile that causes the problem.

Variable.	Meaning.
V.[ch].G.BLKN	Last block executed (number).
V.[ch].G.LINEN	Number of program blocks executed or prepared.
V.[ch].G.PATHFEED	Theoretical feedrate on the tool path.

Then, use the oscilloscope to analyze the following variables, sequentially because it only admits four, to find the cause of the feedrate problem.

Variable.	Meaning.
V.[ch].G.PARLIMF	Cause that limits the feedrate at the block being executed.
V.[ch].G.AXLIMF	Logic number of the axis that limits the feedrate at the block being executed.
V.[ch].G.PARLIMC	Cause that limits the feedrate at the corner in the current block.
V.[ch].G.AXLIMC	Logic number of the axis that limits the feedrate at the corner, in the block being executed.
V.[ch].G.LIMERROR	Value of the error that cancels the splines (CONTERROR mode).
V.[ch].G.PATHFEED	Theoretical feedrate on the tool path.
V.[ch].G.FREAL	Real feedrate on the tool path.

Analyze the various variables of the axes in the problem area to determine if there is an error on the theoretical path, in the control loop or in the feedback. The following variables may be used for this latter analysis.

Variable.	Meaning.
V.[ch].A.IPOPOS.xn	Theoretical position value (coordinate) at the output of the interpolator, before the transformation; i.e. in part coordinates.
V.[ch].A.ADDMANOF.xn	Distance moved with G201.
V.[ch].A.INDPOS.xn	Theoretical coordinate of the independent interpolator.
V.[ch].A.FILTERIN.xn	Theoretical coordinate of the interpolator before the filter.
V.[ch].A.FILTEROUT.xn	Theoretical coordinate of the interpolator after the filter.
V.[ch].A.LOOPTPOS.xn	Theoretical coordinate at the input of the position loop.
V.[ch].A.LOOPPOS.xn	Real coordinate at the input of the position loop.
V.[ch].A.TFEED.xn	Instantaneous theoretical speed value at the input of the position loop.
V.[ch].A.FEED.xn	Instantaneous real speed value at the input of the position loop.
V.[ch].A.POSCMD.xn	Position command for Sercos.
V.[ch].A.FTEO.xn	Velocity command for Sercos (in rpm).
V.[ch].A.POSNC.xn	Position feedback.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

13.4 Summary of the useful variables to analyze the HSC.

13.4.1 Analysis of the loop time (cycle time) at the CNC.

V.G.NCTIMERATE

Percentage of loop time (cycle time) used by the real time portion of the CNC.

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

Percentage of loop time (cycle time) used by the position loop.

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.

13.4.2 Analysis of the loop time (cycle time) in the channel.

V.[ch].G.CHTIMERATE

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.

V.[ch].G.PREPTIMERATE

Percentage of loop time (cycle time) used by the channel for block preparation.

This variable serves to evaluate the load in path preparation and to know whether it is possible to increase the PREPFREQ parameter or not.

V.[ch].G.IPOTIMERATE

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.

13.

CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).
Summary of the useful variables to analyze the HSC.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

13.4.3 Variables associated with feedrate limitation.

V. [ch].G.PERFRATE

Percentage of blocks managed by the CNC in block preparation, with respect to the best possible to reach the maximum feedrate in each section.

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

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.

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.

V. [ch].G.LIMERROR

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.

Special returned values.

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 (CONTERROR mode).

V. [ch].G.AXLIMF

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.

V. [ch].G.PARLIMF

Cause that limits the feedrate at the block being executed.

13.

CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).
 Summary of the useful variables to analyze the HSC.

Special returned values.

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.

V.[ch].G.AXLIMC

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.

V.[ch].G.PARLIMC

Cause that limits the feedrate at the corner in the block being executed.

Special returned values.

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).

13.4.4 Variables associated with the FEEDRATE on the tool path.**V.[ch].G.PATHFEED**

Theoretical feedrate on the tool path.

V.[ch].G.FREAL

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.

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.

(REF: 1709)

13.4.5 Variables associated with the block being executed.

(V.)[ch].G.BLKN

Last block executed (number).

If no label has been executed, the variable will return a -1 value.

(V.)[ch].G.LINEN

Number of program blocks executed or prepared.

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.

13.4.6 Variables associated with the coordinates in the loop.

(V.)[ch].A.IPOPOS.xn
(V.)[ch].A.IPOPOS.sn
(V.)[ch].SP.IPOPOS.sn

Theoretical position value (coordinate) at the output of the interpolator, before the transformation; i.e. in part coordinates.

(V.)[ch].A.ADDMANOF.xn

Distance moved with G201.

(V.)[ch].A.INDPOS.xn
(V.)[ch].A.INDPOS.sn
(V.)[ch].SP.INDPOS.sn

Theoretical coordinate of the independent interpolator.

(V.)[ch].A.FILTERIN.xn
(V.)[ch].A.FILTERIN.sn
(V.)[ch].SP.FILTERIN.sn

Theoretical coordinate of the interpolator before the filter.

(V.)[ch].A.FILTEROUT.xn
(V.)[ch].A.FILTEROUT.sn
(V.)[ch].SP.FILTEROUT.sn

Theoretical coordinate of the interpolator after the filter.

(V.)[ch].A.LOOPTPOS.xn
(V.)[ch].A.LOOPTPOS.sn
(V.)[ch].SP.LOOPTPOS.sn

Theoretical coordinate at the input of the position loop.

(V.)[ch].A.LOOPPOS.xn
(V.)[ch].A.LOOPPOS.sn
(V.)[ch].SP.LOOPPOS.sn

Real coordinate at the input of the position loop.

13.4.7 Variables associated with the velocity in the loop.

(V.)[ch].A.TFEED.xn
 (V.)[ch].A.TFEED.sn
 (V.)[ch].SP.TFEED.sn

Instantaneous theoretical speed value at the input of the position loop.

(V.)[ch].A.FEED.xn
 (V.)[ch].A.FEED.sn
 (V.)[ch].SP.FEED.sn

Instantaneous real speed value at the input of the position loop.

13.4.8 Variables associated with the velocity command and the feedback.

(V.)[ch].A.POSCMD.xn
 (V.)[ch].A.POSCMD.sn
 (V.)[ch].SP.POSCMD.sn

Position command for Sercos.

(V.)[ch].A.POSNC.xn
 (V.)[ch].A.POSNC.sn
 (V.)[ch].SP.POSNC.sn

Position feedback.

(V.)[ch].A.FTEO.xn
 (V.)[ch].A.FTEO.sn
 (V.)[ch].SP.FTEO.sn

Velocity command for Sercos (in rpm).

13.

CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).

Summary of the useful variables to analyze the HSC.

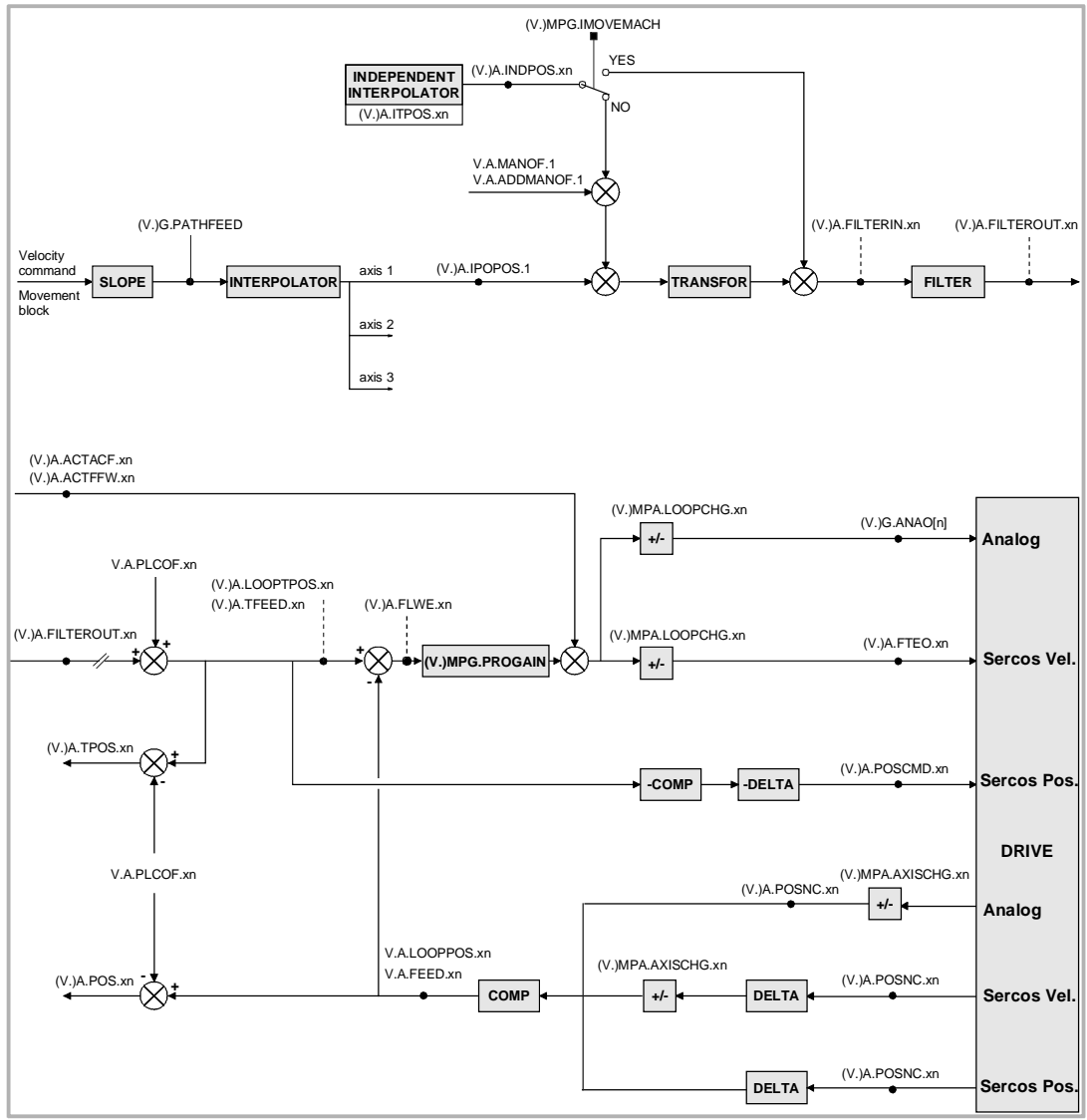


FAGOR AUTOMATION

CNC 8070

(REF: 1709)

13.5 The loops and the variables.



- COMP Coordinate compensation due to leadscrew error compensation or to cross compensation.
- DELTA Offset on power-up to maintain the coordinate when it was turned off.

13.

CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).
The loops and the variables.

13.

CONFIGURING IN HSC MODE (HIGH SPEED CUTTING).

The loops and the variables.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

The CNC has a series of subroutines that must be defined depending on the configuration of the machine (subroutine associated with a tool change, home search, etc.).

Subroutine.	Meaning.
Subroutine associated with the start.	Name: PROGRAM_START PROGRAM_START_C1 PROGRAM_START_C2 PROGRAM_START_C3 PROGRAM_START_C4 Channels: Subroutine per channel. Path: ..\Users\Sub.
Subroutine associated with the reset.	Name: PROGRAM_RESET PROGRAM_RESET_C1 PROGRAM_RESET_C2 PROGRAM_RESET_C3 PROGRAM_RESET_C4 Channels: Subroutine per channel. Path: ..\Users\Sub.
Subroutines associated with the kinematics calibration cycle.	Name: KinCal_Begin.nc KinCal_End.nc Channels: Subroutine common to all the channels. Path: ..\Mtb\Sub.
Subroutines associated with the tool calibration cycle.	Name: Sub_Probe_Tool_Begin.fst Sub_Probe_Tool_End.fst Channels: Subroutine common to all the channels. Path: ..\Mtb\Sub.
Subroutines associated with the part measuring cycle.	Name: Sub_Probe_Piece_Begin.fst Sub_Probe_Piece_End.fst Channels: Subroutine common to all the channels. Path: ..\Mtb\Sub.
Subroutine associated with program execution for the 8055 MC/TC.	Name: 9998 / 9999 Channels: Subroutine common to all the channels. Path: General parameter SUBPATH.
Subroutine associated with the tool change (function T).	Name: General parameter TOOLSUB. Channels: Subroutine per channel. Path: ..\Mtb\Sub.
Subroutine associated with the tool change (function M6).	Name: MPROGNAME parameters of the M function table. Channels: Subroutine common to all the channels. Path: ..\Mtb\Sub.
Subroutine associated with function G74.	Name: General parameter REFPSUB. Channels: Subroutine per channel. Path: ..\Mtb\Sub
Subroutines associated with M functions.	Name: MPROGNAME parameters of the M function table. Channels: Subroutine common to all the channels. Path: ..\Mtb\Sub.
OEM subroutines associated with functions G180 through G189.	Name: General parameter OEMSUB(G180) to OEMSUB(G189). Channels: Subroutine per channel Path: ..\Mtb\Sub.

14.

SUBROUTINES.

Subroutine.	Meaning.
OEM subroutines associated with functions G380 through G399.	Name: General parameter OEMSUB(G380) to OEMSUB(G399). Channels: Subroutine per channel. Path: ..\Mtb\Sub.
Interruption subroutines.	Name: General parameter INIT1SUB to INIT4SUB. Channels: Subroutine per channel. Path: ..\Mtb\Sub.
Subroutine associated with the command #INITIALSUB.	Name: General parameter INITIALSUB. Channels: Subroutine per channel. Path: ..\Mtb\Sub.
Subroutine associated with the command #PIERCING.	Name: General parameter PIERCING. Channels: Subroutine per channel. Path: ..\Mtb\Sub.
Subroutine associated with the command #CUTTING ON.	Name: General parameter CUTTINGON. Channels: Subroutine per channel. Path: ..\Mtb\Sub.
Subroutine associated with the command #CUTTING OFF.	Name: General parameter CUTTINGOFF. Channels: Subroutine per channel. Path: ..\Mtb\Sub.
Subroutine associated with the command #FINALSUB.	Name: General parameter FINALSUB. Channels: Subroutine per channel. Path: ..\Mtb\Sub.
User subroutines associated with functions G500 to G599.	Name: Name of the function (G500.nc, G501.nc, etc.) Channels: Subroutine common to all the channels. Path: ..\Users\Sub.
User subroutines associated with functions G8000 to G8999.	Name: Name of the function (G8000.nc, G8001.nc, etc.) Channels: Subroutine common to all the channels. Path: ..\Users\Sub.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.1 Run subroutines from RAM (extension *.fst).

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. The reserved RAM space for subroutines is 5 Mb.

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, 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.

14.

SUBROUTINES.

Run subroutines from RAM (extension *.fst).

14.2 Subroutine associated with the start.

For each channel, a subroutine can be associated with the execution start, which can be executed by pressing the [START] key, in automatic mode, to start the execution of the entire program; that is if no program start point has been selected. The CNC does not call upon the subroutine either when executing a cycle from the conversational mode. This subroutine may be used, for example, to have certain machining conditions defined to determine the execution of user programs.

If the subroutine exists, the CNC will execute it immediately after pressing the [START] key, before starting the program execution. If that subroutine is missing, the CNC executes the program directly.

Execution of subroutine

During execution, the CNC displays the name of the subroutine on the general status bar. The CNC does not display the blocks under execution, however, it executes the subroutine as a single block, this means that the block-by-block execution is not affected.



14.2.1 Configure the subroutines.

Name and location of the subroutine.

The name of the subroutine must be PROGRAM_START (without an extension) and it will be saved in the folder ..\Users\Sub. If there are several channels, there may be a different subroutine for each channel, whose name must be PROGRAM_START_Cn, where n is the channel number (between 1 and 4).

Name.	Channel.
PROGRAM_START PROGRAM_START_C1	Channel 1. The CNC accepts both names for the subroutine associated with the first channel and where both subroutines exist, the CNC will execute the PROGRAM_START.
PROGRAM_START_C2	Channel 2.
PROGRAM_START_C3	Channel 3.
PROGRAM_START_C4	Channel 4.

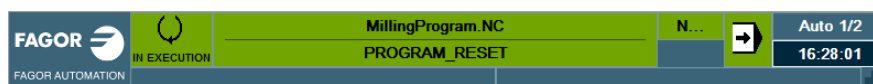
14.3 Subroutine associated with the reset.

For each channel, the reset may have an associated subroutine, which will be executed after pressing the [RESET] key on the operator panel or when the PLC activates the RESETIN mark. This subroutine, for example, establishes any initial conditions that are different from those set by the reset or conditioned to the configuration of the machine, activates operations/modes that disable the reset, etc.

If this subroutine exists, the CNC will execute it immediately after the reset. If this subroutine is missing, the CNC executes the reset directly.

Execution of subroutine

During execution, the CNC displays the name of the subroutine on the general status bar. The CNC does not display the blocks under execution, however, it executes the subroutine as a single block, this means that the block-by-block execution is not affected.



The CNC executes the subroutine with the "Functions G" option of the simulated execution mode.

- The execution takes into account tool radius compensation (functions G41 and G42), hence drawing the tool center path.
- The execution does not send M H S T functions to the PLC.
- The execution does not move the axes of the machine nor starts the spindle.
- The execution takes into account the dwells programmed with G4.
- The execution takes into account the program stops programmed with M00 and M01.

14.3.1 Configure the subroutines.

Name and location of the subroutine.

The name of the subroutine must be PROGRAM_RESET (without an extension) and it will be saved in the folder ..\Users\Sub. If there are several channels, there may be a different subroutine for each channel, whose name must be PROGRAM_RESET_Cn, where n is the channel number (between 1 and 4).

Name.	Channel.
PROGRAM_RESET PROGRAM_RESET_C1	Channel 1. The CNC accepts both names for the subroutine associated with the first channel and where both subroutines exist, the CNC will execute the PROGRAM_RESET.
PROGRAM_RESET_C2	Channel 2.
PROGRAM_RESET_C3	Channel 3.
PROGRAM_RESET_C4	Channel 4.

Executing the reset and the associated subroutine.

Each channel can have a subroutine associated with the reset. The reset process is as follows.

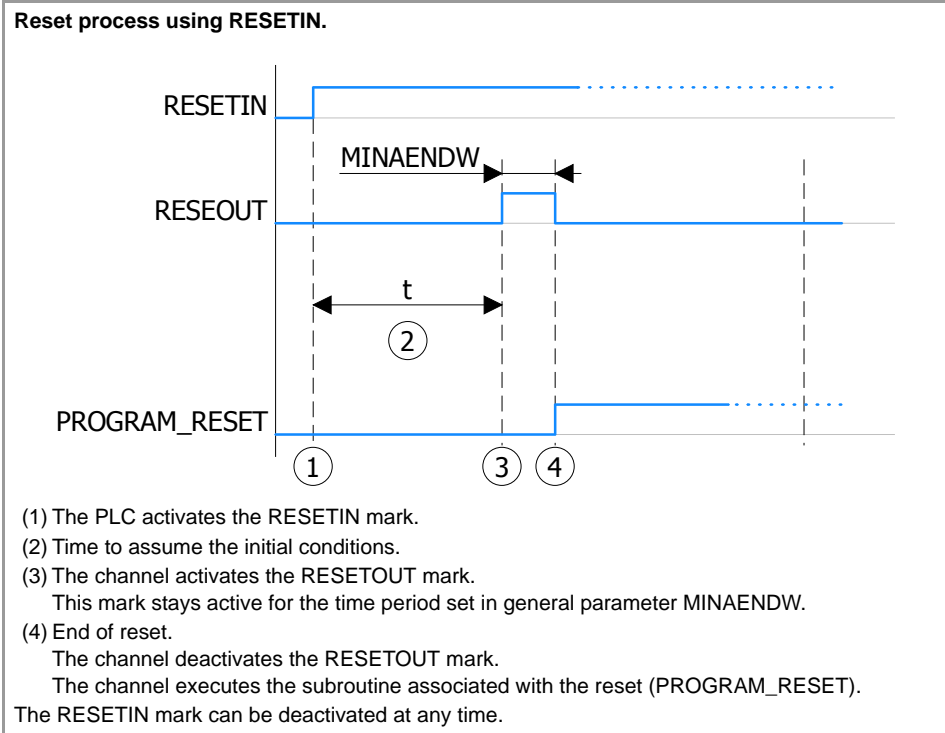
- 1 The user presses the [RESET] key on the operator panel or the PLC activates the RESETIN (RESETIN=1) mark.
- 2 The channel assumes the initial conditions (those defined by the machine parameters).
- 3 The channel activates the RESETOUT (RESETOUT=1) mark to indicate that the reset has finalized. This mark stays active for the time period set in general parameter MINAENDW.

- 4 The channel executes the subroutine associated with the reset (PROGRAM_RESET), should there be one.

14.

SUBROUTINES.

Subroutine associated with the reset.



14.4 Subroutine associated with 8055-MC and 8055-TC programs (subroutines 9998 and 9999).

In order to be able to execute programs of the 8055 MC/TC, the CNC must have two subroutines called 9998 and 9999 written in 8070 CNC language. Each program in the language of the 8055 MC/TC has a call to the corresponding subroutine at the beginning and at the end.

Subroutine.	Meaning.
9998	Routine that the CNC will execute at the beginning of each program of the 8055-MC and 8055-TC.
9999	Routine that the CNC will execute at the end of each program of the 8055-MC and 8055-TC.

Both subroutines must be defined, even if no operation is to be carried out at the beginning or at the end of the part-program, in which case the subroutines will be empty (except for the end-of-subroutine block). If any of these subroutines are missing, the CNC will issue an error message every time when trying to execute a 8055-MC or 8055-TC part-program.

14.4.1 Configure the subroutines.

Name and location of the subroutine.

Parameter.	Meaning.
SUBPATH CHN	This parameter indicates the directory, by default, for the user subroutines and the necessary subroutines to execute 8055-MC and 8055-TC programs.

[CHN]..... General machine parameter of the channel.

The name of the subroutines is 9998 and 9999. Both subroutines must be saved in the folder path indicated by the general parameter SUBPATH. All channels use the same subroutines.

14.

SUBROUTINES.

Subroutine associated with 8055-MC and 8055-TC programs (subroutines 9998 and 9999).



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.5 Subroutines associated with the kinematics calibration cycle.

The kinematics calibration cycle has two associated subroutines (KinCal_Begin.nc and KinCal_End.nc), which the CNC executes before and after the cycle.

Name.	Meaning.
KinCal_Begin.nc	Subroutine associated with the beginning of the kinematics calibration cycle.
KinCal_End.nc	Subroutine associated with the end of the kinematics calibration cycle.

Fagor supplies both subroutines as incomplete and it is the manufacturer's responsibility to define both subroutines. Software updates do not modify any existing subroutines.

14.5.1 Configure the subroutines.

Name and location of the subroutine.

The names of the subroutines are KinCal_Begin.nc and KinCal_End.nc. Both subroutines must be saved in the folder ..Mtb\Sub. All channels use the same subroutines.

14.

SUBROUTINES.

Subroutines associated with the kinematics calibration cycle.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.6 Subroutines associated with the tool calibration cycle.



The subroutines supplied by Fagor provide basic probe management, so they should be adjusted and configured appropriately by the OEM. Fagor provides the subroutine `Sub_Probe_Tool_Begin.fst` associated with probe input 1 and the subroutine `Sub_Probe_Piece_Begin.fst` associated with probe input 2.

The tool calibration cycle, for both the ISO and the editor, has two associated subroutines that the CNC executes before and after the cycle. These subroutines can be used to enable the probe and indicate if it is in safe mode.

Name.	Meaning.
<code>Sub_Probe_Tool_Begin.fst</code>	Subroutine associated with the beginning of the tool calibration cycle.
<code>Sub_Probe_Tool_End.fst</code>	Subroutine associated with the end of the tool calibration cycle.

Subroutines and M functions defined in the cycle.

Up to 4 M functions may be defined in each probing cycle to be executed before the cycle and another 4 M functions to be executed afterwards. All these functions may have a subroutine associated with them. When starting the execution, the cycle first executes the subroutine `Sub_Probe_Tool_Begin.fst` and then the M-before functions with their associated subroutines. At the end of the execution, the cycle executes first the M-after functions with their corresponding associated subroutines and then the subroutine `Sub_Probe_Tool_End.fst`.

Management of the PROBE1ENA/PROBE2ENA marks in the subroutine.

These mark indicate that the probe has is enabled. It is recommended that the OEM manages both marks from the subroutine `Sub_Probe_Tool_Begin.fst`, so that execution waits until the mark is active.

Management of the PROBE1MONIT/PROBE2MONIT marks in the subroutine.

These marks are associated with the safe mode of the probe. It is recommended that the OEM manages both marks from the subroutine `Sub_Probe_Tool_Begin.fst` to warn, should they be deactivated, that the probe is in non-safe mode.

14.6.1 Configure the subroutines.

Name and location of the subroutine.

The name of the subroutine is `Sub_Probe_Tool_Begin.fst` and `Sub_Probe_Tool_End.fst`. Both subroutines must be saved in the folder `.\Mtb\Sub`. All channels use the same subroutines.

Since it is an OEM subroutine with an `.fst` extension, the CNC loads this subroutine into the RAM of the CNC. See "14.1 Run subroutines from RAM (extension `*.fst`)." on page 617.

14.

SUBROUTINES.

Subroutines associated with the tool calibration cycle.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.6.2 Example of a subroutine.

Example of a subroutine Sub_Probe_Tool_Begin.fst supplied by Fagor (to be adjusted by the OEM).

```
#ESBLK
; Activate PROBE1 Hardware by PLC output.

; Check PROBE is READY with PLC Input from Probe Hardware.
#MSG["WAIT FOR ENABLING PROBE1"]
#WAIT FOR[V.PLC.PROBE1ENA==1]
#MSG[""]

; Select PROBE1 input for using in Probing.
#SELECT PROBE[1]
#FLUSH
$IF [V.G.PRBST1==1]
  #WARNING["PROBE1 SIGNAL TOUCHING"]
  MO
$ENDIF

; Check probe in safe mode feature is activated.
$IF [V.PLC.PROBE1MONIT==0]
  #MSG["PROBE NOT IN SAFE MODE"]
$ENDIF
#RETDSBLK

#COMMENT BEGIN
PLC signals to add in the logic:

PROBE1ENA; PROBE1 is enabled confirmation signal to CNC.
If not used in PLC, it is activated by default.

PROBE1MONIT; It activates PROBE1 in SAFE mode.
If probe is activated in no G100/3 motion, CNC will stop motion and shows an error.
If not used in PLC, it is activated by default.

#COMMENT END
```

Example of a subroutine Sub_Probe_Tool_End.fst supplied by Fagor (to be adjusted by the OEM).

```
#ESBLK
; Deactivate PROBE1 Hardware by PLC output

#RETDSBLK
```

14.

SUBROUTINES.

Subroutines associated with the tool calibration cycle.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.7 Subroutines associated with the part measuring cycle.



The subroutines supplied by Fagor provide basic probe management, so they should be adjusted and configured appropriately by the OEM. Fagor provides the subroutine `Sub_Probe_Tool_Begin.fst` associated with probe input 1 and the subroutine `Sub_Probe_Piece_Begin.fst` associated with probe input 2.

The part measuring cycle, for both the ISO and the editor, has two associated subroutines that the CNC executes before and after the cycle. These subroutines can be used to enable the probe and indicate if it is in safe mode.

Name.	Meaning.
<code>Sub_Probe_Piece_Begin.fst</code>	Subroutines associated with the start of the part measuring cycle.
<code>Sub_Probe_Piece_End.fst</code>	Subroutines associated with the end of the part measuring cycle.

Subroutines and M functions defined in the cycle.

Up to 4 M functions may be defined in each probing cycle to be executed before the cycle and another 4 M functions to be executed afterwards. All these functions may have a subroutine associated with them. When starting the execution, the cycle executes first the subroutine `Sub_Probe_Piece_Begin.fst` and then the M-before functions with their associated subroutines. At the end of the execution, the cycle executes first the M-after functions with their corresponding associated subroutines and then the subroutine `Sub_Probe_Piece_End.fst`.

Management of the PROBE1ENA/PROBE2ENA marks in the subroutine.

These mark indicate that the probe has is enabled. It is recommended that the OEM manages both marks from the subroutine `Sub_Probe_Piece_Begin.fst`, so that the execution waits until the mark is active.

Management of the PROBE1MONIT/PROBE2MONIT marks in the subroutine.

These marks are associated with the safe mode of the probe. It is recommended that the OEM manages both marks from the subroutine `Sub_Probe_Piece_Begin.fst` to warn, should they be deactivated, that the probe is in non-safe mode.

14.7.1 Configure the subroutines.

Name and location of the subroutine.

The name of the subroutine is `Sub_Probe_Piece_Begin.fst` and `Sub_Probe_Piece_End.fst`. Both subroutines must be saved in the folder `..\Mtb\Sub`. All channels use the same subroutines.

Since it is an OEM subroutine with an `.fst` extension, the CNC loads this subroutine into the RAM of the CNC. See "[14.1 Run subroutines from RAM \(extension *.fst\).](#)" on page 617.

14.

SUBROUTINES.

Subroutines associated with the part measuring cycle.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.7.2 Example of a subroutine.

Example of a subroutine Sub_Probe_Piece_Begin.fst supplied by Fagor (to be adjusted by the OEM).

```
#ESBLK
; Activate PROBE 2 Hardware by PLC output.

; Check PROBE is READY with PLC Input from Probe Hardware.
#MSG["WAIT FOR ENABLING PROBE2"]
#WAIT FOR[V.PLC.PROBE2ENA==1]
#MSG[""]

; Select PROBE 2 input for using in Probing.
#SELECT PROBE[2]
#FLUSH
$IF [V.G.PRBST2==1]
  #WARNING["PROBE2 SIGNAL TOUCHING"]
  MO
$ENDIF

; Check probe in safe mode feature is activated.
$IF [V.PLC.PROBE2MONIT==0]
  #MSG["PROBE NOT IN SAFE MODE"]
$ENDIF
#RETDSBLK

#COMMENT BEGIN
PLC signals to add in the logic:

PROBE2ENA; PROBE 2 is enabled confirmation signal to CNC.
If not used in PLC, it is activated by default.

PROBE2MONIT; It activates PROBE2 in SAFE mode.
If probe is activated in no G100/3 motion, CNC will stop motion and shows an error.
If not used in PLC, it is activated by default.

#COMMENT END
```

Example of a subroutine Sub_Probe_Piece_End.fst supplied by Fagor (to be adjusted by the OEM).

```
#ESBLK
; Deactivate PROBE2 Hardware by PLC output

#RETDSBLK
```

14.

SUBROUTINES.

Subroutines associated with the part measuring cycle.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.8 Subroutine associated with the tool change (function T).

The subroutine associated with the tool is executed automatically every time a T function (tool selection) is executed. If the M06 function is included in this subroutine, the tool loading process will commence when the "T" code has been executed.

Characteristics of a turret type magazine.

For a turret magazine, it is recommended that the routine associated with T include the M06 function. When the CNC selects a tool, the routine associated with T executes the function M06 (tool change). If the subroutine associated with T does not include the function M06, when the CNC selects a tool, it internally executes a T# M6 block to make the change.

14.8.1 Configure the subroutines.

Assign the subroutine to the command.

Parameter.	Meaning.
TOOLSUB CHN	Subroutine associated with "T".

[CHN]..... General machine parameter of the channel.

Name and location of the subroutine.

This subroutine must be defined in the general parameter TOOLSUB and saved in the folder ..\Mtb\Sub. If there are several channels, there may be a different subroutine for each channel.

14.

SUBROUTINES.
Subroutine associated with the tool change (function T).

14.9 Subroutine associated with the tool change (function M6).

To properly configure the magazines and the tool change, the routine associated with the tool and the function M06 must be programmed. The M06 function executes the tool change.

It is recommended to set this function in the "M" function table so it executes the subroutine for the tool changer installed on the machine. The M06 function must also be programmed within the subroutine so the CNC prompts the tool manager to start making the tool change.

- When there is no tool magazine, the function M6 behaves like a normal function, as it does not execute anything related to the tool change, therefore, it is not necessary to assign this subroutine.
- For a turret magazine, it is recommended that the routine associated with T include the M06 function. When the CNC selects a tool, the routine associated with T executes the function M06 (tool change). If the subroutine associated with T does not include the function M06, when the CNC selects a tool, it internally executes a T# M6 block to make the change.
- For other magazines, it is recommended that this function in the "M" function table be customized, so it executes the subroutine for the tool changer installed on the machine. The M06 function must also be programmed within the subroutine so the CNC prompts the tool manager to start making the tool change.

Considerations and recommendations.

The management of the tool change should be included in the subroutine associated with the M06 and leave the control of the external devices up to the PLC. Use the auxiliary functions to govern the various devices (magazine rotation, magazine movement, tool changer arm, etc) from the M06 subroutine.

14.9.1 Configure the subroutines.

This subroutine must be defined in the "M" function table and saved in the folder ..\Mtb\Sub. See "[14.11 Subroutines associated with M functions.](#)" on page 630.

14.

SUBROUTINES.
Subroutine associated with the tool change (function M6).



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.10 Subroutine associated with function G74.

Function G74 (machine home search) can be programmed in 2 ways, either by indicating the axes and the order in which the axes are referenced or by programming only G74 (without axes). When executing a block that only contains the G74 function (without axes), it calls the subroutine associated with this function. This subroutine must contain the axes and the order in which the axes and the spindle must be referenced.

This subroutine is also called when homing the axis in JOG mode without selecting the axes.

14.10.1 Configure the subroutines.

Assign the subroutine to the function.

Parameter.	Meaning.
REFPSUB CHN	Subroutine associated with function G74.

[CHN]..... General machine parameter of the channel.

Name and location of the subroutine.

This subroutine must be defined in the general parameter REFPSUB (by default, sub_ref.nc) and saved in the folder ..\Mtb\Sub. If there are several channels, there may be a different subroutine for each channel.

14.

SUBROUTINES.
Subroutine associated with function G74.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.11 Subroutines associated with M functions.

The "M" functions may have an associated subroutine that the CNC will execute instead of the function. If, within a subroutine associated with an "M" function, the same "M" function is programmed, the CNC will execute this, but not its associated subroutine.

14.11.1 Configure the subroutines.

M FUNCTION TABLE : M VALUES				
MNUM	SYNCHTYPE	MPROGNAME	MTIME	MPLC
12	Without Sync	sub_M12.nc	500	Yes
15	Before-Before		0	Yes
33	Before-After	sub_M33.nc	100	No
50	After-After		0	No

Assign the subroutine to the function.

Parameter.	Meaning.
MPROGNAME FUNM	Name of subroutine associated with M function.

[FUNM] Machine parameters; M function table.

Name and location of the subroutine.

These subroutines must be defined in the "M" function table, in the MPROGNAME field and saved in the folder ..\Mtb\Sub.

14.11.2 Variables.

The following variables may be accessed from (PRG) the part-program and from the MDI/MDA mode, PLC and (INT) from an external application. For each variable, the table indicates whether the access is reading (R) or writing (W).

Variable.	PRG	PLC	INT
(V.)MPM.MPROGNAME[pos] Name of subroutine associated with M function. Units: Text.	-	-	R

Syntax.

·pos· Position inside the "M" function table.

MPM.MPROGNAME[12] Position ·12· of the "M" function table.

14.

SUBROUTINES.
Subroutines associated with M functions.

14.12 OEM subroutines associated with functions G180 to G189 / G380 to G399.

The CNC allows the machine manufacturer to 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 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.

14.12.1 Configure the subroutines.

Assign the subroutines to the functions.

Parameter.		Meaning.
OEMSUB (G180)	CHN	Subroutines associated with functions G180 through G189.
..		
OEMSUB (G189)		
OEMSUB (G380)	CHN	Subroutines associated with functions G380 through G399.
..		
OEMSUB (G399)		

[CHN]..... General machine parameter of the channel.

Name and location of the subroutine.

These subroutines must be defined in the general parameters OEMSUB and saved in the folder ..\Mtb\Sub. These subroutines can be loaded into RAM (extension fst).

14.12.2 Variables.

The following variables may be accessed from (PRG) the part-program and from the MDI/MDA mode, PLC and (INT) from an external application. For each variable, the table indicates whether the access is reading (R) or writing (W).

Variable.	PRG	PLC	INT
(V.)[ch].MPG.OEMSUB1	-	-	R
..			
(V.)[ch].MPG.OEMSUB10	-	-	R
OEM subroutines associated with functions G180 through G189.			
Units: -.			
(V.)[ch].MPG.OEMSUB11	-	-	R
..			
(V.)[ch].MPG.OEMSUB30			
OEM subroutines associated with functions G380 through G399.			
Units: -.			

14.13 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.

14.

SUBROUTINES.
Interruption subroutines.

14.13.1 Configure the subroutines.

Assign the subroutines to the functions.

Parameter.	Meaning.
INT1SUB CHN	Interruption subroutines.
..	
INT4SUB	

[CHN]..... General machine parameter of the channel.

Name and location of the subroutine.

These subroutines must be defined in the general parameters INIT1SUB to INIT4SU and saved in the folder ..\Mtb\Sub. If there are several channels, there may be a different subroutine for each channel.

14.13.2 Variables.

The following variables may be accessed from (PRG) the part-program and from the MDI/MDA mode, PLC and (INT) from an external application. For each variable, the table indicates whether the access is reading (R) or writing (W).

Variable.	PRG	PLC	INT
(V.)[ch].MPG.INT1SUB (V.)[ch].MPG.INT2SUB (V.)[ch].MPG.INT3SUB (V.)[ch].MPG.INT4SUB Interruption subroutines. Units: Text.	-	-	R

Syntax.

·ch· Channel number.

[2].MPG.INT1SUB Channel ·2·.

14.14 Subroutine associated with the command #INITIALSUB.



Subroutine specifically for the laser model. The subroutines supplied by Fagor provide basic laser management, so they should be configured appropriately by the OEM. The OEM is responsible for ensuring that the subroutine complies with all safety aspects relating to handling the laser.

The command #INITIALSUB has an associated subroutine, which is executed by the CNC along with the command. This command specifically for laser machines, where this subroutine must perform the initial laser operations.

14.14.1 Configure the subroutines.



If this version is updated, the CNC behaves as follows.

- For the two first two installation options, the CNC will only copy the subroutines in the folder ..\Mtb\Sub if they do not already exist.
- In the third installation option, "rename previous version and install it completely", the CNC always updates the subroutines.

Assign the subroutine to the command.

Parameter.	Meaning.
INITIALSUB CHN	Subroutine associated with the command #INITIALSUB.

[CHN]..... General machine parameter of the channel.

Name and location of the subroutine.

This subroutine must be defined in the general parameter INITIALSUB (by default, Initialsub.fst) and saved in the folder ..\Mtb\Sub. If there are several channels, there may be a different subroutine for each channel.

Since it is an OEM subroutine with an .fst extension, the CNC loads this subroutine into the RAM of the CNC. See "14.1 Run subroutines from RAM (extension *.fst)." on page 617.

14.14.2 Variables.

The following variables may be accessed from (PRG) the part-program and from the MDI/MDA mode, PLC and (INT) from an external application. For each variable, the table indicates whether the access is reading (R) or writing (W).

Variable.	PRG	PLC	INT
(V.)[ch].MPG.INITIALSUB Subroutine associated with the command #INITIALSUB. Units: Text.	-	-	R

Syntax.

·ch· Channel number.

[2].MPG.INITIALSUB Channel ·2·.

14.15 Subroutine associated with the command #PIERCING.



Subroutine specifically for the laser model. The subroutines supplied by Fagor provide basic laser management, so they should be configured appropriately by the OEM. The OEM is responsible for ensuring that the subroutine complies with all safety aspects relating to handling the laser.

The command #PIERCING executes the associated subroutine after updating the piercing variables ((V.) TTPIR.name). This command is specifically for laser machines, where this subroutine must perform the initial piercing operations.

Management of the PIERCING mark in the subroutine.

The OEM must manage the PIERCING mark from this subroutine. At the start of the subroutine (or when the OEM deems it necessary), the OEM must activate the PIERCING mark (assigning it the value 1) to indicate to the PLC that it must start the piercing operation. At the end of the subroutine (or when the OEM deems it necessary), the OEM must deactivate this mark (assigning it the value 0) to indicate to the CNC that it has finished the piercing operation.

- The instruction #PLC makes it possible to manage a PLC mark from the part or subroutine program, without interrupting the block preparation.
- This mark shows the state of the variable V.PLC.PIERCING. The OEM can also use this variable in the subroutine associated with the piercing (by default, Piercing.fst) to modify the status of this mark. Entering this variable interrupts the block preparation.

14.15.1 Configure the subroutines.



If this version is updated, the CNC behaves as follows.

- For the two first two installation options, the CNC will only copy the subroutines in the folder ..Mtb\Sub if they do not already exist.
- In the third installation option, "rename previous version and install it completely", the CNC always updates the subroutines.

Assign the subroutine to the command.

Parameter.	Meaning.
PIERCING CHN	Subroutine associated with the command #PIERCING.

[CHN]..... General machine parameter of the channel.

Name and location of the subroutine.

This subroutine must be defined in the general parameter PIERCING (by default, Piercing.fst) and saved in the folder ..Mtb\Sub. If there are several channels, there may be a different subroutine for each channel.

Since it is an OEM subroutine with an .fst extension, the CNC loads this subroutine into the RAM of the CNC. See "[14.1 Run subroutines from RAM \(extension *.fst\).](#)" on page 617.

14.

SUBROUTINES.
Subroutine associated with the command #PIERCING.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.15.2 Variables.

The following variables may be accessed from (PRG) the part-program and from the MDI/MDA mode, PLC and (INT) from an external application. For each variable, the table indicates whether the access is reading (R) or writing (W).

Variable.	PRG	PLC	INT
(V.)[ch].MPG.PIERCING Subroutine associated with the command #PIERCING. Units: Text.	-	-	R

Syntax.

·ch· Channel number.

[2].MPG.PIERCING Channel ·2·.

14.

SUBROUTINES.

Subroutine associated with the command #PIERCING.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.16 Subroutine associated with the command #CUTTING ON and #CUTTING OFF.



Subroutines specifically for the laser model. The subroutines supplied by Fagor provide basic laser management, so they should be configured appropriately by the OEM. The OEM is responsible for ensuring that the subroutine complies with all safety aspects relating to handling the laser.

The command #CUTTINGON executes the associated subroutine after updating the cutting variables ((V.)TTCUT.name). The command #CUTTINGOFF executes the associated subroutine. Both commands are specifically for laser machines, where these subroutines must perform the initial associated cutting operations.

Management of the CUTTING mark in the subroutine.

The OEM must manage the CUTTING mark from these subroutines. At the start of the associated subroutine #CUTTING ON (or when the OEM deems it necessary), the OEM must activate the CUTTING mark (assigning it a value of 1) to indicate to the PLC that it should start the cutting operation. At the end of the associated subroutine #CUTTING OFF (or when the OEM deems it necessary), the OEM must deactivate this mark (assigning it the value 0) to indicate to the CNC that it has finished the piercing operation.

- The instruction #PLC makes it possible to manage a PLC mark from the part or subroutine program, without interrupting the block preparation.
- This mark shows the state of the variable V.PLC.CUTTING. The OEM can also use this variable in the subroutines associated with cutting (by default, Cuttingon.fst / Cuttingoff.fst) to modify the status of this mark. Entering this variable interrupts the block preparation.

14.16.1 Configure the subroutines.



If this version is updated, the CNC behaves as follows.

- For the two first installation options, the CNC will only copy the subroutines in the folder ..Mtb\Sub if they do not already exist.
- In the third installation option, "rename previous version and install it completely", the CNC always updates the subroutines.

Assign the subroutine to the command.

Parameter.		Meaning.
CUTTINGON	CHN	Subroutine associated with the command #CUTTING ON.
CUTTINGOFF	CHN	Subroutine associated with the command #CUTTING OFF.

[CHN]..... General machine parameter of the channel.

Name and location of the subroutine.

These subroutines associated must be defined for the general parameters CUTTINGON (by default, Cuttingon.fst) and CUTTINGOFF (by default, Cuttingoff.fst). Both subroutines must be saved in the folder ..Mtb\Sub. If there are several channels, there may be a different subroutine for each channel.

Since it is an OEM subroutine with an .fst extension, the CNC loads this subroutine into the RAM of the CNC. See "14.1 Run subroutines from RAM (extension *.fst)." on page 617.

14.

SUBROUTINES.

Subroutine associated with the command #CUTTING ON and #CUTTING OFF.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.16.2 Variables.

The following variables may be accessed from (PRG) the part-program and from the MDI/MDA mode, PLC and (INT) from an external application. For each variable, the table indicates whether the access is reading (R) or writing (W).

Variable.	PRG	PLC	INT
(V.)[ch].MPG.CUTTINGON Subroutine associated with the command #CUTTING ON. Units: Text.	-	-	R
(V.)[ch].MPG.CUTTINGOFF Subroutine associated with the command #CUTTING OFF. Units: Text.	-	-	R

Syntax.

·ch· Channel number.

[2].MPG.CUTTINGON	Channel ·2·.
[2].MPG.CUTTINGOFF	Channel ·2·.

14.

SUBROUTINES.

Subroutine associated with the command #CUTTING ON and #CUTTING OFF.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.17 Subroutine associated with the command #FINALSUB.



Subroutine specifically for the laser model. The subroutines supplied by Fagor provide basic laser management, so they should be configured appropriately by the OEM. The OEM is responsible for ensuring that the subroutine complies with all safety aspects relating to handling the laser.

The command #FINALSUB has an associated subroutine, which is executed by the CNC along with the command. This command is specifically for laser machines, where this subroutine must perform the final laser operations.

14.

SUBROUTINES.
Subroutine associated with the command #FINALSUB.

14.17.1 Configure the subroutines.



If this version is updated, the CNC behaves as follows.

- For the two first two installation options, the CNC will only copy the subroutines in the folder `..\Mtb\Sub` if they do not already exist.
- In the third installation option, "rename previous version and install it completely", the CNC always updates the subroutines.

Assign the subroutine to the command.

Parameter.	Meaning.
FINALSUB CHN	Subroutine associated with the command #FINALSUB.

[CHN]..... General machine parameter of the channel.

Name and location of the subroutine.

This subroutine must be defined in the general parameter FINALSUB (by default, `Finalsub.fst`) and saved in the folder `..\Mtb\Sub`. If there are several channels, there may be a different subroutine for each channel.

Since it is an OEM subroutine with an `.fst` extension, the CNC loads this subroutine into the RAM of the CNC. See "[14.1 Run subroutines from RAM \(extension *.fst\)](#)" on page 617.

14.17.2 Variables.

The following variables may be accessed from (PRG) the part-program and from the MDI/MDA mode, PLC and (INT) from an external application. For each variable, the table indicates whether the access is reading (R) or writing (W).

Variable.	PRG	PLC	INT
(V.)[ch].MPG.FINALSUB Subroutine associated with the command #FINALSUB. Units: Text.	-	-	R

Syntax.

·ch· Channel number.

[2].MPG.FINALSUB Channel ·2·.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.18 User subroutines associated with functions G500 to 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.

Subroutines supplied by Fagor.

Subroutine.	Meaning.
G500	HSC cancellation.
G501	HSC activation for roughing operations.

14.18.1 Configure the subroutines.



If this version is updated, the CNC behaves as follows.

- For the two first two installation options, the CNC will only copy the subroutines in the folder `..\Mtb\Sub` if they do not already exist.
- In the third installation option, "rename previous version and install it completely", the CNC always updates the subroutines.

Name and location of the subroutine.

The subroutines associated with functions G500 to G599 will have the same name as the function (without an extension) and they will be saved in the folder `..\Users\Sub`.

- G500 will have subroutine G500 associated with it.
- G501 will have subroutine G501 associated with it.
- ...
- G599 will have subroutine G599 associated with it.

14.19 User subroutines associated with functions G8000 to G8999.

The CNC lets the user define up to 1000 subroutines, common to all channels and associated with functions G8000 through G8999, so when the CNC executes one of these functions, it will execute its associated subroutine. These 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.

Some of these subroutines have a function pre-assigned by Fagor, being ProGTL3 language macros. If one of these G functions is programmed, the CNC executes the corresponding macro and not the subroutine.

14.

SUBROUTINES.
User subroutines associated with functions G8000 to G8999.

14.19.1 Configure the subroutines.



If this version is updated, the CNC behaves as follows.

- *For the two first two installation options, the CNC will only copy the subroutines in the folder ..Mtb\Sub if they do not already exist.*
- *In the third installation option, "rename previous version and install it completely", the CNC always updates the subroutines.*

Name and location of the subroutine.

The subroutines associated with functions G8000 to G8999 will have the same name as the function (without an extension) and they will be saved in the folder ..\Users\Sub.

G8000	The subroutine G8000 will be associated.
G8001	The subroutine G8001 will be associated.
...	
G8999	The subroutine G8999 will be associated.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.20 Assistance for subroutines.

14.20.1 Subroutine help files.

Help files may be associated with each OEM subroutine (G180, G380, etc), user subroutine (G500, G800, etc) and each global subroutine called upon using #MCALL or #PCALL and they will be displayed while editing.

The help window is displayed while editing, after a blank space or tab following a G function or following the name of the subroutine. The help window is only informative, it cannot be accessed with the cursor nor browse through it. 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.

When the help file is displayed, its text may be inserted into the part-program using the [INS] key.

Edit the 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.

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. Also, since the text of the help file can be inserted into the program ([INS] key), the following is recommended.

- 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.

Help file example for a subroutine.

```
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
```

Name and location of files.

Name of the help files.

The name of the files must follow the following rule:

Subroutine.	Name of the help files.
G180-G189 G380-G399 G500-G599 G800-G899 G8000-G8999	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> .

14.

SUBROUTINES.

Assistance for subroutines.

Where to save the help files.

The machine manufacturer can save the help files in the folders `..\Mtb\Sub\Help` and `..\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\Sub\Help` and `..\Users\Sub\Help\{language}`.

The CNC looks for the files in the following order and displays the first that it finds, this is the reason why it is recommended that the user does not define subroutines and/or help files with the same name as those of the OEM. If there are no help files, the CNC will not show any help and it will not display an error.

```

..\Users\Sub\Help\{idioma}
..\Users\Sub\Help
..\Mtb\Sub\Help\{idioma}
..\Mtb\Sub\Help\

```



For versions V1.60 (8060) y V5.60 (8065), the CNC no longer searches for help files in the following folders.

```

..\Users\Session\Help\{idioma}
..\Mtb\Sub\Help\{idioma}.
..\Users\Help\{idioma}.

```



For versions prior to V1.60 (8060) and V5.60 (8065), the CNC first searched for the help files in the manufacturer folders and then in the user folders. For these and later versions, it does so in the opposite order.

14.20.2 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".

List of subroutines.

The list of subroutines must be in a text (txt) file. The file must be edited so each line is the name of a possible subroutine to be called.

Example of a file with a list of subroutines.

```
C:\CNC8070\USERS\SUB\FAGOR.NC
SUBROUTINE.NC
EXAMPLE.NC
POSITIONING.NC
```

Name and location of files.

The name of the file should be *pcall.txt*.

Where to save the list of subroutines.

The machine manufacturer must save the *pcall.txt* file in the folder `..\Mtb\Sub\Help`. 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\Sub\Help`.

The CNC looks for the help files in both files; 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.



For versions V1.60 (8060) y V5.60 (8065), the CNC no longer searches for help files in the following folders.

```
..\Users\Session\Help\{idioma}
..\Mtb\Sub\Help\{idioma}.
..\Users\Help\{idioma}.
```

14.

SUBROUTINES.
Assistance for subroutines.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

14.

SUBROUTINES.

Assistance for subroutines.

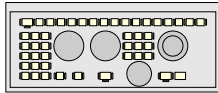


FAGOR AUTOMATION

CNC 8070

(REF: 1709)

15.1 Management of several keyboards.

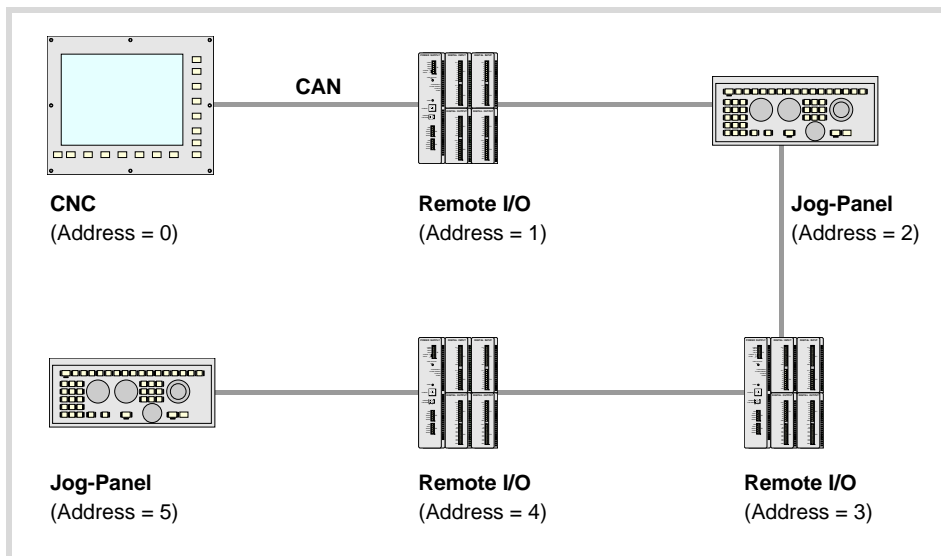


Since the CNC may have several jog panels connected to it, the CNC allows configuring the behavior of each one of them with respect to the channels. A jog panel may be associated with a particular channel (for example, in a TT lathe) or always with the active channel (for example, a machine with two work stations where both jog panels work the same way).

15.1.1 How to configure the feature.

Identify the order of the operator panels in the bus.

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.



Address	Element.	Order number.
0	CNC	First operator panel of the system.
1	Remote I/O	
2	Jog-Panel	Second operator panel of the system.
3	Remote I/O	
4	Remote I/O	
5	Jog-Panel	Third operator panel of the system.

Define the number of jog panels and their relationship with the channels.

Machine parameter.	Meaning.
NKEYBD	Number of jog panels.
KEYBD1CH .. KEYBD8CH	List of jog panels of the system.

Parameter NKEYBD sets the number of jog panels of the system. Parameter KEYBDCH shows the table for assigning the jog panels to the channels, where parameter KEYBD1CH corresponds to the first jog panel, KEYBD2CH to the second one and so on.

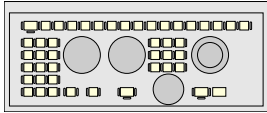
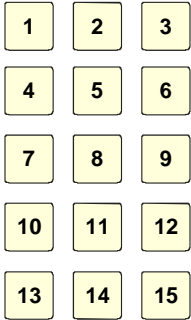
Each jog panel may be assigned to a particular channel or always to the active channel. When a jog panel is assigned to a channel, it is always operative even if the channel is not the active one. When there are several jog panels assigned to the same channel, operations may be carried out from any of them indistinctively.

Configure the jog keys.








Machine parameter.	Meaning.
JOGKEYDEF JOGKEYBD2DEF ... JOGKEYBD8DEF	Tables to configure the jog keys.

These parameters may be used to configure the jog keypad of each panel. Parameter JOGKEYDEF corresponds to the first jog panel, JOGKEYBD2DEF to the second one and so on.

Each parameter contains a table to define the function of each jog key. The CNC always offers 15 parameters; if the jog keypad has fewer keys, the parameters that do not have any keys associated with them will be ignored. The relationship between these parameters and the jog keys is the following.

Hardware module.	Jog keys.
JOG-PANEL 	

The JOG keypad may consist of the following types of keys. Both types of keys may be defined at the same JOG keypad. To define the behavior of each key, assign to them one of the following values.

Keys.	Meaning.
 	Keys to define the axis and the jogging direction. The parameter is set with a value between -1 and +16 (signed). The sign indicates the positive direction (+) or the negative direction (-) and the number corresponds to the logic axis, according to parameter AXISNAME.
 	Keys to define the axis to be jogged. The parameter is set with a value between 1 and 16 (unsigned) that corresponds to the logic axis according to parameter AXISNAME.
 	Keys to define the direction of the movement. The parameter is set with one of the "+" and "-" values to indicate the moving direction.
	Rapid key. The parameter is set with the "R" value.

15.

HARDWARE:
Management of several keyboards.



FAGOR AUTOMATION

CNC 8070

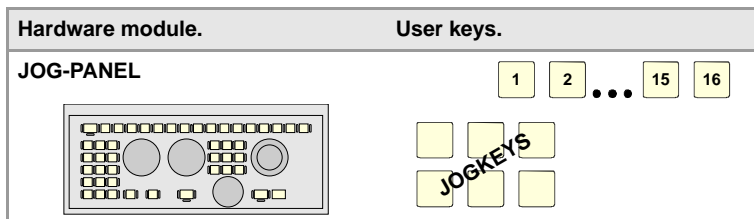
(REF: 1709)

Configure the user keys as jog keys

Machine parameter.	Meaning.
USERKEYDEF USERKEYBD2DEF ... USERKEYBD8DEF	Tables to configure the user keys as jog keys.

These parameters may be used to configure the user keys of each panel as jog keys. Parameter USERKEYDEF corresponds to the first jog panel, USERKEYBD2DEF to the second one and so on.

Each parameter contains a table to define the function of each user key. The CNC always offers 16 parameters; if the jog keypad has fewer keys, the parameters that do not have any keys associated with them will be ignored. The relationship between these parameters and the user keys is the following.



The meaning of this parameter is similar to that of machine parameter JOGKEYBDkbDEF. To define the behavior of each key, assign to them one of the following values:

- They are defined with a value between 1 and 16 (signed) for the keys that define the axis and the direction. The sign indicates the positive direction (+) or the negative direction (-) and the number corresponds to the logic axis, according to parameter AXISNAME.
- They are defined with a value between 1 and 16 (unsigned) for the keys that only define the axis.
- They are defined with the "+" and "-" values for the keys that only define the moving direction.
- They are defined with the "R" value for the rapid key.

The user keys defined this way behave like the jog keys whether they've been defined signed or unsigned and also respect the setting of machine parameter JOGTYPE.

Check the status of the keys.

PLC marks and registers.	Meaning.
KEYBD1 / KEYBD2	These registers are a copy of the map of the keys pressed on the last keyboard used.
KEYBD1_1 / KEYBD2_1 .. KEYBD1_8 / KEYBD2_8	These registers indicate (bit=1) which key has been pressed on each operator panel.

Registers KEYBD1 y KEYBD2 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). If there is only one keyboard, these registers coincide with KEYBD1_1 and KEYBD2_1. When there are several keyboards, the contents of these registers are not always the same as KEYBD1_1 and KEYBD2_1; therefore, they may be used indistinctively.

Registers KEYBD1_n and KEYBD2_n 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.

15.

HARDWARE:
Management of several keyboards.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Disable the keys.

PLC marks and registers.	Meaning.
KEYDIS1 to KEYDIS3	These registers inhibit (bit=1) the keys and the switches on all operator panels at the same time.
KEYDIS1_1 to KEYDIS3_1 .. KEYDIS1_8 to KEYDIS3_8	These registers inhibit (bit=1) the keys and the switches on each operator panel.

Registers KEYDIS1, KEYDIS2 and KEYDIS3 inhibit (bit=1) the keys and the switches on all operator panels at the same time.

Registers KEYDIS1_n, KEYDIS2_n and KEYDIS3_n 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.

When selecting one of the inhibited positions of the feedrate override, the CNC will take the value of the lowest position allowed. If all of them are inhibited, it will take the value of 0%.

Manage the LED's (lamps) of the keys.

PLC marks and registers.	Meaning.
KEYLED1 / KEYLED2	These registers control the LED's (lamps) of the keys of all the operator panels at the same time.
KEYLED1_1 / KEYLED2_1 .. KEYLED1_8 / KEYLED2_8	These registers control the LED's (lamps) of the keys of each operator panel.

Registers KEYLED1 and KEYLED2 control the LED's (lamps) of the keys of all the operator panels at the same time.

Registers KEYLED1_n and KEYLED2_n 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.

Change the default behavior of the keyboards with respect to the channels.

PLC marks and registers.	Meaning.
KEYBD1CH to KEYBD8CH	These registers may be used to change the default behavior of the keyboards with respect to the channels, set by machine parameters.

These registers may be used to change the default behavior of the keyboards with respect to the channels, set by machine parameters. These registers can associate a jog panel with a particular channel, always with the active channel or restore the configuration defined in the machine parameters.

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.

15.

HARDWARE:
Management of several keyboards.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Disable the jog panels.

PLC marks and registers.	Meaning.
PANELOFF PANELOFF1 to PANELOFF8	If the PLC sets this one of these marks high (=1) the CNC disables the corresponding jog panel.

If the PLC activates one of these marks the CNC disables the corresponding jog panel. Each mark may be used to disable the jog panel integrated into the CAN bus; the rest of the elements of the bus are not affected. When the keyboard and the operator panel make up a single element, this mark only disables the jog panel.

There is one mark for each keyboard or jog panel. The PANELOFF1 mark disables the first one of the bus, the PANELOFF2 mark the second one and so on. Both PANELOFF and PANELOFF1 are valid for the first jog panel.

Address	Element	PLC mark
0	CNC	
1	Remote (I/O) group	
2	Jog panel	PANELOFF1
3	Remote (I/O) group	
4	Remote (I/O) group	
5	Jog panel	PANELOFF2

15.

HARDWARE:
Management of several keyboards.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

15.1.2 Operation of the jog panels.

When enabling a keyboard (PANELOFF mark) and every time there is a keyboard change (press a key, change a switch, etc.), the channel assumes the status of the keys and the position of the switches of the new keyboard with the active values. If necessary the PLC routine must include the proper restrictions to avoid sudden feedrate changes due to different positions of the feedrate override switch on the different keyboards. It is also recommended to manage the user keys using up and down flanks to avoid the possible activation of devices when changing keyboards.

Every time the active keyboard is changed, the new keyboard updates at the PLC the information on all the marks and registers associated with the keyboard itself (KEYBD1_n, KEYBD2_n, KEYDIS1_n a KEYDIS3_n, KEYLED1_n, KEYLED2_n) and the generic ones associated with all the keyboards (KEYBD1, KEYBD2, KEYDIS1 to KEYDIS3, KEYLED1, KEYLED2).

Behavior of the keys and the switches.

The keys from the jog panel.

The keys of each jog panel are only effective for each channel with which the keyboard is associated; whether it is the active channel or not. The following keys have a different behavior.

- The [RESET] key affects the channel associated with the keyboard where the key has been pressed and the channels that belong to its group.
- The keys to select the work mode (automatic, jog, etc.) only affect the active channel.

Feedrate override selector.

If the keyboard is associated with a particular channel, an override change only affects that channel, even if it belongs to a group. If the keyboard is associated with the active channel, an override change affects all the channels of the system at the same time, whether they belong to a group or not.

15.

HARDWARE.

Management of several keyboards.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

15.2 Configure a handwheel as "feed handwheel".

Usually, when machining a part for the first time, the feedrate is controlled with the switch on the operator panel. The "feed handwheel" allows using one of the handwheels of the machine to control that feedrate depending on how fast the handwheel is turned.

The "feed handwheel" must be managed by the PLC. Usually, this feature is turned on and off using an external push button or key configured for that purpose.

Know the number of pulses sent by the handwheel.

The following variable reads the number of handwheel pulses.

Variable.	Meaning.
(V.)G.HANDP[hw]	Number of pulses sent by the handwheel since the system was started up.

PLC routine.

The PLC routine must consider the following elements.

- Inhibit via PLC all the positions of the feedrate override switch.
- Detect how much the handwheel has turned (reading - counting - the pulses received).
- Set via PLC the feedrate override depending on the handwheel pulses.

Example of a PLC routine. The machine has a push button to enable and disable the "feed handwheel" (input I71) and the feedrate control is handled by the second handwheel.

```

; Resources used in the PLC program.
; I71 _____ Push button to enable and disable the "feed handwheel"
; R100 _____ Total number of handwheel pulses.
; R101 _____ Number of handwheel pulses until previous reading.
; R102 _____ Number of handwheel pulses since previous reading.
; R103 _____ Calculated percentage of feedrate.
; M1000 _____ "Feed handwheel" on.
; M1001 _____ Auxiliary mark.

CY1
() = ERA R101
    Initialize the register that contains the reading (counting) of handwheel pulses.
END
PRG
DFU I71 = CPL M1000
    Every time the button associated with the "feed handwheel" is pressed, complement
    mark M1000.
NOT M1000
= AND KEYDIS3 $FF800000 KEYDIS3
= JMP L101
    If the "feed handwheel" is not off, inhibit all the positions of the feedrate override
    switch and resume the execution of the program.
M1000 = MSG1
    If the "feed handwheel" is on, show a message.

DFU CLK100
= CNCRD (G.HANDP[2], R100, M1001)
= SBS R101 R100 R102
= MOV R100 R101
= MLS R102 3 R103
= OR KEYDIS3 $7FFFFFFF KEYDIS3
    If the "feed handwheel" is on and an up flank occurs at the clock CLK100, the PLC
    saves in register R100 the handwheel pulses; calculates in register R102 the pulses
    received since the previous reading (count); updates register R101 for the next
    reading; calculates in register R103 the right feedrate override percentage; inhibits
    all the positions of the feedrate override switch (KEYDIS3).
    
```

15.

HARDWARE:

Configure a handwheel as "feed handwheel".



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

15.

HARDWARE.

Configure a handwheel as "feed handwheel".

```
CPS R103 LT 0 = SBS 0 R103 R103
```

```
CPS R103 GT 120 = MOV 120 R103
```

Adjust the value of register R103; ignore the handwheel turning direction (sign) and limit the value to 120%.

```
DFU CLK100
```

```
= CNCWR (R103,PLC.FRO, M1001)
```

Use the up flank at the clock CLK100 to set the calculated feedrate override value.

```
L101
```

```
END
```



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

15.3 Assigning a help text to the graphic softkeys and to the CNC status icon.

The help texts are defined in the "SoftkeyHelper.txt" file and there could be a file per language, they are saved in the "..\MTB\data\Lang" folder. The file is in text format and those texts may be displayed and edited (and thus translated into other languages) using any text editor.

Help text for the graphic softkeys.

The file is divided into sections, one per operating mode, each one of them contains the list of softkeys and the help text associated with it. For example, for the automatic mode.

```
[AUTOMATIC MODE]
AUTO1=Select program
AUTO2=Prg in simulation
AUTO3=Inspection
AUTO4=Quit inspection
AUTO5=Reposition
AUTO6=Reposition block start
AUTO7=Execution mode
AUTO8=Cancel block
AUTO9=Graphics
```

Help text for the CNC status icon.

In the "SoftkeyHelper.txt" file it is also possible to assign a help text to the icons that show the CNC status, at the top of the screen.



These help texts are defined in the [HMI] section.

```
[HMI]
HMI_NOREADY=
HMI_READY=
HMI_INEXECUTION=
HMI_INTERRUPTED=
HMI_INERROR=
```

15.

HARDWARE: Assigning a help text to the graphic softkeys and to the CNC status icon.

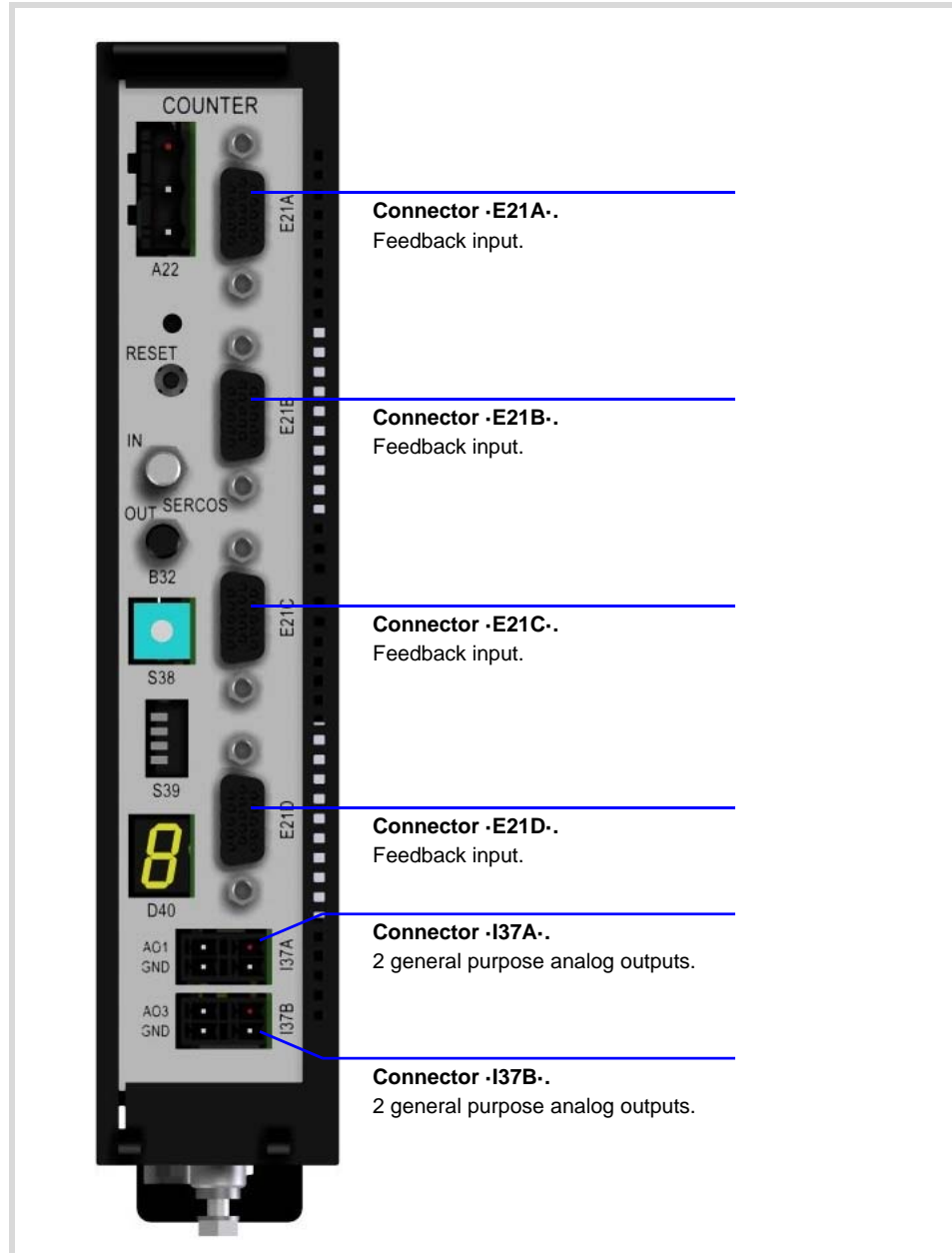
15.4 Remote module RCS-S.

The "RCS-S" counter may be used to get extra feedback (counter) inputs and analog outputs (remote I/O). These modules are connected to the central unit via Sercos-II bus and they are just another bus within the bus.

Module	Description.
RCS-S-4F4AO	4 Feedback inputs. - Incremental TTL, differential TTL and Vpp signal. - SSI, FeeDat, EnDat 2.1 and EnDat 2.2 protocols. 4 general purpose ± 10 V analog outputs (16 bits).

15.

HARDWARE:
Remote module RCS-S.



Connector -E21A-.
Feedback input.

Connector -E21B-.
Feedback input.

Connector -E21C-.
Feedback input.

Connector -E21D-.
Feedback input.

Connector -I37A-.
2 general purpose analog outputs.

Connector -I37B-.
2 general purpose analog outputs.



The FeeDat / EnDat support for the RCS-S module is available from the software version v2.01 and hardware version v2.01.

15.4.1 Configure the module like as a node of the Sercos bus.

The Sercos counter is, like the drives, a node type of the Sercos ring. In terms of parameter setting, Sercos counters have dual identification, physical and logic, as described next.

Physical identification of the module inside the bus.

Each counter must be identified by a number (Sercos ID) using the thumbwheel located on its front plate. That identifier must be unique in the ring, the number is sequential from 1 to n and without skipping any and "n" being the total number of Sercos nodes (including drives and counters).

Logic identification of the module inside the bus.

Parameter.	Meaning.
NSERCOUNT	Number of Sercos counter nodes in the bus.
SERCOUNTID	Thumbwheel identifiers associated with Sercos counters.

The SERCOUNTID table of the general machine parameters allows linking the logic number of the Sercos counters (1..8) to the physical number set by the thumbwheel of each node (enter in each position of the Sercos ID table associated with that logic position of the counter).

Although the Sercos ring may have up to 32 nodes, the Sercos counters always occupy positions from 1 to 15.

15.4.2 Configure the analog outputs.

The analog output is configured via machine parameters. Each axis has the following machine parameters to configure it.

Parameter.	Meaning.
ANAOUTTYPE	Type of analog output associated with the axis.
ANAOUTID	Number of the analog output associated with the axis.

Configure the analog outputs.

The velocity command for an analog axis may be taken from an analog output of the remote modules or of a Sercos drive, or of a Sercos counter. It is selected with parameters ANAOUTID and ANAOUTID.

Parameter.	Value.
ANAOUTTYPE	RCS-S module RCS-S (Sercos counter).
ANAOUTID	1 - 4.

Managing the analog command from the Sercos counter.

In this case, parameter ANAOUTID indicates the number of the analog output used for the command. The Sercos counters are numbered according to their logic order (parameter SERCOUNTID). The analog outputs of the first module will be 1 through 4, those of the second module 5 through 8 and so on.

15.

HARDWARE:
Remote module RCS-S.

FAGOR 

FAGOR AUTOMATION

CNC 8070

(REF: 1709)

15.4.3 Configuration of the feedback inputs.

The feedback inputs are configured via machine parameters. Each axis has the following machine parameters to configure it.

Parameter.	Meaning.
COUNTERTYPE	Type of feedback input for the axis.
COUNTERID	Number of the feedback input for the axis.

Configure the feedback input.

The axis feedback input may be located in the remote modules, at the free feedback of a Sercos drive, in a Sercos counter or at the local feedback inputs of the central unit. It is selected with parameters COUNTERTYPE and COUNTERID.

Parameter.	Value.
COUNTERTYPE	RCS-S module RCS-S (Sercos counter).
COUNTERID	1 - 32.

Feedback input for the Sercos counter.

Parameter COUNTERID indicates the number of the feedback input being used. The Sercos counters are numbered according to their logic order (parameter SERCOUNTID). The feedback inputs of the first module will be 1 through 4, those of the second module 5 through 8 and so on.

15.4.4 Configure the feedback input for a handwheel.

The handwheel is configured via machine parameters of the jog mode. Each handwheel has the following machine parameters to configure it.

Parameter.	Meaning.
COUNTERTYPE	Type of handwheel feedback input.
COUNTERID	Feedback input associated with the handwheel.

Configure the feedback input.

This parameter indicates the location of the handwheel feedback input; at the remote modules, at the keyboard or at the local feedback inputs of the central unit or in a Sercos counter. It is selected with parameters COUNTERTYPE and COUNTERID.

Parameter.	Value.
COUNTERTYPE	RCS-S module RCS-S (Sercos counter).
COUNTERID	1 - 32.

Feedback input for the Sercos counter.

Parameter COUNTERID indicates the number of the feedback input being used. The Sercos counters are numbered according to their logic order (parameter SERCOUNTID). The feedback inputs of the first module will be 1 through 4, those of the second module 5 through 8 and so on.

15.

HARDWARE:
Remote module RCS-S.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

15.4.5 Disabling the feedback input alarms.

The feedback input alarms of the RCS-S module (Sercos counter) can be deactivated using the drive variables (parameter DRIVEVAR). This option is useful, for example, for multi-axes where it is desired to avoid any non-active axis from reporting warnings that are a consequence of the movement of the active axis (axes/spindle sharing the input signal, but with different resolutions).

Define the variable on the DRIVEVAR table.

In the OEM machine parameters, open the DRIVEVAR table.

Field.	Meaning.
MNEMONIC	Mnemonic assigned to the DRV variable. Access to the variable is made by the following: (V.)DRV.{mnemonic}.{node}
AXIS	Node number of the RCS-S module (SERCOUNTID parameter).
ID	Sercos IDN of the alarm mask of the corresponding input. <ul style="list-style-type: none"> • Sercos IDN 33348 for feedback input 1. • Sercos IDN 33349 for feedback input 2. • Sercos IDN 33350 for feedback input 3. • Sercos IDN 33351 for feedback input 4.
TYPE	These variables must be asynchronous.
MODE	Access to the variables must be written.

Example.

To deactivate the alarms for the second feedback input of the module configured as SERCOUNTID=4.

DRIVEVAR : DATA				
MNEMONIC	AXIS	ID	TYPE	MODE
RCSALARM	4	33349	Asynch ▼	Write ▼

Access from the CNC is V.DRV.RCSALARM.4

Access from the PLC is DRV.RCSALARM.4

Enabling and disabling alarms.

Alarms may be disabled via the PLC by writing (CNCWR) the value of 0 in the defined variable. To re-enable the alarms, rewrite the original value of the mask. Before overriding any alarms, it is recommended that the value of the mask be recorded.

15.

HARDWARE:
Remote module RCS-S.

15.4.6 Parameter setting example.

The system has 2 Sercos counters (nodes 3, 5) and 3 Sercos AXD drives (nodes 1, 2, 4). Plus, the system has the following axes.

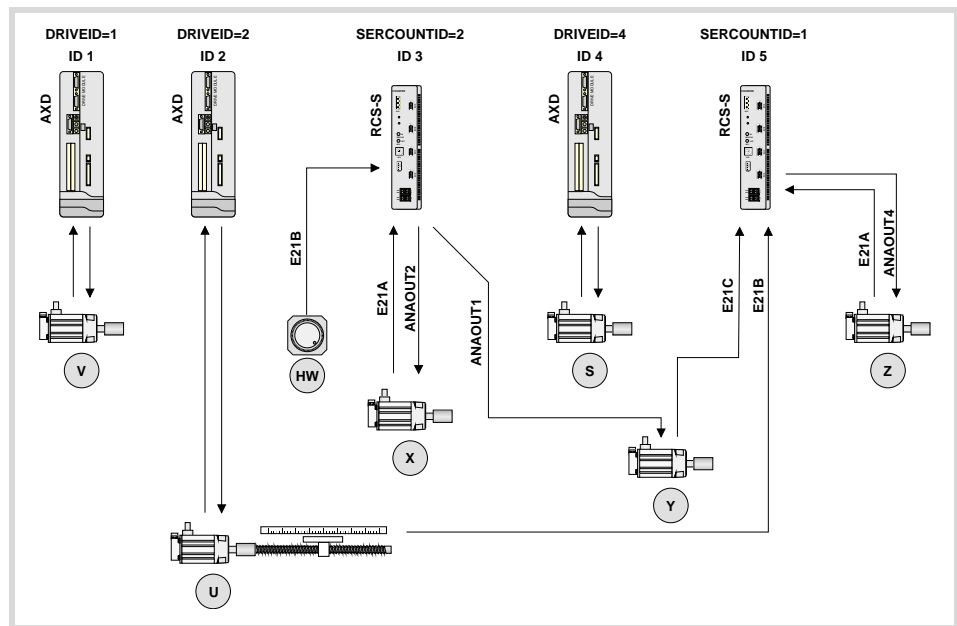
- 3 analog axes X Y Z.
- 1 semi-Sercos axis U.
- 1 Sercos axis V.
- 1 handwheel.
- 1 Sercos spindle S.

The connection is as follows:

NODE 1:	V Sercos axis.	
NODE 2:	U Sercos axis.	
NODE 3:	X analog axis.	E21A / ANAOUT2
	Handwheel 1.	E21B
	Y analog axis.	ANAOUT1
NODE 4:	Sercos spindle S	
NODE 5:	U axis (second feedback)	E21B
	Y analog axis.	E21C
	Z analog axis.	E21A / ANAOUT4

The parameter could be set as follows.

NSERCOUNT	2	Number of Sercos axes in the system.
SERCOUNTID1	5	(No common, but valid)
SERCOUNTID2	3	



15.

HARDWARE:
 Remote module RCS-S.

Parameters SERCOUNTID1 through SERCOUNTID8 set the order (sequence) of the feedback inputs. Since SERCOUNTID1=5 and SERCOUNTID2=3, the first feedback input will be the first one of node 5.

Axis.	Parameter.	Value.
X axis.	DRIVETYPE COUNTERTYPE COUNTERID ANAOUTTYPE ANAOUTID	Analog. Sercos counter. 5. Sercos counter. 6.
Y axis.	DRIVETYPE COUNTERTYPE COUNTERID ANAOUTTYPE ANAOUTID	Analog. Sercos counter. 3. Sercos counter. 5.
Z axis.	DRIVETYPE COUNTERTYPE COUNTERID ANAOUTTYPE ANAOUTID	Analog. Sercos counter. 1. Sercos counter. 4.
U axis.	DRIVETYPE DRIVEID OPMODEP FBACKSRC COUNTERTYPE COUNTERID	Sercos. 2. Speed. External. Sercos counter. 2.
V axis.	DRIVETYPE DRIVEID	Sercos. 1.
Handwheel	MANPG COUNTERTYPE COUNTERID	1. Sercos counter. 6.
Spindle S	DRIVETYPE DRIVEID	Sercos. 4.

15.

HARDWARE:
Remote module RCS-S.

15.5 Number the digital inputs and outputs of the CANopen bus.

By default, the CNC numbers the digital inputs sequentially, according to the order of the modules in the bus. When the bus is CANopen, the CNC allows the digital I/O nodes in logic blocks (virtual modules) to be divided, and then to establish the numbering of the I/Os of each logic block, assigning a base index to the first one.

- In RIO5 and RIOR nodes, each module is a logic block, several modules cannot be grouped in a block or divide a module into blocks. The logic blocks must have the same number and size as the modules physically detected in the bus.
- In the RIO5 nodes, each double module (module with two boards) it counts as two, in other words, two logic blocks.
- In RIO5 nodes and third-party nodes, it is possible to display as many logic blocks as desired, both in number and in size.

Configure the digital inputs.

The digital inputs and outputs are configured via machine parameters.

Parameter.	Meaning.
NDIMOD	Total number of digital input logic blocks. Possible values: From 0 to 42 (by default; 0).
DIMODADDR	Base addresses of the digital input logic blocks.
MNEMONIC	Mnemonic of the logic block.
NODE	Node number of the header to which the logic block belongs. Possible values: From 1 to 126 (by default; 0).
BLOCK	Number of the logic block. Possible values: From 1 to 64 (by default; 0).
ADDRESS	Base address of the logic block. Possible values: From 1 to 1017 (by default, the first valid value).
NDI	Number of digital inputs of the module. Possible values: From 8 to 1024 (by default; 0).

The configuration must meet the following requirements. If the parameters are not set according to these requirements, the CNC will ignore it and it will number the digital inputs sequentially according to the order of the modules in the bus.

- The parameters of all I/O nodes of the bus must be set; no node can be left undefined.
- The parameters of all the resources of a node must be set; no I/O can be left without a number.
- The maximum number of logic blocks in the system will be 64, both for digital inputs and digital outputs.
- The number of I/Os in a logic block must be other than zero and a multiple of 8 (8, 16, 24, 32, ...1024).
- The values of the base index must be comply with the formula " $8n + 1$ " (i.e. 1, 9, 17, 25, etc.).
- The blocks may be distributed at will within the range of resources managed by PLC (1...1024). The PLC admits blanks, but the blocks cannot overlap; in other words, a PLC digital input or output can only belong to a single node-module of the bus.

15.

HARDWARE:

Number the digital inputs and outputs of the CANopen bus.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Configure the digital outputs.

The digital inputs and outputs are configured via machine parameters.

Parameter.	Meaning.
NDOMOD	Total number of digital output logic blocks. Possible values: From 0 to 42 (by default; 0).
DOMODADDR	Base addresses of the digital output logic blocks.
MNEMONIC	Mnemonic of the logic block.
NODE	Node number of the header to which the logic block belongs. Possible values: From 1 to 125 (by default; 0).
BLOCK	Number of the logic block. Possible values: From 1 to 64 (by default; 0).
ADDRESS	Base address of the logic block. Possible values: From 1 to 1017 (by default, the first valid value).
NDO	Number of digital outputs of the module. Possible values: From 8 to 1024 (by default; 0).

The configuration must meet the following requirements. If the parameters are not set according to these requirements, the CNC will ignore it and it will number the digital outputs sequentially according to the order of the modules in the bus.

- The parameters of all I/O nodes of the bus must be set; no node can be left undefined.
- The parameters of all the resources of a node must be set; no I/O can be left without a number.
- The maximum number of logic blocks in the system will be 64, both for digital inputs and digital outputs.
- The number of I/Os in a logic block must be other than zero and a multiple of 8 (8, 16, 24, 32, ...1024).
- The values of the base index must be comply with the formula " $8n + 1$ " (i.e. 1, 9, 17, 25, etc.).
- The blocks may be distributed at will within the range of resources managed by PLC (1...1024). The PLC admits blanks, but the blocks cannot overlap; in other words, a PLC digital input or output can only belong to a single node-module of the bus.

15.

HARDWARE:

Number the digital inputs and outputs of the CANopen bus.



FAGOR AUTOMATION

CNC 8070

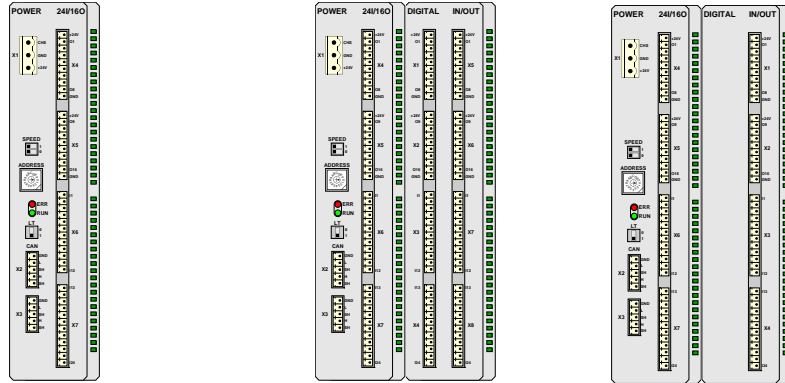
(REF: 1709)

15.5.1 Example. I/O numbering via machine parameters.

Example 1.

A CANopen bus offers the following groups.

- Node 1: Remote RIO5 group with 1 module of 24 digital inputs.
- Node 2: Remote RIO5 group with 3 module of 24 digital inputs each.
- Node 3: Remote RIO5 group with 2 module of 24 digital inputs each.



Group -1-	Group -2-	Group -3-
24 digital inputs.	24 + 48 digital inputs.	24 + 24 digital inputs.
16 digital outputs.	16 + 32 digital outputs.	16 + 16 digital outputs.

In the RIO5 module must be configured as one logic block. Each RIO5 node is to be configured as follows.

- The first RIO5 node as a block of 24 inputs from I1 on.
- The second RIO5 node as three blocks of 24 inputs from I49, I73 and I97 on.
- The third RIO5 node as two blocks of 24 inputs from I257 and I129 on.

Parameter.	Value.	Comment.
NDIMOD	6	In the RIO5 series, each module must be configured as one logic block. Each double module (module with two boards) it counts as two blocks.
DIMODADDR	- - -	See the table below.

DIMODADDR				
MNEMONIC	NODE	BLOCK	ADDRESS	NDI
DIMOD 1	1 (RIO5)	1	1	24
DIMOD 2	2 (RIO5)	2	49	24
DIMOD 3	2 (RIO5)	3	73	24
DIMOD 4	2 (RIO5)	4	97	24
DIMOD 5	3 (RIO5)	5	257	24
DIMOD 6	3 (RIO5)	6	129	24

15.

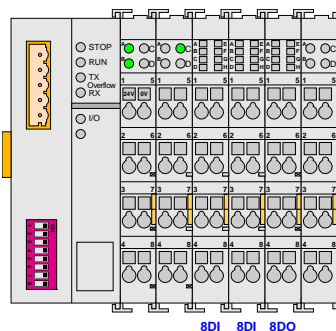
HARDWARE.

Number the digital inputs and outputs of the CANopen bus.

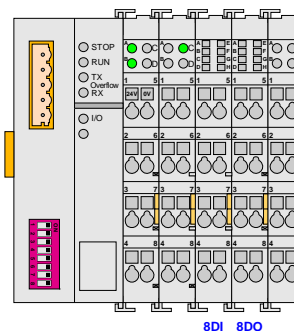
Example 2.

A CANopen bus offers the following groups.

- Node 1: Remote RIOW group with 16 digital inputs and 8 digital outputs.
- Node 2: Remote RIOW group with 8 digital inputs and 8 digital outputs.



Node 1 (RIOW)
16 digital inputs.
8 digital outputs



Node 2 (RIOW)
8 digital inputs.
8 digital outputs

Each RIOW node is to be configured as follows.

- The first RIOW node as 2 blocks, one 8 inputs from I1 on and the other from I33 on. Also a block of 8 digital outputs from O81 on.
- The second RIOW node as a block of 8 inputs from I113 on. Also a block of 8 digital outputs from O49 on.

Parameter.	Value.	Comment.
NDIMOD	3	3 blocks; 2 for the first node and 1 for the second one.
DIMODADDR	- - -	See the table below.

DIMODADDR				
MNEMONIC	NODE	BLOCK	ADDRESS	NDI
DIMOD 1	1 (RIOW)	1	1	8
DIMOD 2	1 (RIOW)	2	33	8
DIMOD 3	2 (RIOW)	3	113	8

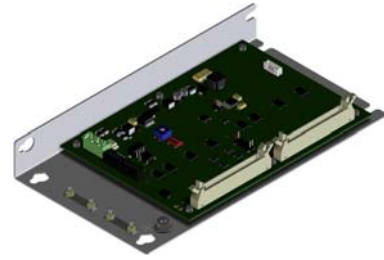
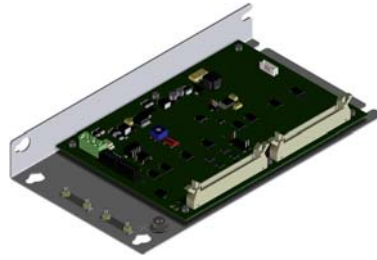
Parameter.	Value.	Comment.
NDOMOD	2	2 blocks, one per each RIOW node.
DOMODADDR	- - -	See the table below.

DOMODADDR				
MNEMONIC	NODE	BLOCK	ADDRESS	NDO
DOMOD 1	1 (RIOW)	1	81	8
DOMOD 2	2 (RIOW)	2	49	8

Example 3.

A CANopen bus offers the following groups.

- Node 1: Remote RIOR group with 48 digital inputs and 32 digital outputs.
- Node 2: Remote RIOR group with 48 digital inputs and 32 digital outputs.



Node 1 (RIOR)
48 digital inputs.
32 digital outputs

Node 2 (RIOR)
48 digital inputs.
32 digital outputs

In the RIOR module must be configured as one logic block. Each RIOR node is to be configured as follows.

- The first RIOR node as a block of 48 inputs from I1 on. Also a block of 32 digital outputs from O1 on.
- The second RIOR node as three blocks of 48 inputs from I97 on. Also a block of 32 digital outputs from O81 on.

Parameter.	Value.	Comment.
NDIMOD	2	2 blocks, one per each RIOR node. In the RIOR series, each module must be configured as one logic block.
DIMODADDR	- - -	See the table below.

DIMODADDR				
MNEMONIC	NODE	BLOCK	ADDRESS	NDI
DIMOD 1	1 (RIOR)	1	1	48
DIMOD 2	2 (RIOR)	2	97	48

Parameter.	Value.	Comment.
NDOMOD	2	2 blocks, one per each RIOR node. In the RIOR series, each module must be configured as one logic block.
DOMODADDR	- - -	See the table below.

DOMODADDR				
MNEMONIC	NODE	BLOCK	ADDRESS	NDO
DOMOD 1	1 (RIOR)	1	1	32
DOMOD 2	2 (RIOR)	2	81	32

15.

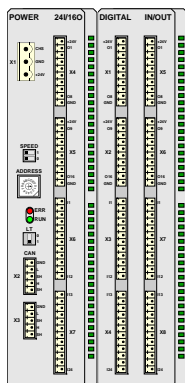
HARDWARE.

Number the digital inputs and outputs of the CANopen bus.

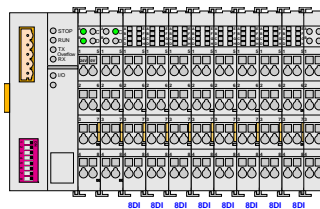
Example 4.

A CANopen bus offers the following groups.

- Node 1: Remote RIO5 group with 3 module of 24 digital inputs each.
- Node 2: Remote RIOW group with 64 digital inputs.



Node 1 (RIO5)
24 + 48 digital inputs.



Node 2 (RIOW)
64 digital inputs.

The first RIOW node is to be configured in 2 blocks, one 24 inputs from I104 on and the other from I200 on.

Parameter.	Value.	Comment.
NDIMOD	5	2 blocks for the RIOW and 3 blocks for the RIO5. In the RIO5 series, each module must be configured as one logic block. Each double module (module with two boards) it counts as two blocks.
DIMODADDR	- - -	See the table below.

DIMODADDR				
MNEMONIC	NODE	BLOCK	ADDRESS	NDI
DIMOD 1	1 (RIO5)	1	1	24
DIMOD 2	1 (RIO5)	2	25	24
DIMOD 3	1 (RIO5)	3	49	24
DIMOD 4	2 (RIOW)	4	104	24
DIMOD 5	2 (RIOW)	5	200	40

15.

HARDWARE: Number the digital inputs and outputs of the CANopen bus.

15.6 Configure the PT100 inputs.

To use the PT100 sensors, the CNC must activate the input to which it is connected. These inputs are deactivated by default and must only be activated if they have a sensor connected to them. Likewise, if the input is active in the machine parameter and there is no sensor connected, the CNC returns the corresponding error.

Configure the PT100 inputs.

General machine parameters.

Parameter.	Meaning.
NPT100	Total number of active PT100 inputs. Possible values: From 0 to 10 (by default; 0).
PT100	Table of active PT100 inputs.
PT100 n	Analog input associated with the PT100 input. Possible values: From 0 to 32 (by default; 0).

How the CNC numbers the PT100 inputs.

When numbering the analog inputs, the CNC considers the PT100 inputs as analog inputs. This way, for numbering, the CNC considers the following:

- The RIO5 modules have 6 analog inputs; four analog inputs (AI1..AI4) plus two PT100 inputs (AI5..AI6).
- The RIOR modules have 4 analog inputs; the two analog inputs (AI1..AI2) and the two PT100 inputs (AI3..AI4).

Example.

In a system with an RIO5 module (node 1) and an RIOR (node 2), the CNC identifies the analog inputs as follows.

- (address = 0) CNC
- (address = 1) RIO5
 - Analog inputs. - 1..4
 - PT100 inputs. - 5..6
- (address = 2) RIOR
 - Analog inputs. - 7..8
 - PT100 inputs. - 9..10

To have 3 PT100 inputs active (the two of the first module and the first one of the second module), the PT100 parameters may be set as follows.

```
NPT100 = 3
PT100 1 = 5
PT100 2 = 6
PT100 3 = 9
```

Enable/disable a PT100 sensor from the PLC.

The PT100 inputs are activated via the machine parameters. In order to be able to temporarily disable the sensor (for example, during a spindle change), the PLC has the following marks.

Parameter.	Meaning.
PT100OFF1 .. PT100OFF20	If the PLC activates one of these marks, the CNC disables the corresponding sensor (parameter PT100 n).

15.

HARDWARE:
Configure the PT100 inputs.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

15.6.1 CNC 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 and writing these variables interrupts block preparation.

Variable.	PRG	Meaning.
(V.)G.ANAI[n]	R	Voltage in volts of analog input [n] or temperature in tenths of a degree if it is a temperature sensor PT100. Units: Volts / tenths of a degree.
(V.)G.ANAO[n]	R/W	[n] output voltage (in volts). Units: Volts.

Syntax of the variables.

n· Number of the analog input or output.

V.G.ANAI[3]	Voltage of the analog input ·3·.
V.G.ANAO[3]	Voltage of the analog output ·3·.

15.

HARDWARE:
Configure the PT100 inputs.

15.

HARDWARE:

Configure the PT100 inputs.



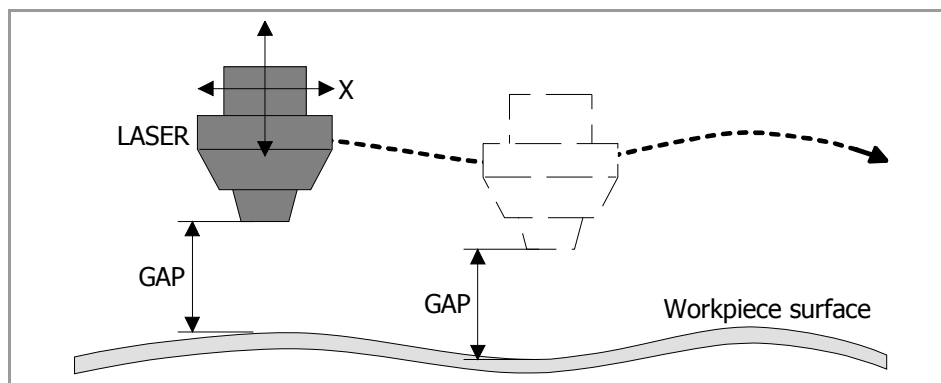
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

16.1 Laser. Gap control.

Gap control makes it possible to maintain a set distance between the laser nozzle and the surface of the sheet. This distance is calculated by a sensor connected to the CNC, so that it offsets the sensor variations on the distance programmed with additional movements in the longitudinal axis.



The CNC allows only one gap control. When trying to activate the gap control from a channel when it is already active in another channel, the CNC will issue the corresponding error message.



For the gap control, the software option "GAP CONTROL" should be available.



The CNC does not allow work when the three functionalities are active simultaneously; leapfrog (#LEAP), gap control (#GAPCTRL), and synchronous switching (#SWTOUT).

The PLC can activate the gap control by executing the corresponding CNC block with the CNCEX command. The instruction executed from the PLC updates the instruction programmed from the CNC and vice-versa.

Sensor calibration.

For correct following by the sensor, it is recommended that the sensor signal be calibrated with the maximum number of points possible.

Programming from the CNC.

The #GAPCTRL ON instruction may be used to activate the gap control. The #GAPCTRL OFF instruction disables the gap control. The programming format is the following; the arguments appear between curly brackets and the optional ones, between angle brackets. Refer to the operating manual for more information.

16.

LASER.

Laser. Gap control.

Activating gap control.

#GAPCTRL ON [<X..C> / <X..C{pos}> <, GAP{gap}>]

<X..C>	Optional (by default , the last programmed value). Axis associated with the sensor.
<X..C{pos}>	Optional (by default , the last programmed value). Axis associated with the sensor and approach coordinate. Units: mm or inches.
<GAP{gap}>	Optional (by default , the last programmed value). Distance (gap) to be kept from the surface. Units: mm or inches.

```
#GAPCTRL ON [Z, GAP 1]
#GAPCTRL ON [Z-11, GAP 2]
#GAPCTRL ON [GAP 5]
#GAPCTRL ON [Z-10]
#GAPCTRL ON
```

Canceling gap control.

#GAPCTRL OFF

```
#GAPCTRL OFF
```

Programming from the PLC.

The PLC can activate the gap control by executing the corresponding CNC block with the CNCEX command.

Programming format.

The programming format is the following; the arguments appear between curly brackets and the optional ones, between angle brackets.

```
{condition} = CNCEX [{block}, {plc_mark} <,{channel}>]
```

{condition}	Condition to execute the CNCEX command.
{block}	Block to be executed.
{plc_mark}	Control mark. The PLC activates the mark when the operation begins and keeps it active until the operation ends.
{channel}	Optional (by default, the first channel). Channel where the PLC must execute the block.

```
{condition} = CNCEX [#GAPCTRL ON [Z, GAP 2], M123]
```

```
{condition} = CNCEX [#GAPCTRL OFF], M123]
```

Execution.

After activating gap control, the axis associated with the sensor moves in programmed feed (if there is no programmed feed, the feed defined in MAXFEED), until detecting the part with the sensor or the programmed coordinate (POS). If the axis reaches the programmed coordinate, without receiving the sensor signal, the CNC will show the corresponding error or not, depending on the GAPTOLCANCEL machine parameter.

The CNC considers that the approach block is complete when the sensor reaches the gap with the tolerance defined in the GAPTOL machine parameter. Once the part is detected, the CNC will keep the defined gap, including if it detects the part when executing #GAPCTRL, without lowering.

During the approach movement, the CNC is able to stop the movement (signals _STOP o _FEEDHOL). In this case, the axis will remain stationary and without gap control until the movement is resumed.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

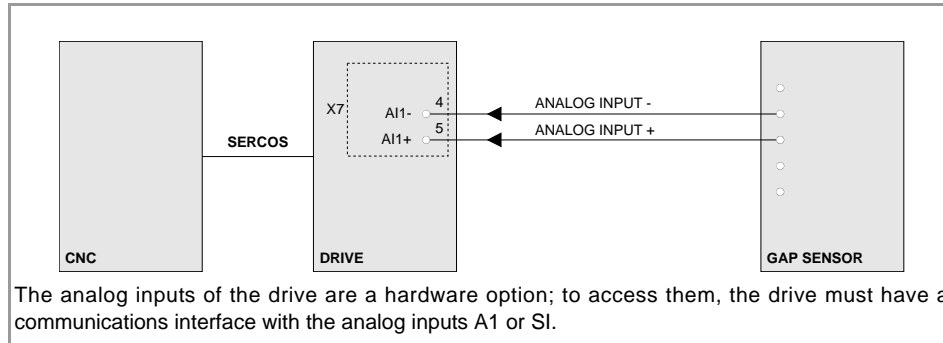
16.1.1 Connecting the gap sensor.

The analog input for the sensor can be installed in remote CAN modules or on the Sercos drive of any axis. Fagor Automation recommends using a drive input. Use the machine parameters GAPANAINTYPE and GAPANAINID to define the CNC where the sensor input is located. See "16.1.4 Configuring the gap sensor." on page 674.

Electrical diagram.

Connecting the sensor to an analog input of the Sercos drive.

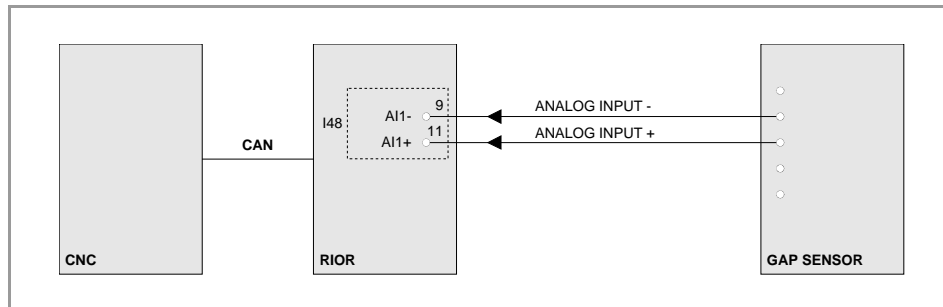
The example uses the first analog input of the drive (pins ·4· and ·5·); to use the second input, use pins ·2· and ·3·.



The analog inputs of the drive are a hardware option; to access them, the drive must have a communications interface with the analog inputs A1 or SI.

Connecting a sensor to an analog input on remote modules.

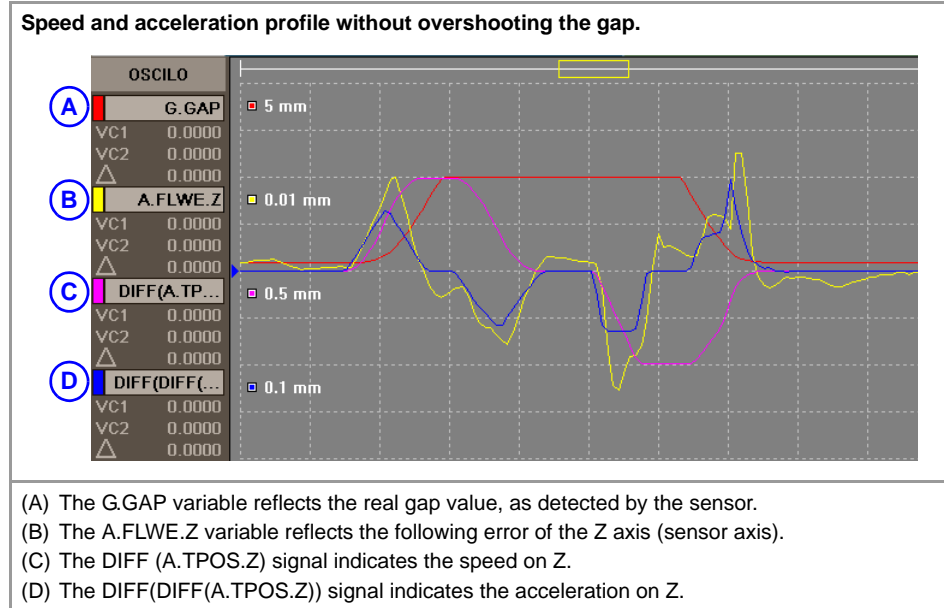
The analog inputs are available on modules RIO5 and RIOR.



16.1.2 Adjusting the axis that controls the gap.

The axis controlling the gap must not be adjusted in the standard way, rather it must be done with approach movements with the gap control active. The parameters GAPGAIN, PROGAIN, FFGAIN and ACFGAIN will be obtained from these movements. As a reference to prevent overshooting the gap, it is recommended to work with the following values.

Sercos speed Analog.	Sercos position.
<ul style="list-style-type: none"> GAPGAIN = 0.20. FFGAIN = 90 %. ACFGAIN = 30 %. 	<ul style="list-style-type: none"> GAPGAIN = 0.20. It is recommended to reduce the FFGAIN values by between 0.5 and 1% of the standard adjustment values. The ACFGAIN values can match those of a standard adjustment.



The axis that controls the gap should be adjusted so that it can reach zero follow-up errors (parameters FFGAIN and ACFGAIN). When the axes are Sercos, it is possible to make a prior adjustment using the FineTune application.

Parameter.	Meaning.
FFGAIN	Percentage of Feed-Forward in automatic.
ACFGAIN	Percentage of AC-Forward in automatic.

Loop time.

It is advisable to use a cycle time of 2 milliseconds (LOOPTIME=2).

Parameter.	Meaning.
LOOPTIME	Cycle time of the CNC.

16.

LASER.

Laser. Gap control.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

16.1.3 Configuring the gap.

Configuring the gap limits.

Parameter.	Meaning.
GAPMIN	Minimum permitted value for the gap; below this value, the CNC will show an error or not as defined in the parameter GAPERRORCANCEL.
GAPMAX	Maximum permitted value for the gap; above this value, the CNC will show an error or not as defined in the parameter GAPERRORCANCEL. This parameter should be lower than GAPDISTLIMIT.
GAPTOL	This parameter defines a positioning window on the programmed gap, to finish the approach block.

If the gap exceeds the range defined in the parameters GAPMIN and GAPMAX, the PLC deactivates the INPOSGAP mark. The CNC will display an error or not, depending on how the parameter GAPERRORCANCEL has been set. In the event of an error, the CNC stops the movement of the axes, according to the braking ramp and controlling the gap during the ramp.

If the axis exceeds the tolerance set in the GAPTOL parameter, the PLC deactivates the INTOL mark. The CNC will display an error or not, depending on how the parameter GAPTOLCANCEL has been set. In the event of an error, the CNC stops the movement of the axes, according to the braking ramp and controlling the gap during the ramp.

Configure the CNC response when exceeding the gap limits.

Parameter.	Meaning.
GAPERRORCANCEL	This parameter indicates if the CNC should cancel the error for gap outside the range, when the gap exceeds the GAPMIN - GAPMAX range. Regardless of the value of this parameter, the CNC keeps monitoring the INPOSGAP mark.
GAPTOLCANCEL	This parameter indicates if the CNC should cancel the error for gap outside the tolerance range, when the gap exceeds the margin defined in the parameter GAPTOL.

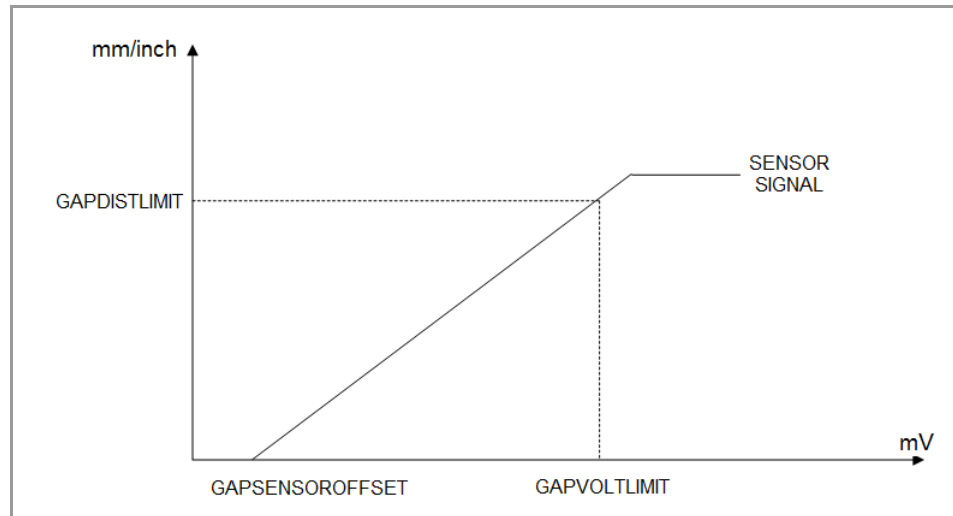
16.1.4 Configuring the gap sensor.

Configuring the analog input to which the sensor is connected.

Parameter.	Meaning.
GAPANAINTYPE	This parameter indicates the location of the analog input connected to the sensor; either in the CAN remote modules or in a Sercos drive. It is advisable to use a drive input.
GAPANAINID	This parameter identifies the analog input where the sensor is connected: 1-60 for CAN remote and 101-132 or 201-232 for drives. Analog input of the remote modules. In this case, this parameter indicates the number of the analog input using the gap sensor. Analog input of a Sercos drive. Recommended option. In this case, this parameter indicates the number of the analog input and the drive to be used. The parameter is set with a three-digit number; the first digit indicates the number of the analog input to be used (1 or 2) and the other two digits indicate the logic address of the drive (1 through 32).

Configure the gap signal (relationship between distance and voltage).

Parameter.	Meaning.
GAPSENSOROFFSET	Offset (in millivolts) to apply to the sensor from the CNC. If a Fagor analog input is used, this parameter is not required, as each has its own offset.
GAPSENSORCH	Change the sensor signal sign.
GAPDISTLIMIT	Distance corresponding to the limit analog signal of the sensor (parameter GAPVOLTLIMIT).
GAPVOLTLIMIT	Voltage (in millivolts) corresponding to the limit path of the sensor (parameter GAPDISTLIMIT).



Adjusting the sensor collision signal.

PLC signals.	Meaning.
GAPCOLLISIONMODE	The gap control sensor may have a collision signal connected to the CNC, which should be managed by the PLC via the GAPCOLLISION mark. If the PLC activates the GAPCOLLISION mark, the CNC will display an error or not, depending on how this parameter has been set. The error does not activate the emergency.

16.

LASER.

Laser. Gap control.



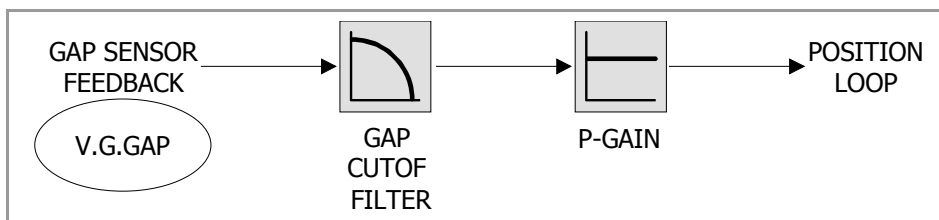
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

16.1.5 Adjusting the sensor position loop.

GAPCONTROL	
GAPGAIN	Proportional gain applied to the signal sensor, in position. To use values greater than 0.2 (close to 1), it is advisable to use the parameter GAPSENSORFILTER.
GAPAPPROACHDYN	Dynamic response during the final part of the movement approaching the plate.
GAPSENSORFILTER	Low step filter; it is advisable to define a filter of order 2 and cutoff frequency of 30 Hz.
ORDER	Filter order.
FREQUENCY	Break frequency.



Adjust with the aid of the oscilloscope.

- 1 Executing a program when gap control is active.
- 2 Selecting the variables (V.)G.GAP y (V.)A.GAPCOMP.Z (sensor axis).
 - (V.)G.GAP Actual gap value, detected by the sensor.
 - (V.)A.GAPCOMP.Z Compensation introduced by gap control into the axis associated with the sensor to maintain the gap.
- 3 Attempting to reduce the gap (variable G.GAP) while incrementing the proportional gain (parameter GAPGAIN). Increasing the proportional gain with small increments (e.g. 0.05). Using the GAPAPPROACHDYN parameter to adjust the GAPGAIN value during the final part of the plate approach, when the CNC has detected the sensor signal.
- 4 Reducing the filter frequency (machine parameter FREQUENCY) to obtain a softer response. Since the gap value increases, increase the proportional gain (machine parameter GAPGAIN).

16.1.6 PLC signals.

Logic consultation signals.

PLC signals.	Meaning.
INPOSGAP	The CNC activates this mark if the gap is within the range defined by the parameters GAPMIN-GAPMAX. If the gap exceeds the range defined in the parameters GAPMIN-GAPMAX, the CNC disables the INPOSGAP mark and stops the movement of the axes. The CNC will display an error or not, depending on how the parameter GAPERRORCANCEL has been set. In the event of an error, the CNC stops the movement of the axes, according to the braking ramp and controlling the gap during the ramp.
INTOL	The CNC activates this mark if the gap is within the tolerance limit defined by the parameter GAPTOL regarding the gap value programmed. If the axis exceeds the tolerance set in the GAPTOL parameter, the CNC deactivates the INTOL mark. The CNC will display an error or not, depending on how the parameter GAPTOLCANCEL has been set. In the event of an error, the CNC stops the movement of the axes, according to the braking ramp and controlling the gap during the ramp.

Modifiable logic signals.

PLC signals.	Meaning.
_FEEDHOL	If the PLC sets this mark at a high logic level, the CNC channel momentarily interrupts the movement of the axes. When the mark returns to the high logic level, the movement of the axes continues. All the stops and starts of the axes are carried out with the corresponding acceleration and deceleration. <ul style="list-style-type: none"> This mark does not affect the independent movement of an axis (independent interpolator). On hirth axes, if the axis does not stop in a particular position, the CNC does not activate the MATCH(axis) mark. This mark does not turn off the laser. If the power control is active, the laser removes the minimum defined power. If the PLC sets the _FEEDHOL at a low logic level in a motionless block, the CNC continues executing the program until it detects a motion block. The "Freal" text of the screens of the automatic and jog modes shows the status of this mark. The text appears in red when the _FEEDHOL mark is active. The status of the mark is not displayed if the screen does not show this text.
_STOP	If the PLC sets this mark at a low logic level, the CNC channel interrupts the execution of the part program. To resume the execution of the program, besides setting this mark at a high logical level, the PLC should activate the CYSTART mark. <ul style="list-style-type: none"> This mark does not affect the independent movement of an axis (independent interpolator), which also does not affect the key [STOP]. This mark does not turn off the laser. This mark does not disable gap control, except in approach movement. In this case, the axis will remain stationary and without gap control until the movement is resumed. The treatment received by the signal _STOP is similar to that given to the [STOP] key in the CNC. With this signal at a low logic level, all keys remain enabled.
ENABLEGAP	This mark is active by default. The PLC deactivates this mark to disable the active gap control. The PLC activates this mark to enable the active gap control in the CNC; if no gap control is active, this mark does nothing.
GAPCOLLISION	The PLC activates this mark to indicate that the collision sensor has been activated. This mark is not kept in memory. When this mark is active, the CNC will display an error or not, depending on how the parameter GAPCOLLISIONMODE has been set.

16.

LASER.

Laser. Gap control.



FAGOR AUTOMATION

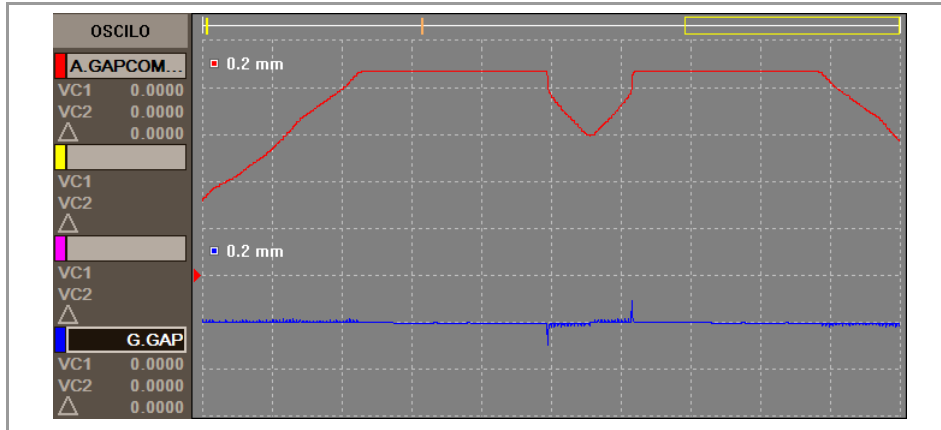
CNC 8070

(REF: 1709)

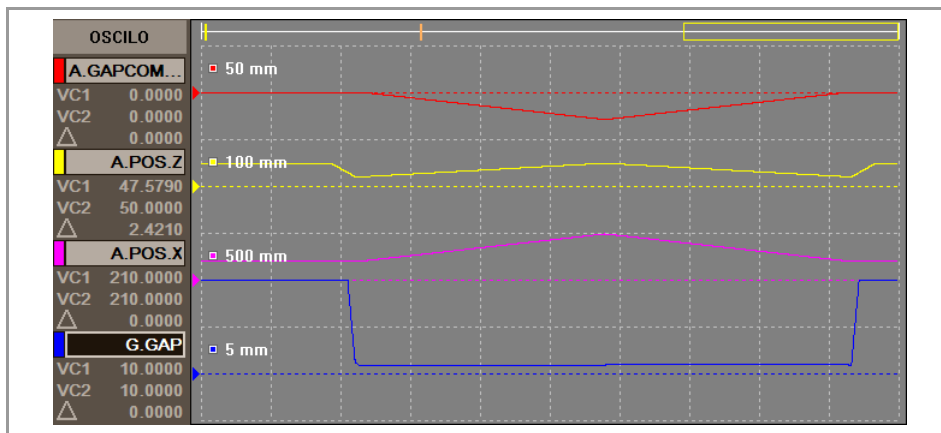
16.1.7 Gap compensation examples.

- (V.)G.GAP Actual gap value, detected by the sensor.
- (V.)A.POS.Z Z axis. Actual machine coordinators of the sensor.
- (V.)A.POS.X Actual machine coordinates for X-axis.
- (V.)A.GAPCOMP.Z Compensation introduced by gap control into the axis associated with the sensor to maintain the gap.

Part with a profile irregularity.



Part with a rising profile.



16.1.8 Operating the PLC to remove the axis associated with the gap control.

16.

LASER.

Laser. Gap control.

```
DFU {conditions} AND NOT M666 AND NOT INCYCLE
= RES ENABLEGAP
= RES ENABLELEAP
= SET IRESETZ
= TG1 333 12 ; Dwell, 12 milliseconds.
```

```
T333 = M666
```

```
DFD M666 = RES IRESETZ
= MOVE ABS (Z, 513000,50000000,NULL)
```

```
DFDB B10KEYLED1 = SET ENABLEGAP = SET ENABLELEAP
```

CALCULATION OF THE KINEMATICS DIMENSIONS.

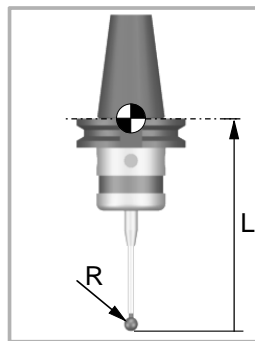
17

The OEM may set up to 6 different kinematics for the same machine. The CNC offers a number of pre-defined kinematics that may be easily configured by machine parameters. Besides these kinematics, the OEM can integrate 6 additional kinematics.

The dimensions to be set by parameter depend on the type of kinematics, set in parameter TYPE. The following sections show, as examples, the methods to calculate the dimensions of a kinematics with the help of a probe or a dial indicator.

17.1 Angular spindle. Calculation of the dimensions using a probe.

This example calculates the dimensions of a YZ 45° angular spindle, parameter TYPE = 6. Then next example uses a probe and a sample part secured to the table, with its sides aligned with the main axes of the machine.



The machine parameters associated with the probe movement must be set and the probe must be properly calibrated. You need to know the length and radius of the probe to calculate the dimensions of the spindle.

The CNC can have configured two probes. Before any probing move, the CNC must know which is the active probe, or, which is the same, which of the two probes it must attend to. It is selected via part-program or MDI using the instruction #SELECT PROBE.

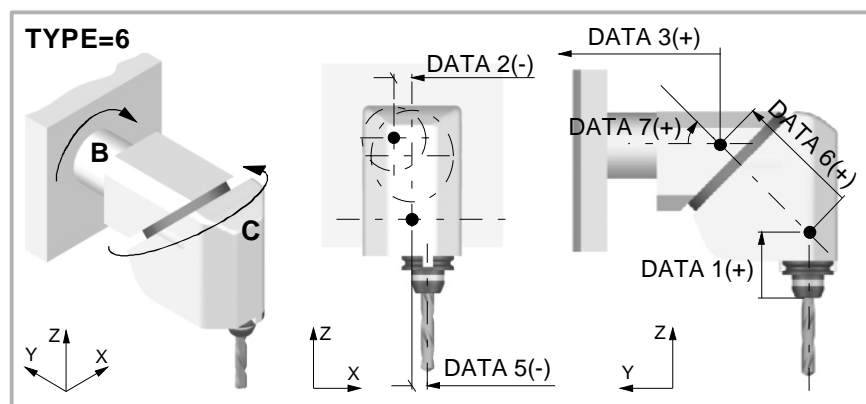


If a probing move is executed without activating the probe, it will not send any signal to the CNC when it makes contact. This can cause the probe to break because the probing move will not be stopped.

Dimensions to calculate at a YZ angular spindle (TYPE=6).

In this type of kinematics the following dimensions must be defined. All the parameters may be set with a positive or negative value. The (+) sign in the illustrations indicates that the direction is assumed as positive.

- TDATA1 It indicates the distance between the tip of the quill and the secondary rotary axis along the Z axis.
- TDATA2 It indicates the distance between the secondary rotary axis and the main axis along the X axis.
- TDATA3 It indicates the distance between the secondary rotary axis and the main axis along the Y axis.
- TDATA5 It indicates the distance between the tool axis and the secondary rotary axis along the X axis.
- TDATA6 It indicates the distance between the tool axis and the secondary rotary axis along the Y axis.
- TDATA7 It indicates the angle between the main and secondary rotary axes on rotary spindles (in this case TDATA7 = 45).



17.

CALCULATION OF THE KINEMATICS DIMENSIONS. Angular spindle. Calculation of the dimensions using a probe.



FAGOR AUTOMATION

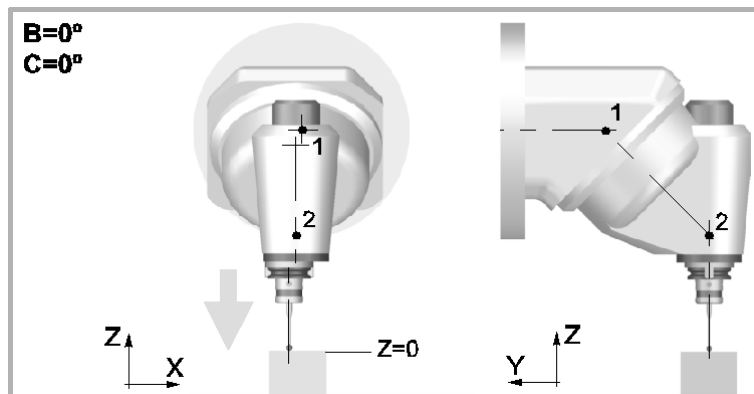
CNC 8070

(REF: 1709)

Calculation of parameter TDATA11.

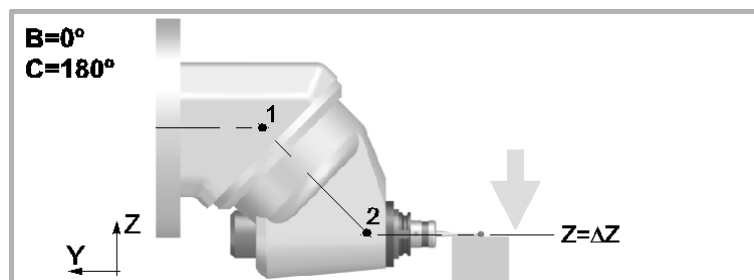
First probing move.

With rotary axes B and C positioned at 0°, make a probing movement on the Z side and preset the position Z = 0.



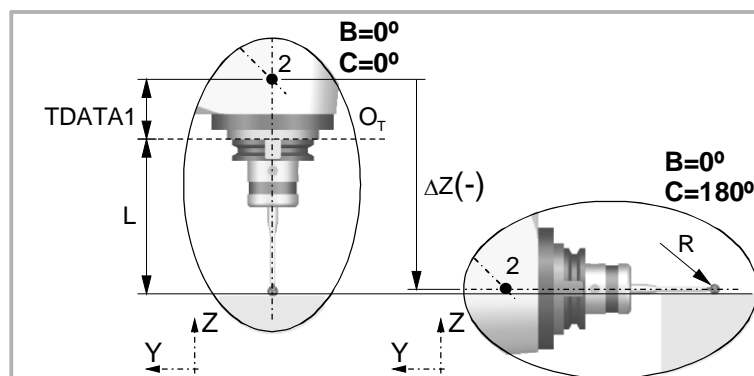
Second probe.

With rotary axis B positioned at 0° and the C axis at 180°, make a probing movement on the same side. Jot down the position value (coordinate) shown at the CNC, appearing in the drawing as ΔZ.



Mathematical calculations.

After changing the position with respect to the center of rotation ·2·, the only unknown data after both probing movements is TDATA1.



Solution.

$$|TDATA1| = |\Delta Z| + R - L$$

TDATA1	Dimensions to calculate.
OT	Reference point of the tools.
ΔZ	Coordinate shown by the CNC.
L	Probe length.
R	Probe ball radius.

17.

CALCULATION OF THE KINEMATICS DIMENSIONS.

Angular spindle. Calculation of the dimensions using a probe.

FAGOR

FAGOR AUTOMATION

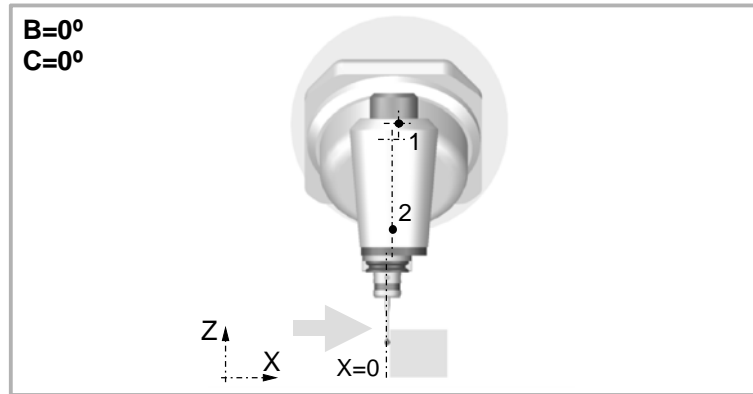
CNC 8070

(REF: 1709)

Calculation of parameter TDATA5.

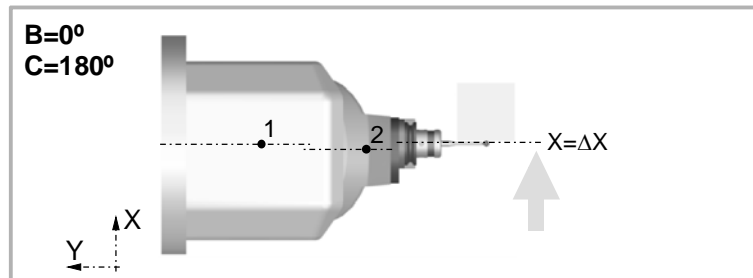
First probing move.

With rotary axes B and C positioned at 0°, make a probing movement on the X side closest to the origin and preset the position X = 0.



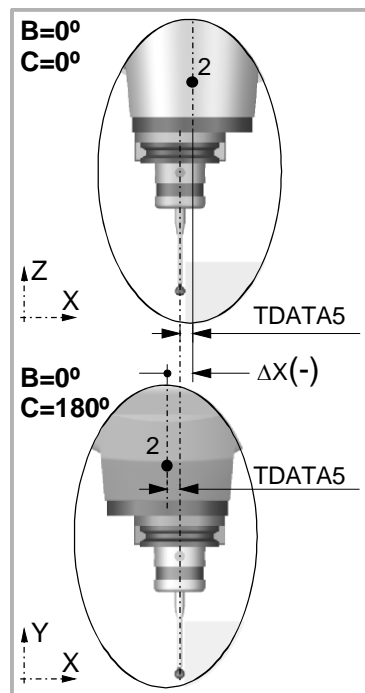
Second probe.

With rotary axis B positioned at 0° and the C axis at 180°, make a probing movement on the same side. Jot down the position value (coordinate) shown at the CNC, appearing in the drawing as ΔX .



Mathematical calculations.

After changing the position with respect to the center of rotation ·2·, the only unknown data after both probing movements is TDATA5.



Solution.	
$ TDATA5 = 0,5 \times \Delta X $	
TDATA5	Dimensions to calculate.
ΔX	Coordinate shown by the CNC.

17.

CALCULATION OF THE KINEMATICS DIMENSIONS.

Angular spindle. Calculation of the dimensions using a probe.



FAGOR AUTOMATION

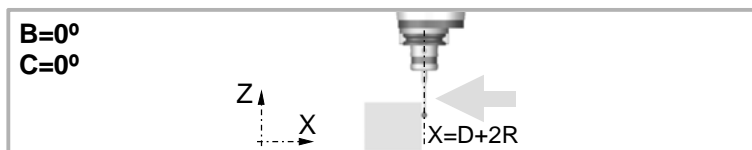
CNC 8070

(REF: 1709)

Calculation of parameter TDATA6.

First probing move.

With the B and C axes positioned at 0°, make a probing movement on the other X side to measure the part whose dimension will be the coordinate shown at the CNC minus the diameter of the probe ball.

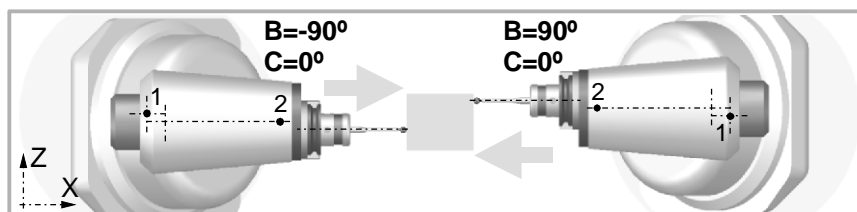


Second probe.

With the C rotary axis positioned at 0° and B at -90°, make a probing movement on the first X side. Take note of the coordinate displayed on the CNC, depicted in the diagram as $\Delta X'$.

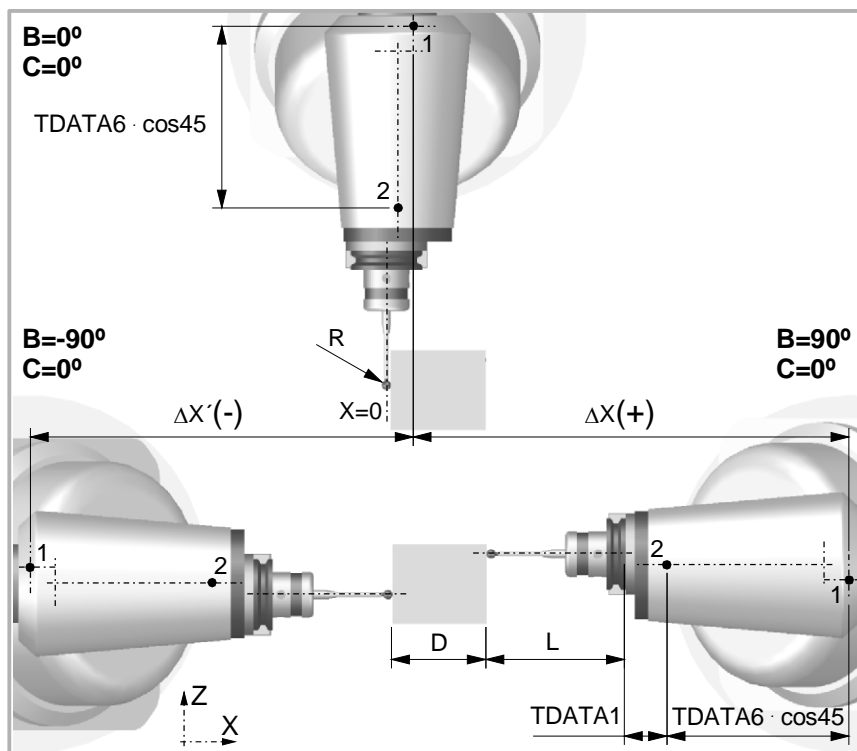
Third probing move.

With the C rotary axis positioned at 0° and B at 90°, make a probing movement on the second X side. Take note of the coordinate displayed on the CNC, depicted in the diagram as ΔX .



Mathematical calculations.

After changing the position with respect to the center of rotation ·1·, the only unknown data after both probing movements is TDATA6.



17.

CALCULATION OF THE KINEMATICS DIMENSIONS.
Angular spindle. Calculation of the dimensions using a probe.

Solution.

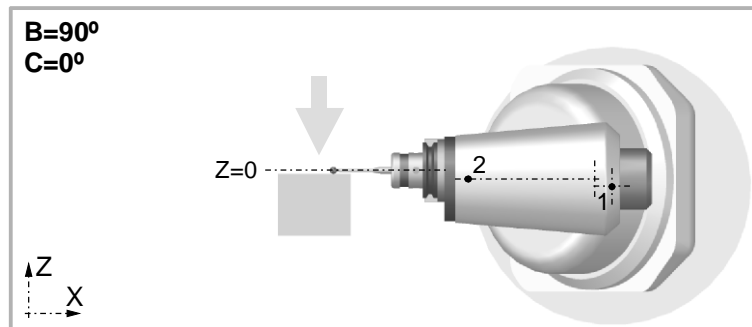
$$|TDATA6| = \frac{|\Delta X| + |\Delta X'| - D - 2L - 2|TDATA1|}{2 \cdot \cos 45}$$

TDATA1	Known dimension.
TDATA6	Dimensions to calculate.
D	Length of the part.
$\Delta X'$ ΔX	Coordinates shown by the CNC.
L	Probe length.
R	Probe ball radius.

Calculation of parameter TDATA2.

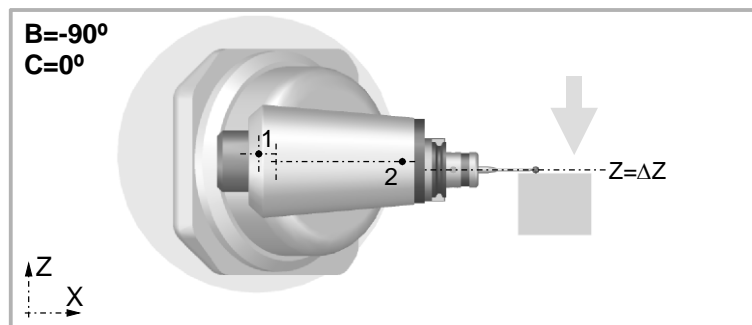
First probing move.

With the C rotary axis positioned at 0° and the B axis at 90°, make a probing movement on the Z side and preset the position Z = 0.



Second probe.

With the C rotary axis positioned at 0° and the B axis at -90°, make a probing movement on the Z side. Jot down the position value (coordinate) shown at the CNC, appearing in the drawing as ΔZ .



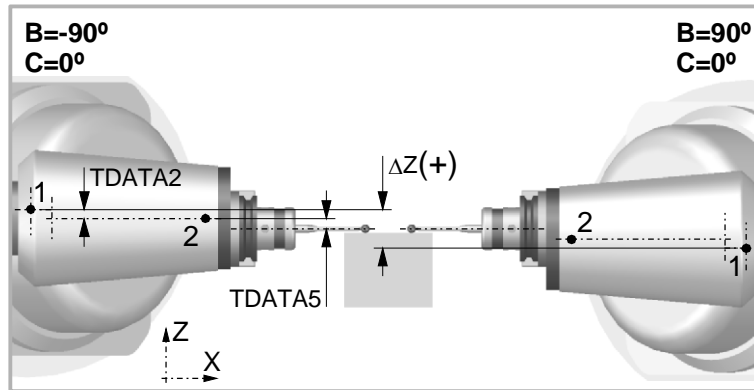
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Mathematical calculations.

After changing the position with respect to the center of rotation ·1·, the only unknown data after both probing movements is TDATA2.



Solution.	
$ TDATA2 = 0,5 \cdot \Delta Z - TDATA5 $	
TDATA2	Dimensions to calculate.
TDATA5	Known dimension.
ΔZ	Coordinate shown by the CNC.

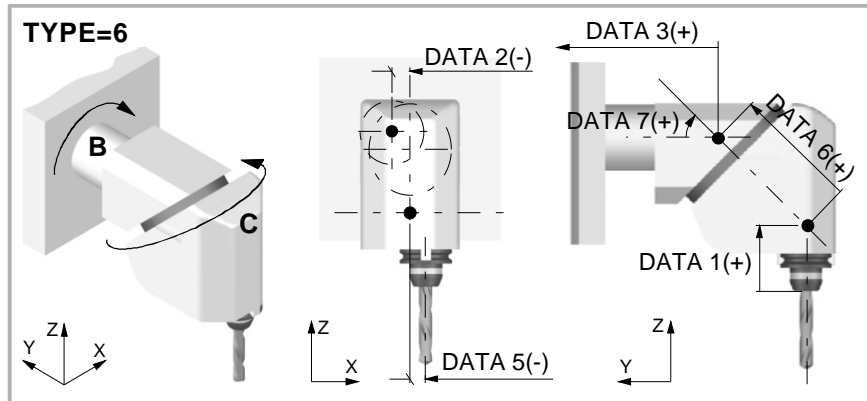
17.

CALCULATION OF THE KINEMATICS DIMENSIONS.

Angular spindle. Calculation of the dimensions using a probe.

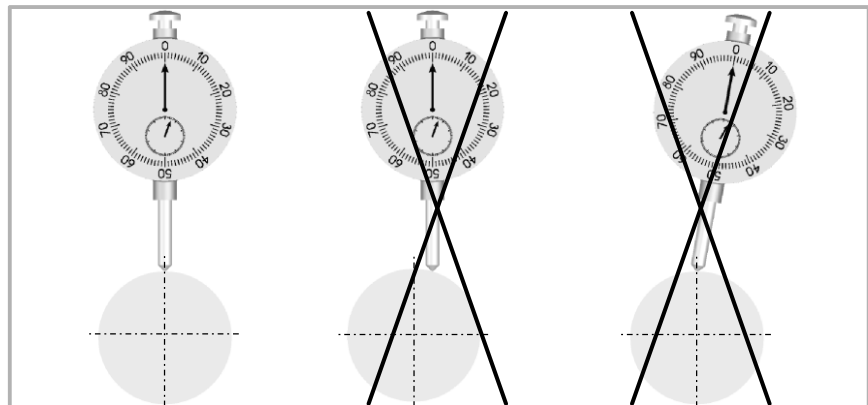
17.2 Angular spindle. Calculation of the dimensions using a dial indicator.

This example calculates the dimensions of a YZ 45° angular spindle (same as in the previous example), parameter TYPE = 6. The next example uses a dial indicator and a cylinder of known diameter placed in the spindle.



How to use the dial indicator.

When touching a cylinder with a dial indicator, contact must be made on the outside line of the cylinder. Otherwise, there will be a measuring error when presetting the value of the cylinder value. The dial indicator should be as perpendicular as possible to the tangent of the surface being measured.



A simple method to place the dial indicator on the most external line of the cylinder is to move the dial indicator on the circumference of the cylinder. To do that, preset $-0\cdot$ on the axis to be moved. When moving the axis, the needle of the dial indicator will rotate in one direction and after passing the most external point of the cylinder, the needle will rotate in the opposite direction. Continue with the movement until the needle of the dial indicator reaches its initial value. The most external point will be in the middle of the distance traveled by the axis.

17.

CALCULATION OF THE KINEMATICS DIMENSIONS.
 Angular spindle. Calculation of the dimensions using a dial indicator.



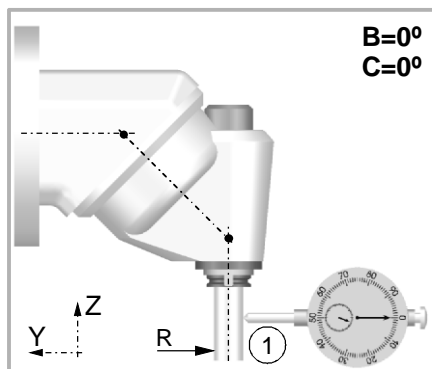
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

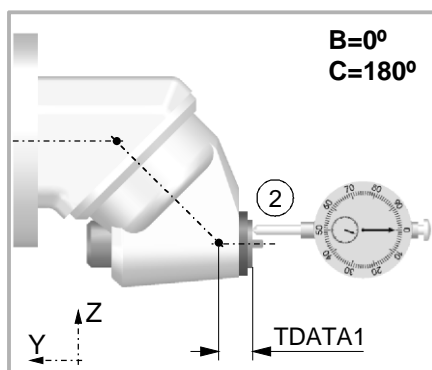
Calculation of parameter TDATA11.

First contact.



- 1 Position rotary axes B and C at 0°.
- 2 Place the dial indicator on the table, in the direction of the Y axis.
- 3 Move the table in the Y axis until the dial indicator and the cylinder touch each other at point 1.
- 4 Preset Y=R so Y=0 coincides with the shaft of the cylinder placed in the spindle.

Second contact.



- 1 Remove the dial indicator, without loosening it and position the rotary C axis at 180°.
- 2 Touch with the dial indicator at point 2.
- 3 Jot down the position value (coordinate) shown at the CNC ΔY .

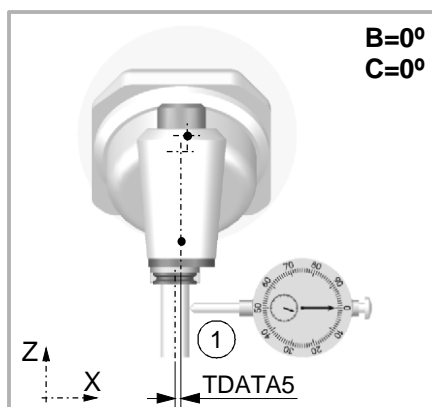
Solution.

$$|TDATA1| = |\Delta Y|$$

TDATA1	Dimensions to calculate.
ΔY	Coordinate shown by the CNC.

Calculation of parameter TDATA5.

First contact.



- 1 Position rotary axes B and C at 0°.
- 2 Place the dial indicator on the table, in the direction of the X axis.
- 3 Move the table in the X axis until the dial indicator and the cylinder touch each other at point 1.
- 4 Preset X=0.

17.

CALCULATION OF THE KINEMATICS DIMENSIONS.

Angular spindle. Calculation of the dimensions using a dial indicator.

FAGOR 

FAGOR AUTOMATION

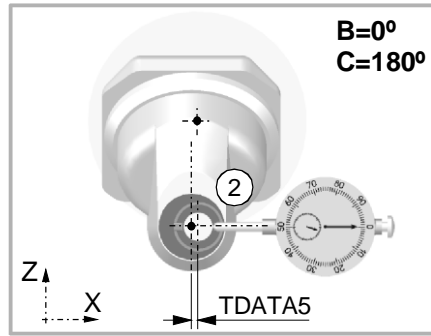
CNC 8070

(REF: 1709)

17.

CALCULATION OF THE KINEMATICS DIMENSIONS.
Angular spindle. Calculation of the dimensions using a dial indicator.

Second contact.



- 1 Remove the dial indicator, without loosening it and position the rotary C axis at 180° .
- 2 Touch with the dial indicator at point 2.
- 3 Jot down the position value (coordinate) shown at the CNC ΔX .

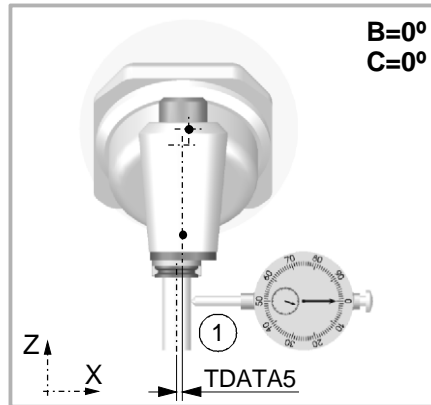
Solution.

$$|TDATA5| = 0,5 \cdot |\Delta X|$$

TDATA5	Dimensions to calculate.
ΔX	Coordinate shown by the CNC.

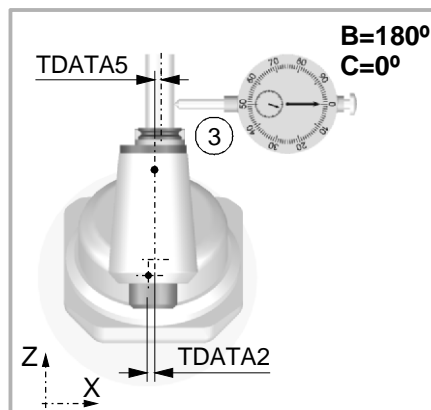
Calculation of parameter TDATA2.

First contact (made when calculating TDATA5).



- 1 Keep previous preset.

Second contact.



- 1 Remove the dial indicator, without loosening it and position the rotary C axis at 0° .
- 2 Position B rotary axis at 0° .
- 3 Touch with the dial indicator at point 2.
- 4 Jot down the position value (coordinate) shown at the CNC ΔX .

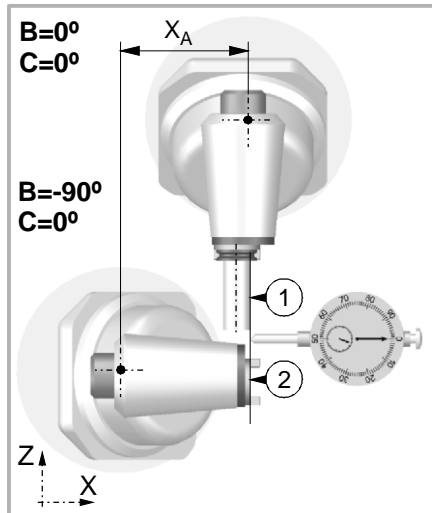
Solution.

$$|TDATA2| = 0,5 \cdot |\Delta X| - |TDATA5|$$

TDATA1	Dimensions to calculate.
TDATA5	Dimensions to calculate.
ΔX	Coordinate shown by the CNC.

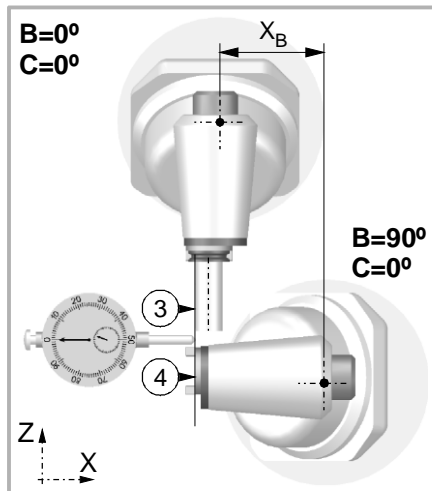
Calculation of parameter TDATA6.

First and second contacts.

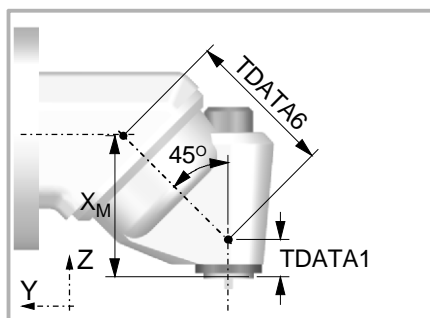


- 1 Position rotary axes B and C at 0°.
- 2 Move the table in the X axis until the dial indicator and the cylinder touch each other at point 1.
- 3 Preset X=R so X=0 coincides with the shaft of the cylinder placed in the spindle.
- 4 Remove the dial indicator, without loosening it and position the B rotary axis at -90°. Touch with the dial indicator at point 2.
- 5 Jot down the position value (coordinate) shown at the CNC (Xa).

Third and fourth contact.



- 1 Position rotary axes B and C at 0°.
- 2 Move the table in the X axis until the dial indicator and the cylinder touch each other at point 3.
- 3 Preset X=R so X=0 coincides with the shaft of the cylinder placed in the spindle.
- 4 Remove the dial indicator, without loosening it and position the B rotary axis at 90°. Touch with the dial indicator at point 4.
- 5 Jot down the position value (coordinate) shown at the CNC (Xb).



Solution.	
$ X_M = 0,5 \cdot (X_B + X_A)$	
$ TDATA6 = \sqrt{2} \cdot (X_M - TDATA1)$	
TDATA5	Dimensions to calculate.
ΔX	Coordinate shown by the CNC.

17.

CALCULATION OF THE KINEMATICS DIMENSIONS.

Angular spindle. Calculation of the dimensions using a dial indicator.



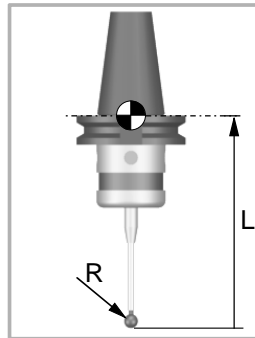
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

17.3 Rotary table. Calculation of the dimensions using a probe.

This example calculates the dimensions of an AB rotary table with a moving range for the A axis between -90° and 90° and for the B axis between 0° and -90° , parameter TYPE = 9. The next example uses a probe and a cylinder of known diameter secured to the table.



The machine parameters associated with the probe movement must be set and the probe must be properly calibrated. You need to know the length and radius of the probe to calculate the dimensions of the spindle.

The CNC can have configured two probes. Before any probing move, the CNC must know which is the active probe, or, which is the same, which of the two probes it must attend to. It is selected via part-program or MDI using the instruction #SELECT PROBE.

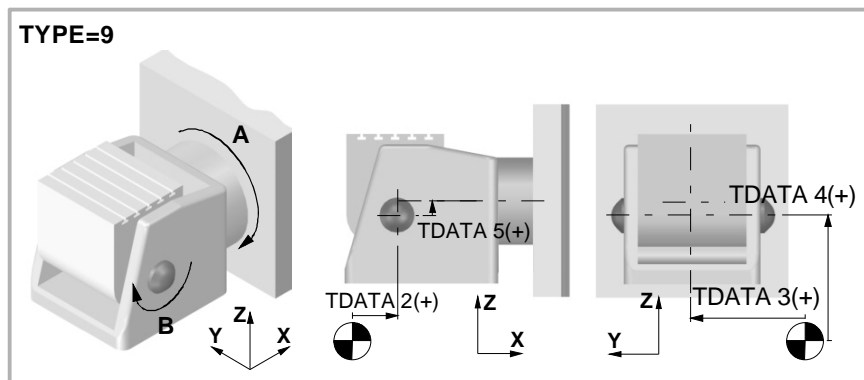


If a probing move is executed without activating the probe, it will not send any signal to the CNC when it makes contact. This can cause the probe to break because the probing move will not be stopped.

Dimensions to calculate on an AB rotary table (TYPE=9).

In this type of kinematics the following dimensions must be defined. All the parameters may be set with a positive or negative value. The (+) sign in the illustrations indicates that the direction is assumed as positive.

- TDATA2 It indicates the position of the secondary rotary axis or the intersection with the primary axis along the X axis.
- TDATA3 It indicates the position of the secondary rotary axis or the intersection with the primary axis along the Y axis.
- TDATA4 It indicates the position of the secondary rotary axis or the intersection with the primary axis along the Z axis.
- TDATA5 It indicates the distance between the secondary and the main rotary tables.



17.

CALCULATION OF THE KINEMATICS DIMENSIONS.
Rotary table. Calculation of the dimensions using a probe.



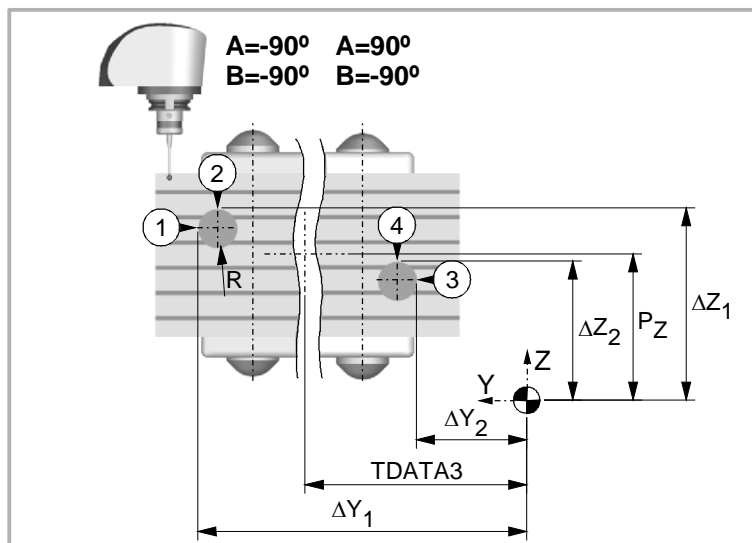
FAGOR AUTOMATION

CNC 8070

(REF: 1709)

Calculation of the coordinates of the A axis.

The diagram shows two different positions that have the center of axis A in common. The coordinate for the Y axis is the parameter TDATA3 and the coordinate for the Z axis is Pz. Both coordinates are necessary to calculate the rest of the parameters.



17.

CALCULATION OF THE KINEMATICS DIMENSIONS.
 Rotary table. Calculation of the dimensions using a probe.

First position.

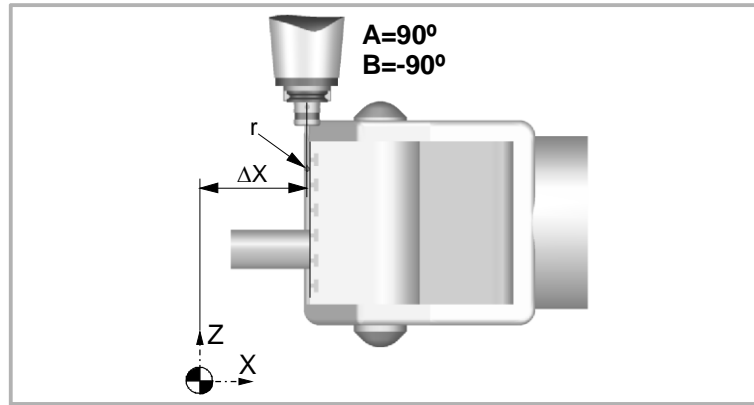
- 1 Position rotary axes A and B at -90°.
- 2 Make a probing movement along the Y axis at point 1. Jot down the position value (coordinate) shown at the CNC, appearing in the drawing as ΔY1.
- 3 Make a probing movement along the Z axis at point 1. Jot down the position value (coordinate) shown at the CNC, appearing in the drawing as ΔZ1.

Second position.

- 1 Position the A rotary axis at 90° and the B axis at -90°.
- 2 Make a probing movement along the Y axis at point 3. Jot down the position value (coordinate) shown at the CNC, appearing in the drawing as ΔY2.
- 3 Make a probing movement along the Z axis at point 4. Jot down the position value (coordinate) shown at the CNC, appearing in the drawing as ΔZ2.

Mathematical calculations.

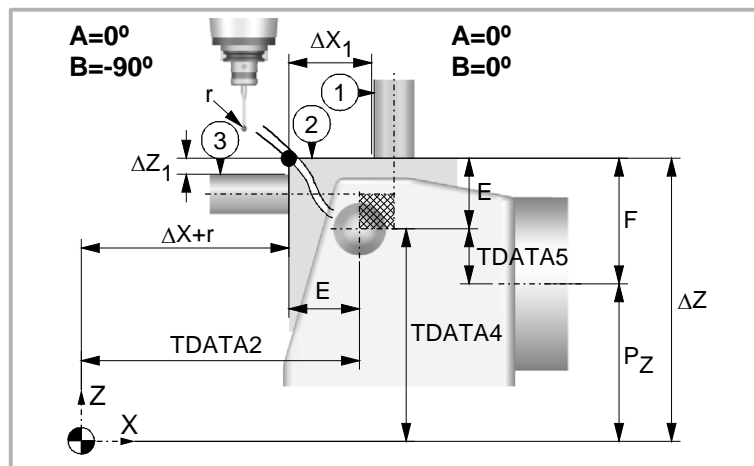
Solution.	
$ TDATA3 = 0,5 \cdot (\Delta Y_1 + \Delta Y_2)$ $ Pz = 0,5 \cdot (\Delta Z_1 + \Delta Z_2 - R)$	
TDATA3 Pz	Coordinates of the A axis.
ΔY1, ΔZ1 ΔY2, ΔZ2	Coordinates shown by the CNC.
R	Cylinder radius.

Origin preset on the X axis.

- 1 Keep previous position, $A=90^\circ$ and $B=-90^\circ$.
- 2 Probe on the table, in the direction of the X axis.
- 3 Jot down the position value (coordinate) shown at the CNC, appearing in the drawing as ΔX .
- 4 Preset $X = -r$ so $X=0$ coincides with the surface of the table.

Calculation of the rest of the parameters.

The drawing shows two different positions of the B axis while keeping the position of the A axis.

**First position.**

- 1 Position rotary axes A and B at 0° .
- 2 Make a probing movement along the X axis at point 1. Jot down the position value (coordinate) shown at the CNC, appearing in the drawing as ΔX_1 .
- 3 Make a probing movement along the Z axis at point 2. Jot down the position value (coordinate) shown at the CNC, appearing in the drawing as ΔZ_1 .
- 4 Preset $Z=0$.

Second position.

- 1 Position the A rotary axis at 0° and the B axis at -90° .
- 2 Make a probing movement along the Z axis at point 3. Jot down the position value (coordinate) shown at the CNC, appearing in the drawing as ΔZ_1 .

17.

CALCULATION OF THE KINEMATICS DIMENSIONS.
Rotary table. Calculation of the dimensions using a probe.

Mathematical calculations.

Solution.	
According to the shaded frame of the drawing.	
$ E = 0,5 \cdot (\Delta X_1 + \Delta Z_1 + 2R + r)$	
Calculation of parameters.	
$ F = \Delta Z - Pz $	
$ TDATA5 = F - E $	
$ TDATA4 = TDATA4 - Pz $	
$ TDATA2 = \Delta X + r + E $	
TDATA2 TDATA4 TDATA5	Parameters to calculate.
Pz	Coordinate of the A axis.
$\Delta Z, \Delta Z1$ $\Delta X, \Delta X1$	Coordinates shown by the CNC.
R	Cylinder radius.
r	Probe ball radius.
E	Distance from the table surface to the B axis.
F	Distance from the table surface to the A axis.

17.

CALCULATION OF THE KINEMATICS DIMENSIONS.

Rotary table. Calculation of the dimensions using a probe.



FAGOR AUTOMATION

CNC 8070

(REF: 1709)

17.

CALCULATION OF THE KINEMATICS DIMENSIONS.

Rotary table. Calculation of the dimensions using a probe.

The CNC can work with its own PLC, with a remote OpenPCS or with both. The OpenPCS must be installed at an external PC that may be connected to the CNC through a network or crossed cable. The CNC must be running and with the IEC option active in order to communicate with the OpenPCS and be able to execute the remote PLC program (IEC-61131).

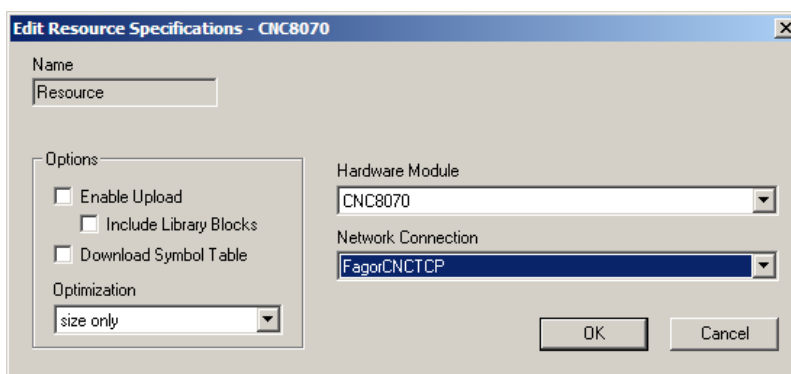
Configure the CNC to use the remote PLC.

Machine parameter PLCTYPE selects the type of PLC to be used; the Fagor one, the standard IEC-61131 or both. The following table shows who is affected by the startup (RUN) and stop (STOP) of the PLC depending on this parameter as well as the status of the mark PLCREADY.

PLCTYPE	RUN		STOP		PLCREADY	
	IEC	Fagor	IEC	Fagor	IEC	Fagor
IEC	Yes	No	Yes	No	ON	- - -
IEC+Fagor	Yes	Yes	Yes	Yes	ON	ON
Fagor	No	Yes	No	Yes	OFF	ON

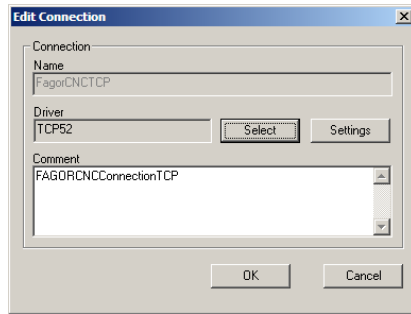
Create the hardware module of the remote OpenPCS.

The information needed by the OpenPCS to generate the proper PLC for the CNC is in the folder c:\cnc8070\fagor\plc\openpcs of the CNC itself. The folder contains a script that must be copied and executed at the work PC. This scripts creates the hardware module for that the PLC program will be compiled for and it also creates a network connection for downloading the program at the CNC, monitor it and debug it.

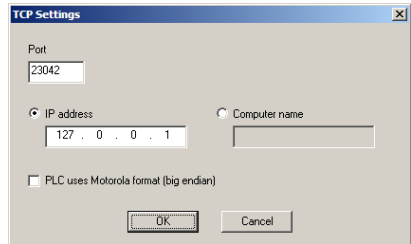


Configure the IP direction of the connection at the hardware module.

The script creates a network connection with a default IP (the loopback IP 127.0.0.1). To connect to the CNC, configure the connection with its IP.



To change the connection, select the FagorCNCTCP on the PLC menu and press Edit.



In the previous dialog box, press the Settings button to show the connection data. Set the proper IP (that of the CNC).

18.

REMOTE OPENPCS.



FAGOR AUTOMATION

Fagor Automation S. Coop.

Bº San Andrés, 19 - Apdo. 144
E-20500 Arrasate-Mondragón, Spain

Tel: +34 943 719 200

+34 943 039 800

Fax: +34 943 791 712

E-mail: info@fagorautomation.es

www.fagorautomation.com