FAGOR AUTOMATION

(Soft M: V15.3x)
(Soft T: V16.3x)

CNC 8035

Ref. 1106

INSTALLATION MANUAL

(·M· & ·T· MODELS)



FAGOR

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

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

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

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

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

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

This product uses the following source code, subject to the terms of the GPL license. The applications *busybox* V0.60.2; *dosfstools* V2.9; *linux-ftpd* V0.17; *ppp* V2.4.0; *utelnet* V0.1.1. The library *grx* V2.4.4. The linux kernel V2.4.4. The linux boot *ppcboot* V1.1.3. If you would like to have a CD copy of this source code sent to you, send 10 Euros to Fagor Automation for shipping and handling.

INDEX

About the product 9
Declaration of conformity 11
Version history (M) 13
Version history (T) 19
Safety conditions 25
Warranty terms 29
Material returning terms 31
Additional remarks 33
Fagor documentation 35

CHAPTER 1 CNC CONFIGURATION

1.1 CNC structure 38
1.1.1 Connectors 40

CHAPTER 2 HEAT DISSIPATION

2.1 Heat dissipation by natural convection 58
2.2 Heat dissipation by forced convection with inside fan 59
2.3 Heat dissipation by air flow to the outside using a fan 60

CHAPTER 3 MACHINE AND POWER CONNECTION

3.1 Digital inputs and outputs. 64
3.2 Analog inputs and outputs 65
3.3 Setup 66
3.4 Connection of the emergency input and output 70

CHAPTER 4 MACHINE PARAMETERS

4.1 Parameters that may be modified from the OEM program or OEM subroutine 76
4.2 General machine parameters 78
4.3 Axis parameters 116
4.4 Spindle parameters 144
4.5 Drive parameters 161
4.6 Serial line parameters 163
4.7 Ethernet parameters 166
4.8 PLC Parameters 170
4.9 Tables 173
4.9.1 Miscellaneous (M) function table 173
4.9.2 Leadscrew error compensation table 175
4.9.3 Cross compensation parameter table 177

CHAPTER 5 CONCEPTS

5.1 Axes and coordinate systems 179
5.1.1 Rotary axes 182
5.1.2 Gantry axes 184
5.1.3 Incline axis 185
5.2 Jog 187
5.2.1 Relationship between the axes and the JOG keys 187
5.2.2 Path-jog mode 188
5.3 Movement with an electronic handwheel 190
5.3.1 Standard handwheel 191
5.3.2 Path handwheel 192
5.3.3 Feed handwheel mode 193
5.3.4 "Additive handwheel" mode 195
5.4 feedback system 197
5.4.1 Counting speed limitation 198
5.4.2 Resolution 199

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

5.5 Axis adjustment ... 203
5.5.1 Drive setting ... 204
5.5.2 Gain setting ... 205
5.5.3 Proportional gain setting ... 206
5.5.4 Feed-forward gain setting ... 207
5.5.5 Derivative (AC-forward) gain setting ... 208
5.5.6 Leadscrew backlash compensation. ... 209
5.5.7 Leadscrew error compensation ... 210
5.6 Reference systems ... 212
5.6.1 Home search ... 213
5.6.2 Setting on systems without distance-coded feedback ... 216
5.6.3 Setting on systems with distance-coded feedback ... 218
5.6.4 Axis travel limits (software limits) ... 219
5.7 Unidirectional approach ... 220
5.8 Auxiliary M, S, T function transfer ... 221
5.8.1 Transferring M, S, T using the AUXEND signal ... 223
5.8.2 Transferring the auxiliary (miscellaneous) M functions without the AUXEND signal 224
5.9 Spindle ... 225
5.9.1 Spindle types ... 225
5.9.2 Spindle speed (S) control ... 226
5.9.3 Spindle gear change ... 228
5.9.4 Spindle in closed loop ... 230
5.10 Treatment of emergency signals ... 236
5.11 Digital CAN servo ... 239
5.11.1 Communications channel ... 239
5.12 Fagor handwheels: HBA, HBE and LGB ... 242
5.13 Machine safety related functions ... 246
5.13.1 Maximum machining spindle speed ... 246
5.13.2 Cycle start disabled when hardware errors occur. ... 248
5.14 Tool magazine ... 249
5.14.1 Tool change via PLC ... 249
5.14.2 Tool change management ... 250
5.15 Gear ratio management on axes and spindle ... 251
5.15.1 Example of axes: outside feedback without gear box ... 252
5.15.2 Example of axes: encoder at the motor ... 255
5.15.3 Example of axes: outside feedback with gear box ... 256
5.15.4 Example of spindle: external encoder without gear box ... 258
5.15.5 Example of spindle: encoder at the motor ... 260
5.15.6 Example of spindle: external encoder with gear box ... 262

CHAPTER 6 PLC RESOURCES

6.1 Inputs ... 265
6.2 Outputs ... 266
6.3 Marks ... 267
6.4 Registers ... 269
6.5 Timers ... 270
6.5.1 Monostable mode. TG1 input ... 273
6.5.2 Delayed activation mode. TG2 input ... 275
6.5.3 Delayed deactivation mode. TG3 input ... 277
6.5.4 Signal limiting mode. TG4 Input ... 279
6.6 Counters ... 281
6.6.1 Operating mode of a counter ... 284

CHAPTER 7 INTRODUCTION TO THE PLC

7.1 PLC resources ... 286
7.2 PLC program execution ... 287
7.3 Loop time ... 290
7.4 Modular structure of the program ... 291
7.4.1 First cycle module (CY1) ... 292
7.4.2 Main module (PRG) ... 293
7.4.3 Periodic execution module (PE t) ... 294
7.4.4 Priority of execution of the PLC modules ... 295

CHAPTER 8 PLC PROGRAMMING

8.1 Module structure ... 298
8.2 Directing instructions ... 299
8.3 Consulting instructions ... 303
8.4 Operators and symbols ... 305

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

8.5 Action instruction. 306
8.5.1 Binary assignment instructions 307
8.5.2 Conditional binary action instructions 308
8.5.3 Sequence breaking action instructions 309
8.5.4 Arithmetic action instructions 310
8.5.5 Logic action instructions 312
8.5.6 Specific action instructions 314

CHAPTER 9 CNC-PLC COMMUNICATION

9.1 Auxiliary M, S, T functions 318
9.2 Auxiliary M, S, T function transfer 321
9.2.1 Transferring M, S, T using the AUXEND signal 322
9.2.2 Transferring the auxiliary (miscellaneous) M functions without the AUXEND signal 323
9.3 Displaying messages, errors and screens 324
9.4 Access to the PLC from the CNC 326
9.5 Access to the PLC from a PC, via DNC 327

CHAPTER 10 LOGIC CNC INPUTS AND OUTPUTS

10.1 General logic inputs 330
10.2 Axis logic inputs. 338
10.3 Spindle logic inputs. 343
10.4 Key inhibiting logic inputs. 348
10.5 Logic inputs of the PLC channel 351
10.6 General logic outputs 353
10.7 Logic outputs of the axes 359
10.8 Spindle logic outputs. 361
10.9 Logic outputs of key status 363

CHAPTER 11 ACCESS TO INTERNAL CNC VARIABLES

11.1 Variables associated with tools. 369
11.2 Variables associated with zero offsets. 373
11.3 Variables associated with machine parameters 374
11.4 Variables associated with work zones 375
11.5 Variables associated with feedrates 377
11.6 Variables associated with coordinates 379
11.7 Variables associated with electronic handwheels 381
11.8 Variables associated with feedback 383
11.9 Variables associated with the main spindle 383
11.10 Variables associated with local and global parameters 386
11.11 Operating-mode related variables 387
11.12 Other variables 389

CHAPTER 12 AXES CONTROLLED FROM THE PLC

12.1 PLC execution channel 398
12.1.1 Considerations 399
12.1.2 Blocks which can be executed from the PLC 401
12.1.3 Control of the PLC program from the CNC 405
12.2 Action CNCEX1 407

CHAPTER 13 PLC PROGRAMMING EXAMPLE

13.1 Definition of symbols (mnemonics) 410
13.2 First cycle module. 412
13.3 Main module. 413

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

APPENDIX

A	CNC technical characteristics	425
B	Probe connection	429
C	Summary of internal CNC variables.	431
D	Summary of PLC commands	437
E	Summary of PLC inputs and outputs	441
F	2-digit BCD code output conversion table	447
G	Key codes	449
H	Logic outputs of key status	451
I	Key inhibiting codes	453
J	Machine parameter setting chart	455
K	M functions setting chart	461
L	Leadscrew error compensation table	463
M	Cross compensation table	465
N	Maintenance	467

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

ABOUT THE PRODUCT

Basic characteristics.

RAM memory	256 Kb
PLC cycle time	3 ms / 1000 instructions
RS-232 serial line	Standard
DNC (via RS232)	Standard
5 V or 24 V probe inputs	2
Digital inputs and outputs.	40 I / 24 O
Feedback inputs for the axes and spindle	4 TTL/1Vpp inputs
Feedback inputs for handwheels	2 TTL inputs

Software options.

	Model					
	M-MON	**M-MON-R**	**M-COL**	**M-COL-R**	**T-MON**	**T-COL**
Number of axes	3	3	3	3	2	2
Hard disk	Opt	Opt	Opt	Opt	Opt	Opt
Electronic threading	Stand	Stand	Stand	Stand	Stand	Stand
Tool magazine management:	Stand	Stand	Stand	Stand	Stand	Stand
Machining canned cycles	Stand	Stand	Stand	Stand	Stand	Stand
Multiple machining	Stand	Stand	Stand	Stand	-----	-----
Rigid tapping	Stand	Stand	Stand	Stand	Stand	Stand
DNC	Stand	Stand	Stand	Stand	Stand	Stand
Tool radius compensation	Stand	Stand	Stand	Stand	Stand	Stand
Retracing	-----	Stand	-----	Stand	-----	-----
Color monitor	-----	-----	Stand	Stand	-----	Stand

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

DECLARATION OF CONFORMITY

The manufacturer:

Fagor Automation S. Coop.

Barrio de San Andrés Nº 19, C.P. 20500, Mondragón -Guipúzcoa- (SPAIN).

Declares:

Under their responsibility that the product:

8035 CNC

Consisting of the following modules and accessories:

8035-M, 8035-T

Note. Some additional characters may follow the references mentioned above. They all comply with the directives listed. However, check that that's the case by checking the label of the unit itself.

Referred to by this declaration with following directives:

Low voltage regulations.

EN 60204-1: 2006 Electrical equipment on machines — Part 1. General requirements.

Regulation on electromagnetic compatibility.

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

As instructed by the European Community Directives 2006/95/EEC on Low Voltage and 2004/108/EC on Electromagnetic Compatibility and its updates.

In Mondragón, July 27th, 2010.

Fagor Automation, S. Coop.

Director Gerente
Pedro Ruiz de Aguirre

VERSION HISTORY (M)

(mill model)

Here is a list of the features added in each software version and the manuals that describe them.

The version history uses the following abbreviations:

INST	Installation manual
PRG	Programming manual
OPT	Operating manual

Software V07.1x — July 2003

First version.

Software V09.0x — February 2004

List of features	Manual
Incline axis.	INST / PRG
Machine parameters. TOOLTYPE (P167): Stop block preparation when executing a new "T". TOOLTYPE (P167): Execute the stop signal when done with the "T" change. FEEDTYPE (P169): Select the behavior of the feedrate for F0. TYPCROSS (P135): On Gantry axes, cross compensation is also applied to the slave axis. RAPIDEN (P130): Rapid key controlled by PLC. General parameters that may be modified from OEM subroutine/program: CODISET. Axis parameters that may be modified from OEM subroutine/program: MAXFLWE1, MAXFLWE2.	INST
PLC marks. Name the logic inputs and outputs with the axis name BLOABOR: Ending the execution of a block using a PLC mark (main channel). BLOABORP: Ending the execution of a block using a PLC mark (PLC channel). ELIMIS: Park the spindle.	INST
While compiling the PLC program, the outputs are initialized to zero.	
Variables. SELPRO: Variable to select the active probe input. DIAM: Variable to select the programming mode, radius or diameter.	INST / PRG
G2/G3. There is no need to program the center coordinates if their value is zero.	PRG
M41-M44: These functions admit subroutines when the gear change is automatic.	PRG

Software V09.1x — December 2004

List of features	Manual
Calculation of central unit heat dissipation .	INST
New board "Axes2".	INST
Automatic keyboard type identification.	INST
Frequency filters for axes and spindles.	INST
Machine parameters. COMPMODE (P175). New tool radius compensation methods. Axis parameters that may be modified from OEM subroutine/program: REFVALUE, REFDIREC, FLIMIT. Spindle parameters that may be modified from OEM subroutine/program: REFVALUE, REFDIREC, SLIMIT.	INST
Variables. DNCSTA: DNC communication status. TIMEG: Status of the timer count programmed with G4 HANDSE: Handwheel's axis selector button pressed. ANAI(n): Value of the analog inputs. APOS(X-C): Real coordinates of the tool base, referred to part zero. ATPOS(X-C): Theoretical coordinates of the tool base, referred to part zero.	INST / PRG
Retracing function. If RETRACAC=2 , the retrace function does not stop at the M functions. The RETRACAC parameter is initialized with [SHIFT][RESET]. The number of blocks being retraced has been increased to 75.	INST
When activating tool radius compensation in the first motion block even if there is no movement of the plane axes.	INST
Manual intervention with additive handwheel.	INST / OPT
G46. Maintain G46 when the home search does not involve any axis of the angular transformation.	INST / PRG
MEXEC. Execute a modal part-program.	PRG
Up to 319 G functions now available.	PRG
The simulations without axis movement ignore the G4.	OPT
Maintain the feedrate selected in simulation.	OPT

Software V9.12 — February 2005

List of features	Manual
Look-ahead	INST / PRG

Software V09.13 — April 2005

List of features	Manual
Hirth axis pitch may be set in degrees via parameters.	INST
Rollover positioning axis. Movement in G53 via the shortest way.	INST

Software V09.15 — June 2005

List of features	Manual
CAN servo system.	INST

Software V11.01 — August 2005

List of features	Manual
The CNC supports Memkey Card + Compact Flash or KeyCF.	OPT
File explorer to show the contents of the storage devices.	INST / OPT
Loading the version from the Memkey card o from the hard disk.	OPT
New way to search home that may be selected through g.m.p. I0TYPE=3.	INST
Improved block search. Switching from simulation to execution.	INST / OPT
New repositioning mode that is activated by setting g.m.p. REPOSTY=1.	INST/PRG/OPT
Square-sine ramps on open-loop spindle.	INST
Numbering of the local inputs/outputs of the expansion modules using plc machine parameters.	INST
Default value of axis and spindle machine parameter ACFGAIN = YES.	INST
Setting axis parameters FFGAIN and FFGAIN2 with two decimals.	INST
Up to 400 (DEF) symbols now available at the PLC.	INST
New HTOR variable that indicates the tool radius being used by the CNC.	INST / PRG
Longitudinal axis definition with G16.	INST / PRG

Software V11.11 — February 2006

List of features	Manual
Handwheel feedback taken to a free feedback connector.	INST
New variables: RIP, GGSE, GGSF, GGSG, GGSH, GGSI, GGSJ, GGSK, GGSL, GGSM, PRGSP and PRBMOD	INST
G04 K0. Block preparation interruption and coordinate update.	PRG

Software V11.13 — June 2006

List of features	Manual
Smooth stop when homing the axes, it may be selected with a.m.p. I0TYPE.	INST

Software V11.14 — August 2006

List of features	Manual
Selecting the additive handwheel as handwheel associated with the axis.	INST

Software V11.18 — June 2007

List of features	Manual
Copy and execute programs on Hard Disk (KeyCF)	OPT

Software V13.01x — December 2006

List of features	Manual
Display of PLC or CNC messages in Russian and Chinese.	INST
New FAGOR filters.	INST
Leadscrew backlash compensation. Compensation peak cutting criterion.	INST
Home search on Gantry axes (managing two home switches).	INST
Automatic spindle homing with the first M3/M4.	OPT
Allow two "switched" axes to have different gear ratios.	INST
Look-Ahead. Angle below which, it machines in square corner mode.	PRG
Teach-in. Execution of the edited block.	OPT
Improvements to the oscilloscope and direct access from jog and execution modes.	OPT
Editing on hard disk (KeyCF)	OPT
Data safety backup. Backup - Restore.	OPT
New set of gains and accelerations.	INST
Withdraw or skip a drilling or mill type threading cycle.	INST / PRG
MSGFILE: Number of PLC messages and errors expanded to 255 and 128 respectively.	INST / OPT
Faster rigid tapping without sending M functions to the PLC.	INST

Software V13.02x — March 2007

List of features	Manual
Tool inspection. Resume the interrupted cycle.	OPT / PRG

Software V15.01 — May 2007

List of features	Manual
Do not execute a program sent via DNC until pressing START.	INST
Select the set of gains and accelerations to be used in a home search.	INST
Prevent motion blocks from being executed in square corner mode.	INST / PRG
There are now more zero offsets.	PRG
G86. Boring with rapid withdrawal and spindle orientation.	PRG
The labels can now have 8 digits.	PRG
Maintain the longitudinal axis when changing the work plane.	INST / PRG
Editing on hard disk (KeyCF) on CNC's without memory expansion.	OPT

Software V15.11x — March 2008

List of features	Manual
Spindle home search on the next revolution after detecting that the home switch has been pressed.	INST
Defining a helical interpolation without programming the final coordinate of the axes of the plane.	PRG
Starting the CNC up while FAGOR filters are active.	INST
Larger numeric format to define the arc center in a G2/G3.	PRG
Monitoring the offset between the spindle and the longitudinal axis during rigid tapping.	INST / OPT
Hysteresis in the reversal movement compensation command.	INST
G210. Bore milling cycle.	PRG
G211/G212. Thread milling cycles.	PRG
Manual part centering without a probe.	OPT
New default value for a.m.p. INPOSW2 (P51).	INST
Setting the CNC in Turkish.	INST

Software V15.12 — May 2008

List of features	Manual
Improved Look-Ahead function: • Advanced look-ahead algorithm (integrating FAGOR filters). • Look-ahead operation with FAGOR filters active • Smoother machining speed.	INST / PRG

Software V15.31 — January 2009

List of features	Manual
Saving the last 10 MDI instructions..	OPT
Turn the "retrace with Look-Ahead" function on.	INST
Monitoring of the difference between the first and second feedback.	INST
Improved safety with absolute feedback.	INST
Improved tool changer.	INST
Gear ratio management on axes and spindle.	INST
Threading without spindle orientation.	PRG
PLC monitoring in ladder diagram language.	OPT
Profile editor: Polar and incremental coordinates.	OPT

Software V15.32 — July 2009

List of features	Manual
DISBLO variable: total distance programmed in blocks with look-ahead	INST

Software V15.33 — May 2010

List of features	Manual
Incompatibilities in tool change.	INST

VERSION HISTORY (T)

(lathe model)

Here is a list of the features added in each software version and the manuals that describe them.

The version history uses the following abbreviations:

INST	Installation manual
PRG	Programming manual
OPT	Operating manual

Software V08.1x — July 2003

First version.

Software V10.0x — February 2004

List of features	Manual
Incline axis.	INST / PRG
Machine parameters. TOOLTYPE (P167): Stop block preparation when executing a new "T". TOOLTYPE (P167): Execute the stop signal when done with the "T" change. FEEDTYPE (P169): Select the behavior of the feedrate for F0. TYPCROSS (P135): On Gantry axes, cross compensation is also applied to the slave axis. RAPIDEN (P130): Rapid key controlled by PLC. General parameters that may be modified from OEM subroutine/program: CODISET. Axis parameters that may be modified from OEM subroutine/program: MAXFLWE1, MAXFLWE2.	INST
PLC marks. Name the logic inputs and outputs with the axis name BLOABOR: Ending the execution of a block using a PLC mark (main channel). BLOABORP: Ending the execution of a block using a PLC mark (PLC channel). ELIMIS: Park the spindle.	INST
While compiling the PLC program, the outputs are initialized to zero.	
Variables. SELPRO: Variable to select the active probe input. DIAM: Variable to select the programming mode, radius or diameter.	INST / PRG
G2/G3. There is no need to program the center coordinates if their value is zero.	PRG
M41-M44: These functions admit subroutines when the gear change is automatic.	PRG

Software V10.1x — December 2004

List of features	Manual
Calculation of central unit heat dissipation .	INST
New board "Axes2".	INST
Automatic keyboard type identification.	INST
Frequency filters for axes and spindles.	INST
Machine parameters. COMPMODE (P175). New tool radius compensation methods. Axis parameters that may be modified from OEM subroutine/program: REFVALUE, REFDIREC, FLIMIT. Spindle parameters that may be modified from OEM subroutine/program: REFVALUE, REFDIREC, SLIMIT.	INST
Variables. DNCSTA: DNC communication status. TIMEG: Status of the timer count programmed with G4 HANDSE: Handwheel's axis selector button pressed. ANAI(n): Value of the analog inputs. APOS(X-C): Real coordinates of the tool base, referred to part zero. ATPOS(X-C): Theoretical coordinates of the tool base, referred to part zero.	INST / PRG
Retracing function. If RETRACAC=2 , the retrace function does not stop at the M functions. The RETRACAC parameter is initialized with [SHIFT][RESET]. The number of blocks being retraced has been increased to 75.	INST
When activating tool radius compensation in the first motion block even if there is no movement of the plane axes.	INST
Manual intervention with additive handwheel.	INST / OPT
G46. Maintain G46 when the home search does not involve any axis of the angular transformation.	INST / PRG
G151-G152. Programming in diameter or radius.	PRG
MEXEC. Execute a modal part-program.	PRG
Up to 319 G functions now available.	PRG
The simulations without axis movement ignore the G4.	OPT
Maintain the feedrate selected in simulation.	OPT

Software V10.12 — February 2005

List of features	Manual
Look-ahead.	INST / PRG

Software V10.13 — April 2005

List of features	Manual
Hirth axis pitch may be set in degrees via parameters.	INST
Rollover positioning axis. Movement in G53 via the shortest way.	INST

Software V10.15 — June 2005

List of features	Manual
CAN servo system.	INST

Software V12.01 — August 2005

List of features	Manual
The CNC supports Memkey Card + Compact Flash or KeyCF.	OPT
File explorer to show the contents of the storage devices.	INST / OPT
Loading the version from the Memkey card o from the hard disk.	OPT
New way to search home that may be selected through g.m.p. I0TYPE=3.	INST
Improved block search. Switching from simulation to execution.	INST / OPT
New repositioning mode that is activated by setting g.m.p. REPOSTY=1.	INST/PRG/OPT
Square-sine ramps on open-loop spindle.	INST
Numbering of the local inputs/outputs of the expansion modules using plc machine parameters.	INST
Default value of axis and spindle machine parameter ACFGAIN = YES.	INST
Setting axis parameters FFGAIN and FFGAIN2 with two decimals.	INST
Up to 400 (DEF) symbols now available at the PLC.	INST
New HTOR variable that indicates the tool radius being used by the CNC.	INST / PRG
Spindle override in the whole threading cycle at 100%.	PRG

Software V12.11 — February 2006

List of features	Manual
Handwheel feedback taken to a free feedback connector.	INST
New variables: RIP, GGSE, GGSF, GGSG, GGSH, GGSI, GGSJ, GGSK, GGSL, GGSM, PRGSP and PRBMOD	INST
G04 K0. Block preparation interruption and coordinate update.	PRG

Software V12.13 — June 2006

List of features	Manual
Smooth stop when homing the axes, it may be selected with a.m.p. I0TYPE.	INST

Software V12.14 — August 2006

List of features	Manual
Selecting the additive handwheel as handwheel associated with the axis.	INST

Software V12.18 — June 2007

List of features	Manual
Copy and execute programs on Hard Disk (KeyCF)	OPT

Software V14.01x — December 2006

List of features	Manual
Display of PLC or CNC messages in Russian and Chinese.	INST
New FAGOR filters.	INST
Leadscrew backlash compensation. Compensation peak cutting criterion.	INST
Home search on Gantry axes (managing two home switches).	INST
Automatic spindle homing with the first M3/M4.	OPT
Allow two "switched" axes to have different gear ratios.	INST
Look-Ahead. Angle below which, it machines in square corner mode.	PRG
Teach-in. Execution of the edited block.	OPT
Improvements to the oscilloscope and direct access from jog and execution modes.	OPT
Editing on hard disk (KeyCF)	OPT
Data safety backup. Backup - Restore.	OPT
New set of gains and accelerations.	INST
MSGFILE: Number of PLC messages and errors expanded to 255 and 128 respectively.	INST / OPT
Faster rigid tapping without sending M functions to the PLC.	INST
Withdrawal of axes when interrupting a threading operation.	INST / PRG
Spindle override change while threading.	INST / PRG
Threading in blind threads (without thread exit)	OPT / PRG
Jogging in G95.	PRG

Software V16.01 — May 2007

List of features	Manual
Do not execute a program sent via DNC until pressing START.	INST
Select the set of gains and accelerations to be used in a home search.	INST
Prevent motion blocks from being executed in square corner mode.	INST / PRG
There are now more zero offsets.	PRG
The labels can now have 8 digits.	PRG
Editing on hard disk (KeyCF) on CNC's without memory expansion.	OPT

Software V16.11x — March 2008

List of features	Manual
Spindle home search on the next revolution after detecting that the home switch has been pressed.	INST
Starting the CNC up while FAGOR filters are active.	INST
Larger numeric format to define the arc center in a G2/G3.	PRG
Monitoring the offset between the spindle and the longitudinal axis during rigid tapping.	INST / OPT
Hysteresis in the reversal movement compensation command.	INST
New default value for a.m.p. INPOSW2 (P51).	INST
Setting the CNC in Turkish.	INST
G86/G87. Threading cycles with variable pitch.	PRG

Software V16.12 — May 2008

List of features	Manual
Improved Look-Ahead function: • Advanced look-ahead algorithm (integrating FAGOR filters). • Look-ahead operation with FAGOR filters active • Smoother machining speed.	INST / PRG

Software V16.31 — January 2009

List of features	Manual
Saving the last 10 MDI instructions..	OPT
Turn the "retrace with Look-Ahead" function on.	INST
Monitoring of the difference between the first and second feedback.	INST
Improved safety with absolute feedback.	INST
Improved tool changer.	INST
Gear ratio management on axes and spindle.	INST
Threading without spindle orientation.	PRG
PLC monitoring in ladder diagram language.	OPT
Profile editor: Polar and incremental coordinates.	OPT

Software V16.32 — July 2009

List of features	Manual
DISBLO variable: total distance programmed in blocks with look-ahead	INST

Software V16.33 — May 2010

List of features	Manual
Incompatibilities in tool change.	INST
Zig-zag grooving mode.	OPT-TC

SAFETY CONDITIONS

Read the following safety measures in order to prevent harming people or damage to this product and those products connected to it.

This unit may only be repaired by authorized personnel at Fagor Automation.

Fagor Automation shall not be held responsible of any physical damage or defective unit resulting from not complying with these basic safety regulations.

Precautions against personal damage

- Interconnection of modules

  Use the connection cables provided with the unit.
- Use proper Mains AC power cables

  To avoid risks, use only the Mains AC cables recommended for this unit.
- Avoid electrical overloads

  In order to avoid electrical discharges and fire hazards, do not apply electrical voltage outside the range selected on the rear panel of the central unit.
- Ground connection.

  In order to avoid electrical discharges, connect the ground terminals of all the modules to the main ground terminal. Before connecting the inputs and outputs of this unit, make sure that all the grounding connections are properly made.
- Before powering the unit up, make sure that it is connected to ground

  In order to avoid electrical discharges, make sure that all the grounding connections are properly made.
- Do not work in humid environments

  In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45 ºC (113º F).
- Do not work in explosive environments

  In order to avoid risks or damages, do no work in explosive environments.

Precautions against product damage

- Working environment

  This unit is ready to be used in industrial environments complying with the directives and regulations effective in the European Community.

  Fagor Automation shall not be held responsible for any damage suffered or caused when installed in other environments (residential or homes).

- 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 complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as:

  - Powerful loads connected to the same AC power line as this equipment.
  - Nearby portable transmitters (Radio-telephones, Ham radio transmitters).
  - Nearby radio/TV transmitters.
  - Nearby arc welding machines.
  - Nearby High Voltage power lines.
  - Etc.

- Enclosures

  The manufacturer is responsible of assuring that the enclosure involving the equipment meets all the currently effective directives of the European Community.

- Avoid disturbances coming from the machine tool

  The machine-tool must have all the interference generating elements (relay coils, contactors, motors, etc.) uncoupled.

  - DC relay coils. Diode type 1N4000.
  - AC relay coils. RC connected as close to the coils as possible with approximate values of R=220 Ω / 1 W and C=0,2 µF / 600 V.
  - AC motors. RC connected between phases, with values of R=300 Ω / 6 W and C=0,47 µF / 600 V.

- Use the proper power supply

  Use an external regulated 24 Vdc power supply for the inputs and outputs.

- Grounding of the power supply

  The zero volt point of the external power supply must be connected to the main ground point of the machine.

- Analog inputs and outputs connection

  It is recommended to connect them using shielded cables and connecting their shields (mesh) to the corresponding pin.

- Ambient conditions

  The working temperature must be between +5 ºC and +40 ºC (41ºF and 104º F)

  The storage temperature must be between -25 ºC and +70 ºC. (-13 ºF and 158 ºF)

- Central unit enclosure (8055i CNC)

  Make sure that the needed gap is kept between the central unit and each wall of the enclosure. Use a DC fan to improve enclosure ventilation.

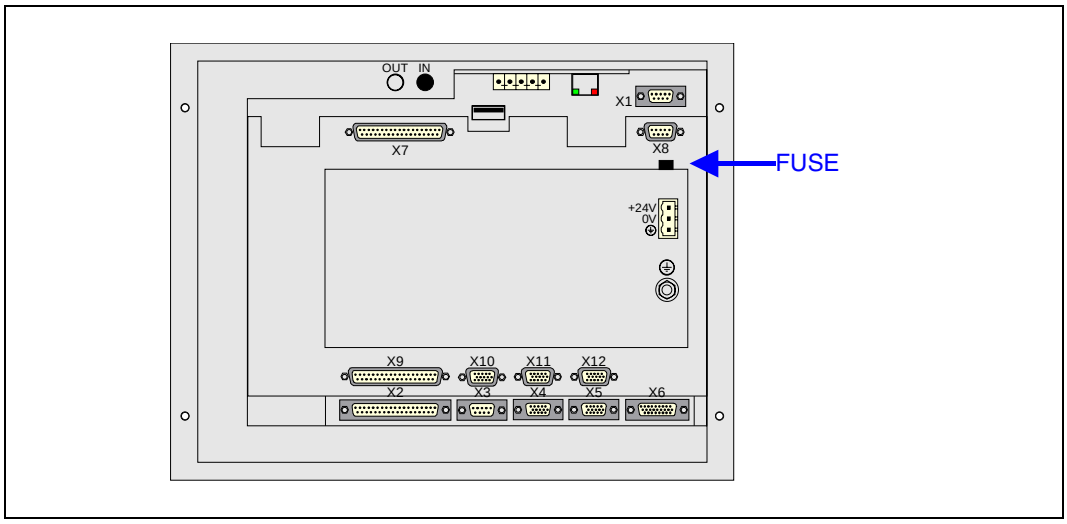
- Power switch

  This power switch must be mounted in such a way that it is easily accessed and at a distance between 0.7 meters (27.5 inches) and 1.7 meters (5.5ft) off the floor.

Protections of the unit itself

- Central unit

  It has a 4 A 250V external fast fuse (F).



- Inputs-Outputs

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

Precautions during repair

Do not open this unit. Only personnel authorized by Fagor Automation may open this unit.

Do not handle the connectors with the unit connected to mains. Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

Safety symbols

- Symbols which may appear on the manual.

Danger or prohibition symbol.

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

Warning symbol.

It indicates situations that may be caused by certain operations and the actions to be taken to prevent them.

Obligation symbol.

It indicates actions and operations that must be carried out.

Information symbol.

It indicates notes, warnings and advises.

WARRANTY TERMS

Initial warranty

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

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

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

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

FAGOR has exclusive competence in deciding whether the repair enters within the term defined as the warranty period.

Excluding clauses

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

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

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

Warranty on repairs

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

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

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

For sold parts the warranty is 12 moths length.

Maintenance contracts

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

MATERIAL RETURNING TERMS

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

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

ADDITIONAL REMARKS

Mount the CNC away from coolants, chemical products, blows, etc. which could damage it. Before turning the unit on, verify that the ground connections have been properly made.

In case of a malfunction or failure, disconnect it and call the technical service. Do not get into the inside of the unit.

FAGOR DOCUMENTATION

OEM manual

It is directed to the machine builder or person in charge of installing and starting-up the CNC.

USER-M manual

Directed to the end user.
It describes how to operate and program in M mode.

USER-T manual

Directed to the end user.
It describes how to operate and program in T mode.

USER-TC manual

Directed to the end user.
It describes how to operate and program in ISO mode and Fagor conversational mode.

FAGOR

CNC 8035

CNC CONFIGURATION

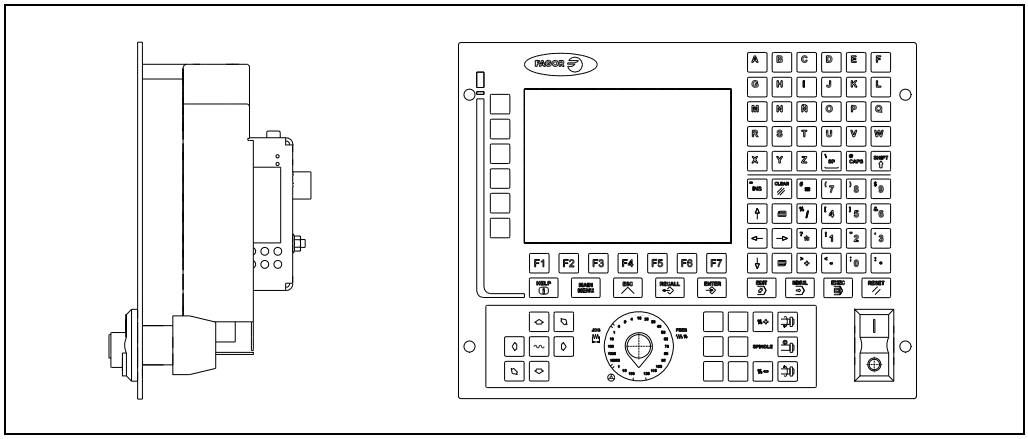
1

The CNC is prepared to be used in industrial environments, especially on milling machines, lathes, etc.

The CNC can control machine movements and devices.

1.1 CNC structure

The central unit is located on the rear of the monitor.



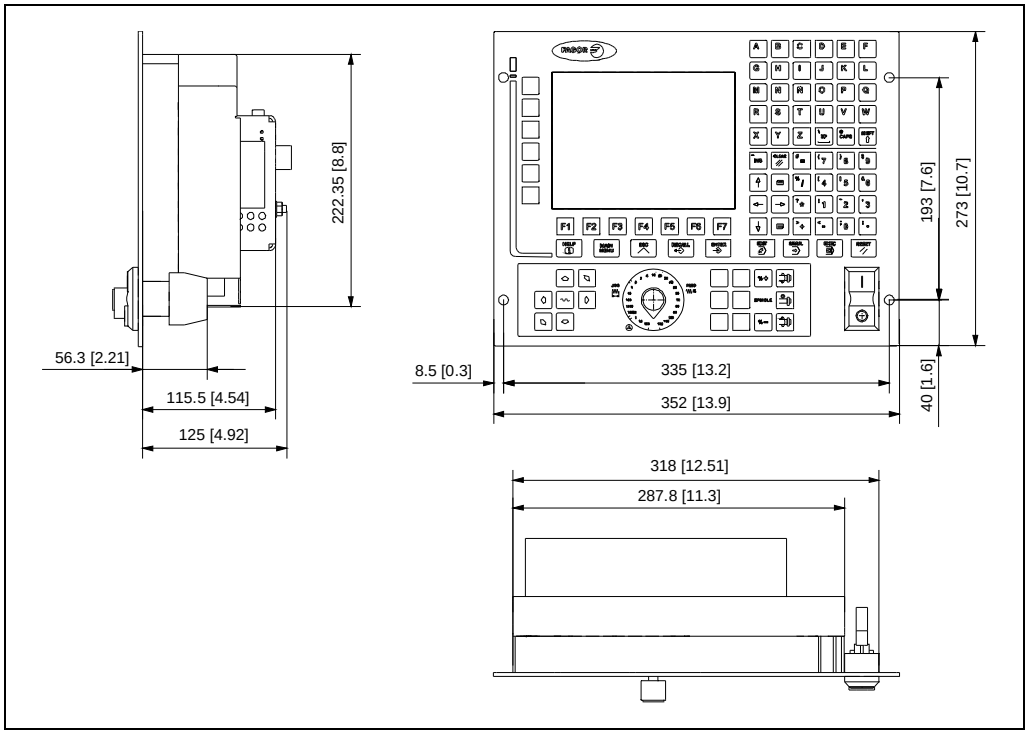
Keyboard auto-identification

The keyboard has an auto-identification system that updates g.m.p. CUSTOMTY (P92) automatically.

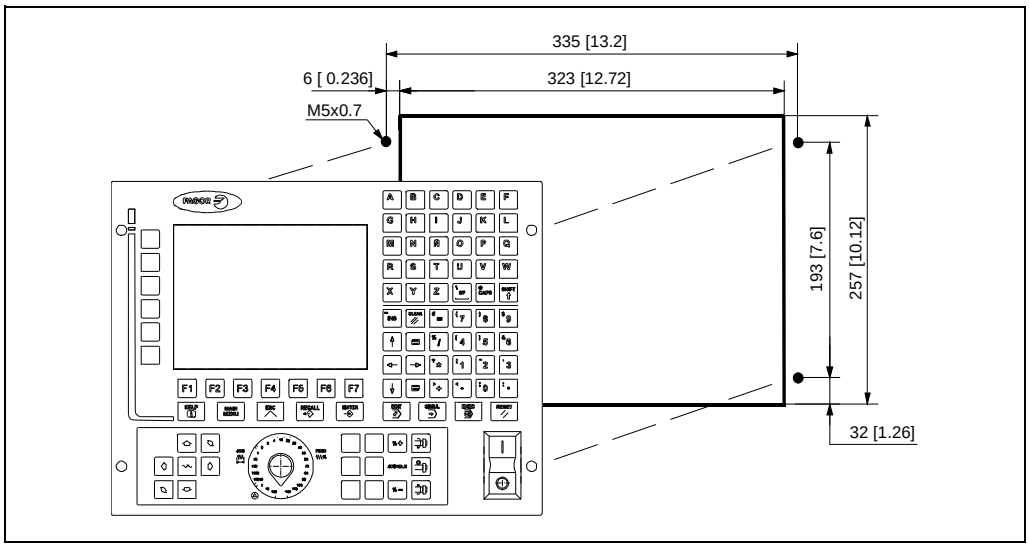
The auto-identification system of the keyboards is recognized from versions V9.11 and V10.11 on.

If an auto-identifying keyboard is connected to a CNC that has an older software version, the keyboard will beep. In this case, disable the auto-identification hardware of the keyboard by setting the identification switch to zero.

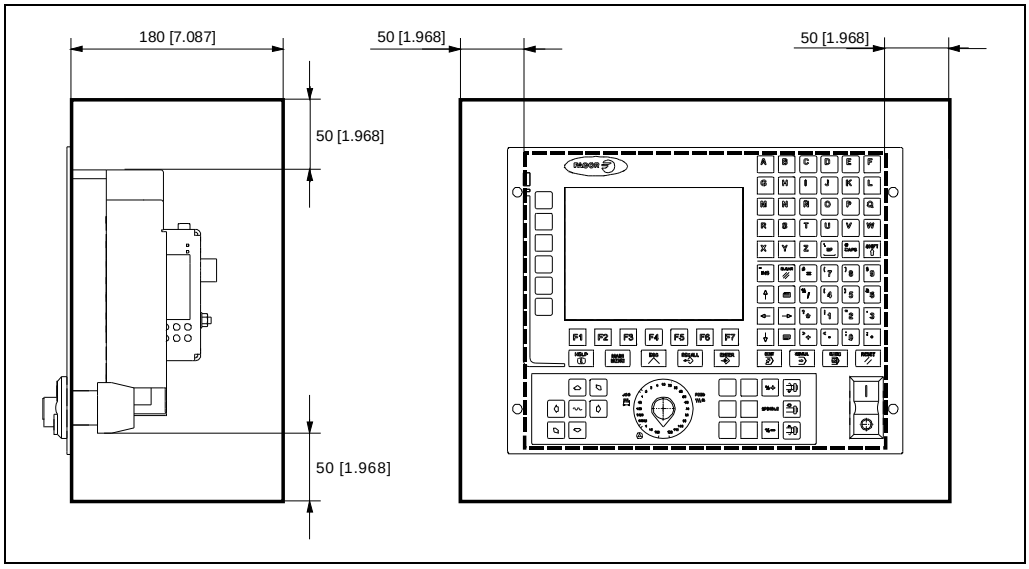
Dimensions



Enclosure



The minimum distance from each side of the monitor to its enclosure in order to guarantee the required ambient conditions is shown below:



It is up to the installer to make sure that the enclosure has forced ventilation or ventilation grooves in order to prevent the inside temperature to exceed the specified ambient temperature.

Between 5º C and +50º C (41º F and 122º F)

Relative humidity between 5% and 95% non condensing

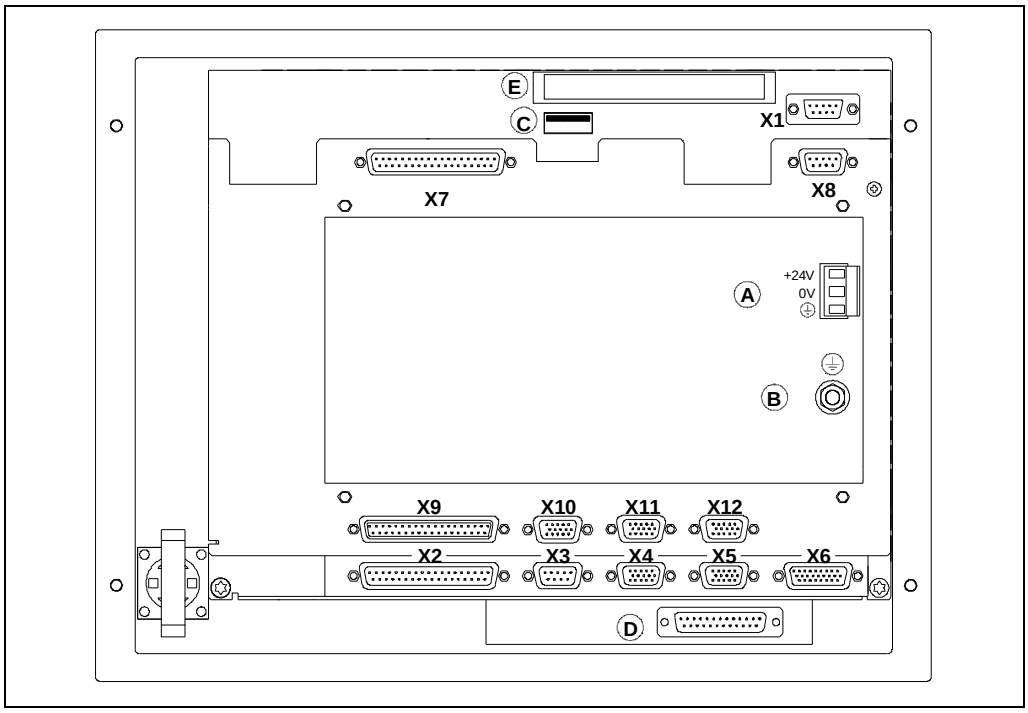
When using a fan to better ventilate the enclosure, a DC fan must be used since an AC fan may generate electromagnetic interference resulting in distorted images being displayed by the CRT.

Brightness and contrast may be adjusted on monochrome monitors. See the Operating manual, chapter on Diagnosis, section on Hardware configuration.

1.1.1 Connectors

From versions V11.1x and V12.1x on, there is a new axes board that includes the recognizance of 24V at the inputs and outputs. This board will appear in Diagnosis > Configuration > Hardware with the name of "Axes 3". This board is not compatible with previous software versions.

The connectors are located in the rear of the CNC.



(A)	Power supply.
(B)	Ground connection.
(C)	To connect the USB hard disk (Pen Drive).
(D)	Operator panel.
(E)	To connect the communications board.
X1	For RS232 serial line connection.
X2	For digital I/O connection (I1 through I16 and O1 through O8).
X3	For probe connection.
X4	For analog spindle connection.
X5	For electronic handwheel connection.
X6	For Operator Panel connection.
X7	For digital output connection (O33 to O48).
X8	For axis velocity command connection.
X9	For digital input connection (I65 to I88).
X10	For feedback connection of the first axis.
X11	For feedback connection of the second axis.
X12	For feedback connection of the third axis.
COMPACT FLASH	Slot for the local hard disk (KeyCF).

Do not open this unit. Only personnel authorized by Fagor Automation may open this module.

Do not handle the connectors with the unit connected to mains. Before doing it, make sure that the unit is disconnected.

The machine manufacturer must comply with the EN 60204-1 (IEC-204-1) standard in terms of protection against electrical shock due to faulty I/O contacts with external power supply.

Signal adapters

There are the following signal adapters.

SA-TTL-TTLD	Adapter for "Non-differential TTL" to "differential TTL" signals
SA-FS-P	Adapter for Fagor sinusoidal signals to Vpp signals

Technical characteristics of the feedback inputs

Feedback inputs for the axes and spindle

Power supply consumption of +5 V 1 A (250 mA per axis).

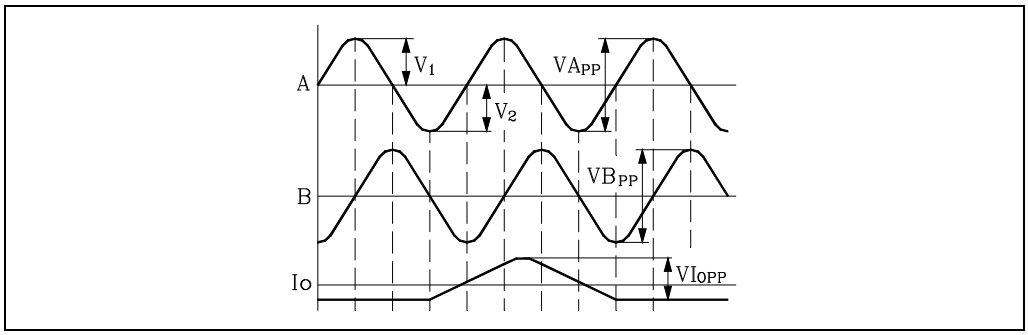
Work levels for differential square signal (axes and spindle).

Maximum frequency:	1000 kHz.
Maximum gap between flanks:	460 ns.
Phase shift:	90º ± 20º.
Vmax in common mode:	± 7 V.
Vmax in differential mode:	± 6 V.
Hysteresis:	0,2 V.
Maximum differential input current:	3 mA.

Work levels for non-differential square signal (axes and spindle).

Maximum frequency:	400 kHz.
Maximum gap between flanks:	460 ns.
Phase shift:	90º ± 20º.
High threshold (logic level "1") V_{IH}:	1.25 V < V_{IH} < 7 V.
Low threshold (logic level "0") V_{IL}:	-7 V < V_{IL} < 1 V.
Vmax:	± 7 V.
Hysteresis:	0,25 V.
Maximum differential input current:	3 mA.

Work levels for sinusoidal signal (only for axes).



Maximum frequency: 500 kHz

A and B signals	Amplitude: 0.6 ÷ 1.2 Vpp Centered: |V1-V2| / 2 Vpp =< 6.5% Relationship: VApp / VBpp = 0.8 ÷ 1.25 Phase shift: 90º ± 10º
Reference mark (I0)	Amplitude: 0.2 ÷ 0.85 V Width: T-90º =< I0 =< T+180º

Feedback input for the handwheels

Power supply consumption of +5 V 1 A (250 mA per axis).

Work levels for differential square signal.

Maximum frequency:	200 kHz.
Maximum gap between flanks:	460 ns.
Phase shift:	90º ± 20º.
Vmax in common mode:	± 7 V.
Vmax in differential mode:	± 6 V.
Hysteresis:	0,2 V.
Maximum differential input current:	3 mA.

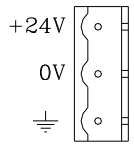
Work levels for non-differential square signal.

Maximum frequency:	200 kHz.
Maximum gap between flanks:	460 ns.
Phase shift:	90º ± 20º.
High threshold (logic level "1") V_{IH}:	1.25 V < V_{IH} < 7 V.
Low threshold (logic level "0") V_{IL}:	-7 V < V_{IL} < 1 V.
Vmax:	± 7 V.
Hysteresis:	0,25 V.
Maximum differential input current:	3 mA.

Connectors and connection

Power supply

3-prong male Phoenix connector, 7.65 mm pitch.

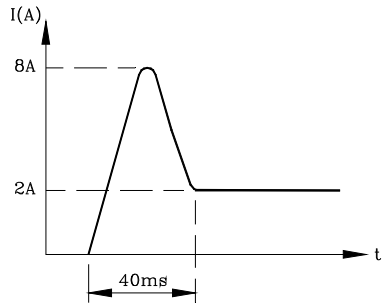
	Pin	Signal and function	
	1	+ 24 V	Power supply.
	2	0 V	Power supply.
	3	Chassis	Shield.

Use an independent external power supply with the following specifications:

Nominal voltage	20 V minimum	30 V maximum
Ripple:	4 V	
Nominal current:	2 A	
Current peak on power-up:	8 A	

The central unit has a protection against overvoltage that activates at 36 V.

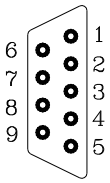
The supply current has the following shape on power-up:



Connector X1 **RS232**

It is a 9-pin SUB-D type male connector to connect the RS 232 C serial port.

The cable shield must be connected to the metallic hood at each end.

	Pin	Signal
	1	DCD
	2	RxD
	3	TxD
	4	DTR
	5	GND ISO
	6	- - -
	7	RTS
	8	CTS
	9	- - -

All the pins of this connector are opto-isolated.

Cable length

EIA RS232C standards specify that the capacitance of the cable must not exceed 2500pF; therefore, since average cables have a capacitance between 130pF and 170pF per meter, the maximum length of the cable should not be greater than 15m (49ft).

Shielded cables with twisted-pair wires should be used to avoid communication interference when using long cables.

Use shielded 7 conductor cable of 0.14 mm^2 section.

Transmission speed

The CNC can operate at up to 115,200 Baud.

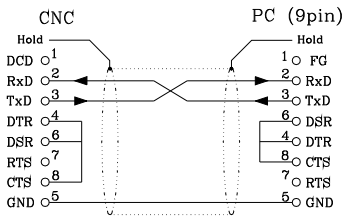
It is recommended to ground the unused pins in order to avoid erroneous control and data signal interpretations.

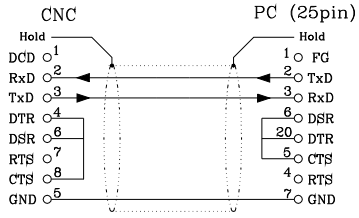
Ground connection

It is suggested to reference all control and data signals to the same ground cable (pin -GND-) thus, avoiding reference points at different voltages especially in long cables.

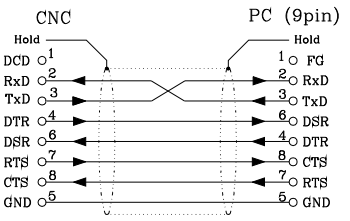
Recommended RS232C interface connection

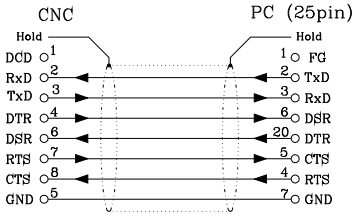
Simplified connection





Full connection.





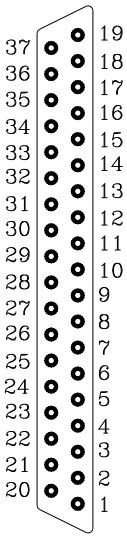
Connector X2 Digital inputs (I1 to I16) and digital outputs (O1 to O8)

It is a 37-pin normal density SUB-D type female connector.

Connect both 24V and 0V of the power supply used for these inputs and outputs must be connected to pins 18 and 19 (for 0V) and pins 1 and 20 (for the 24V) of the connector.

Since the response time of the emergency signal must be very short, the CNC has assigned input I1 for this purpose; thus, the CNC will treat this input immediately regardless of how the PLC program uses it.

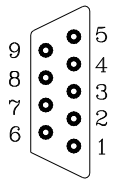
The emergency output, which coincides with O1 of the PLC, will be activated (change from logic level 1 to 0) when an ALARM or ERROR occurs at the CNC or when the PLC output O1 is set to 0 (logic level 0).



Pin		Signal and function	
	1	24 V	External power supply.
	2	O1	/ Emergency output.
	3	O3	
	4	O5	
	5	O7	
	6	- - -	
	7	- - -	
	8	- - -	
	9	- - -	
	10	I1	
	11	I3	
	12	I5	
	13	I7	
	14	I9	
	15	I11	
	16	I13	
	17	I15	
	18	0 V	External power supply.
	19	0 V	External power supply.
	20	24 V	External power supply.
	21	O2	
	22	O4	
	23	O6	
	24	O8	
	25	- - -	
	26	- - -	
	27	- - -	
	28	- - -	
	29	I2	
	30	I4	
	31	I6	
	32	I8	
	33	I10	
	34	I12	
	35	I14	
	36	I16	
	37	Chassis	Shield.

Connector X3 For probe connection

9-pin normal density SUB-D type female connector.

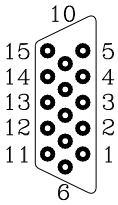
Pin		Signal and function	
	1	Chassis	Shield.
	2	+5 V	Probe 1. +5 V output for the probe.
	3	PRB1_5	Probe 1. 5 V TTL input.
	4	PRB1_24	Probe 1. 24 V DC input.
	5	GND	Probe 1. Probe's 0 V input.
	6	+5 V	Probe 2. +5 V output for the probe.
	7	PRB2_5	Probe 2. 5 V TTL input.
	8	PRB2_24	Probe 2. 24 V DC input.
	9	GND	Probe 2. Probe's 0 V input.

Up to 2 probes may be connected. There are 2 feedback inputs for each one (5V and 24V).

All shields must only be connected to ground at the CNC end through pin 1 of the connector leaving the other end free. The wires of the shielded cables cannot be unshielded for more than 75mm (about 3 inches).

Connector X4 For analog spindle connection

15-pin high density SUB-D type female connector.

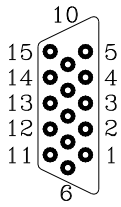
Pin		Signal and function	
	1	A	
	2	/ A	
	3	B	Feedback signals.
	4	/ B	
	5	I0	
	6	/ I0	
	7	- - -	
	8	- - -	
	9	+5 V	+5 V output for feedback.
	10	ana_out	Velocity command output.
	11	GND	0 V output for feedback.
	12	GND	0 V output for velocity command.
	13	- - -	
	14	- - -	
	15	Chassis	Shield.

It admits 1Vpp and differential TTL feedback.

The cable shield must be connected to the metallic hood at each end.

Connector X5 For electronic handwheel connection

15-pin high density SUB-D type female connector.

Pin		Signal and function	
	1	A1	Feedback signals of first handwheel.
	2	/A1	
	3	B1	
	4	/B1	
	5	A2	Feedback signals of second handwheel.
	6	/A2	
	7	B2	
	8	/B2	
	9	+5 V	Supply output.
	10	+5 V	Supply output.
	11	GND	Supply output.
	12	GND	Supply output.
	13	100P	Push button of Fagor 100P handwheel.
	14	- - -	
	15	Chassis	Shield

It admits differential (double-ended) and non-differential (single-ended) TTL feedback.

The cable must have overall shielding. The rest of the specifications depend on the feedback system used and the cable length required.

The cable shield must be connected to the metallic hood at each end.

It is highly recommended to run these cables as far as possible from the power cables of the machine.

When using a FAGOR 100P model handwheel, connect it as first handwheel and connect the axis selecting signal (button) to pin 13.

Connector X6 For Operator Panel connection

26-pin high density SUB-D type female connector.

FAGOR AUTOMATION provides the cable necessary for this connection. This cable has two 26-pin male connectors of the high density SUB-D type.

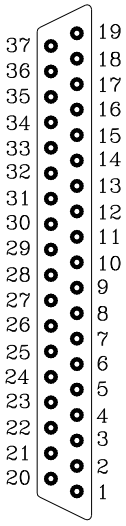
Both connectors have a latching system by means of two screws UNC4.40.

It is a straight connection, 1 to 1, 2 to 2, 3 to 3 and so on. The cable hose shield is soldered to the metal hoods covering both connectors.

Connector X7 Digital outputs (O33 to O48)

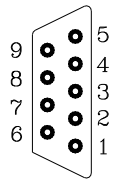
It is a 37-pin normal density SUB-D type female connector.

Connect both 24V and 0V of the power supply used for these inputs and outputs must be connected to pins 18 and 19 (for 0V) and pins 1 and 20 (for the 24V) of the connector.

Pin		Signal and function	
	1	24 V	External power supply.
	2	O33	
	3	O35	
	4	O37	
	5	O39	
	6	O41	
	7	O43	
	8	O45	
	9	O47	
	10	- - -	
	11	- - -	
	12	- - -	
	13	- - -	
	14	- - -	
	15	- - -	
	16	- - -	
	17	- - -	
	18	0 V	External power supply.
	19	0 V	External power supply.
	20	24 V	External power supply.
	21	O34	
	22	O36	
	23	O38	
	24	O40	
	25	O42	
	26	O44	
	27	O46	
	28	O48	
	29	- - -	
	30	- - -	
	31	- - -	
	32	- - -	
	33	- - -	
	34	- - -	
	35	- - -	
	36	- - -	
	37	Chassis	Shield.

Connector X8 **For connecting the outputs for the velocity command of the axes**

9-pin normal density SUB-D type female connector.

Pin		Signal and function	
	1	Chassis	Shield.
	2	Cons 1	Velocity command output for the first axis.
	3	Cons 2	Velocity command output for the second axis.
	4	Cons 3	Velocity command output for the third axis.
	5	Cons 4	Not being used
	6	GND	Velocity command reference signals.
	7	GND	
	8	GND	
	9	GND	

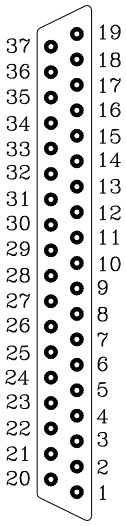
The cable shield must be connected to the metallic hood at each end.

The axis nomenclature is set when setting machine parameters AXIS1 (P0) to AXIS4 (P3).

Connector X9 **Digital inputs (I65 to I88)**

It is a 37-pin normal density SUB-D type male connector.

Connect the 0V of the power supply used for these inputs to pins 18 and 19 (for 0V) of the connector.



Pin	Signal and function	
1	- - -	
2	I65	
3	I67	
4	I69	
5	I71	
6	I73	
7	I75	
8	I77	
9	I79	
10	I81	
11	I83	
12	I85	
13	I87	
14	- - -	
15	- - -	
16	- - -	
17	- - -	
18	0 V	External power supply.
19	0 V	External power supply.
20	- - -	
21	I66	
22	I68	
23	I70	
24	I72	
25	I74	
26	I76	
27	I78	
28	I80	
29	I82	
30	I84	
31	I86	
32	I88	
33	- - -	
34	- - -	
35	- - -	
36	- - -	
37	Chassis	Shield.

Connectors X10, X11, X12

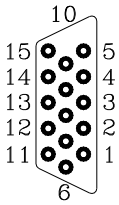
Feedback inputs for the axes

X10 For feedback connection of the first axis.

X11 For feedback connection of the second axis.

X12 For feedback connection of the third axis.

15-pin high density SUB-D type female connectors.

Pin		Signal and function	
	1	A	Feedback signals.
	2	/ A	
	3	B	
	4	/ B	
	5	I0	
	6	/ I0	
	7	- - -	
	8	- - -	
	9	+5 V	Voltage supply for the feedback system.
	10	+5 V	
	11	GND	
	12	GND	
	13	100P	
	14	- - -	
	15	Chassis	Shield

Admits differential TTL and 1Vpp sinusoidal feedback.

The cable shield must be connected to the metallic hood at each end.

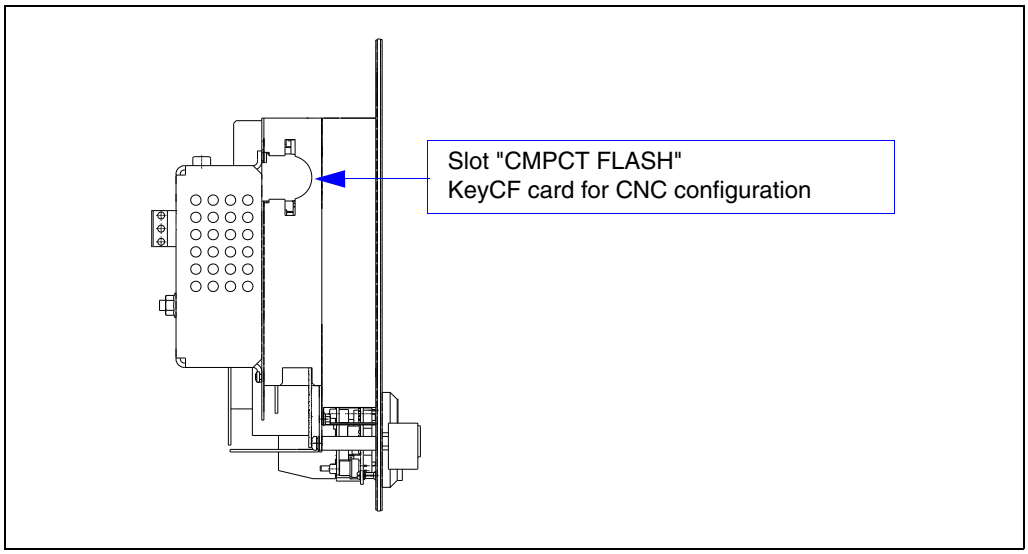
Protection at the connectors

It detects over-currents or short-circuits at the feedback of the handwheels, spindle and probe and it issues the relevant error message.

Slot "CMPCT FLASH

Compartment of the Key Compact Flash (KeyCF card for CNC configuration)

The CMPCT FLASH slot is located on the left side of the CNC.



This slot is used for the KeyCF that may be used to update the software versions among other operations.

The KeyCF supplied by Fagor with each CNC has an identification code corresponding to:

- The card id (all the cards are different).
- The software features that have been purchased for that unit

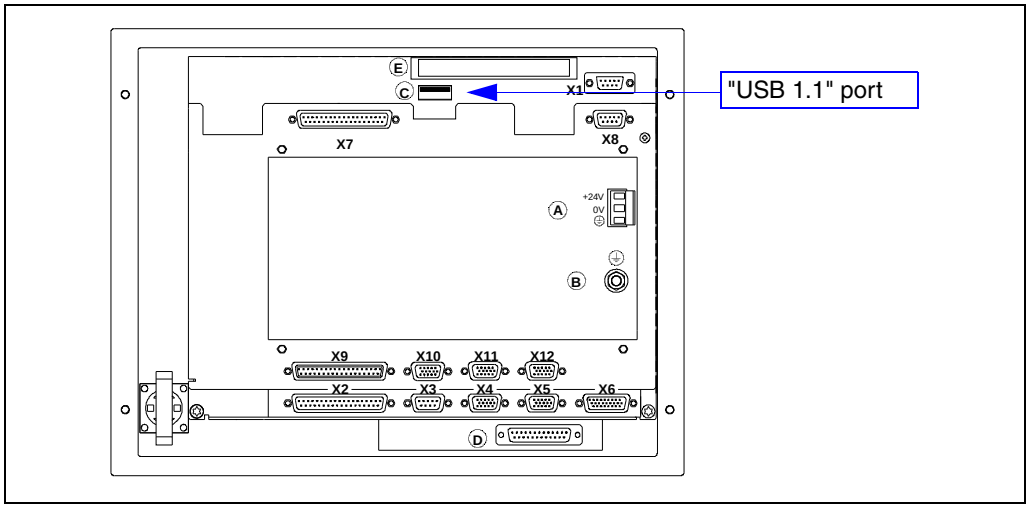
The id code only needs very little memory space. The rest of memory space of the KeyCF may be used to store data on machine customizing (user screens, PLC program backup and/or machine parameters, etc.) as well as user part-programs.

The KeyCF cannot be accessed manually from the outside, but it can via DNC. The CNC will recognize it as <Hard Disk>. This may be observed by accessing the left panel of the <explorer>.

"USB" port **USB hard disk (Pen Drive) connection.**

The USB 1.1 port admits connecting a "Pen Drive" type memory device. These memory devices are commercially available (off-the-shelf) and they're all valid regardless of their size, brand name or model.

This port is located in the top rear of the CNC.



Do not connect a multi-hub USB adapter to connect several devices at the same time. It will only recognize the first Pen Drive that is connected. Nor will it recognize other types of devices such as keyboards, mice, recorders, etc.

When using a USB cable, it should not be more than 3 m long.

The CNC recognizes this device as USB Hard Disk. Even if the CNC is turned on, when the USB device is either inserted or extracted, it will be recognized immediately. When it is connected, it will be shown as <USB hard disk> on the left panel of the <explorer>. To see its contents, press the <update> (refresh) softkey.

Within the USB device, the CNC will only recognize files with extensions *fgr (software version), *fpg (FPGA files) and part-programs. The CNC will not recognize any other type of file. Check it by selecting <USB hard disk> on the left panel of the explorer. The right panel only shows the files stored with the extensions mentioned earlier.

Only software versions can be transferred to the CNC's hard disk (KeyCF) through this USB device. It can also be transferred from the USB to the hard disk (KeyCF).

WARNING: Part-programs cannot be edited or executed from the USB hard disk.

To install a new software version stored in the USB hard disk, first copy the *fgr file into the hard disk (KeyCF).

Once the software version has been copied into the KeyCF, the transferred version may be installed. This is done using the tools of the <explorer>. See the section "Loading the version from the hard disk" in the 8035 CNC manual.

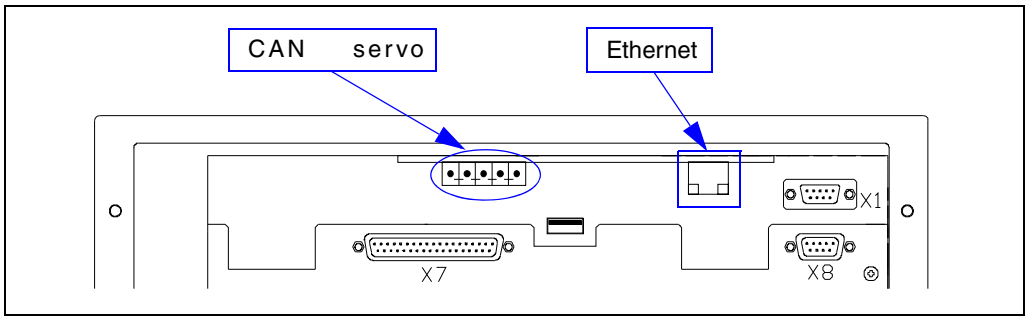
WARNING:A new software version cannot be installed directly from the USB hard disk.

From versions V11.1x and V12.1x on, the CNC will manage the hard disk (KeyCF) and the USB hard disk at the same time.

CAN and Ethernet communications board

There is a new communications board:

CAN - Ethernet:



This board has the following connections:

- CAN servo system bus.
- Ethernet bus

Ethernet CNC configuration in a local network

The Ethernet option permits configuring the CNC as another node within the local area network. This makes it possible to communicate with other PC's to transfer files.

Use a standard shielded 10BASE-T cable for this connection. It must not be longer than 100 meters.

Once the connection to Ethernet has been configured, the following types of connections are possible:

- PC connection through WinDNC (it requires WinDNC version 4.0 or higher).
- Connection from a PC through an FTP client.
- Connection to a remote hard disk.

Remote hard disk

The Ethernet connection may be used to use a PC directory (server) as a hard disk. This memory space may be shared by several CNC's or each may have its own memory space.

The interface and the softkeys of the CNC will the same as if it were a local hard disk. When accessing the CNC through WinDNC or FTP, the remote hard disk behaves like a local hard disk.

The remote hard disk is configured by machine parameters. The PC that makes its hard disk (server) public must be connected to the local network.

The NFS protocol is used to communicate with the remote hard disk. This protocol must be available at the PC that is used as server.

DIGITAL DRIVES
Digital CAN servo system

Digital servo is being used to communicate with Fagor drives.

- CAN field bus and standard CanOpen communication protocol.

Module identification at the bus

Each one of the elements integrated into the CAN bus is identified by the 16-position rotary switch (0-15) "Address" (also referred to as "Node_Select"). This rotary switch selects the address (node) occupied by each element integrated in the bus.

Although the switch has 16 positions, only positions 1 through 8 are valid. The CNC does not have a switch, The drives occupy consecutive positions (recommended) starting from ·1·.

The corresponding drive must be turned off and back on (or press the Reset button) for the address change to be assumed.

The "Line_Term" switch

The "Line_Term" switch identifies which are the elements that occupy the ends of the CAN bus; i.e. the first and last physical element in the connection.

The central unit must always be at one end of the line. The other end will be the last one of the remote module groups.

The switch position of the terminating elements must be "1" and that of the rest of the elements "0". The CNC does not have a switch and always has the terminating resistor activated.

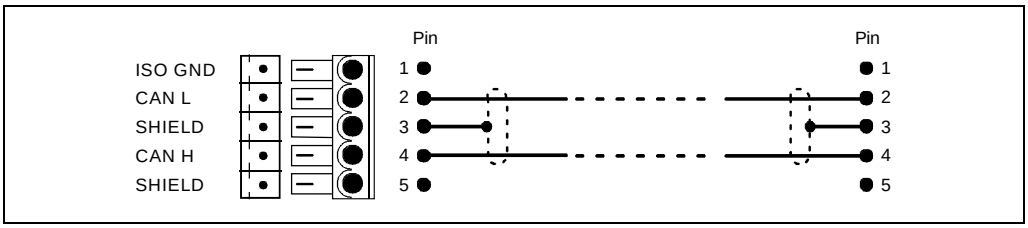
Characteristics of the CAN cable

Use a specific CAN cable. The ends of all the wires and the shield must be protected by the corresponding pin. Also use the pins to secure the cable to the connector.

Type:	Shield. Twisted pairs (1 x 2 x 0,22 mm^2).
Flexibility:	Superflexible. Minimum static bending radius of 50 mm and a dynamic radius of 95 mm.
Cover:	PUR
Impedance:	Cat.5 (100 Ω - 120 Ω)

CAN connector pinout

5-pin male Phoenix minicombicon connector (3.5 mm pitch).

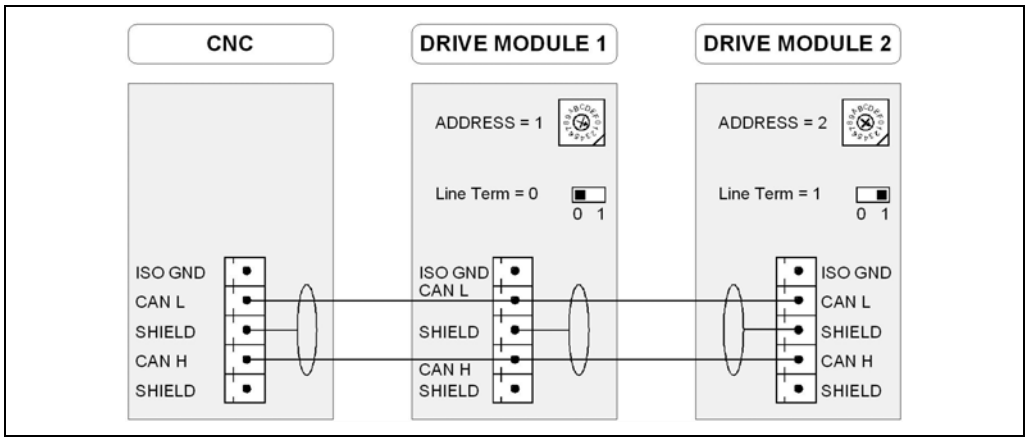


Signal	Description
ISO GND	Ground / 0 V.
CAN L	Bus signal (LOW).
SHIELD	CAN shield.
CAN H	Bus signal (HIGH).
SHIELD	CAN shield.

The connector has two shield pins. Both pins are equivalent; the CAN shield may be connected to either one.

Interconnection of modules

It is connected in series. The figure shows the CAN connection between the central unit and 2 drives.



FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

HEAT DISSIPATION

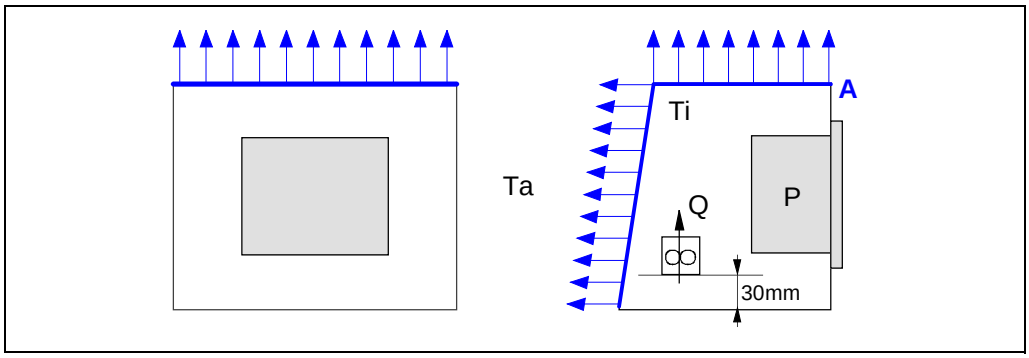
2

The working temperature of the central unit enclosure must not exceed 45 ºC (113ºF). To ensure that this temperature is not exceeded, the enclosure must have enough surface to evacuate the heat generated inside and maintain the ambient conditions within the working temperature range.

Calculating the surface needed to dissipate the heat

The expressions have been obtained for an enclosure having a 2 mm wall and made out of aluminum. When using internal cooling, the fan is located at 30 mm from the bottom.

To calculate the required total surface of the enclosure in order to dissipate the heat generated in it, the following data must be taken into account.



A	(m^2)	Total surface required.
P	(W)	Total power dissipated by all the elements that generate heat inside the enclosure, including the power supply and the fan if there is one.
Ta	(ºC)	Ambient temperature outside the enclosure.
Ti	(ºC)	Temperature inside the enclosure.
Δt	(ºC)	Temperature difference (Ti-Ta).
Q	(m^3/h)	Air flow provided by the fan, if there is one.

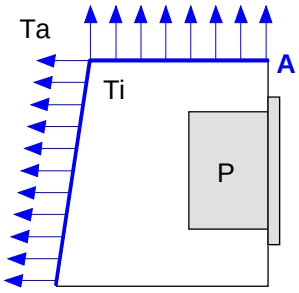
Dissipating surface

Only surfaces dissipating heat by convection will be considered, the top and the rear of the enclosure. The rest of the surfaces are not to be considered when calculating the total surface.

Power dissipated by the CNC

The maximum power dissipated by the CNC is 55 W, power supply not included.

2.1 Heat dissipation by natural convection



Surface without paint.

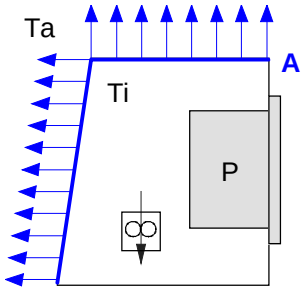
$$A = \frac{P}{5 \cdot \Delta T}$$

Surface with smooth metallic enamel.

$$A = \frac{P}{5{,}7 \cdot \Delta T}$$

2.2 Heat dissipation by forced convection with inside fan

Fan whose air flow is Q = 13.6 m^3/h facing down.



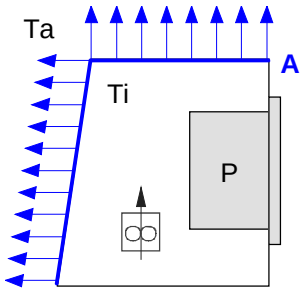
Surface without paint.

$$A = \frac{P}{5{,}6 \cdot \Delta T}$$

Surface with smooth metallic enamel.

$$A = \frac{P}{7{,}6 \cdot \Delta T}$$

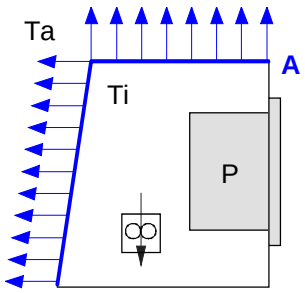
Fan whose air flow is Q = 13.6 m^3/h facing up.



Surface without paint.

$$A = \frac{P}{5{,}8 \cdot \Delta T}$$

Fan whose air flow is Q = 30 m^3/h facing down.



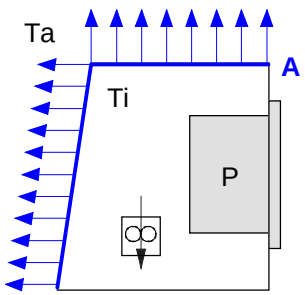
Surface without paint.

$$A = \frac{P}{6{,}75 \cdot \Delta T}$$

Surface with smooth metallic enamel.

$$A = \frac{P}{9{,}1 \cdot \Delta T}$$

Fan whose air flow is Q = 102 m^3/h facing down.



Surface without paint.

$$A = \frac{P}{7{,}5 \cdot \Delta T}$$

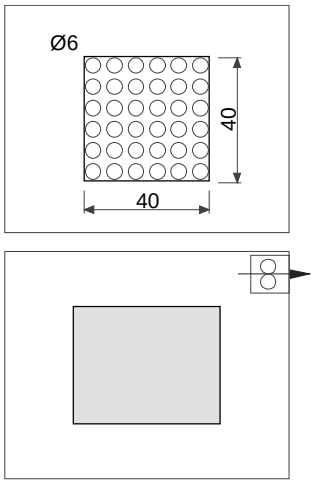
Surface with smooth metallic enamel.

$$A = \frac{P}{9{,}8 \cdot \Delta T}$$

2.3 Heat dissipation by air flow to the outside using a fan

Heat dissipation by convection forcing hot air flow to flow outside with a fan and ambient air input through the holes of the bottom surface of the enclosure.

For this case, calculate the necessary air flow that the fan must supply to dissipate the heat generated inside the enclosure. The fan's air flow is calculated according to the power dissipated by the CNC and the fan itself as well as the inside and outside temperatures.



Surface without paint.

$$V = \frac{3{,}8 \cdot P}{\Delta T}$$

Bear in mind that this air flow through the unit extracts hot air to the outside, but it allows dirt into the enclosure. Thus, a filter should be installed to maintain the ambient conditions allowed.

MACHINE AND POWER CONNECTION

3

Power switch.

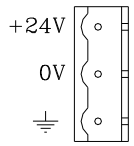
- *This power switch must be mounted in such a way that it is easily accessed and at a distance between 0.7 meters (27.5 inches) and 1.7 meters (5.5ft) off the floor.*

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.*

Mains connection of the central unit

The "Central Unit + Monitor" set has a three-prong male Phoenix connector with a 7.62 mm pitch.

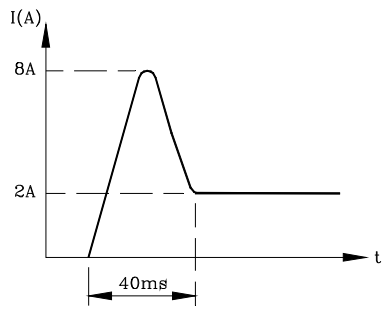
	Pin	Signal and function	
	1	+ 24 V	Power supply.
	2	0 V	Power supply.
	3	Chassis	Shield.

Use an independent external power supply with the following specifications:

Nominal voltage	20 V minimum	30 V maximum
Ripple:	4 V	
Nominal current:	2 A	
Current peak on power-up:	8 A	

The central unit has a protection against overvoltage that activates at 36 V.

The supply current has the following shape on power-up:



Machine connection

The machine-tool must have all the interference generating elements (relay coils, contactors, motors, etc.) uncoupled.

- DC relay coils.

Diode type 1N4000.

- AC relay coils.

  RC connected as close as possible to the coils. Their approximate values should be:

  R 220 Ω / 1 W C 0.2 µF / 600 V

- AC motors.

  RC connected between phases with values:

  R 300 Ω / 6 W C 0,47 µF / 600 V

Ground connection

It is imperative to carry out a proper ground connection in order to achieve:

- Protection of anybody against electrical shocks caused by a malfunction.
- Protection of the electronic equipment against interference generated by the proper machine or by other electronic equipment near by which could cause erratic equipment behavior.

Thus, it is essential to connect all metallic parts to a point and it to ground in order to achieve this. Therefore, it is crucial to install one or two ground points where the above mentioned elements must be connected.

Use large section cables for this purpose in order to obtain low impedance and efficiently avoid any interference. This way all parts of the installation will have the same voltage reference.

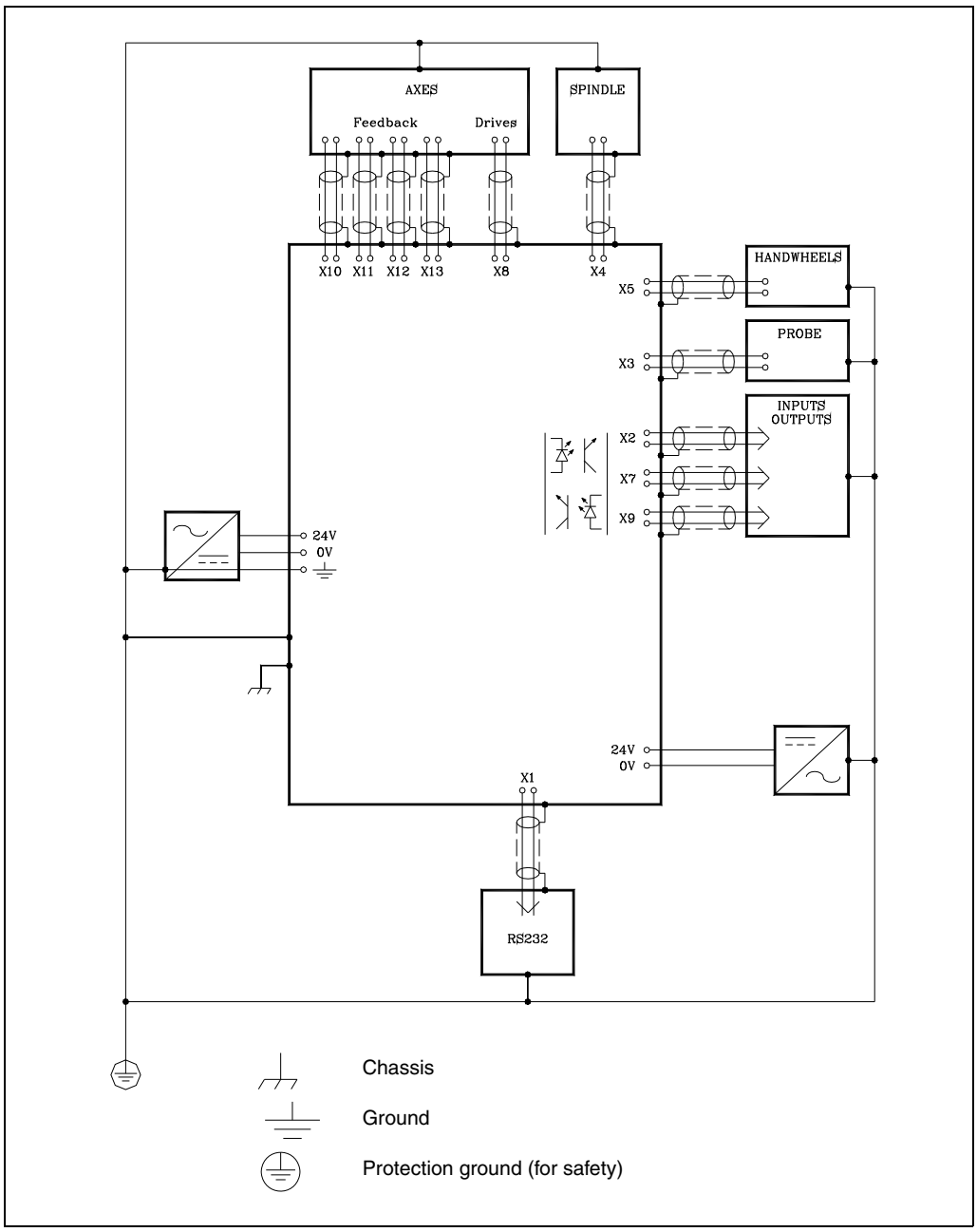
Proper ground installation reduces the effects of electrical interference. But, signal cables also require additional protections. This is generally achieved by using twisted-pair cables that are also covered with antistatic shielding mesh-wire. This shield must be connected to a specific point avoiding ground loops that could cause undesired effects. This connection is usually done at one of CNC's ground point.

Each element of the machine-tool/CNC interface must be connected to ground via the established main points. These points will be conveniently set close to the machine-tool and properly connected to the general ground (of the building).

When a second point is necessary, it is recommended to join both points with a cable whose section is no smaller than 8 mm^2.

Verify that the impedance between the central point of each connector housing and the main ground point is less than 1 Ω.

Ground connection diagram



3.1 Digital inputs and outputs.

Digital outputs

The CNC system offers a number of optocoupled digital PLC outputs which can be used to activate relays, deacons, etc.

The electrical characteristics of these outputs are:

Nominal voltage value	+24 V DC.
Maximum voltage value	+30 V.
Minimum voltage value	+18 V.
Output voltage	2 V less than the supply voltage.
Maximum output current	100 mA.

All outputs are protected by means of:

- Galvanic isolation by optocouplers.
- The CNC has protection against short-circuits, overvoltage of the external power supply (over 33 Vdc) and against reverse connection of the power supply (up to –30 Vdc).

Digital inputs

The digital PLC inputs offered by the CNC system are used to read external devices, etc.

The electrical characteristics of these inputs are:

Nominal voltage value	+24 V DC.
Maximum voltage value	+30 V DC.
Minimum voltage value	+18 V DC.
High threshold voltage (logic level 1)	from +18 V up.
Low threshold voltage (logic level 0)	Under +5 V.
Typical consumption of each input	5 mA.
Maximum consumption of each input	7 mA.

All inputs are protected by means of:

- Galvanic isolation by optocouplers.
- Protection against reversal of power supply connection up to -30 V.



The external 24Vdc power supply used for the PLC's inputs and outputs MUST be regulated.

The 0V point of this power supply must be connected to the main ground point of the electrical cabinet.

3.2 Analog inputs and outputs

Analog outputs

They may be used with axis and spindle drives. The electrical characteristics of these outputs are:

Command voltage within range	± 10 V.
Minimum impedance of the connected drive:	10 Kw.
Maximum cable length without shield:	75 mm.

Shielded cables should be used connecting the shield at each connector as shown here. See chapter *"1 CNC configuration"*.



It is recommended to adjust the servo drives so the maximum feedrate (G00) is obtained at a velocity command of +9.5V .

3.3 Setup

Some general points to consider

Inspect the whole electrical cabinet verifying the ground connections BEFORE powering it.

This ground connection must be done at a single machine point (Main Ground Point) and all other ground points must be connected to this point.

The power supply used for the digital inputs and outputs must be regulated and its zero volts must be connected to the main ground point.

Check the connection of the cables and connectors. DO NOT connect or disconnect these cables to/from the CNC when the CNC is on.

Without powering the electrical cabinet on, check all the pins of the connectors for short-circuits.

Precautions

It is recommended to reduce the axis travel installing the limit switches closer to each other or detaching the motor from the axis until they are under control.

Verify that there is no power going from the servo drives to the motors.

Verify that the connectors for the digital inputs and outputs are disconnected.

Verify that the E-STOP button is pressed.

Connection

Verify that the A.C. power is correct.

With the CNC completely disconnected from the electrical cabinet, power the electrical cabinet and verify that it responds properly:

Verify that there is proper voltage between the pins corresponding to external 0V and 24V of the connectors for the digital inputs and outputs.

Apply 24V to each one of the terminals of the electrical cabinet being used that correspond to the digital outputs of the CNC and verify their correct performance. Check that the electrical cabinet responds properly.

With the motors being decoupled from the axes, verify that the system consisting of drive, motor and tacho is operating properly.

Connect the A.C. power to the CNC. If there is any problem, the CNC will display the corresponding error.

Select the PLC monitoring mode at the CNC and activate the digital outputs (O1=1) one by one to verify their proper operation.

With power turned off, connect the I/O and feedback connectors to the CNC.

Connect the CNC and the electrical cabinet to A.C. power and confirm the counting direction of each axis.

Introduction to machine parameters

The machine parameters relate the CNC to the particular machine. The values that the CNC assigns to each one of them by default are described in the relevant chapter. See chapter *"4 Machine parameters"*.

These values, shown in the parameter tables, may be modified manually from the CNC's keyboard or from a peripheral (cassette reader, floppy disk reader, computer, etc.) via the RS 232C serial line.

Some characters appear next to certain parameters indicating when the CNC assumes the new value assigned to that parameter.

// It is necessary to press the keystroke sequence: "Shift - Reset" or turn the CNC off and back on.

/ Just press Reset.

The rest of the parameters (those unmarked) will be updated automatically, only by changing them.

Setting of the machine parameters for the axes

Once the active axes have been assigned by means of g.m.p. "AXIS1" (P0) thru "AXIS8" (P7), the CNC will enable the relevant axes parameter tables.

The values to be assigned to the parameters of each of these tables will depend on the results obtained when adjusting each machine axis.

Before making this adjustment, position the axes near the middle of their travel and place the hard stops (monitored by the electrical cabinet) near these mid-travel points in order to prevent any possible damage to the machine.

Verify that the PLC Mark "LATCHM" is OFF. Then, after selecting the parameters of the desired axes, go on to adjusting them following these advises:

- Adjust the axes one by one.
- Connect the power output of the drive corresponding to the axis being adjusted.
- Selecting the Jog mode at the CNC, jog the axis to be adjusted.

  In case of runaway, the CNC will display the relevant following error and the machine parameter labelled LOOPCHG (corresponding to the sign of the velocity command output of the CNC) will have to be changed.
- If the axis does not run away; but the direction of the move is not the desired one, parameters labelled AXISCHG (P13) (axis feedback counting direction) and LOOPCHG (P26) (sign of the velocity command output) will have to be changed.

Machine reference point (home) adjustment for each axis

Once the movement of the axes has been properly adjusted, place the travel-limit switches back where they should be.

The following adjusting sequence is one of the many that could be used:

- This adjustment should be done one axis at a time.
- Indicate in the a.m.p. REFPULSE (P32) the type of marker pulse Io being used for Home Search.
- Set a.m.p. REFDIREC (P33) to indicate the direction of the axis when searching Home.
- Set g.m.p. REFEED1 (P34) and REFEED2 (P35) to indicate the feedrates for Home search.
- a.m.p. REFVALUE (P36) will be set to “0”.
- Once in the JOG mode and after positioning the axis in the right area, start homing the axis. When done, the CNC will assign a "0" value to this point.
- If the machine reference zero is in a different physical location from the machine reference point (location of the marker pulse), proceed as follows:

  After moving the axis to a known position (with respect to Machine Reference Zero), observe the position reading of the CNC for that point.

  This will be the distance away from the machine reference point; thus, the value to be assigned to a.m.p. REFVALUE (P36) will be:

  Machine coordinate of the measured point - CNC reading at that point.

Example:

If the point whose known position is located 230 mm from Machine Reference Zero and the CNC reads -123.5 mm as the coordinate value for this point, the coordinate of the Machine Reference Point with respect to Machine Reference Zero will be:

REFVALUE = 230 - (-123.5) = 353.5 mm.

Assign this new value and press [RESET] so it is assumed by the CNC.

It is also necessary to search Home once again in order for this axis to assume the correct reference values.

Axis travel limits (software limits)

Once all the axes have been referenced, their software limits must be measured and set.

This is achieved a single axis at a time as follows:

- Move the axis in the positive direction towards the end of the axis travel stopping at a safe distance from the mechanical end-of-travel stop.
- Assign the coordinate shown by the CNC for that point to a.m.p. LIMIT+ (P5).
- Repeat these steps in the negative direction assigning the resulting coordinate to a.m.p. LIMIT- (P6).
- Once this process is completed, hit SHIFT RESET or turn the CNC off and back on in order for it to assume the new values.

Adjustment of the drift (offset) and maximum feedrate (G00)

These adjustments are performed on servo drives of the axes and on spindle drives.

Offset (drift) adjustment

Disconnect the velocity command input and short-circuit it with a wire jumper.

Turn the offset potentiometer of the drive until the voltage on the tach terminals is 0mVdc. Check this with a volt meter set at a range of 200 mV.

Remove take the wire jumper that short-circuited the velocity command input.

Maximum feedrate adjustment

It is recommended to adjust the drives so the maximum feedrate is obtained with a velocity command voltage of 9.5V. If they are adjusted to a different voltage, it must be indicated in the a.m.p. or s.m.p. MAXVOLT (P37).

Also, the maximum feedrate must be indicated in the a.m.p. G00FEED (P38).

The maximum feedrate can be calculated from the motor rpm, the gear ratios and the type of leadscrew being used.

Example:

A motor can turn at 3000 rpm and it is attached to a 5 pitch screw (5 mm/turn).

The maximum feedrate will be:

3000 rpm x 5 mm/turn = 15000 mm/minute

This will be the value to be assigned to a.m.p. G00FEED (P38).

Once these values are assigned to the relevant parameters, the drives must be adjusted.

To do so, a CNC program can be executed which will move the axis back and forth continuously at G00 feedrate. One such program could be:

N10 G00 G90 X200

X -200

(GOTO N10)

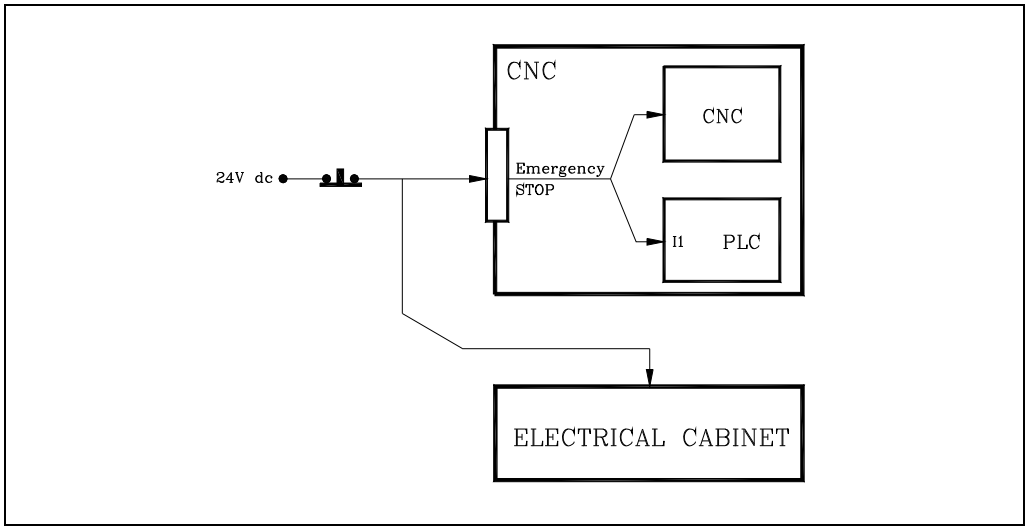
If the Tach in use provides 20V per 1000 rpm, its voltage should be:

(20 V / 1000 rpm) x 3000 rpm. = 60 V

3.4 Connection of the emergency input and output

The emergency input of the CNC corresponds with the PLC input I1 (pin 10 of connector X2) and must be supplied with 24V.

Since the CNC also processes this signal directly, if the 24V disappear, the CNC will display EXTERNAL EMERGENCY ERROR and will deactivate all axes enables and will cancel all velocity commands.



During the initializing process carried out by the CNC on power-up, the EMERGENCY OUTPUT of the CNC (pin 2 of connector X10) remains at low (at “0”) in order to avoid a premature activation of the electrical cabinet.

If this process is successful, the CNC will set the real value of PLC output O1 to “1”. Otherwise, it will keep the /EMERGENCY OUTPUT signal active (low) and it will display the corresponding error message.

Once the initialization process is over, the PLC will execute the PLC program stored in memory. If none is available, it wait for one to be entered and executed.

When the execution of the first cycle (CY1) (or the first program scan) is finished the PLC will assign the value of output O1 to physical output “/EMERGENCY OUTPUT”.

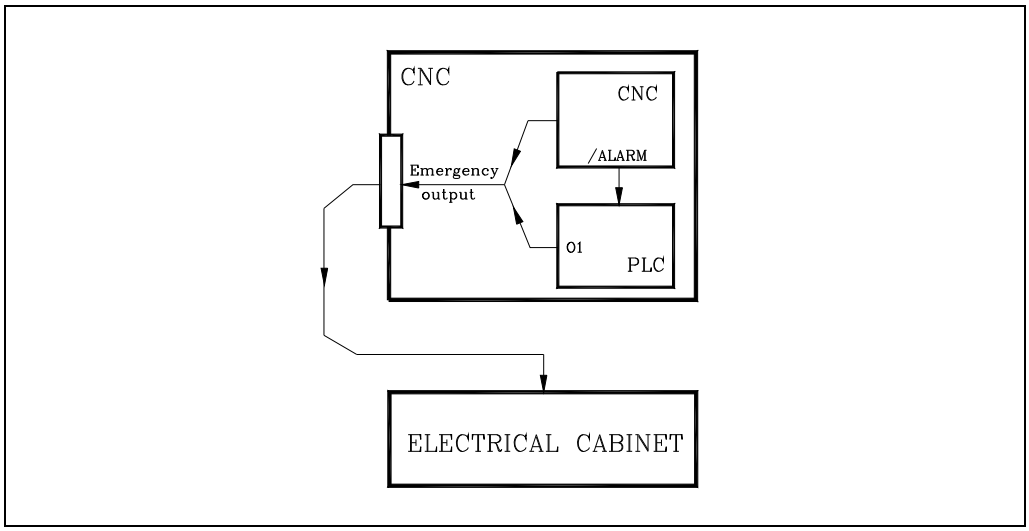
It is recommended to program the CY1 cycle of the PLC program assigning a value of 1 to O1 when everything checks out fine and a value of 0 when there is an error.

The interface of the electrical cabinet will take into account all the elements that could cause this type of error. Among such elements are:

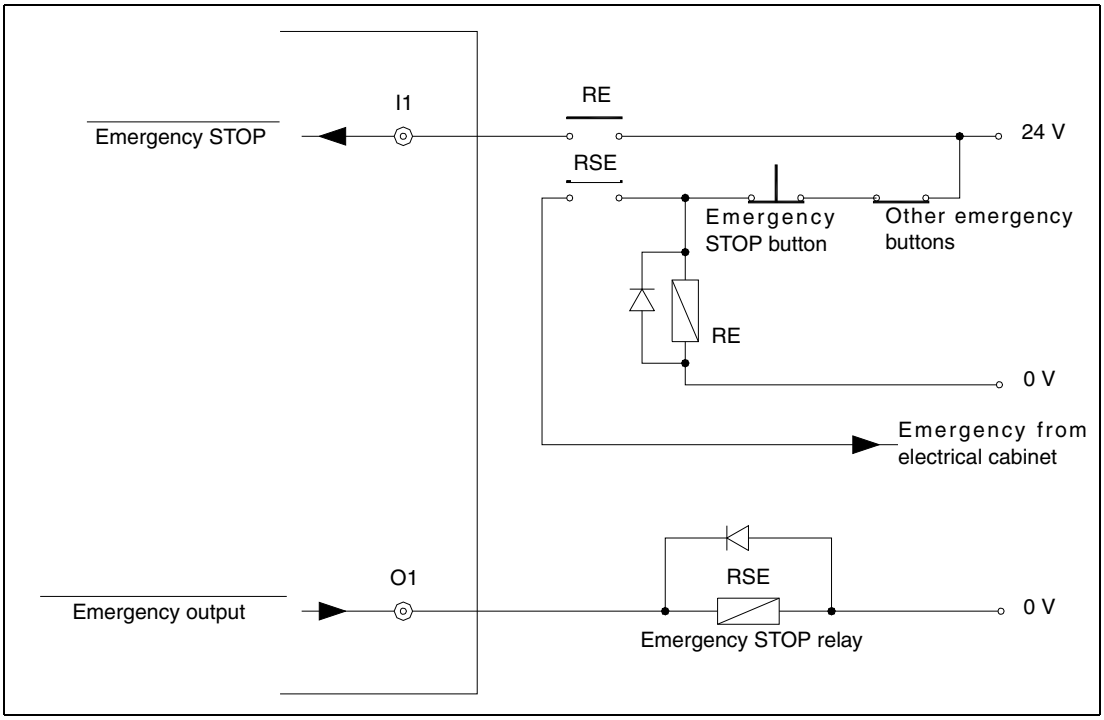
- E-stop has been pressed.
- The travel limit of any axis has be exceeded.
- There is a malfunction on a drive or it is locked without velocity command signal.

When the CNC detects an error, it will indicate it to the PLC with the general logic output "/ALARM" and it will set the emergency output low (pin 2 of connector X2).

Since this signal corresponds to the PLC output O1, it can also be activated by the PLC program.



The recommended connection diagram is the following:



3.

MACHINE AND POWER CONNECTION

Connection of the emergency input and output

FAGOR

CNC 8035

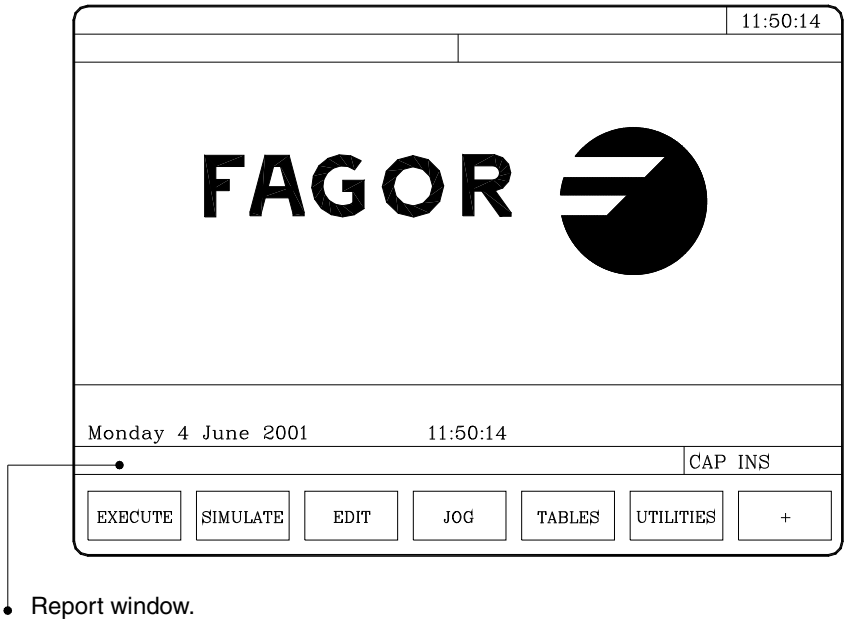
(SOFT M: V15.3X)
(SOFT T: V16.3X)

MACHINE PARAMETERS

4

It is recommended to save the CNC machine parameters into the hard disk (KeyCF) or in a peripheral or PC to avoid losing them.

On power-up, the CNC performs a system autotest and when this is over, it displays the following screen:



The CNC allows the display of a previously defined screen instead of the Fagor logo. Refer to the operating manual.

During the autotest, if any error occurs, its relevant message will be displayed in the report window.

The main menu for the various operating modes will appear at the bottom of the CRT. These options will be selected using the softkeys F1 through F7.

Since it is possible to have more than 7 options to choose from at one time, use the "+" softkey to display the rest of them.

Once the "Machine Parameters" operating mode has been selected, the CNC shows the machine parameter tables that are saved in the hard disk (KeyCF).

The available machine parameter tables are:

- General machine parameters.
- Machine parameters for the axes (one table per axis)
- Spindle parameters.
- Serial line parameters.
- PLC Parameters.
- Auxiliary (miscellaneous) M functions.
- Leadscrew backlash Compensation (one table per axis).
- Cross compensation.

To access each one of them, use the softkeys shown at the bottom of the screen.

Some characters appear next to certain parameters indicating when the CNC assumes the new value assigned to that parameter.

Character	Type of update
//	It is necessary to press the keystroke sequence: [SHIFT] + [RESET] or turn the CNC off and back on.
/	Just do a reset.
	The rest of the parameters (those unmarked) will be updated automatically, only by changing them.

On each table, it is possible to move the cursor line by line using the [⬆] [⬇] keys or page by page using the Page-up and Page-down keys.

Abbreviations used in this manual

The manual uses the following abbreviations to identify the type of machine parameter.

Abbreviation	Machine parameter	Example
g.m.p.	General machine parameter.	g.m. p. CUSTOMTY (P92)
a.m.p.	Axis machine parameter.	a.m.p. AXISTYPE (P0)
s.m.p.	Spindle machine parameter.	s.m.p. MAXGEAR1 (P2)
plc.m.p.	PLC machine parameter.	plc.m.p. WDGPRG (P0)

Operation with parameter tables

Once one of the table lines has been selected, the user can move the cursor over this line by means of the [⬅] [➡] keys .

It is also possible to perform other functions by using the following keys:

Key	Function
[CL]	Deletes characters.
[INS]	Switches between insert and overwrite (replace) writing modes.
[CAP]	Switches between upper case and lower case letters; when the CRT shows CAP, it will indicate that the upper case mode has been selected. Make sure this mode is selected since all characters entered in these tables must be upper case.
[ESC]	Quits line editing.
[ENTER]	Assumes the edited line and ends the editing of the line.

The CNC offers the following options when working with each parameter of these tables:

EDIT	Edit a parameter. The CNC will indicate the proper format by means of the softkeys.
MODIFY	Modify a parameter. Position the cursor on the desired parameter and press the Modify softkey. Once the modification is done, press [ENTER] for the CNC to assume the new value.
FIND	Look for a parameter. The cursor will be positioned over the indicated parameter. With this function it is also possible to "find" the beginning or the end of the table.
INITIALIZE	Initialize the table assuming the default values.
LOAD	Load into memory the tables saved in the hard disk (KeyCF), a peripheral device or a PC.
SAVE	Save the tables into the hard disk (KeyCF), a peripheral device or a PC.
MM/INCHES	See the parameter values in the desired units. Only those parameters affected by this conversion will be altered. It will not change the g.m.p. INCHES (P8) that indicates machine units.

Machine parameter setting

In order for the machine-tool to be able to properly execute the programmed instructions as well as interpret the different elements connected to it, the CNC must "know" the specific data of the machine, such as: feedrates, accelerations, feedback, automatic tool change, etc..

This data is determined by the machine builder and can be introduced either from the CNC's keyboard or via the CNC's two serial ports.

First, the general machine parameters must be set since they determine the machine axes.

There are some parameters to indicate whether the machine has cross compensation or not. This compensation table will be generated by the CNC from the values assigned to those parameters.

The general machine parameters also determine the number of elements at the tables for tools, tool magazine, tool offsets and M functions (miscellaneous).

The axes parameters will define the leadscrew compensation tables and they will only be generated for those axes which require them.



When selecting the drive parameters at the CNC, it is possible to display and modify the parameters stored at each drive.

The CNC does not have parameters of the drive although their copies may be stored in the hard disk (KeyCF).

4.1 Parameters that may be modified from the OEM program or OEM subroutine

Here is a list of the machine parameters that may be modified either from the oscilloscope or from an OEM program/subroutine. The variables associated with the machine parameters must be used to modify these parameters from an OEM program/subroutine. See *"11.3 Variables associated with machine parameters"* on page 374.

General machine parameters:

Parameter	Number	Update
TLOOK	P161	Beginning of program execution
CODISET	P147	Immediate

Machine parameters of an axis:

Parameter	Number	Update
BACKLASH	P14	Immediate
ACCTIME	P18	Beginning of the next block
INPOSW	P19	Immediate
MAXFLWE1	P21	Immediate
MAXFLWE2	P22	Immediate
PROGAIN	P23	Immediate
DERGAIN	P24	Immediate
FFGAIN	P25	Immediate
BAKANOUT	P29	Immediate
BAKTIME	P30	Immediate
REFDIREC	P33	Immediate
REFVALUE	P36	Immediate
MAXVOLT	P37	Immediate
G00FEED	P38	Beginning of the next block
MAXFEED	P42	Beginning of the next block
JOGFEED	P43	Beginning of the next block
ACCTIME2	P59	Beginning of the next block
PROGAIN2	P60	Immediate
DERGAIN2	P61	Immediate
FFGAIN2	P62	Immediate
JERKLIM	P67	Beginning of the next block
FLIMIT	P75	Beginning of the next block
TORQDIST	P78	Immediate
PRELOAD	P79	Immediate
TPROGAIN	P81	Immediate
TINTTIME	P82	Immediate
TCOMPLIM	P83	Immediate

Spindle machine parameters:

Parameter	Number	Update
MAXGEAR1	P2	Beginning of the next block
MAXGEAR2	P3	Beginning of the next block
MAXGEAR3	P4	Beginning of the next block
MAXGEAR4	P5	Beginning of the next block
ACCTIME	P18	Beginning of the next block
INPOSW	P19	Immediate
PROGAIN	P23	Immediate
DERGAIN	P24	Immediate
FFGAIN	P25	Immediate
REFDIREC	P33	Immediate
REFVALUE	P36	Immediate
MAXVOLT1	P37	Immediate
MAXVOLT2	P38	Immediate
MAXVOLT3	P39	Immediate
MAXVOLT4	P40	Immediate
OPLACETI	P45	Immediate
ACCTIME2	P47	Beginning of the next block
PROGAIN2	P48	Immediate
DERGAIN2	P49	Immediate
FFGAIN2	P50	Immediate
SLIMIT	P66	Immediate
JERKLIM	P80	Beginning of the next block

A modification in the MAXGEAR(1··4) parameters sets the square corner mode even if a round corner has been programmed.

4.2 General machine parameters

AXIS1 (P0) **AXIS2 (P1)** **AXIS3 (P2)** **AXIS4 (P3)**
AXIS5 (P4) **AXIS6 (P5)** **AXIS7 (P6)** **AXIS8 (P7)**

They permit associating axes, handwheels, spindles or live tools with each feedback input and analog output according to the following code:

Value	Meaning	Value	Meaning
0	Free; not associated.	12	Handwheel with axis selector button
1	X axis.	13	
2	Y axis.	14	
3	Z axis.	21	Handwheel associated with X.
4	U axis.	22	Handwheel associated with Y.
5	V axis.	23	Handwheel associated with Z.
6	W axis.	24	Handwheel associated with U.
7	A axis.	25	Handwheel associated with V.
8	B axis.	26	Handwheel associated with W.
9	"C" axis.	27	Handwheel associated with A.
10	Main spindle.	28	Handwheel associated with B.
11	Handwheel.	29	Handwheel associated with C.

The following table shows the feedback input, the velocity command output and the default values associated with each parameter.

Parameter		Feedback (connector)	Command	Default value - M -	Default value - T -
AXIS1 (P0)	1st axis	X10	X8 - Pin 2	1 (X axis)	1 (X axis)
AXIS2 (P1)	2nd axis	X11	X8 - Pin 3	2 (Y axis)	3 (Z axis)
AXIS3 (P2)	3rd axis	X12	X8 - Pin 4	3 (Z axis)	0 (free)
AXIS4 (P3)	Not being used	----	----	0 (free)	0 (free)
AXIS5 (P4)	Spindle	X4	X4 - Pins 10 and 12	10 (spindle)	10 (spindle)
AXIS6 (P5)	1st handwheel	X5	----	11 (handwheel)	11 (handwheel)
AXIS7 (P6)	2nd handwheel	X5	----	0 (free)	0 (free)
AXIS8 (P7)	Not being used	----	----	0 (free)	0 (free)

About the handwheels

Depending on their configuration, the available handwheels are:

- General handwheel.

  It can be used to jog any axis one by one. Select the axis and turn the handwheel to move it.

- Individual handwheel.

  It replaces the mechanical handwheels. Up to 2 handwheels can be used (one per axis). It only moves the axis it is associated with.

When using a Fagor 100P handwheel, no other handwheels may be used and it must be connected as first handwheel. See *"5.3 Movement with an electronic handwheel"* on page 190.

INCHES (P8)

It defines the measuring units assumed by the CNC for machine parameters, tool tables and programming on power-up and after executing M02,M30, EMERGENCY or RESET. The code is:

Value	Meaning
0	Millimeters (G71)
1	Inches (G70)

By default: 0

IMOVE (P9)

Indicates which function G00 (rapid traverse) or G01 (linear interpolation) is assumed on power-up, after executing M02,M30, EMERGENCY or RESET. The code is:

Value	Meaning
0	G00 (rapid traverse).
1	G01 (linear interpolation).

Default value: 0

ICORNER (P10)

Indicates which function, G05 (round corner) or G07 (square corner) is assumed on power-up, after executing M02,M30, EMERGENCY or RESET. The code is:

Value	Meaning
0	G07 (square corner).
1	G05 (round corner).

Default value: 0

IPLANE (P11)

Indicates which function: G17 (XY plane) or G18 (ZX plane) is assumed on power-up, after executing M02,M30, EMERGENCY or RESET. The code is:

Value	Meaning
0	G17 (XY plane).
1	G18 (ZX plane).

Default value: 0 (for the M model)
Default value: 1 (for the T model)

ILCOMP (P12)

It is only used in the Mill model CNC and indicates which function: G43 (tool length compensation ON) or G44 (tool length compensation OFF) is assumed on power-up, after executing M02,M30, EMERGENCY or RESET; the code is:

Value	Meaning
0	G44 (tool length compensation ON).
1	G43 (tool length compensation ON).

Default value: 0

ISYSTEM (P13)

Indicates which function: G90 (absolute programming) or G91 (incremental programming) is assumed on power-up, after executing M02,M30, EMERGENCY or RESET.

Value	Meaning
0	G90 (absolute programming).
1	G91 (incremental programming).

Default value: 0

IFEED (P14)

Indicates which function: G94 (feedrate in mm/min or inch/min) or G95 (mm/rev or inch/rev) is assumed on power-up, after executing M02,M30, EMERGENCY or RESET.

Value	Meaning
0	G94 (mm/min or inches/min).
1	G95 (mm/rev or inches/rev).

Default value: 0

THEODPLY (P15)

Indicates whether the CNC will display real or theoretical position values according to the following code:

Value	Meaning
0	Real position values.
1	Theoretical position values.

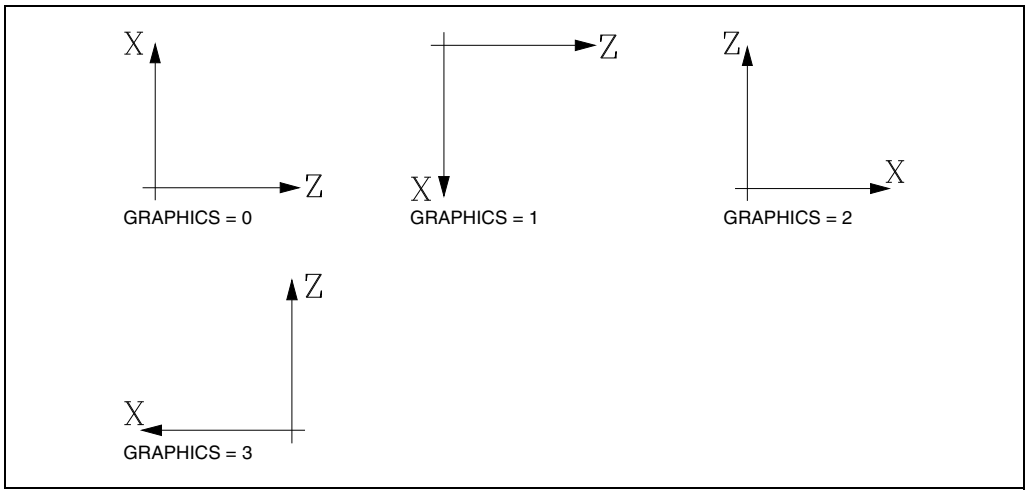
Default value: 1

GRAPHICS (P16)

On the T model, it indicates the axes coordinates system to be used for the graphic representation. In this model, it also defines the layout of the keys of the X-Z axis on the jog keypad; on vertical lathes, the X axis keys are swapped with those of the Z axis and vice versa.

Possible values
Integer numbers between 0, 1, 2, 3.

Default value: 0

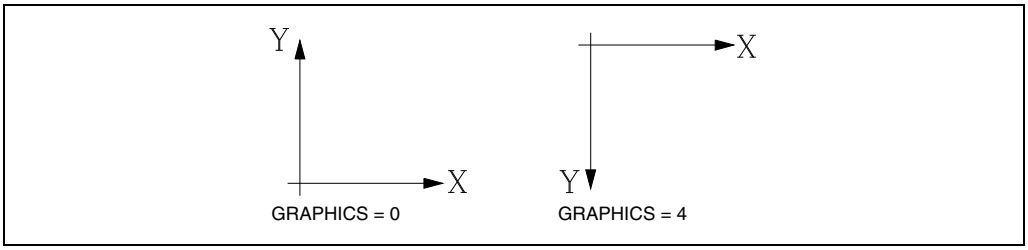


On the M model, it indicates the axis system being used for the graphic representation as well as the motion possibilities for the W axis added to those of the Z axis in the graphic representation (W additive).

Value	Meaning
0	Mill graphics.
1	Mill graphics with added W axis.
2	Boring Mill graphics.
3	Boring Mill graphics with added W axis.
4	Mill graphics (changed line graphics).

Default value: 0

The ·4· value is only available when having PowerPC.



RAPIDOVR (P17)

Indicates whether it is possible to vary the feedrate override between 0% and 100% when working in G00.

Value	Meaning
YES	It may be modified.
NO	It cannot be modified; it is set to 100 %.

Default value: NO

The feedrate override % may be changed from the operator panel switch, from the PLC, via DNC or by program.

The feedrate % can always be changed in JOG movements.

MAXFOVR (P18)

Indicates the maximum value of the feedrate override % applicable to the programmed feedrate.

Possible values
Integer numbers between 0 and 255.

Default value: 120

From the operator panel switch, it may be varied between 0% and 120% and from the PLC, DNC or by program between 0% and 255%.

CIRINLIM (P19)

Indicates the maximum angular feedrate value for circular interpolations.

This limitation prevents circular interpolations resulting in polygons instead of arcs when the radius is very small. The CNC adjusts the angular feedrate in order not to exceed the selected maximum angular feedrate.

Possible values
Integer numbers between 0 and 65535.

Default value: 0 (it is not limited)

Example

If "CIRINLIN" = 1500 and an arc of a radius = 0.5mm at F=10000mm/min.

The theoretical angular speed is:

10000 mm/min / 0.5 mm = 20000 min^{-1}

But, since the speed was limited to 1500, the CNC adjusts the feedrate in the following manner:

Feedrate to be applied = 1500 x 0.5 = 750 mm/min.

CIRINERR (P20)

Indicates the maximum error allowed when calculating the end point of an arc.

From the programmed path, the CNC will calculate the radius for both the starting point and end point of the arc. Although both of them should be "exactly" the same, This parameter allows a certain calculation tolerance by establishing the maximum difference between these two radii.

Possible values
Between 0.0001 and 99999.9999 millimeters. Between 0.00001 and 3937.00787 inches.

Default value: 0.01 mm.

PORGMOVE (P21)

Indicates whether the CNC assumes or not as the new polar coordinate origin the center of the last G02 or G03 programmed.

Value	Meaning
YES	It assumes the arc center.
NO	It is not affected by G02 and G03.

Default value: NO

BLOCKDLY (P22)

It indicates the delay between motion blocks when operating in G7 (square corner).

This dwell can be very useful when some devices have to activated after the execution of each block.

Possible values
Integers between 0 and 65535 ms.

Default value: 0 (there is no delay)

NTOOL (P23)

Indicates the number of tools in the tool magazine. On the other hand, the CNC adjusts the length of the tool table to that value.

Possible values
Integer numbers between 0 and 255.

Default value: 100

NPOCKET (P24)

Indicates the number of pockets in the tool magazine. On the other hand, the CNC adjusts the length of the tool magazine table to that value.

Possible values
Integer numbers between 0 and 255.

Default value: 100 (for the M model)
Default value: 0 (for the T model)

RANDOMTC (P25)

Indicates whether the tool magazine is random or not.

- On a random magazine, the tools may occupy any position (pocket). If this machine parameter is set for random magazine, g.m.p. TOFFM06 (P28) must be set for machining center.
- On a non-random magazine, the tool always occupies its own pocket. The magazine position number is the same as the tool number.

Value	Meaning
YES	It is a random tool magazine.
NO	It is not a random tool magazine.

Default value: NO

In a non-random magazine, the tools must be placed in the tool magazine table in the pre-established order (P1 T1, P2 T2, P3 T3, etc.). Optionally, g.m.p. TOOLMATY (P164) may be used to assign several different tools to each tool position. In this case, the magazine position number may be different from the tool number.

TOOLMONI (P26)

Selects the display units of the tool's nominal and real lives.

Value	Meaning
0	Tool life in minutes
1	Tool life in number of operations.

Default value: 0

NTOFFSET (P27)

Indicates the number of tool offsets available in the tool offset table. On the other hand, the CNC adjusts the length of the tool offset magazine table to that value.

Possible values
Integer numbers between 0 and 255.

Default value: 100

TOFFM06 (P28)

Indicates if the machine is a machining center.

If it is, the CNC will select, at the tool magazine, the tool indicated when executing the "T" function and it will be necessary to execute M06 afterwards in order to carry out the tool change.

Value	Meaning
YES	Yes, it is a machining center.
NO	It is not a machining center.

Default value: NO

It is recommended to associate he subroutine corresponding to the tool changer with the M06.

NMISCFUN (P29)

Indicates the number of M functions available in the M function table.

Possible values
Integer numbers between 0 and 255.

Default value: 32

MINAENDW (P30)

Indicates the minimum time period that the AUXEND signal must remain activated so the CNC will interpret it as a valid signal. AUXEND is a PLC signal which indicates to the CNC that functions M,S or T have been executed.

If the corresponding M function has been set in the M table not to wait for the AUXEND signal, the time period indicated in this parameter will be the duration of the MSTROBE signal.

Possible values
Integers between 0 and 65535 ms.

Default value: 100

See *"5.8 Auxiliary M, S, T function transfer"* on page 221.

NPCROSS (P31)

Indicates the number of points available in the first cross compensation table.

This compensation is used when the movement of one axis causes a position change on another axis. The CNC offers a table where one could enter the position variations of one axis for the particular positions of the other axis.

Possible values
Integer numbers between 0 and 255.

Default value: 0 (not available)

MOVAXIS (P32)

Used in the first cross compensation table, it indicates the axis causing position variations on another axis. The definition code is:

Value	Meaning	Value	Meaning
0	None.	5	V axis.
1	X axis.	6	W axis.
2	Y axis.	7	A axis.
3	Z axis.	8	B axis.
4	U axis.	9	"C" axis.

Default value: 0 (none)

COMPAXIS (P33)

Used in the first cross compensation table, it indicates the axis suffering the position variations caused by another axis. The compensation is applied onto this axis. The definition code is:

Value	Meaning	Value	Meaning
0	None.	5	V axis.
1	X axis.	6	W axis.
2	Y axis.	7	A axis.
3	Z axis.	8	B axis.
4	U axis.	9	"C" axis.

Default value: 0 (none)

Example

If NPCROSS=20, MOVAXIS=X and COMPAXIS=W, the CNC will allow access to the cross compensation table.

Each one of these 20 points (NPCROSS) of this table will indicate the X position value and the error suffered by the W axis when the X axis is positioned at this point.

This way, the CNC will apply the compensation of the X axis table onto the W axis.

REFPSUB (P34)

Indicates the number of the subroutine associated with function G74 (machine reference zero or home search). This subroutine will be executed automatically when G74 is programmed alone in a block or, also, when searching home in the JOG mode by pressing the softkey "ALL AXES".

Possible values
Integer numbers between 0 and 9999.

Default value: 0 (no associated subroutine)

INT1SUB (P35) INT2SUB (P36) INT3SUB (P37) INT4SUB (P38)

They indicate the number of the subroutine associated with the corresponding general logic input: "INT1" (M5024), "INT2" (M5025), "INT3" (M5026)", "INT4" (M5027).

When one of these inputs is activated, the program currently being executed is interrupted and the CNC jumps to execute the associated subroutine whose number is indicated in the corresponding parameter.

These interruption subroutines do not change the nesting level of local parameters, thus only global parameters must be used in them.

Once the CNC completes the execution of the subroutine, it will continue running the original program.

Possible values
Integer numbers between 0 and 9999.

Default value: 0 (no associated subroutine)

PRBPULSE (P39)

Indicates whether the probe functions of the CNC react to the up-flank (leading edge) or down-flank (trailing edge) of the signals provided by the probes connected through connector X3.

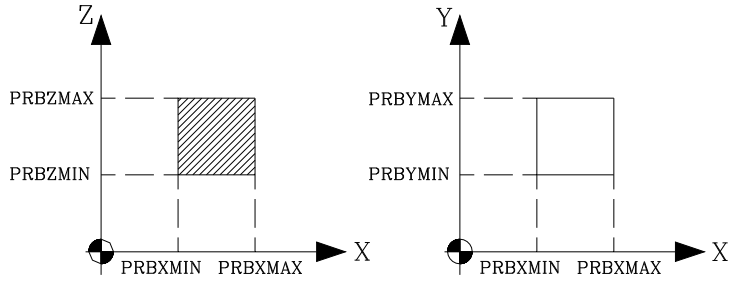
Value	Meaning
+ sign	Positive pulse (24 V or 5 V).
- sign	Negative pulse (0 V).

Default value: + sign

PRBXMIN (P40) **PRBXMAX (P41)** **PRBYMIN (P42)**
PRBYMAX (P43) **PRBZMIN (P44)** **PRBZMAX (P45)**

Indicate the position of the tabletop probe used for tool calibration.

These position values must be absolute and with respect to machine reference zero (home). If a lathe model CNC, these values must be in radius.



PRBXMIN	Probe's minimum X coordinate.
PRBXMAX	Probe's maximum X coordinate.
PRBYMIN	Probe's minimum Y coordinate.
PRBYMAX	Probe's maximum Y coordinate.
PRBZMIN	Probe's minimum Z coordinate.
PRBZMAX	Probe's maximum Z coordinate.

Possible values
±99999.9999 mm or ±3937.00787 inches.

Default value: 0

PRBMOVE (P46)

Indicates the maximum distance the tool can travel when calibrating it with a probe in JOG mode.

Possible values
Between 0.0001 and 99999.9999 millimeters. Between 0.00001 and 3937.00787 inches.

Default value: 50 mm.

USERDPLY (P47)

Indicates the number of the user program associated with the execution mode. This program will be executed via the user channel when pressing the softkey USER in the EXECUTE mode.

Possible values
Integer numbers between 0 and 65535.

Default value: 0 (none)

USEREDIT (P48)

Indicates the number of user program associated with the Edit mode. This program will be executed via the user channel when pressing the softkey USER in the EDIT mode.

Possible values
Integer numbers between 0 and 65535.

Default value: 0 (none)

USERMAN (P49)

Indicates the number of the user program associated with the JOG mode. This program will be executed via the user channel when pressing the softkey USER in the JOG mode.

Possible values
Integer numbers between 0 and 65535.

Default value: 0 (none)

USERDIAG (P50)

Indicates the number of the user program associated with the Diagnosis mode. This program will be executed via the user channel when pressing the softkey USER in the DIAGNOSIS mode.

Possible values
Integer numbers between 0 and 65535.

Default value: 0 (none)

ROPARMIN (P51) ROPARMAX (P52)

They indicate the upper limit "ROPARMAX " and lower limit "ROPARMIN" of the global arithmetic parameter group (P100-P299), user arithmetic parameters (P1000-P1255) or OEM arithmetic parameters (P2000-P2255) to be write-protected. There are no restrictions to read these parameters.

Possible values
Integer numbers between 0 and 9999.

Default value: 0 (it is not protected)

The parameters write-protected from the CNC may be modified from the PLC.

PAGESMEM (P53) NPCROSS2 (P54) MOVAXIS2 (P55) COMAXIS2 (P56) NPCROSS3 (P57) MOVAXIS3 (P58) COMAXIS3 (P59)

Not being used.

TOOLSUB (P60)

Indicates the number of the subroutine associated with the tools. This subroutine will be executed automatically every time a T function is executed.

Possible values
Integer numbers between 0 and 9999.

Default value: 0 (none)

CYCATC (P61)

This parameter must be used when having a machining center, g.m.p. TOFFM06 (P28) = YES.

Indicates whether a cyclic tool changer is being used or not.

A "cyclic tool changer" is an automatic tool changer which requires an M06 command (tool change) after searching for a tool and before searching for 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).

Value	Meaning
YES	It is a cyclic changer.
NO	It is not a cyclic changer.

Default value: YES

TRMULT (P62) **TRPROG (P63)** **TRDERG (P64)**
MAXDEFLE (P65) **MINDEFLE (P66)** **TRFBAKAL (P67)**

Not being used.

TIPDPLY (P68)

Indicates whether the CNC displays the position of the tool tip or that of the tool base when working with tool length compensation.

Value	Meaning
0	It displays the coordinate of the tool base.
1	It displays the coordinate of the tool tip.

Default value: 0 (for the M model)
Default value: 1 (for the T model)

On the Mill model, it is necessary to execute G43 in order to work with tool length compensation. When not working with tool length compensation (G44), the CNC displays the tool base position.

On the Lathe model, it always works with tool length compensation. Therefore, by default, the CNC always displays the tool tip position.

ANTIME (P69)

It is used on punch presses that have an eccentric cam as a punching system.

It indicates how far in advance the general logic output ADVINPOS (M5537) is activated before the axes reach position.

This reduces idle time, thus increasing the number of punches per minute.

Possible values
Integers between 0 and 65535 ms.

Default value: 0

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

If ANTIME = 0, the anticipation signal ADVINPOS will never be activated.

PERCAX (P70)

Not being used.

TAFTERS (P71)

g.m.p. TOOLSUB (P60) indicates the number of the subroutine associated with the tool.

The TAFTERS parameter determines whether the tool selection is carried out before or after executing that subroutine.

Value	Meaning
YES	After executing the subroutine.
NO	Before executing the subroutine.

Default value: NO

LOOPTIME (P72)

It sets the sample period used by the CNC and, consequently, affects the block processing time.

Value	Meaning
0	4 ms period (standard).
4	Period of 4 milliseconds.
5	Period of 5 milliseconds.
6	Period of 6 milliseconds.

Sampling periods shorter than 2 msec. are not allowed when not using the CPU-TURBO option.

Likewise, the CNC configuration limits the sample period. The shorter the sample period, the less time will the CPU have to process data. Therefore, bear in mind that:

- *Sinewave feedback requires more calculation time.*
- *More axes means more calculation time.*
- *if the user channel is active, more calculation time is required.*

IPOTIME (P73)

It sets the interpolation period used by the CNC and, consequently it affects its block processing time.

Value	Meaning
0	IPOTIME = LOOPTIME.
1	IPOTIME = 2 * LOOPTIME.

COMPTYPE (P74)

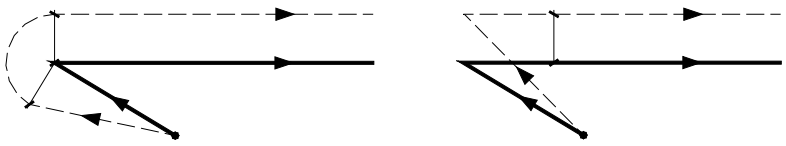
It determines how tool radius compensation is applied. This parameter has three digits.

(units) Type of tool radius compensation beginning and end.

The units set the type of beginning/end of tool radius compensation applied by the CNC.

Value	Meaning
xx0	It approaches the starting point going around the corner.
xx1	It goes directly perpendicular to the point; without going around the corner.

Default value: 0



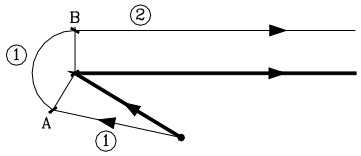
COMPTYPE= x0 COMPTYPE= x1

(tens) Additional compensation block.

The tens indicate whether the additional compensation block is executed at the end of the current block or at the beginning of the next block with compensation.

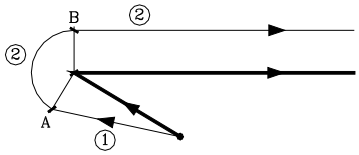
Value	Meaning
x00	It is executed at the end of the current block.
x10	It is executed at the beginning of the next block with compensation.

Default value: 00



COMPTYPE = 00

Executing block by block (single block mode), the first movement ends at point "B".

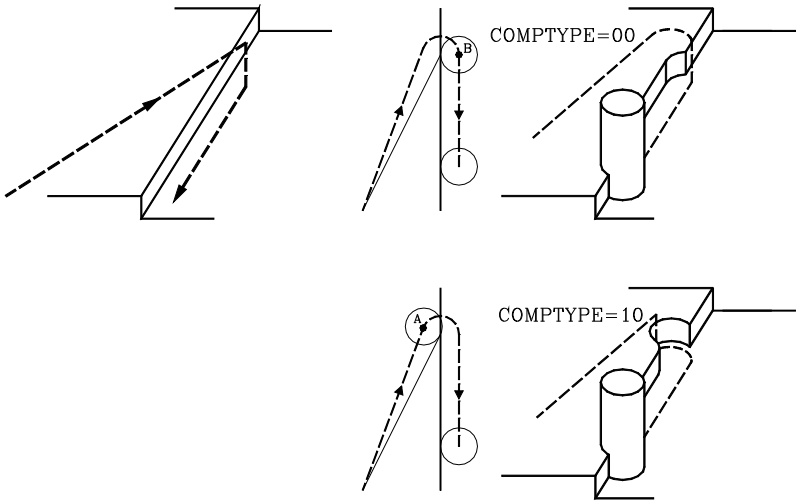


COMPTYPE = 10

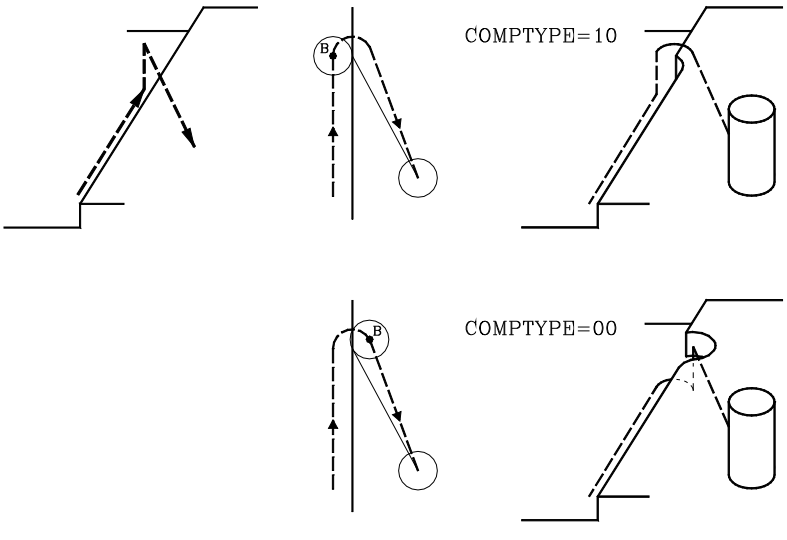
Executing block by block (single block mode), the first movement ends at point "A".

When the beginning or the end of the compensation takes place in a different plane (there is an intermediate vertical movement) and with angle greater than 270º, one should be analyze the CNC's behavior as shown next:

- At the beginning of the compensation, the tool should be positioned before penetrating into the part. The additional block must be executed in the upper plane and, consequently, together with the first block (COMPTYPE=00).



- At the end of the compensation, the tool should withdraw from the part without penetrating into it. The additional block must be executed in the upper plane and, consequently, together with the second block (COMPTYPE=10).



(hundreds) Activate the compensation in the first motion block.

The hundreds indicate whether the compensation is activated in the first motion block or not, even if the plane axes are not involved. The same criteria also applies when turning the compensation off.

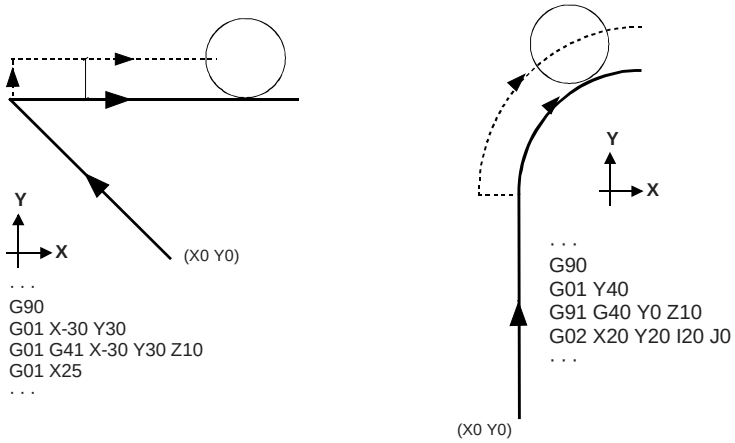
Value	Meaning
0xx	The compensation is activated in the first block having a movement of the plane axes.
1xx	The compensation is activated in the first motion block even if there is no movement of the plane axes.

Default value: 000

After activating the compensation, it could happen that the plane axes do not get involved in the first motion block either because they have not been programmed or because the same point as the tool position has been programmed or because a null incremental move has been programmed. In this case, the compensation is applied in the current tool position; depending on the first movement programmed in the plane, the tool moves perpendicular to the path on its starting point.

The first movement programmed in the plane may be either linear or circular.

Example of beginning of compensation (COMPTYPE=1x1)



FPRMAN (P75)

It is only used on lathe model CNC's and it indicates whether feedrate per revolution is permitted or not.

Value	Meaning
YES	Admitted.
NO	Not admitted.

Default value: NO

MPGAXIS (P76)

It is only used on the Lathe model CNCs and it indicates which axis the handwheel is assigned to. It is set according to the following codes:

Value	Meaning	Value	Meaning
0	None.	5	V axis.
1	X axis.	6	W axis.
2	Y axis.	7	A axis.
3	Z axis.	8	B axis.
4	U axis.	9	"C" axis.

Default value: 0 (shared)

DIRESET (P77)

It is used on the lathe model CNC. It indicates whether the RESET is effective with or without a previous CYCLE STOP.

Value	Meaning
YES	The CNC accepts the RESET any time.
NO	Only if the STOP condition occurs.

Default value: NO

If DIRESET=YES, the CNC first carries out an internal CYCLE STOP to interrupt program execution and, then, executes the RESET.

Obviously, if it is performing a threadcutting or similar operation, not admitting a CYCLE STOP, it will wait for the operation to be concluded before interrupting the program.

PLACOMP (P78)

Not being used.

MACELOOK (P79)

When using "Look-Ahead" the operator sets the percentage of acceleration being applied in Look-Ahead by means of function G51.

With g.m.p. MACELOOK (P79) the OEM can limit the maximum percentage of acceleration that the user may set with G51.

Possible values
Integer numbers between 0 and 255.

Default value: 0 (there is no limit)

MPGCHG (P80) MPGRES (P81) MPGNPUL (P82)

These parameters must be used when having an electronic handwheel to jog the axes.

MPGCHG (P80)

Parameter MPGCHG (P80) indicates the turning direction of the electronic handwheel. If correct, leave it as is. Otherwise, select YES is there was a NO before or vice versa.

Possible values
NO / YES.

Default value: NO

MPGRES (P81)

Parameter MPGRES (P81) indicates the counting resolution of the electronic handwheel and depends on the display format selected for the corresponding a.m.p. DFORMAT (P1).

Possible values
0, 1 and 2.

Default value: 0

Format	Resolution		
DFORMAT (P1)	MPGRES=0	MPGRES=1	MPGRES=2
5.3 mm 4.4"	0.001 mm 0.0001"	0.010 mm 0.0010"	0.100 mm 0.0100"
4.4 mm 3.5"	0.0001 mm 0.00001"	0.0010 mm 0.00010"	0.0100 mm 0.00100"
6.2 mm 5.3"	0.01 mm 0.001"	0.10 mm 0.010"	1.00 mm 0.100"

MPGNPUL (P82)

Parameter MPGNPUL (P82) indicates the number of pulses per turn of the electronic handwheel.

Possible values
Integer numbers between 0 and 65535.

Default value: 0 (means 25)

Example

Having a Fagor electronic handwheel (25 pulses per turn) we would like to move 1 mm per handwheel turn.

1. Set the a.m.p. for the feedback input of the electronic handwheel AXIS1 (P0) through AXIS7 (P6), to a value of 12 (Fagor 100P handwheel). Also set g.m.p. MPGAXIS (P76) to indicate which axis has been assigned this handwheel.
2. Set parameter MPGNPUL=25 or 0 meaning 25 pulses per turn of the Fagor handwheel.
3. Since the handwheel outputs square signals and the CNC applies a x4 multiplying factor to them, we get 100 pulses per turn.
4. The value to be assigned to parameter MPGRES depends on the axis resolution format.

   With 5.3mm type display format, set MPGRES=1

   With 4.4mm type display format, set MPGRES=2

   With 6.2mm type display format, set MPGRES=0

Format		Resolution		
		MPGRES=0	MPGRES=1	MPGRES=2
5.3 mm	Resolution Pulses / turn	0.001 mm 0.100 mm	0.010 mm 1.000 mm	0.100 mm 10.000 mm.
4.4 mm	Resolution Pulses / turn	0.0001 mm 0.0100 mm	0.0010 mm 0.1000 mm	0.0100 mm 1.0000 mm
6.2 mm	Resolution Pulses / turn	0.01 mm 1.00 mm	0.10 mm 10.000 mm	1.00 mm 100.000 mm

MPG1CHG (P83) **MPG1RES (P84)** **MPG1NPUL (P85)** **MPG2CHG (P86)**
MPG2RES (P87) **MPG2NPUL (P88)** **MPG3CHG (P89)** **MPG3RES (P90)**
MPG3NPUL (P91)

These parameters must be used when the machine has several electronic handwheels, one per axis.

Set the a.m.p. for the feedback input of the electronic handwheel AXIS1 (P0) through AXIS7 (P6), to one of the following values:

Value	Meaning	Value	Meaning
21	Handwheel associated with X.	26	Handwheel associated with W.
22	Handwheel associated with Y.	27	Handwheel associated with A.
23	Handwheel associated with Z.	28	Handwheel associated with B.
24	Handwheel associated with U.	29	Handwheel associated with C.
25	Handwheel associated with V.		

Parameters "MPG1***" correspond to the first handwheel, "MPG2***" to the second one and "MPG3***" to the third one.

The CNC uses the following order to know which one is the first, second and third handwheel: X, Y, Z, U, V, W, A, B, C.

The meaning of parameters MPG*CHG, MPG*RES and MPG*NPUL is similar to the meaning of parameters MPGCHG (P80), MPGRES (P81) and MPGNPUL (P82).

CUSTOMTY (P92)

It indicates the configuration being used.

Possible values
250.

Default value: 0

Keyboard auto-identification

The keyboard has an auto-identification system that updates this parameter automatically.

The auto-identification system of the keyboards is recognized from versions V9.11 and V10.11 on.

If an auto-identifying keyboard is connected to a CNC that has an older software version, the keyboard will beep. In this case, disable the auto-identification hardware of the keyboard by setting the identification switch to zero.

XFORM (P93) **XFORM1 (P94)** **XFORM2 (P95)**

Not being used.

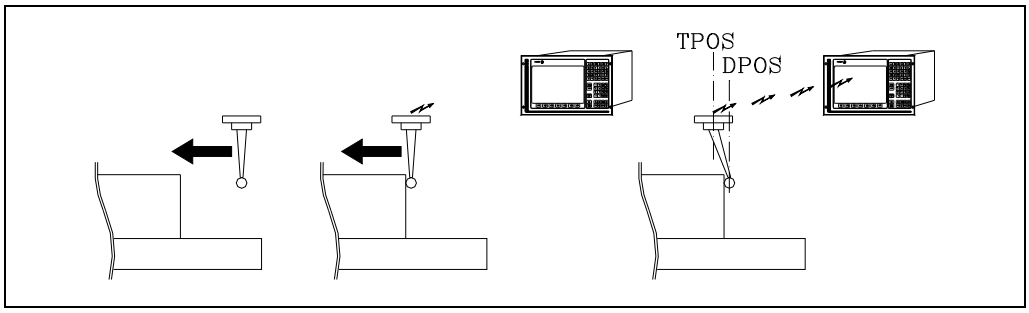
XDATA0 (P96) **XDATA1 (P97)** **XDATA2 (P98)** **XDATA3 (P99)**
XDATA4 (P100) **XDATA5 (P101)** **XDATA6 (P102)** **XDATA7 (P103)**
XDATA8 (P104) **XDATA9 (P105)**

Not being used.

PRODEL (P106)

The CNC takes this parameter into account when probing, functions G75 and G76.

When the digital probe communicates with the CNC via infrared beams, there could be some delay (milliseconds) from the time the probe touches the part to the instant the CNC receives the probe signal.



The probe keeps moving until the CNC receives the probe signal.

Parameter PRODEL indicates, in milliseconds, the delay mentioned earlier.

Possible values
Integer numbers between 0 and 255.

Default value: 0

While probing, the CNC always takes into account the value assigned to parameter PRODEL and provides the following information (variables associated with the coordinates).

TPOS Actual position of the probe when the CNC receives the probe signal.

DPOS Theoretical position of the probe when the probe touched the part.

With "PRODEL=0", the DPOS variable has the same value as the TPOS variable.

MAINOFFS (P107)

Indicates whether the CNC maintains the tool offset number (D) on power-up and after an EMERGENCY or RESET.

Value	Meaning
0	It does not maintain it. It always assumes offset D0.
1	It maintains it.

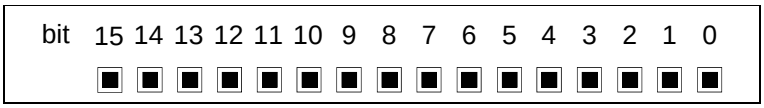
Default value: 0

ACTGAIN2 (P108)

The axes and the spindle can have 3 sets of gains and accelerations. By default, the CNC always assumes the first set indicated by the parameters of the axis or of the spindle ACCTIME, PROGAIN, DERGAIN and FFGAIN.

Parameter ACTGAIN2 indicates when the CNC assumes the second set of gains and accelerations, indicated by the parameters of the axis or of the spindle ACCTIME2, PROGAIN2, DERGAIN2 and FFGAIN2.

This parameter has 16 bits counted from right to left.



Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning	Bit	Meaning
0	Threading in blind threads (for lathe only)	8	G51
1	G34	9	G50
2		10	
3	G74	11	
4	JOG	12	
5	Rigid tapping	13	G33
6	G95	14	G01
7	G75 / G76	15	G00

Default value in all the bits: 0

Every time each of the functions or work modes assigned to the bits of g.m.p. ACTGAIN2 (P108) or ACTGAINT (P185) is activated, the CNC checks the value assigned to the bit corresponding to that function in these parameters and acts as follows:

- If the bits of ACTGAIN2 is set to ·0· and the bit of ACTGAINT is set to ·0·, it applies the first set "ACCTIME, PROGAIN, etc".
- If the bit of ACTGAINT2 is set to ·1· and the bit of ACTGAINT is set to ·0·, it applies the third set "ACCTIME2, PROGAIN2, etc".
- If the bits of ACTGAINT is set to ·1· and the bit of ACTGAIN2 is set to ·0·, it applies the third set "ACCTIMET, PROGAINT, etc".

When that function or work mode is deactivated, the CNC applies the first of the sets "ACCTIME, PROGAIN".

Example

When setting ACTGAIN2 = 1000 0000 0001 0000 and ACTGAINT = 0000 0000 0000 0000, the CNC applies the second set to all the axes and the spindle whenever function G0 or the JOG mode is selected.

Considerations to bear in mind

The change of gains and accelerations is always made at the beginning of the block. When working in round corner (G5), the change does not take place until G07 is programmed.

Example ·1·		**Example ·2·**	
G2 X10 Y10 I10 J0	(Set 1)	G05 G2 X10 Y10 I10 J0	(Set 1)
G1 X20	(Set 2)	G1 X20	(Set 1)
G3 X30 Y20 I0 J10	(Set 1)	G3 X30 Y20 I0 J10	(Set 1)
G1 Y30	(Set 2)	G7 G1 Y30	(Set 2)

The gains and accelerations may also be changed from the PLC. To do that, there is a general logic CNC input ACTGAIN2 (M5013). Every time this input is activated, the CNC selects the second set of gains and accelerations regardless of the active operating mode or function.

TRASTA (P109)

Not being used.

DIPLCOF (P110)

This parameter indicates whether the CNC takes into consideration or not the additive zero offset when displaying the coordinates of the axes on the screen and when accessing the POS(X-C) and TPOS(X-C) variables.

Value	Meaning
0	When displaying the position of the axes referred to home, it only takes into account the additive offset when displaying the coordinates referred to machine reference zero. The coordinate returned by the POS(X-C) and TPOS(X-C) variables takes into account the additive zero offset.
1	When displaying the position of the axes, it ignores the additive offset. The coordinate returned by the POS(X-C) and TPOS(X-C) variables ignores the additive zero offset.
2	When displaying the position of the axes, the CNC takes into account the additive offset except when showing the Command - Actual -To Go coordinates. The coordinate returned by the POS(X-C) and TPOS(X-C) variables takes into account the additive zero offset.

Default value: 0

The additive zero offset can be originated as follows:

- With variable PLCOF(X-C), it is possible to set an additive zero offset for each CNC axis from the PLC.
- With the additive handwheel.

HANDWIN (P111) HANDWHE1 (P112) HANDWHE2 (P113) HANDWHE3 (P114) HANDWHE4 (P115)

Not being used.

STOPTAP (P116)

Indicates whether the general inputs /STOP (M5001), /FEEDHOL (M5002) and /XFERINH (M5003) are enabled (P116=YES) or not (P116=NO) while executing function G84, regular tapping or rigid tapping.

INSFEED (P117)

Sets the tool inspection feedrate.

When accessing tool inspection, the CNC assumes this feedrate as the new one, and it resumes the execution of the program at the previous feedrate (the one used in the program or set via MDI while in tool inspection) when tool inspection is over.

Possible values
Between 0.0001 and 199999.9999 degrees/min or mm/min. Between 0.00001 and 7874.01574 inches/min.

Default value: NO

If set to "0" (by default), tool inspection will be carried out at the feedrate currently used for machining.

DISTYPE (P118)

Only to be used by Fagor Automation technical personnel.

PROBERR (P119)

Indicates whether the CNC issues an error message when the axes reach the programmed position without having received the probe signal while executing function G75 or G76.

Value	Meaning
YES	It issues the error message.
NO	It does NOT issue the error message.

Default value: NO

SERSPEED (P120) SERPOWSE (P121)

Not being used.

LANGUAGE (P122)

Defines the work language.

Value	Meaning	Value	Meaning
0	English	7	Czech
1	Spanish	8	Polish
2	French	9	Mainland Chinese
3	Italian	10	Basque
4	German	11	Russian
5	Dutch	12	Turkish
6	Portuguese		

Default value: 0

GEOMTYPE (P123)

It indicates whether the cutter geometry is associated with the tool (T) or with the tool offset (D).

The "T" function, tool number, indicates the magazine position it occupies.

The "D" function, offset, indicates the tool dimensions.

Value	Meaning
0	It is associated with the tool.
1	It is associated with the tool offset.

Default value: 0

When using a tool holding turret, the same turret position is usually used by several tools. In those cases, the "T" function refers to the turret position and the "D" function to the dimensions and geometry of the tool occupying that position. Thus, "GEOMTYPE=1".

SPOSTYPE (P124) AUXSTYPE (P125)

Not being used.

FOVRG75 (P126)

It indicates whether function G75 ignores the feedrate override switch of the front panel or not.

Value	Meaning
NO	It ignores the setting of the switch. Always at 100%.
YES	It is affected by the % of the switch.

Default value: NO

CFGFILE (P127)

Not being used.

STEODISP (P128)

It indicates whethe the CNC displays the real or theoretical RPM (affected by the %) of the main spindle.

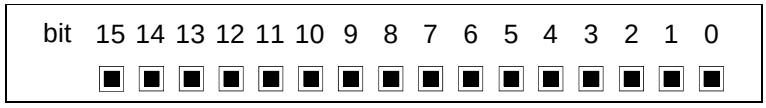
Value	Meaning
0	It displays the real RPM.
1	It displays the theoretical RPM.

Default value: 0

When not having spindle encoder (NPULSES=0), it is recommended to set P128=1 so it displays theoretical value.

HDIFFBAC (P129)

This parameter has 16 bits counted from right to left.



Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning	Bit	Meaning
0	Handwheel ·1·.	8	
1	Handwheel ·2·.	9	
2		10	
3		11	
4		12	
5		13	
6		14	
7		15	It limits the movement.

Default value in all the bits: 0

Bit 15 indicates how the CNC acts when requesting a feedrate greater than the maximum allowed depending on the handwheel turning speed and the position of the switch.

(0) It limits the feedrate to the maximum allowed but it moves the indicated distance.

(1) It limits the feedrate and the distance to the maximum allowed. The movement stops when the handwheel stops. It does not move the indicated distance.

The individual handwheels, those associated with each axis, always limit the feedrate and the distance.

bits 0 and 1 indicate whether the handwheels output differential signals (1) or not (0).

RAPIDEN (P130)

It indicates how the rapid movements are carried out. The behavior of this key is managed using the EXRAPID mark.

Value	Meaning
0	It has no effect.
1	When the mark is activated, the movements are executed in rapid. There is no need to press the key.
2	The "rapid" key is enabled when the mark is activated or when the key is pressed. The key must be pressed to make the movements.

Default value: 0

The rapid key is treated in execution and simulation as follows:

- The movements are carried out in rapid traverse (G00) while the rapid key is pressed.
- The rapid key is ignored while threading, while look-ahead is active.
- If G95 is active, it switches to G94 mode. When releasing the rapid key, it goes back to G95 mode.
- It only affects the main channel. It is ignored in the PLC channel.

MSGFILE (P131)

Number of the program that contains the OEM texts in several languages.

By default, the CNC sets this parameter to "0" (there is no program).

If programmed with a value of "0", the texts defined by the OEM are in a single language and stored in several programs:

PLCMSG Texts for PLC messages

PLCERR Texts for PLC errors

The MSGFILE program may be in user memory, in the Memkey card or in the hard disk (KeyCF). If it is in several places, it takes the one in user memory.

If any of these texts is in Russian or mainland Chinese, the file format must be converted from Unicode to a special Unicode customized for the Fagor CNC. Use WinDNC to do this conversion.

Converting standard Unicode files into Fagor Unicode format requires WinDNC version 5.1. This conversion is not possible with previous WinDNC versions.

To display these messages, besides doing this conversion, the CNC must be set in one of the following languages:

- English.
- Chinese.
- Russian.

Note: Russian or Chinese comments cannot be displayed in a program even if that program has been converted using WinDNC.
Having the MSGFILE in the hard disk (KeyCF) requires the memory expansion option.

FLWEDIFA (P132)

Not being used.

RETRACAC (P133)

It indicates whether retracing is allowed or not

Value	Meaning
0	It is not permitted.
1	It is permitted. The withdrawal stops at the M functions.
2	It is permitted. The withdrawal does not stop at the M functions.

Default value: 0

If RETRACAC = 2, only the M0 is executed; the rest of the M functions are not sent out to the PLC, it neither executes them nor interrupts the withdrawal. The [CYCLE START] key must be pressed after executing M0.

Retracing is activated and deactivated with the RETRACE (M5051) signal. This function can also be activated while function G51 (look-ahead) is active.

If while executing a part program, the PLC sets this signal high, the CNC interrupts the execution of the program and starts executing backwards what has executed so far.

When the PLC sets the RETRACE signal back low and retracing is canceled. The CNC starts executing forward what was done backwards and it will go on to execute the part of the program that was not machined.

G15SUB (P134)

Not being used.

TYPCROSS (P135)

It indicates how cross compensation is applied. This parameter has two digits.

(units) Cross compensation with theoretical or real coordinates.

The units indicate whether cross compensation is applied with theoretical coordinates or with real ones.

Value	Meaning
x0	With real coordinates.
x1	With theoretical coordinates.

Default value: 0

(tens) The cross compensation on Gantry axis affects the slave axis

The tens indicate whether cross compensation on Gantry axes affects only the master axis or both.

Value	Meaning
0x	It affects the master axis.
1x	It affects both axes.

Default value: 0

AXIS9 (P136) **PAXIS9 (P137)** **AXIS10 (P138)**
PAXIS10 (P139) **AXIS11 (P140)** **PAXIS11 (P141)**
AXIS12 (P142) **PAXIS12 (P143)**

If a CNC is configured so that any of the feedback inputs of its axes or spindles is free (because they are digital axes or spindle without feedback connection to the CNC), these free connectors could be configured as electronic or mechanical handwheels.

AXIS9 ... AXIS12:

Define the handwheel type. The values to be assigned to these parameters are:

Value	Meaning	Value	Meaning
11	Handwheel.	12	Handwheel with axis selector button
21	Handwheel associated with X.	22	Handwheel associated with Y.
23	Handwheel associated with Z.	24	Handwheel associated with U.
25	Handwheel associated with V.	26	Handwheel associated with W.
27	Handwheel associated with A.	28	Handwheel associated with B.
29	Handwheel associated with C.		

PAXIS9 ... PAXIS12:

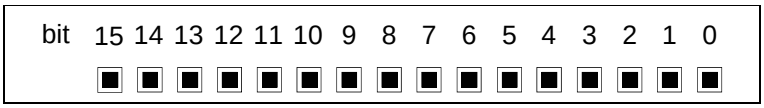
Define which connector is each handwheel associated with. Values 1 through 8 must be assigned to these parameters depending on the connector that the handwheel is associated with.

When detecting any incompatibility, on power-up, it will issue the messages "Feedback busy" or "Feedback not available".

ACTBACKL (P144)

It is related to a.m.p. BACKLASH (P14), leadscrew backlash compensation due to change of direction.

This parameter has 16 bits counted from right to left.



Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning	Bit	Meaning
0		8	
1		9	
2		10	
3		11	
4		12	
5		13	G2 / G3
6		14	
7		15	

Default value in all the bits: 0

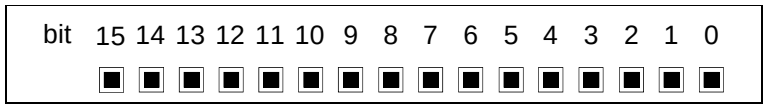
Bit 13. Backlash compensation on arcs G2/G3.

This bit indicates whether the compensation is applied only on circular paths G2/G3 (bit=1) or in any other type of movement (bit=0).

ACTBAKAN (P145)

It is related to a.m.p. BAKANOUT (P29) and BAKTIME (P30), additional analog command pulse to recover the possible leadscrew backlash when reversing the movement.

This parameter has 16 bits counted from right to left.



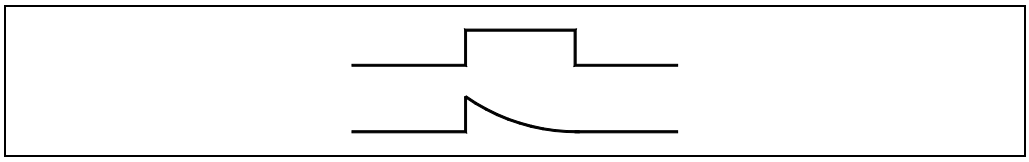
Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning	Bit	Meaning
0	Exponential backlash peak.	8	
1		9	
2		10	
3		11	
4		12	
5		13	Apply the additional pulse with G2 / G3
6		14	
7		15	

Default value in all the bits: 0

Bit 0. Exponential backlash peak.

The additional command pulse used to make up for the possible leadscrew backlash in movement reversals may be rectangular or exponential. This bit indicates whether it applies a rectangular backlash peak (bit=0) or an exponential backlash peak (bit=1).



If the duration of the rectangular pulse is adjusted for low speed, it may be too high for high speed or not enough for low speed when adjusted for high speed. In this cases, it is recommended to use the exponential type that applies a strong pulse at the beginning and decreases in time.

Bit 13. Additional pulse only in circular paths G2/G3.

This bit indicates whether the additional pulse of velocity command is applied only on circular paths G2/G3 (bit=1) or in any other type of movement (bit=0).

STPFILE (P146)

It defines the number of the program where the oscilloscope configuration will be saved. This program will be saved in the hard disk (KeyCF).

Possible values
Integer numbers between 0 and 65535.

Default value: 0

CODISET (P147)

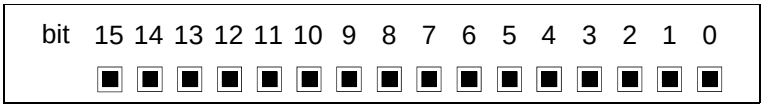
Not being used.

COCYF1 (P148) **COCYF2 (P149)** **COCYF3 (P150)** **COCYF4 (P151)**
COCYF5 (P152) **COCYF6 (P153)** **COCYF7 (P154)** **COCYZ (P155)**
COCYPOS (P156) **COCYPROF (P157)** **COCYGROO (P158)** **COCYZPOS (P159)**

Not being used.

LOOKATYP (P160)

This parameter has 16 bits counted from right to left.



Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning
0	It may be used to apply jerk control in look-ahead.
1...12	Not being used.
13	It makes it possible to use Fagor filters with Look-ahead (not advanced look-ahead algorithm).
14	It activates/cancels the smoothing of machining speed.
15	It activates/deactivates the advanced look-ahead algorithm (integrating Fagor filters).

Bit 0. Apply jerk control in look-ahead.

That bit indicates whether jerk control is to be applied (bit=1) or not (bit=0) in look-ahead.

Using jerk in look-ahead, a trapezoidal acceleration profile is applied with a ramp slope equivalent to the maximum jerk of the axis. The maximum jerk depends on the value assigned to a.m.p. "JERKLIM (P67)" of that axis and of the number of axes involved in the programmed path. For the axes whose JERKLIM parameter has been set to zero, the CNC assumes the jerk value recommended for that parameter.

Bit 13. Use Fagor filters with look-ahead.

Value 0: Fagor filters are not used with look-ahead, even if these filters have been activated by machine parameters for the axes.

Value 1: Fagor filters will be used in all movements. In not-advanced look-ahead, if the Fagor filters have been set by machine parameter, the CNC will assume the values set in these parameters; otherwise, it will assume the default values of these filters.

Default value: 0.

Bit 14. Smoother machining speed.

This bit activates (bit=1) or cancels (bit=0) the smoothing of machining speed. Default value: 0.

Bit 15. Advanced look-ahead algorithm (integrating Fagor filters).

This bit activates (bit=1) or cancels (bit=0) the advanced look-ahead algorithm (integrating Fagor filters). Default value: 0.

TLOOK (P161)

Real block processing time for look-ahead.

Possible values
Integer numbers between 0 and 65535 microseconds.

Default value: 0

If assigned a value smaller than the real one, the machine will vibrate and if assigned a value greater than the real one the machining slows down. The value of this parameter may be calculated as follows:

1. Execute in G91 and G51 E0.1 a program made up of lots of blocks (at least 1000) with very short moves; for example X0.1 Y0.1 Z0.1.
2. Measure the program execution time, making sure that the machine does not vibrate. Divide the execution time by the number of blocks executed and assigned the resulting value (in microseconds) to this parameter.
3. To optimize the parameter, decrease the calculated value and execute the same program until the machine starts vibrating. To avoid damaging the machine, it is recommended to start the execution with the feedrate override switch low and increase its value gradually.
4. We recommend the use of the oscilloscope function and verify that the internal variable VLOOKR remains constant which means that there is no vibration. From the oscilloscope, it is possible to change the value of parameter TLOOK, but the new value is only assumed when executing function G51 from the program.

MAINTASF (P162) CAXGAIN (P163)

Not being used.

TOOLMATY (P164)

When using a non-random tool magazine (e.g. a turret), it indicates how many tools may be assigned to each turret position.

When defined with a ·0· value in a non-random magazine, the tools must be placed in the tool magazine table in the pre-established order (P1 T1, P2 T2, P3 T3, etc.).

Value	Meaning
0	One tool per position. (P1 T1, P2 T2, etc.).
1	The tool can occupy any position.

Default value: 0

MAXOFFI (P165)

Tool wear offsets may be modified from tool inspection mode. This parameter indicates the maximum amount of wear that may be entered for "I" (it is programmed in mm or in inches).. It is defined in diameter at the lathe model.

Default value: 0.5

MAXOFFK (P166)

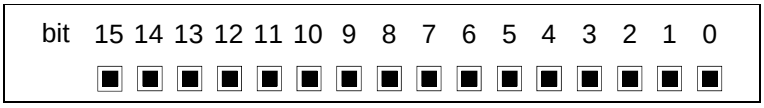
Tool wear offsets may be modified from tool inspection mode. This parameter indicates the maximum amount of wear that may be entered for "K" (it is programmed in mm or in inches)..

Default value: 0.5

TOOLTYPE (P167)

It defines the behavior of the tool or of the tool offset.

This parameter has 16 bits counted from right to left.



Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning
0 - 12	Not being used.
13	The STOP signal is always executed after the "T" function.
14	Machining in round corner mode when changing the tool offset.
15	Stop block preparation when executing a new "T".

Default value in all the bits: 0

Bit 13. The stop signal is taken into account after the "T" function is completed.

This function is applied when the tool change subroutine has been defined so it is executed as a single block and the stop signal has been disabled.

If the stop signal ([CYCLE STOP] key or PLC signal) is received while executing a subroutine, the CNC memorizes it until enabling the stop signal. In this situation, it does not finish the subroutine and it does not consider the T as done, which could cause irregularities at the tool magazine. To avoid this situation, it is possible to consider the stop signal after executing the "T" function.

This bit determines whether the stop signal is considered after completing the "T" function (bit=1) or not (bit=0). If the bit is set to ·0·, the stop signal is considered in the following cases.

- If the stop signal has been disabled, when it is re-enabled.
- If the stop signal has not been disabled, when pressing the [CYCLE STOP] key.

It should be borne in mind that the DSTOP instruction disables both the [CYCLE STOP] key and the signal coming from the PLC. Both may be re-enabled with the ESTOP instruction.

Bit 14. Type of corner when changing tool offsets.

When changing a tool offset, the change takes place at the end of the path. The corner where the tool offset is changed may be machined either with rounding or without it.

This bit indicates whether that point is machined as a round corner (bit=1) or as a square corner (bit=0).

This bit is only taken into account when round corner machining is active; when working in square corner, the corner where the change takes place is always machined as a square corner.

Bit 15. Stop block preparation when executing the "T" function.

If while executing the "T" function, the block preparation detects a programming error, this function might not be executed completely This means that the tool change may have concluded correctly but the requested tool has not been assumed by the CNC. To avoid this situation, it is possible to stop block preparation during the execution of the "T" function.

This bit determines whether block preparation is interrupted (bit=1) or not (bit=0) while executing a "T" function.

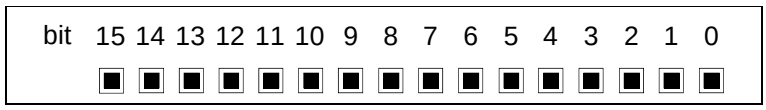
Remember that when having a subroutine associated with the "T" function, the tool change is carried out as follows:

1. It executes the associated subroutine.
2. The "T" function is executed without using the M06 function.
3. The CNC assumes the change.

PROBEDEF (P168)

Defines the behavior of the probe.

This parameter has 16 bits counted from right to left.



Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning
0	Smooth stop of the probe.
1 - 15	Not being used.

Default value in all the bits: 0

Bit 0. Smooth stop of the probe (G75/G76).

This bit permits defining a smooth stop for probing moves (bit=1). When the probe pulse is detected, the following error is not reset, thus making the probe stop more smoothly.

When setting the smooth stop, a.m.p "DERGAIN (P25)" and s.m.p. "FFGAIN (P25)" should be set to zero. This may be done by setting the set of gains through g.m.p. "ACTGAIN2 (P108)" with the bit corresponding to G75/G76.

CANSPEED (P169)

CAN bus transmission speed for the digital drives.

The transmission speed depends on the length of the cable or total CAN connection distance.

Value	Meaning
0	1 Mbit/s. Maximum distance: 20 meters.
1	800 kbit/s. Maximum distance: 45 meters.
2	500 kbit/s. Maximum distance: 95 meters.

Default value: 0 (1 Mbit/s)

FEEDTYPE (P170)

Behavior of the feedrate when programming F0.

Value	Meaning
0	Movement at maximum feedrate possible.
1	F0 cannot be programmed.

Default value: 0

If set to 0, F0 may be programmed and the motion blocks will be executed at the maximum feedrate allowed.

If set to 0, it is not possible to program F0 or execute motion blocks with F0 active.

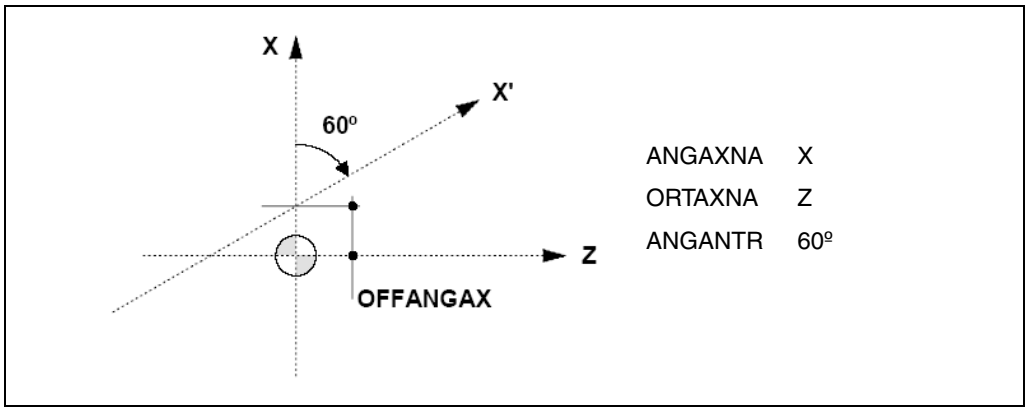
ANGAXNA (P171)

Cartesian axis associated with the incline axis.

Value	Meaning	Value	Meaning
0	None.	5	V axis.
1	X axis.	6	W axis.
2	Y axis.	7	A axis.
3	Z axis.	8	B axis.
4	U axis.	9	"C" axis.

Default value: 0 (none)

With the angular transformation of an incline axis, it is possible to make movements along an axis that is not perpendicular to another. In order to program it in the Cartesian system (Z-X), activate the incline axis transformation to convert the movements to the non-perpendicular real axes (Z-X').



The axes defined in parameters "ANGAXNA" and "ORTAXNA"must exist and must be linear. Those axes may have Gantry axes associated with them.

While searching home, the movements are carried out on the incline axes of the machine. PLC mark "MACHMOVE" determines how the manual movements with handwheels or with the keyboard will be carried out.

The inclined plane is activated from the part-program (function G46). If the inclined plane is active, the displayed coordinates will be those of the Cartesian system. Otherwise, it will display the coordinates of the real axes.

ORTAXNA (P172)

Axis perpendicular to the Cartesian axis associated with the inclined plane.

Value	Meaning	Value	Meaning
0	None.	5	V axis.
1	X axis.	6	W axis.
2	Y axis.	7	A axis.
3	Z axis.	8	B axis.
4	U axis.	9	"C" axis.

Default value: 0 (none)

ANGANTR (P173)

Angle between the Cartesian angle and the angular axis it is associated with. If its value is 0º, there is no need to do an angular transformation.

Positive angle when the angular axis has been rotated clockwise and negative if counterclockwise.

Possible values
Within ±90 degrees.

Default value: 0

OFFANGAX (P174)

Distance between machine zero and the origin of the coordinate system of the incline axis.

Possible values
Within ± 99999.9999 millimeters. Within ±3937.00787 inches.

Default value: 0

COMPMODE (P175)

It defines how to apply tool radios compensation.

Value	Meaning
0	With an angle between paths of up to 300º, both paths are joined with straight sections. In the rest of the cases, both paths are joined with arcs.
1	Both paths are joined with arcs.
2	With an angle between paths of up 300º, it calculates the intersection. In the rest of the cases such as COMPMODE = 0.

Default value: 0

COMPMODE = 0.

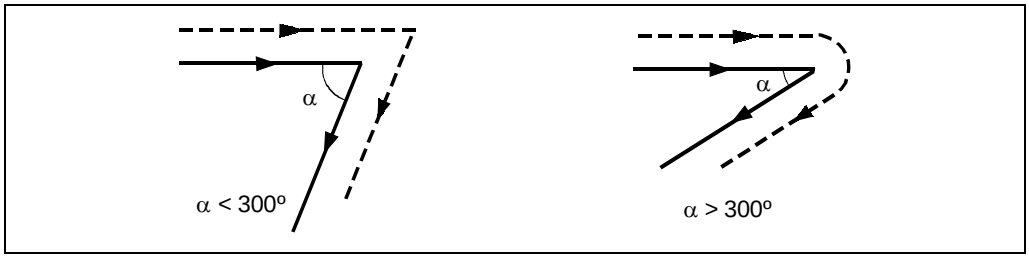
The compensation method depends on the angle between paths.

- For angles up to 300º, it compensates joining both paths with straight sections.
- For angles over 300º, it compensates joining both paths with arcs.

COMPMODE = 2.

The compensation method depends on the angle between paths.

- For angles up to 300º, it calculates the intersection between the compensated paths.
- For angles over 300º, it resolves it like when COMPMODE = 0.

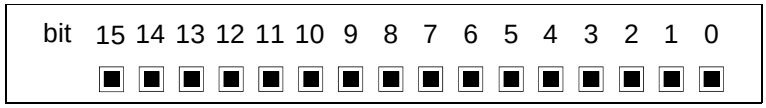


ADIMPG (P176)

This parameter enables manual intervention with an additive handwheel.

This function allows jogging the axes while a program is being executed. This movement will be applied as if it were another zero offset.

This parameter has 16 bits counted from right to left.



Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning
0 - 10	Not being used.
11	Selecting the additive handwheel as handwheel associated with the axis.
12	The resolution of the handwheel is set by g.m.p. ADIMPRES.
13	Manual intervention enabled with look-ahead.
14	Cancel the additive offset after M02, M30, emergency or Reset.
15	Manual intervention with additive handwheel is available.

Default value in all the bits: 0

When enabling the additive handwheel, the following must be borne in mind.

- If the DWELL parameter of an axis has been set and it is not previously in motion, it activates the ENABLE mark of the axis and waits a time period indicated in DWELL to check whether its SERVOON has been activated or not.
- The acceleration applied to the additive handwheel movement is that of parameter. ACCTIME of the axis.
- On Gantry axes, the movement of the master axis using an additive handwheel is also applied to the slave axis.
- The mirror image by PLC is not applied to the additive handwheel movement.
- When testing the software limits during block preparation, it checks the theoretical coordinate ignoring the additional movement of the additive handwheel.

Bit 11. Selecting the additive handwheel as handwheel associated with the axis.

If this bit is set to 1, even if there is a general handwheel, the additive handwheel will always be the one associated with the axis.

Bit 12. The resolution of the handwheel is set by g.m.p. ADIMPRES.

This bit indicates whether the handwheel resolution is set by parameter ADIMPRES (bit01) or not. Otherwise (bit=0), the resolution of the handwheel is set with the switch of the operator panel. If the switch is not in the handwheel position, it assumes a x1 factor.

Bit 13. Manual intervention enabled with look-ahead.

This bit indicates whether manual intervention is available (bit=1) or not (bit=0) when look-ahead is active.

Bit 14. Cancel the additive offset after M02, M30, emergency or Reset.

This bit determines (bit=1) that the additive offset is canceled after executing M02/M30 or after an emergency or reset.

Bit 15. Manual intervention with additive handwheel is available.

This bit indicates whether manual intervention with an additive handwheel is available (bit=1) or not (bit=0). If set to ·0·, the rest of the bits are ignored.

The additive handwheel is activated and deactivated with PLC signal MANINT.

ADIMPRES (P177)

Resolution of the additive handwheel.

Value	Meaning
0	0.001 mm or 0.0001 inch.
1	0.01 mm or 0.001 inch.
2	0.1 mm or 0.01 inch.

Default value: 0

These values are only applied when bit 12 of parameter ADIMPG has been set to ·1·.

SERCDEL1 (P178) SERCDEL2 (P179)

Not being used.

EXPLORER (P180)

It sets how to access the explorer.

Value	Meaning
0	It is accessed using the < explorer> softkey of the utilities, execute, simulate or edit modes.
1	It is accessed directly from the utilities, execute, simulate or edit modes.

Default value: 0

REPOSTY (P181)

It may be used to select the repositioning mode:

Value	Meaning
0	It activates the basic repositioning mode
1	It activates the extended repositioning mode

Default value: 1

MAXOFFJ (P182)

This parameter indicates the maximum incremental value allowed for Y axis wear compensation (it is programmed in mm or in inches). Default value: 0.5.

DISSIMUL (P184)

It may be used to disable the simulation modes and the block search modes when selecting blocks in execution. Setting the relevant bit to 1 disables it and removes from the menu the softkey assigned to this bit. This parameter has 16 bits counted from right to left.

For the block search: Disabling in execution:

DISSIMUL = x x x x x x x x 0/1 x x x x x x x x

bit 7 = 1	EXEC G SEARCH
bit 6 = 1	EXEC GMST SEARCH

For simulation: Disabling in simulation:

DISSIMUL = 0/1 x x x x x x x x x x x x x x x x

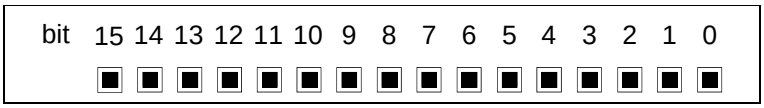
bit 10 = 1	RAPID [S0]
bit 11 = 1	RAPID
bit 12 = 1	MAIN PLANE
bit 13 = 1	G, M, S, T FUNCTIONS
bit 14 = 1	G FUNCTIONS
bit 15 = 1	THEORETICAL PATH

ACTGAINT (P185)

The axes and the spindle can have 3 sets of gains and accelerations. By default, the CNC always assumes the first set indicated by the parameters of the axis or of the spindle ACCTIME, PROGAIN, DERGAIN and FFGAIN.

Parameter ACTGAINT indicates when the CNC assumes the third set of gains and accelerations, indicated by the parameters of the axis or of the spindle ACCTIMET, PROGAINT, DERGAINT and FFGAINT.

This parameter has 16 bits counted from right to left.



Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning	Bit	Meaning
0	Threading in blind threads (for lathe only)	8	G51
1	G34	9	G50
2		10	G49
3	G74	11	G48
4	JOG	12	G47
5	Rigid tapping	13	G33
6	G95	14	G01
7	G75 / G76	15	G00

Default value in all the bits: 0

Every time each of the functions or work modes assigned to the bits of g.m.p. ACTGAIN2 (P108) or ACTGAINT (P185) is activated, the CNC checks the value assigned to the bit corresponding to that function in these parameters and acts as follows:

- If the bits of ACTGAIN2 is set to ·0· and the bit of ACTGAINT is set to ·0·, it applies the first set "ACCTIME, PROGAIN, etc".
- If the bit of ACTGAINT2 is set to ·1· and the bit of ACTGAINT is set to ·0·, it applies the third set "ACCTIME2, PROGAIN2, etc".
- If the bits of ACTGAINT is set to ·1· and the bit of ACTGAIN2 is set to ·0·, it applies the third set "ACCTIMET, PROGAINT, etc".

When that function or work mode is deactivated, the CNC applies the first of the sets "ACCTIME, PROGAIN".

Example

When setting ACTGAINT = 1000 0000 0001 0000 and ACTGAIN2 = 0000 0000 0000 0000, the CNC applies the third set to all the axes and the spindle whenever function G0 or the JOG mode is selected.

Considerations to bear in mind

The change of gains and accelerations is always made at the beginning of the block. When working in round corner (G5), the change does not take place until G07 is programmed.

Example ·1·		**Example ·2·**	
G2 X10 Y10 I10 J0	(Set 1)	G05 G2 X10 Y10 I10 J0	(Set 1)
G1 X20	(Set 2)	G1 X20	(Set 1)
G3 X30 Y20 I0 J10	(Set 1)	G3 X30 Y20 I0 J10	(Set 1)
G1 Y30	(Set 2)	G7 G1 Y30	(Set 2)

The gains and accelerations may also be changed from the PLC. To do that, there is a general logic CNC input ACTGAINT (M5063). Every time this input is activated, the CNC selects the third set of gains and accelerations regardless of the active operating mode or function.

RETRACTE (P186)

It enables or disables the various retraction (withdrawal) options in a drilling or a mill type threading.

0 Value: disabled.

1 Value: enabled.

Bit	Meaning
0	Enables / disables the threading withdrawal in the threading cycles (G86 and G87). Only for the lathe model.
1	Enables / disables the threading withdrawal in the drilling cycles (G69, G81, G82 and G83). Only for the mill model.
2	Enables / disables the threading withdrawal in the tapping cycle (G84). Only for the mill model.
3	Enables / disables the threading withdrawal in the rigid tapping cycle (G84). Only for the mill model.

Default value 0

TAPTYPE (P188)

This version improves the rigid tapping speed. This is especially useful in processes where it is continuously thread cutting.

If bit 0 of g.m.p. TAPTYPE (P188) = 1, the M3, M4 and M5 that are executed in rigid tapping appear in the history, but they are not sent to the PLC. Since these M's are not sent to the PLC, the dwells associated with them are eliminated and, consequently, the cycle is faster.

Bit 0	Meaning
0	Regular rigid tapping.
1	Rigid tapping and without sending M functions to the PLC.

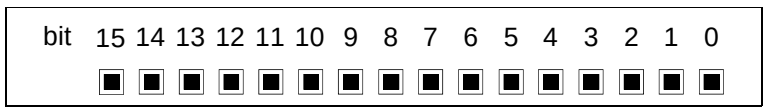
Default value 1

MANTFCON (P198)

While executing in look-ahead (G51), some blocks of the program cause the feedrate to slow down almost to zero causing a square-corner effect. To avoid this effect, when G05 or G51 have been programmed, it is necessary to maintain the machining feedrate of the blocks that cause it.

To maintain the machining feedrate of these blocks and avoid the square-corner effect, change bit 0 of general machine parameter MANTFCON (P189).

This parameter has 16 bits counted from right to left.



Bit 0 of g.m.p. MANTFCON (P189):

Value	Meaning
0	Motionless blocks are executed in square corner mode.
1	Motionless blocks maintain the feedrate and are not executed in square corner mode.

Default value: 0

Blocks affected by the square-corner effect

If bit 0 of g.m.p. MANTFCON (P189) = 1, the CNC will not execute the following blocks in square-corner mode:

- An F programmed alone in the block.
- Blocks consisting one or more of the following G functions:

  G0, G1, G2, G3 (without programaming coordinates)

  G5

  G6

  G10, G11, G12, G13

  G32, G94, G95 (when not changing from one to another)

  G40, G41, G42, G43, G44

  G70, G71

  G90, G91

  G92 Sxxx

  G96, G97 (when not changing from one to another)

  G151, G152

Special cases

If Bit 0 of g.m.p. MANTFCON (P189) = 1:

- When executing auxiliary M, S, T functions, the CNC will do it in square-corner mode.
- If a block contains coordinates that coincide with the position of the previous block, the CNC will not execute it in square-corner mode.

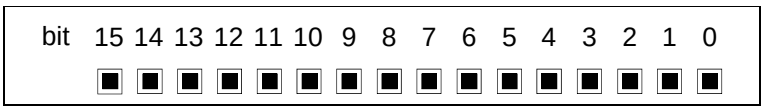
STARTDIS (P190)

When sending an infinite program from a PC to the CNC through Windnc to be executed, there are the following choices:

1. Once the program has been transmitted, it is executed without pressing any key at the CNC.
2. Once the program has been transmitted, it is not executed until the START key is pressed at the CNC.

The new general machine parameter STARTDIS (P190) determines whether the program may be executed or not without pressing START.

This parameter has 16 bits counted from right to left.



Bit 0 of g.m.p. STARTDIS (P190):

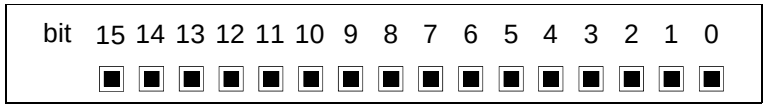
Value	Meaning
0	The program is sent to the CNC and is executed.
1	The program is sent to the CNC and is not executed until the START key is pressed.

Default value: 1

LCOMPTYP (P191):

It may be used to set whether the longitudinal axis will be maintained or changed when changing the work plane(G17, G18 or G19).

This parameter has 16 bits counted from right to left.



Bit 0 of g.m.p. LCOMPTYP (P191):

Value	Meaning
0	The longitudinal axis changes when changing planes.
1	The longitudinal axis does not change when changing planes.

Default value: 0

4.3 Axis parameters

AXISTYPE (P0)

It sets the type of axis and whether it is governed by the CNC or by the PLC.

Value	Meaning
0	Normal linear axis.
1	Rapid positioning linear axis (G00).
2	Normal rotary axis.
3	Rapid positioning rotary axis (G00).
4	Rotary axis with Hirth toothing (positioning in whole degrees).
5	Normal linear axis commanded from the PLC.
6	Rapid positioning linear axis (G00) commanded from the PLC.
7	Normal rotary axis commanded from the PLC.
8	Rapid positioning rotary axis (G00) commanded from the PLC.
9	Rotary axis with Hirth toothing (positioning in whole degrees) commanded from the PLC.

Default value: 0

By default, rotary axes are Rollover and are displayed between 0º and 359.9999º. If rollover is not desired, set a.m.p. ROLLOVER (P55)=NO. The axis position will be displayed in degrees.

Positioning-only and/or Hirth axes follow the shortest path when programmed in absolute (G90). In other words, if its current position is 10º, and its target position is 350º, the axis will go through, 10º, 9º, ... 352, 351, 350.

*See **"5.1 Axes and coordinate systems"** on page 179.*

DFORMAT (P1)

Indicates the work units (radius or diameter) and the display format used for the axis.

Value	Work units	Data format		
		degrees	mm.	inch.
0	radius	5.3	5.3	4.4
1	radius	4.4	4.4	3.5
2	radius	5.2	5.2	5.3
3	radius	It is not displayed		
4	diameters	5.3	5.3	4.4
5	diameters	4.4	4.4	3.5
6	diameters	5.2	5.2	5.3

GANTRY (P2)

Indicates, if it is a Gantry axis, which axis is this one associated with. This parameter is to be set only on the slaved axis according to the following code.

Value	Meaning	Value	Meaning
0	Not Gantry.	5	With the V axis.
1	With the X axis.	6	With the W axis.
2	With the Y axis.	7	With the A axis.
3	With the Z axis.	8	With the B axis.
4	With the U axis.	9	With the C axis.

Default value: 0 (it is not Gantry)

The position of the Gantry axis is displayed next to its associated axis unless machine parameter "DFORMAT(P1)=3".

Example:

If the X and U axes form a Gantry pair, the U axis being the slave axis, program as follows:

Parameter GANTRY (P2) for the X axis = 0

Parameter GANTRY (P2) for U axis = 1 (associated with X axis)

This way, When programming an X axis move, the U axis will also move the same distance.

SYNCHRO (P3)

Not being used.

DROAXIS (P4)

Indicates whether it is a normal axis or it only works as a Digital Read Out

Value	Meaning
NO	It is a normal axis.
YES	It only works as a Digital Read Out.

Default value: NO

LIMIT+ (P5) LIMIT - (P6)

They set the travel limits for the axis (positive and negative). Each one of them indicates the distance from machine reference zero to the relevant travel limit.

Possible values
Within ±99999.9999 degrees or millimeters.
Within ±3937.00787 inches.

Default value: For LIMIT+ (P5) = 8000 mm.
For LIMIT- (P6) = -8000 mm.

On linear axes, if both parameters are set to 0, the travel limits will be ignored.

On the rotary axes, act as follows:

- When both parameters are set to "0", the axis may be moved indefinitely in any direction (rotary tables, indexers, etc.)
- When working with positioning axes and Hirth axes, try to program in incremental coordinates to avoid mistakes. For example, C axis with P5=0, P6=720 and the positioning axis in 700 (340 on the screen), when programming G90 C10, the CNC tries to move the axis via the shortest path (701, 702, etc.) but it will issue an error message for overrunning the travel limits.
- If the travel of positioning axes and hirth axes is limited to less than a turn, they cannot move via the shortest path.
- When the travel is limited to less than a revolution and a positive and negative display is desired, for example P5=-120, P6=120, it is possible to program G90 with positive and negative values.

PITCH (P7)

Defines the pitch of the ballscrew or the resolution of the linear feedback device being used.

It must be set when the feedback is handled through the CNC connector; analog servo or digital with DRIBUSLE = 0.

Possible values
Between 0.0001 and 99999.9999 degrees or millimeters. Between 0.00001 and 3937.00787 inches.

Default value: 5 mm.

Analog servo system.

The meaning of parameter PITCH depends on the type of axis and encoder used.

- On linear axis with rotary encoder, it sets the leadscrew pitch per encoder turn.
- On linear axis with linear encoder, it sets the resolution of the encoder.
- On rotary axis, it sets the number of degrees the shaft rotates per encoder turn.

With this type of servo system, parameter PITCHB (P86) has no meaning.

Axis type	Encoder type	PITCH (P7)	NPULSES (P8)
Linear axis.	Linear encoder.	Encoder resolution.	0
	Rotary encoder.	Leadscrew pitch per encoder turn.	Number of pulses (lines) per encoder turn.
Rotary axis.	Rotary encoder.	Degrees that the shaft rotates per encoder turn.	Number of pulses (lines) per encoder turn.

When using gear reduction on the shaft, only the whole assembly must be taken into account when setting one of parameters PITCH or NPULSES.

Linear axis with a 5 mm pitch leadscrew.	PITCH = 5 mm.
Axis with a 20µm-pitch Fagor linear encoder	PITCH = 0.020 mm.
Rotary axis with 1/10 gear ratio	PITCH = 36º.

CAN servo system.

The meaning of parameter PITCH depends on the type of axis regardless of the type of encoder used.

- On linear axis, it sets the resolution of the encoder.
- On rotary axis, it sets the number of degrees the shaft rotates per encoder turn.

In this type of servo system, the leadscrew pitch is set through parameter PITCHB (P86).

Axis type	Encoder type	PITCH (P7)	PITCHB (P86)	NPULSES (P8)
Linear axis.	Linear encoder.	Encoder resolution.	0	0
	Rotary encoder.	Encoder resolution.	Leadscrew pitch per encoder turn.	Number of pulses (lines) per encoder turn.
Rotary axis.	Rotary encoder.	Degrees that the shaft rotates per encoder turn.	0	Number of pulses (lines) per encoder turn.

When using gear reduction on the shaft, only the whole assembly must be taken into account when setting one of parameters PITCH or NPULSES.

NPULSES (P8)

Indicates the number or pulses/rev provided by the rotary encoder. When using a linear encoder, it must be set to ·0·.

It must be set when the drive's velocity command is analog or it is sent via CAN (DRIBUSLE = 0 or 1).

When using gear reduction on the shaft, only the whole assembly must be taken into account when setting one of parameters PITCH or NPULSES.

Possible values
Integer numbers between 0 and 65535.

Default value: 1250

When using CAN servo, if both parameters NPULSES and PITCHB are set to ·0·, the CNC will assume the equivalent values of the drive.

DIFFBACK (P9)

Indicates whether the feedback device uses differential signals (double ended) or not.

Value	Meaning
NO	It does NOT use differential signals.
YES	It uses differential signals.

Default value: YES

SINMAGNI (P10)

Indicates the multiplying factor (x1, x4, x20, etc.) that the CNC must apply only to sinusoidal feedback signal.

For square feedback signals, this parameter must be set to 0 and the CNC will always apply a multiplying factor of x4.

Possible values
Integer numbers between 0 and 255.

Default value: 0

The axis feedback resolution must be set using a.m.p. PITCH (P7), NPULSES (P8) and SINMAGNI (P10) as shown in the following table:

	PITCH (P7)	NPULSES (P8)	SINMAGNI (P10)
Square signal encoder	Leadscrew pitch	Nr of pulses	0
Sinusoidal signal encoder	Leadscrew pitch	Nr of pulses	multiplying factor
Square signal linear encoder	linear encoder pitch	0	0
Sinusoidal signal linear encoder	linear encoder pitch	0	multiplying factor

FBACKAL (P11)

This parameter is to be used only when the feedback signals are sinusoidal or differential (double ended).

Indicates whether the feedback alarm for this axis will be ON or OFF.

Value	Meaning
OFF	No feedback alarm desired, it is canceled.
ON	Feedback alarm is being used.

Default value: ON

FBALTIME (P12)

It indicates the maximum time that the axis may stay without properly responding to the CNC's command.

Depending on the command for an axis, the CNC calculates the number of feedback pulses that it must receive for each sample period.

The axis will be considered that it is working fine when the number of pulses received is between 50% and 200% of the calculated number.

If at any time, the number of feedback pulses received is out of this range, the CNC will keep checking that axis until it detects that the number of pulses received has come back to normal. But if more time elapses than the one indicated in this parameter without the axis coming back to normal, the CNC will issue the relevant error message.

Possible values
Integers between 0 and 65535 ms.

Default value: 0 (it is not checked)

AXISCHG (P13)

Indicates the counting direction. If correct, leave it as is, but to change it, select YES if it was set to NO and viceversa. When changing this parameter, also change a.m.p. LOOPCHG (P26).

Possible values
NO / YES.

Default value: NO

BACKLASH (P14)

Indicates the amount of backlash. Enter 0 when using linear encoders.

Possible values
Within ±99999.9999 degrees or millimeters. Within ±3937.00787 inches.

Default value: 0

LSCRWCOM (P15)

Indicates whether the CNC should apply leadscrew error compensation or not.

Value	Meaning
OFF	Leadscrew compensation not being used.
ON	Leadscrew compensation being used.

Default value: OFF

NPOINTS (P16)

Indicates the number of leadscrew error compensation points available in the table. The values in this table will be applied if a.m.p. "LSCRWCOM" (P15) is ON.

Possible values
Integer numbers between 0 and 255.

Default value: 30

DWELL (P17)

Indicates the dwell from the moment the "ENABLE" signal is activated until the velocity command is sent out.

Possible values
Integers between 0 and 65535 ms.

Default value: 0 (none)

ACCTIME (P18)

Defines the acceleration stage or the time it takes the axis to reach the feedrate selected with a.m.p. GOFFED (P38). This time is also valid for the deceleration stage.

Possible values
Integers between 0 and 65535 ms.

Default value: 0 (none)

INPOSW (P19)

Indicates the width of the IN POSITION zone (dead band) where the CNC considers the axis to be in position.

Possible values
Between 0 and 99999.9999 degrees or millimeters. Between 0 and 3937.00787 inches.

Default value: 0.01 mm.

INPOTIME (P20)

Indicates the time period that the axis must remain in the "IN POSITION" zone in order to consider it to be in position.

On axes that are only controlled during the interpolation or the positioning (dead axes), this prevents the CNC from considering the block completed before the axis has stopped and could get out of the in-position zone.

Possible values
Integers between 0 and 65535 ms.

Default value: 0

MAXFLWE1 (P21)

Indicates the maximum following error allowed when this axis moves.

Possible values
Between 0 and 99999.9999 degrees or millimeters. Between 0 and 3937.00787 inches.

Default value: 30 mm.

MAXFLWE2 (P22)

Indicates the maximum following error allowed when this axis is stopped.

Possible values
Between 0 and 99999.9999 degrees or millimeters. Between 0 and 3937.00787 inches.

Default value: 0.1 mm.

PROGAIN (P23)

Indicates the value of the proportional gain. It sets the command in millivolts desired for a following error of 1 mm.

Velocity command (mV)

= Following error (mm) x PROGAIN

Possible values
Integers between 0 and 65535 mV/mm.

Default value: 1000 mV/mm.

Example:

A feedrate of 20000 mm/min is selected by a.m.p. G00FEED (P38) to obtain 1 mm of following error (axis lag) for a feedrate of F = 1000 mm/min.

Command from the drive: 9.5V for a feedrate of 20000 mm/min.

Command for a feedrate of F = 1000 mm/min:

Command = (9.5/20000) x 1000 = 475 mV

Therefore "PROGAIN" = 475

DERGAIN (P24)

Indicates the value of the derivative gain. It indicates the analog command (in millivolts) corresponding to a change in following error of 1mm (0.03937 inches) in 10 milliseconds.

This velocity command will be added to the one calculated for the proportional gain.

Command

$$= \left(\xi \cdot \mathrm{PROGAIN} + \frac{\xi \cdot \mathrm{DERGAIN}}{10 \cdot t}\right)$$

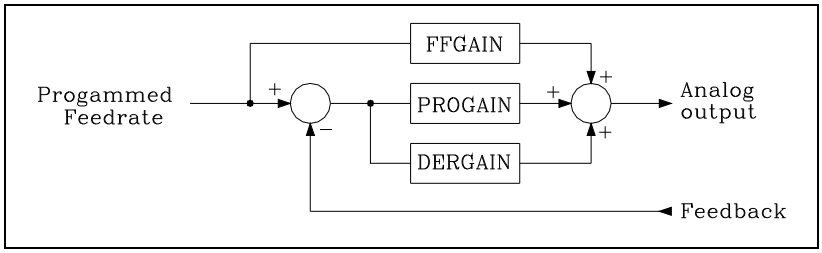
To apply this gain to an axis, that axis should be working with acc/dec [a.m.p. ACCTIME (P18) other than 0].

Possible values
Integer numbers between 0 and 65535.

Default value: 0 (derivative gain not applied)

FFGAIN (P25)

Indicates the % of velocity command due to the programmed feedrate. The rest will depend upon the following error. Both the proportional and derivative gains will be applied onto this following error.



Command

$$= \left(\xi \cdot \text{PROGAIN} + \frac{\dot{\xi} \cdot \text{DERGAIN}}{10 \cdot t} + \frac{\text{FFGAIN} \times \text{Fprog} \times \text{MAXVOLT}}{100 \cdot \text{G00FEED}}\right)$$

The feed-forward gain improves the position loop minimizing the following error, but it should only be used when working with acceleration/deceleration.

Possible values
Between 0 and 100.99 rpm (numbers with 2 decimals).

Default value: 0 (feed-forward gain not applied)

Usually, a value between 40% and 80% is assigned depending mainly on the type of machine and its characteristics.

When having a CNC with a version V11.01 or later where FFGAIN or FFGAIN2 have two decimals, when passing them to an older version, those parameters lose their decimals.

LOOPCHG (P26)

Indicates the sign of the velocity command. If correct, leave it as is, but to change it, select YES if it was set to NO and viceversa.

Possible values
NO / YES.

Default value: NO

MINANOUT (P27)

Indicates the minimum velocity command for the axis.

Possible values
It is given in D/A converter units and it admits integer values between 0 and 32767 which corresponds to an analog command of 10V.

Default value: 0

MINANOUT	Minimum velocity command
1	0.3 mV.
- - -	- - -
3277	1 V.
- - -	- - -
32767	10 V.

SERVOFF (P28)

Defines the velocity command offset value for the drive.

Possible values
It is given in D/A converter units and it admits integer values between 0 and ±32767 which corresponds to an analog command of 10V.

Default value: 0 (not applied)

SERVOFF	Command
-32767	-10 V.
- - -	- - -
-3277	-1 V.
- - -	- - -
1	0.3 mV.
- - -	- - -
3277	1 V.
- - -	- - -
32767	10 V.

BAKANOUT (P29)

Additional velocity command pulse to make up for the possible leadscrew backlash when reversing the moving direction.

Possible values
It is given in D/A converter units and it admits integer values between 0 and 32767 which corresponds to an analog command of 10V.

Default value: 0 (not applied)

BAKANOUT	Additional velocity command
1	0.3 mV.
- - -	- - -
3277	1 V.
- - -	- - -
32767	10 V.

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. This additional velocity command will be applied for a period of time indicated in the a.m.p. BAKTIME (P30).

BAKTIME (P30)

It indicates the duration of the additional velocity command pulse to make up for backlash in movement reversals.

Possible values
Integers between 0 and 65535 ms.

Default value: 0

DECINPUT (P31)

Indicates whether or not this axis has a home switch for machine reference search.

Value	Meaning
NO	It has no home switch.
YES	It has a home switch.

Default value: YES

REFPULSE (P32)

It indicates the type of flank of the I0 signal that is used for home search.

Value	Meaning
+ sign	Positive flank (leading edge); change from 0V to 5V.
- sign	Negative flank (trailing edge); change from 5V to 0V.

Default value: + sign

REFDIREC (P33)

Indicates the direction of the home search in this axis.

Value	Meaning
+ sign	Positive direction.
- sign	Negative direction.

Default value: + sign

REFEED1 (P34)

Indicates the axis feedrate when searching home until it hits the home switch.

Possible values
Between 0.0001 and 199999.9999 degrees/min or mm/min. Between 0.00001 and 7874.01574 inches/min.

Default value: 1000 mm/min.

REFEED2 (P35)

Indicates the axis feedrate when searching home after hitting the home switch until it finds the marker pulse (Io).

Possible values
Between 0.0001 and 99999.9999 degrees/min or mm/min. Between 0.00001 inches/min and 3937.00787 inches/min.

Default value: 100 mm/min.

REFVALUE (P36)

Indicates the position value of the machine reference point (physical location of the marker pulse) with respect to machine reference zero.

Possible values
Within ±99999.9999 degrees or millimeters. Within ±3937.00787 inches.

Default value: 0

The machine reference point is a point that the OEM sets on the machine to synchronize the system. The CNC positions the axis at this point, instead of moving it to the machine zero point.

When the machine uses semi-absolute scales (with coded marker pulses), the axis may be homed anywhere within its travel. Thus, this parameter must only be set when applying leadscrew error compensation. The amount of leadscrew error of the machine reference point may have any value.

MAXVOLT (P37)

Defines the value of the velocity command corresponding to the maximum feedrate of the axis indicated by a.m.p. G00FEED (P38).

Possible values
Integer numbers between 0 mV and 9999 mV.

Default value: 9500 (9.5 V)

G00FEED (P38)

Indicates the maximum feedrate G00 (rapid traverse) of this axis.

Possible values
Between 0.0001 and 199999.9999 degrees/min or mm/min. Between 0.00001 and 7874.01574 inches/min.

Default value: 10000 mm/min.

UNIDIR (P39)

Indicates the direction of the unidirectional approach in G00 moves.

Value	Meaning
+ sign	Positive direction.
- sign	Negative direction.

Default value: + sign

OVERRUN (P40)

Indicates the distance to be kept between the approach point and the programmed point. If it is a Lathe model, this distance must be in radius.

Possible values
Between 0.0001 and 99999.9999 degrees/min or mm/min. Between 0.00001 inches/min and 3937.00787 inches/min.

Default value: 0 (not unidirectional)

UNIFEED (P41)

Indicates the feedrate to be used from the approach point to the programmed point.

Possible values
Between 0.0001 and 99999.9999 degrees/min or mm/min. Between 0.00001 inches/min and 3937.00787 inches/min.

Default value: 0

MAXFEED (P42)

Indicates the maximum programmable feedrate (F0).

Possible values
Between 0.0001 and 199999.9999 degrees/min or mm/min. Between 0.00001 and 7874.01574 inches/min.

Default value: 5000 mm/min.

JOGFEED (P43)

Indicates the feedrate F assumed in the JOG mode if no feedrate is active.

Possible values
Between 0.0001 and 199999.9999 degrees/min or mm/min. Between 0.00001 and 7874.01574 inches/min.

Default value: 1000 mm/min.

PRBFEED (P44)

Indicates the probing feedrate when calibrating a tool in "JOG" mode.

Possible values
Between 0.0001 and 99999.9999 degrees/min or mm/min. Between 0.00001 inches/min and 3937.00787 inches/min.

Default value: 100 mm/min.

MAXCOUPE (P45)

It indicates the maximum difference allowed between the following errors of the Gantry axes.

This value is only assigned to the slave axis.

Possible values
Between 0.0001 and 99999.9999 degrees or millimeters. Between 0.00001 and 3937.00787 inches.

Default value: 1 mm.

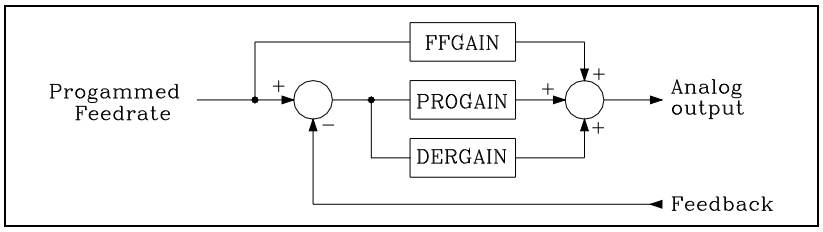
ACFGAIN (P46)

Indicates whether or not the value assigned to a.m.p. DERGAIN (P24) is applied onto the variations of the programmed feedrate (AC-forward).

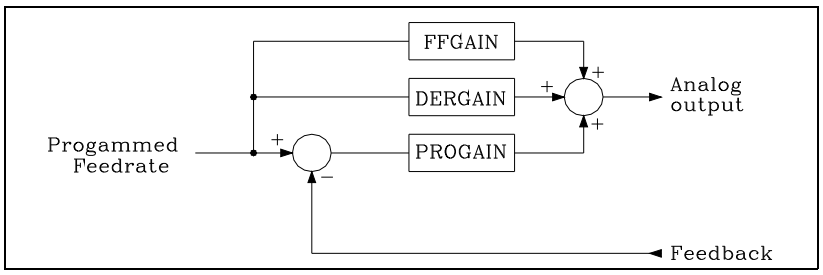
Value	Meaning
NO	It is applied on variations of following error (derivative gain).
YES	It is applied on the variations of the programmed feedrate that are due to acceleration/deceleration (AC-forward).

Default value: YES

ACFGAIN = NO



ACFGAIN = YES



REFSHIFT (P47)

This parameter is used when once the machine has been all set up, it is necessary to reinstall the feedback system and the new machine reference point (home) no longer coincides physically with the previous one.

It indicates the difference existing between the two reference points, the previous one and the current one.

Possible values
Within ±99999.9999 degrees or millimeters. Within ±3937.00787 inches.

Default value: 0

If this parameter has a value other than 0, once the home search has been carried out (the reference mark of the feedback device has been detected) , the CNC moves the distance indicated by a.m.p. REFSHIFT (P47). This way, the machine reference point will always be the same.

This movement is carried out at the feedrate indicated by a.m.p. REFEED2 (P35).

STOPTIME (P48) STOPMOVE (P49)

These parameters are used in conjunction with a.m.p. "STOPAOUT (P50)" with function G52 (move to hardstop).

STOPTIME (P48)

The CNC considers that the hardstop has been run into when a certain time period elapses without the axis moving. This time period is indicated, in thousands of a second, by parameter STOPTIME (P48).

Possible values
Integers between 0 and 65535 ms.

Default value: 0

STOPMOVE (P49)

The CNC considers the axis to be stopped when its movements do not exceed the value set by STOPMOVE (P49) during the time period set by STOPTIME (P48).

Possible values
Between 0.0001 and 99999.9999 millimeters. Between 0.00001 and 3937.00787 inches.

Default value: 0

STOPAOUT (P50)

This parameter is used with function G52 (move to hardstop) and it indicates the residual velocity command supplied by the CNC to exert pressure once contact has been detected.

Possible values
It is given in D/A converter units and it admits integer values between 0 and 32767 which corresponds to an analog command of 10V.

Default value: 0

STOPAOUT	Minimum velocity command
1	0.3 mV.
- - -	- - -
3277	1 V.
- - -	- - -
32767	10 V.

This parameter is especially designed for hydraulic devices.

When using servo motors, first reduce the maximum torque of the drive by means of an "M" function in order to prevent the motor from overheating.

INPOSW2 (P51)

This parameter is used when function G50 (controlled round corner) is active.

It defines the area before the programmed coordinate where the CNC considers the axis to be in position and goes on to execute the next block.

Possible values
Between 0 and 99999.9999 degrees or millimeters. Between 0 and 3937.00787 inches.

Default value: 0.1 mm.

It should be assigned a value 10 times the value of "INPOSW"

I0TYPE (P52)

Axis machine parameter I0TYPE has two digits:

Units:

It indicates the type of Io signal (marker pulse) provided by the feedback device.

Value	Meaning
x0	normal I0.
x1	"A" type distance-coded I0.
x2	Type B distance-coded reference mark (only linear encoder COVS).
x3	Normal I0 (search with retraction)

When using linear encoders with distance-coded reference marks (I0), set a.m.p. I0CODI1 (P68) and I0CODI2 (P69).

Tenths:

It defines whether it stops smoothly or not when detecting the reference mark of the axes or not.

Value	Meaning
0x	Normal stop on I0.
1x	Smooth stop on I0.

When setting the smooth stop, parameters "DERGAIN" and "FFGAIN" should be set to zero.

ABSOFF (P53)

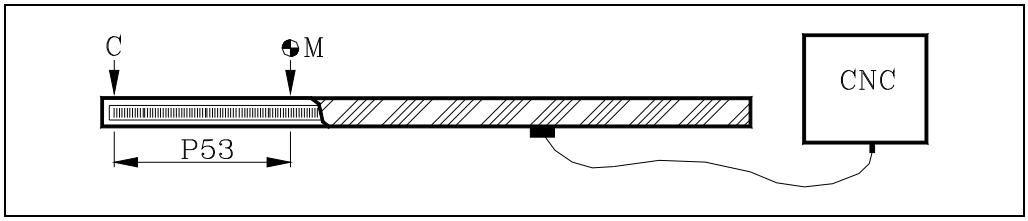
The CNC takes this parameter into account when a.m.p. I0TYPE (P52) has been set with a value other than 0.

Linear encoders having a distance-coded reference mark indicate the machine position with respect to the "zero" of the linear encoder.

Possible values
Within ± 99999.9999 millimeters.
Within ±3937.00787 inches.

Default value: 0

In order for the CNC to show the position of the axes with respect to the machine reference zero (home), this parameter must be assigned the position value (coordinate) of the machine reference zero (point "M") with respect to the "zero" of the linear encoder (C).



MINMOVE (P54)

This parameter has to do with the axis logic outputs "ANT1" through "ANT3".

If the axis move is smaller than the value indicated by this a.m.p. MINMOVE (P54), the corresponding axis logic output "ANT1 through "ANT3" goes high.

Possible values
Within ±99999.9999 degrees or millimeters.
Within ±3937.00787 inches.

Default value: 0

ROLLOVER (P55)

The CNC takes this parameter into account when the axis has been set as rotary "AXISTYPE (P0)=2 or 3". It indicates whether the rotary axis is Rollover or not.

Value	Meaning
NO	It is NOT Rollover.
YES	It is Rollover.

Default value: YES

DRIBUSID (P56)

It indicates the address of the digital drive (CAN) associated with the axis. It corresponds with the value of the drive's rotary switch (address, device select).

Value	Meaning
0	Analog axis.
1 - 8	Address of the digital drive.

Default value: 0

It is recommended (not necessary) that the Can addresses of the various axes and spindles be consecutive and start from number ·1· (the address of the CNC is always ·0·). For example, with 3 CAN axes and 1 CAN spindle, the values of this parameter must be 1, 2, 3 and 4.

EXTMULT (P57)

This parameter is to be used when utilizing a distance-coded feedback system. It indicates the ratio between the mechanical period or the graduation pitch on the glass or steel tape and the electrical period or period of the feedback signal supplied to the CNC.

Possible values

$$\text{EXTMULT (P57)} = \frac{\text{Period of the graduation on the glass (mechanical period or}}{\text{Period of the feedback signal (electrical period)}}$$

Default value: 0

Example:

E.g. Fagor linear encoder "FOT" has a graduation pitch of 100 µm and a feedback signal period of 20 µm.

EXTMULT = 100 / 20 = 5

Values to be assigned for Fagor encoders with distance-coded I0.

Linear encoders					I0CODI1 (P68)	I0CODI2 (P69)	EXTMULT (P57)
SOP SVOP	GOP	MOT MOC MOP	COT COC COP	FOP	1000	1001	1
SOX SVOX	GOX	MOX	COX	FOT	1000	1001	5
		MOY	COY		1000	1001	10
				LOP	2000	2001	1
				LOX	2000	2001	10
				FOX	1000	1001	25

Rotary encoder			I0CODI1 (P68)	I0CODI2 (P69)	EXTMULT (P57)
HO	SO	90000 pulses	1000	1001	5
HO	SO	180000 pulses	1000	1001	10
HOP	SOP	18000 pulses	1000	1001	1

SMOTIME (P58)

Sometimes the axis does not respond as desired on particular movements (handwheel movements, etc.).

In these cases, the axis response may be smoothed by applying a filter to speed variations. This filter is set by parameter SMOTIME that indicates the duration of the filter in milliseconds, value given by g.m.p. LOOPTIME (P72).

Possible values
Integers between 0 and 64 times the value assigned to g.m.p. LOOPTIME (P72). If LOOPTIME = 0 (4 ms), the maximum value that could be assigned to SMOTIME will be 64 x 4 = 256 ms.

Default value: 0

To obtain a better response, parameter SMOTIME of the axes interpolating with each other should be set with the same value.

ACCTIME2 (P59) PROGAIN2 (P60) DERGAIN2 (P61) FFGAIN2 (P62)

These parameters define the second set of gains and accelerations. They must be set like the parameters that define the first set.

First set	Second set
ACCTIME (P18)	ACCTIME2 (P59)
PROGAIN (P23)	PROGAIN2 (P60)
DERGAIN (P24)	DERGAIN2 (P61)
FFGAIN (P25)	FFGAIN2 (P62)

To select the second set of gains and accelerations, set g.m.p. ACTGAIN2 (P108) correctly or activate the CNC's general logic input ACTGAIN2 (M5013).

DRIBUSLE (P63)

The CNC considers this parameter when using a digital drive (CAN). Axis parameter DRIBUSID (P56) other than 0.

Even when the data exchange between the CNC and the drive is done via digital CAN bus, one must define whether the feedback is also handled via bus or through the corresponding connector for the axis or spindle.

Value	Meaning
0	The feedback is done via connector.
1	The feedback is done via CAN. First feedback (motor feedback).

DRIBUSLE = 0	The CNC controls the position loop. The axis feedback is done via connector. The command to the drive is sent out via CAN.
DRIBUSLE = 1	The CNC controls the position loop. The axis feedback is done via CAN. First feedback (motor feedback). The command to the drive is sent out via CAN.

POSINREF (P64)

Not being used.

SWITCHAX (P65)

When having 2 axes controlled by a single servo drive, machine parameter SWITCHAX of the secondary axis indicates which one is the main axis it is associated with.

Value	Meaning	Value	Meaning
0	None.	6	With the W axis.
1	With the X axis.	7	With the A axis.
2	With the Y axis.	8	With the B axis.
3	With the Z axis.	9	With the C axis.
4	With the U axis.	10	Spindle.
5	With the V axis.		

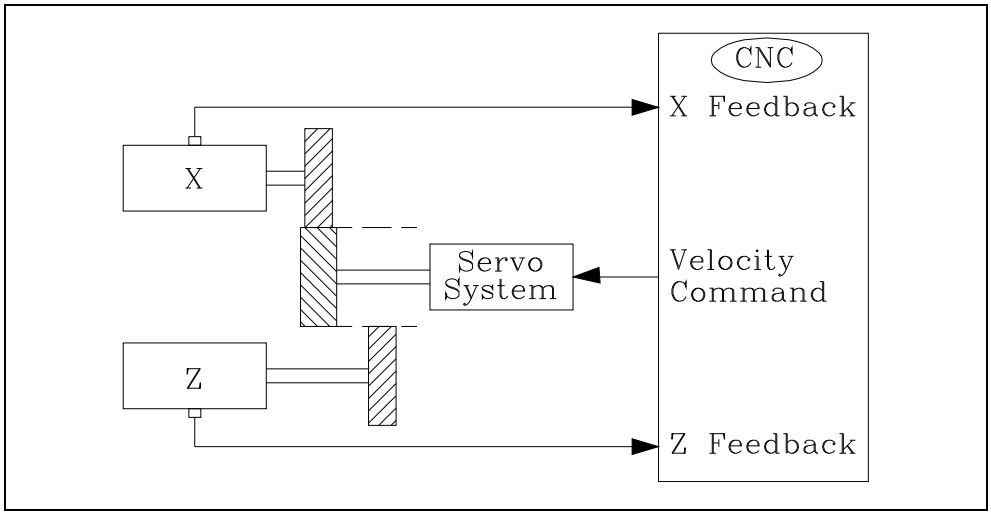
Default value: 0

Example:

On a machine where the X and Z axes cannot move at the same time, the X axis is the main axis and the Z axis is the secondary (associated with the X axis).

SWITCHAX for X = 0.

SWITCHAX for Z = 1.



SWINBACK (P66)

When having 2 axes controlled by a single servo drive, machine parameter SWINBACK of the secondary axis indicates whether it has its own feedback device or it uses that of the main axis it is associated with.

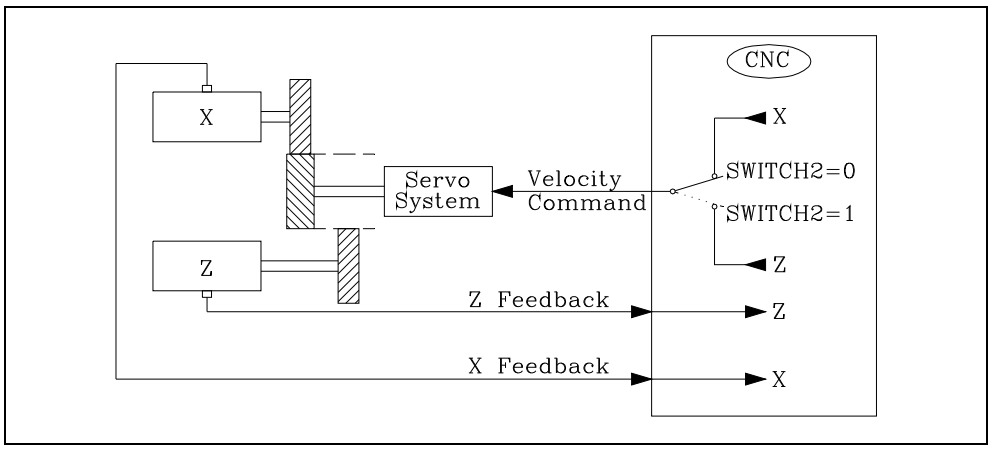
Value	Meaning
0	It assumes the feedback of the main axis.
1	It has its own feedback device.

Default value: 0

The following examples show several possibilities. In all of them, the toggling of the velocity command must be done from the PLC using the SWTCH2 mark.

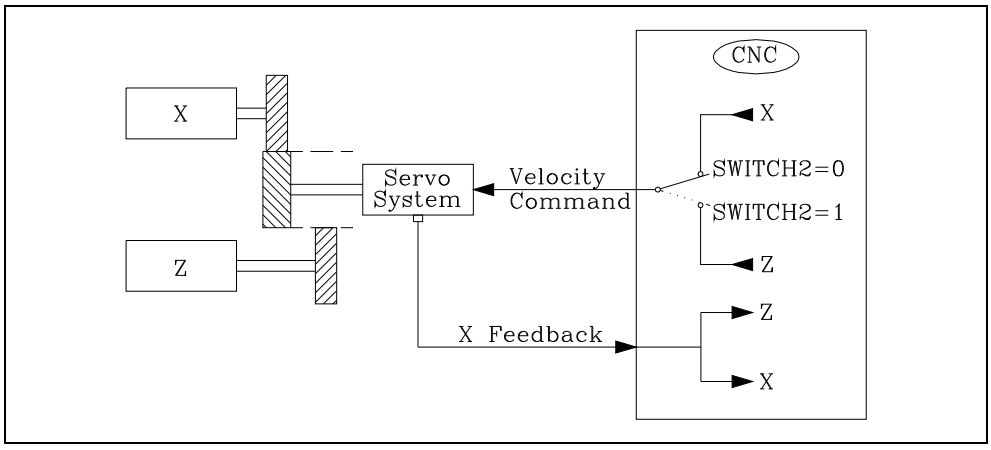
A. Each axis has its own feedback device.

X axis (main)	SWINBACK of the X axis = 0.
Z axis (secondary)	SWINBACK of the Z axis = 1.



B. The two axes share the same feedback device. It must be connected to the feedback connector of the main axis.

X axis (main)	SWINBACK of the X axis = 0.
Z axis (secondary)	SWINBACK of the Z axis = 0.



JERKLIM (P67)

It defines the derivative of the acceleration. It may be used to limit the acceleration changes to smooth the machine movements on small speed increments or decrements and with FFGAIN values close to 100%.

The CNC ignores this parameter when moving with electronic handwheels, mechanical handwheels, look ahead, threading (G33) and rigid tapping.

The smaller the value assigned to JERKLIM, the smoother the machine's response, but the acc/dec time will be longer. When increasing the value of JERKLIM, it decreases the acc/dec time but the machine response worsens.

Possible values
Between 0 and 99999.9999 m/s^3.

Default value: 0

Recommended values:

In millimeters	JERKLIM = 82*G00FEED / ACCTIME**2
In inches	JERKLIM = 2082*G00FEED / ACCTIME**2

Use parameter ACCTIME2 when adjusting the second set of parameters.

If the stability of the machine is affected by the values mentioned earlier, the JERKLIM value should be lowered to half as much.

I0CODI1 (P68) I0CODI2 (P69)

The CNC takes this parameter into account when a.m.p. I0TYPE (P52) has been set with a value other than 0. Parameter I0CODD1 (P68) indicates the gap between 2 fixed distance-coded I0's and parameter I0CODD2 (P69) indicates the gap between 2 variable distance-coded I0's.

It is set in number of waves.

Possible values
Between 0 and 65535 waves.

Default value: For I0CODD1 (P68) = 1000.
Default value: For I0CODD2 (P69) = 1001.

Example with Fagor linear encoder	
Gap between two fixed distance-coded I_0's	20 000 µm
Gap between two variable distance-coded I_0's	20 020µm
Signal period	20 µm
Number of waves between fixed I_0's	20000/(20 x EXTMULT) = 1000
Number of waves between variable I_0's	20020/(20 x EXTMULT) = 1001

Values to be assigned for Fagor encoders with distance-coded I0.

Linear encoders					I0CODI1 (P68)	I0CODI2 (P69)	EXTMULT (P57)
SOP SVOP	GOP	MOT MOC MOP	COT COC COP	FOP	1000	1001	1
SOX SVOX	GOX	MOX	COX	FOT	1000	1001	5
		MOY	COY		1000	1001	10
				LOP	2000	2001	1
				LOX	2000	2001	10
				FOX	1000	1001	25

Rotary encoder			I0CODI1 (P68)	I0CODI2 (P69)	EXTMULT (P57)
HO	SO	90000 pulses	1000	1001	5
HO	SO	180000 pulses	1000	1001	10
HOP	SOP	18000 pulses	1000	1001	1

ORDER (P70)

Filter order. The down ramp is dampened down; the larger the number the greater the drop.

Value	Filter type
[0 - 4]	Low passing filter
[0 - 4]	Notch filter (anti-resonance)
[0 - 30]	FAGOR filter

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

When applying a filter, it must be set with an order of ·3·. Before setting it to another value, consult with Fagor Automation's technical service.

If the filter has been designed wrong, it will not be applied.

The filters are not applied while moving with an electronic handwheel or a mechanical handwheel or while tracing. It is recommended not to activate these filters on machines carrying out movements against a hard stop.

Using a FAGOR filter does not admit values greater than 15.

When detecting that the FAGOR filter order is too high for the filter configuration (according to parameters FREQUEN and LOOPTIME), on power-up it will issue the message: "It is recommended to lower the order of the frequency filter".

It is recommended to start from low values (e.g.: ORDER=5) and go on increasing this value until that message is displayed.

When using filters, it is recommended to use low values of PLC parameter PLC CPUTIME (e.g.: 0, 1), to assign to the CPU the strictly necessary amount of power.

TYPE (P71)

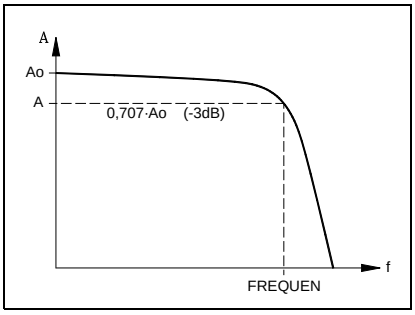
Filter type. Three types of filters may be used, namely "low passing", "anti-resonance (band rejection, notch filter)" and "FAGOR (low passing)". To obtain a good machining quality, all the axes and the spindle interpolating with each other should be defined with the same type of filter and with the same frequency.

Value	Meaning
0	"Low passing" filter.
1	"Anti-resonance" (notch) filter.
2	"FAGOR (low passing)" filter.

Default value: 0

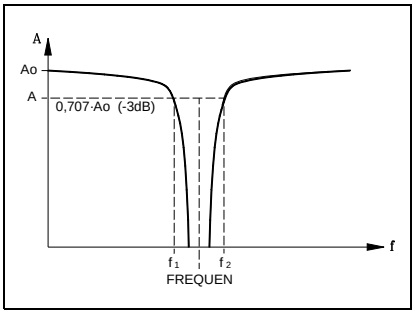
When defining anti-resonance filters, parameters NORBWIDTH and SHARE must also be set.

"Low passing" filter.



The "low passing" filter is used to limit the jerk by making the movements smoother although it has the drawback that it rounds the corners slightly.

Anti-resonance filter (notch filter).



The "anti-resonance" (notch) filter must be used when the machine has a resonance frequency to be eliminated.

Starting the CNC up while Fagor filters are active.

When starting the CNC up, if the Fagor filters are active on any of the axes and g.m.p IPOTIME (P73) =1, the CNC will display the following error message:

- Parameter TYPE=2 is incompatible with general parameter IPOTIME.

When starting the CNC up, if the Fagor filters are active on any of the axes and a.m.p SMOTIME (P58) is other than 0, the CNC will display the following error message:

- Parameter TYPE=2 is incompatible with general parameter SMOTIME.

After the start-up, if the value of that parameter is not changed, the CNC will cancel that parameter automatically.

FREQUEN (P72)

The meaning of this parameter depends on the type of filter being applied.

On "low passing" and "FAGOR" filters, it indicates the break point frequency or frequency where the amplitude drops 3 dB or it reaches 70% of the nominal amplitude.

-3dB = 20 log (A/Ao) ==> A = 0.707 Ao

For the "anti-resonance" (notch) filter, it indicates the mid frequency or frequency at which the resonance reaches its maximum value.

Possible values
Between 0 and 500.0 Hz.

Default value: 30

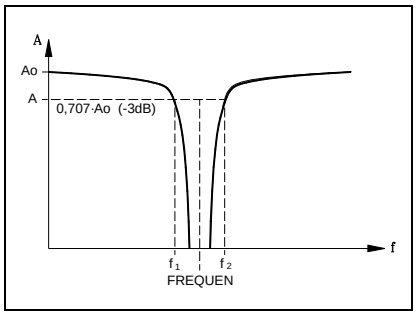
NORBWID (P73)

Standardized bandwidth.

This parameter is only taken into account for the "anti-resonance (notch)" filter type.

Possible values
between 0 and 100.0

Default value: 1



It is calculated with the following formula.

Points f1 and f2 correspond to the cutoff frequency or frequency at which its amplitude drops 3 dB or reaches 70% of the nominal amplitude.

$$NORBWID = \frac{FREQUEN}{(f_2 - f_1)}$$

SHARE (P74)

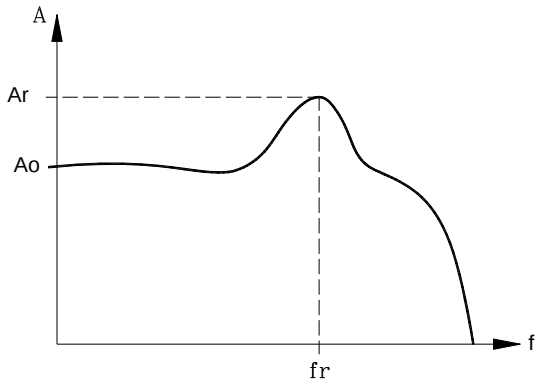
Signal percentage that passes through the filter. This value must be equivalent to the percentage overshooting of the resonance because it has to make up for it.

This parameter is only taken into account for the "anti-resonance (notch)" filter type.

Possible values
between 0 and 100

Default value: 100

Calculation example for a particular response of the machine.



SHARE=100(Ar-Ao)/Ao

FLIMIT (P75)

Maximum safety limit for the axis feedrate. This limit is activated from the PLC and is applied to all the work modes, including the PLC channel.

Possible values
Between 0 and 99999.9999 degrees/min or mm/min.
Between 0 inches/min and 3937.00787 inches/min.

Default value: 0

This limit is activated for all the axes using the mark FLIMITAC (M5058). When the limit is canceled, the CNC recovers the programmed feedrate.

This limit permits clearing the axis feedrate temporarily via PLC, e.g. when opening the doors, etc.

TANSLAID (P76) **TANSLANA (P77)** **TORQDIST (P78)**
PRELOAD (P79) **PRELFITI (P80)** **TPROGAIN (P81)**
TINTTIME (P82) **TCOMPLIM (P83)**

Not being used.

ADIFEED (P84)

Maximum feedrate allowed, due to the additive handwheel.

Possible values
Between 0 and 99999.9999 degrees/min or mm/min.
Between 0 inches/min and 3937.00787 inches/min.

Default value: 1000

FRAPIDEN (P85)

Maximum axis feedrate when activating the EXRAPID mark and when pressing the rapid key in execution or in simulation with motion.

If set to 0, it assumes the feedrate set by parameter G00FEED. If it is set with a value higher than G00FEED, the feedrate will be limited to G00FEED.

Possible values
Between 0 and 199999.9999 degrees/min or mm/min.
Between 0 and 7874.01574 inches/min.

Default value: 0

This limit does not affect the rapid jog which will still have the value of parameter G00FEED.

PITCHB (P86)

Leadscrew pitch.

This parameter must only be set when using CAN servo system. When not using CAN servo system, the leadscrew pitch is defined with parameter PITCH (P7).

When using gear reduction on the shaft, only the whole assembly must be taken into account when setting one of parameters PITCHB or NPULSES.

When using CAN servo, if both parameters NPULSES and PITCHB are set to ·0·, the CNC will assume the equivalent values of the drive.

INPREV (P87) OUTPREV (P88)

Parameters that indicate the input revolutions (INPREV) and the output revolutions (OUTPREV) of each axis. These parameters are used to handle the gear ratios on the axes. The default value for both parameters is 0.

Both axis machine parameters INPREV and OUTPREV must be 0 or both other than zero. Do not program one with a 0 value and the other one with a value other than 0.

HPITCH (P89)

On Hirth axes, it indicates its pitch in degrees. When set to ·0·, it assumes a pitch value of 1º.

Possible values
Between 0 and 99999.9999 degrees. (the remainder of 360/HPITCH must necessarily be zero)

Default value: 1

It admits values other than 1º and decimal values. When HPITCH is set with a decimal value, the screen will show the coordinates with decimals.

Any stop or continuous jog movement will stop the axis in coordinates multiple of HPITCH. The incremental jog movements will be similar to the ones carried out with a 1 degree pitch.

- For incremental switch positions of 1, 10, 100 or 1000, it will move 1 step.
- For an incremental switch position 10000, the movement will be multiple of the closest pitch to 10º (and under 10º). If the pitch value is greater than 10º, it will move a single step.

Even if the position of a Hirth axis does not coincide with its Hirth pitch, any other axis may be moved to a valid position in both automatic and jog modes. An error message will be issued if the position to move the axis does not coincide with the pitch. In any case, it is possible to move any other axis in both automatic and jog modes.

AXISDEF (P90)

It allows customizing the movement of the axis.

This parameter has 16 bits counted from right to left.

bit	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
	■	■	■	■	■	■	■	■	■	■	■	■	■	■	■	■

Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning
0 - 14	Not being used.
15	Rollover axis. Movement in G53 via the shortest way.

Default value in all the bits: 0

Bit 15. Rollover axis. Movement in G53 via the shortest way.

This bit indicates how the movements in G53 are carried out for a positioning-only rotary axis without travel limits.

AXISTYPE = 3 or 4 ROLLOVER = YES LIMIT+ = 0 LIMIT- =0

When set to ·1·, G53 movements are carried out along the shortest path. When several presets have been made, the axis may rotate several complete turns.

DRISET (P91)

It defines from which set of the drive will the following drive parameters be read:

- NP 121: Input rpm.
- NP 122: Output rpm.

This parameter is used to apply a different gear ratio to each axis when having two CAN axes (only with AXD drives) that share the same drive. This allows controlling two completely different axes with the same motor.

Bit	Meaning
0 - 7	Drive set from which drive parameters NP121 and NP 122 will be read.

Default value: 0

A.m.p. DRISET (P91) is only taken into account when two CAN axes share the same drive with switch parameters. Otherwise, it will read the data of set 0.

ACCTIMET (P92) PROGAINT (P93) DERGAINT (P94) FFGAINT (P95)

These parameters define the third set of gains and accelerations. They must be set like the parameters that define the first set.

First set	Second set	Third set
ACCTIME (P18)	ACCTIME2 (P59)	ACCTIMET (P92)
PROGAIN (P23)	PROGAIN2 (P60)	PROGAINT (P93)
DERGAIN (P24)	DERGAIN2 (P61)	DERGAINT (P94)
FFGAIN (P25)	FFGAIN2 (P62)	FFGAINT (P95)

To select the third set of gains and accelerations, set g.m.p. ACTGAINT (P185) correctly or activate the CNC's general logic input ACTGAINT (M5063).

DIFFCOMP (P96)

It indicates whether the position difference between the master and the slave will be corrected or not so that difference is zero once both axes of a Gantry pair have been homed.

Value	Meaning
0	The position difference between the master and the slave is not corrected.
1	The position difference between the master and the slave is corrected.

Default value: 1

MAXDIFF (P97)

It indicates the maximum position difference in mm between the master and the slave once both axes of a Gantry pair have been homed; the position difference will not be corrected over this value.

Value	Meaning
0	There is no maximum limit up to which the position difference will be compensated.
0.0001 - 99999.9999	From this value on, the position difference will not be compensated.

Default value: 0

This axis machine parameter is taken into account when it is about correct the position difference.

PEAKDISP (P98)

Every time the axis movement is inverted, the CNC will apply to that axis the velocity command corresponding to the movement plus an additional velocity command (to make up for backlash). This additional command is eliminated (compensation peak cutoff) depending on the values of the following parameters:

G.m.p. BAKTIME (P30), g.m.p. ACTBAKAN (P145) and a.m.p. PEAKDISP (P98).

Axis machine parameter PEAKDISP (P98) defines the actual distance traveled along the corresponding axis after the theoretical reversal, from which the reversal peak is cut off on that axis.

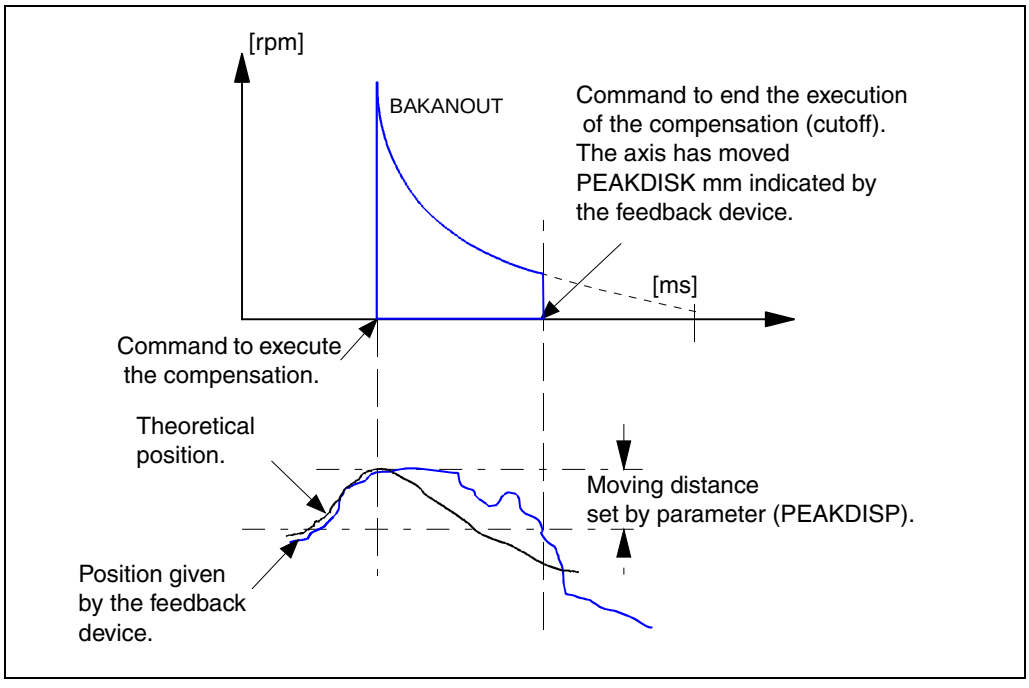
Possible values
Between 0 and 99999.9999 millimeters.

Default value: 0.005

This parameter is only taken into account when bit 1 of g.m.p. ACTBAKAN (P145) is set to ·1· if the peak is exponential or if it is square.

If the value of a.m.p. PEAKDISP (P98) = 0 and bit 1 of g.m.p. ACTBAKAN (P145) = 1, the compensation peak is cut off with the second consecutive loop where the counting reversal has been detected.

Example: cutting the exponential compensation off.



REVEHYST (P99)

This parameter is used to be able to control when the compensation should really be applied after detecting a movement reversal and not applying it every time a reversal command is received.

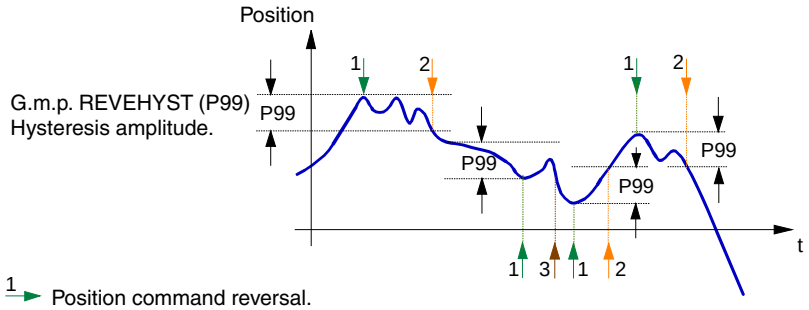
This a.m.p. should be set with the value that the position must vary after the first movement reversal (hysteresis) so it is considered that the command to compensate has been issued, hence prevent it from issuing the compensations every time it receives the command to reverse the moving direction if that margin has not been exceeded. The value entered in this parameter must be in mm for linear axes and in degrees for rotary axes. Default value 0.0000.

Example

If REVEHYST= 5 dµm, the CNC will not activate the reversal compensation in all the reversals after the first one as long as the position does not change at least a value equal to the setting of a.m.p. REVEHYST since the first command to reverse the position command was issued.

In other words, if a reversal command is sent when the position command has varied 2 dµm from the position where the first reversal command took place, it does not issue the compensation (it has not exceeded the value given by a.mp. REVHYST) and it just reverses the movement.

Only when the position command variation reaches 5 dµm, it will issue the compensation and the next command to reverse will be taken as the new reference for evaluating the position variation to determine when it reaches the value given in a.m.p. REVEHYST again and it compensates for it again.



1 → Position command reversal.
2 → Maximum limit given by P99. Issuing the compensation.
3 → Compensation cancellation limit.

Considerations

- Being a.m.p. REVEHYST (P99) =0, the backlash compensation by reversal peak or backlash will always be applied in each reversal.
- Having set a.m.p. REVEHYST (P99) with a value other than 0, to set a.m.p. PEAKDISP (P14) to cut the backlash peak, we recommend to use a smaller REVEHYST value than that of PEAKDISP so as not to apply the backlash peak.
- If they have been set as DRO axes, the value of a.m.p. BACKLASH (P14) will be taken into account for these axes. In these cases, especially if they have sinusoidal feedback, we recommend to use a value of a.m.p. REVEHYST (P99) other than 0 in order to apply backlash compensation.

FBACKDIF (P100)

Axis machine parameter that sets the maximum difference allowed between the first and the second feedback:

Possible values
Between 0 and 99999.9999 degrees or millimeters. Between 0 and 3937.00787 inches.

Default value: 1 mm (for linear axes)
Default value: 1º (for rotary axes)

This position difference may be monitored on the oscilloscope using the read variable FBDIF(X-C). If the value of FBACKDIF (P100)=0, the feedback difference is not monitored. It is recommended to set a.m.p. FBACKDIF (P100) to a value other than 0.

If the difference between the two feedback readings exceeds the value set in a.m.p. FBACKDIF (P100), the CNC will show the relevant error message.

MAXDIFAB (P101)

This parameter sets the maximum position difference between that of the CNC and the one shown by the absolute feedback on power-up.

Possible values
Between 0 and 99999.9999 degrees or millimeters. Between 0 and 3937.00787 inches.

Default value: 1 mm (for linear axes)
Default value: 1º (for rotary axes)

When using absolute feedback, a.m.p. MASDIFAB (P101)=0, the CNC will show a warning on power-up indicating that safety is disabled.

If the position value received from the absolute feedback is not the same as that of the CNC and it is even greater than the value of a.m.p. MAXDIFAB (P101), the CNC will show an error window on power-up (this window only comes up once in every power-up).

To eliminate this error, select the option "REMOVE ERROR" and press [ENTER]. This way, the axis will assume the value indicated by the absolute feedback.

If the [EXIT] option is selected or the [ESC] is pressed, the CNC will show the error "Feedback error on the axis" and will prevent the machine from moving. This error can only be eliminated by starting the CNC up again and selecting the option "REMOVE ERROR".

Once the error has been removed, if the axis is out of the permitted limits, the CNC will only allow to move the axes towards the area within the limits.

This error comes up the first time an absolute feedback device is connected or when changing the offsets of the feedback device. In these cases, once the error has been removed as described earlier, that error will not come up again.

4.4 Spindle parameters

SPDLTYPE (P0)

Indicates the type of spindle output being used.

Value	Meaning
0	±10 V analog output.
1	2-digit BCD coded "S" output. See *"2-digit BCD code output conversion table"* on page 447.
2	8-digit BCD coded "S" output.

Default value: 0

DFORMAT (P1)

Indicates the display format for the spindle.

Value	Meaning
0	In 4 digits.
1	In 5 digits.
2	In 4.3 format
3	In 5.3 format
4	It is not displayed.

Default value: 0

MAXGEAR1 (P2) MAXGEAR2 (P3) MAXGEAR3 (P4) MAXGEAR4 (P5)

They indicate the maximum spindle speed assigned to each gear. When using an automatic gear change, these values will be used to make the change.

MAXGEAR1	for gear 1 (M41).
MAXGEAR2	for gear 2 (M42).
MAXGEAR3	for gear 3 (M43).
MAXGEAR4	for gear 4 (M44).

Possible values
Integers between 0 and 65535 rpm.

Default value: For MAXGEAR1 (P2) = 1000 rpm.
For MAXGEAR2 (P3) = 2000 rpm.
For MAXGEAR3 (P4) = 3000 rpm.
For MAXGEARa4 (P5) = 4000 rpm.

When not using all 4 gears, use the lower ones and set the unused ones to the same value as the highest one used.

AUTOGEAR (P6)

Indicates whether the gear change is generated automatically or not by the CNC activating the M functions M41, M42, M43 and M44.

Value	Meaning
NO	There is no automatic gear change.
YES	There is automatic gear change.

Default value: NO

POLARM3 (P7) **POLARM4 (P8)**

Indicates the sign of the spindle velocity command for M03 and M04.

Value	Meaning
+ sign	Positive command.
- sign	Negative command.

Default value: For POLARM3 (P7) = + sign.
For POLARM4 (P8) = - sign.

If the same value is assigned to both parameters, the CNC will output a single polarity (0V to 10V) command with the indicated sign.

SREVM05 (P9)

This parameter is used with a Mill model CNC.

Indicates whether it is necessary or not to stop the spindle (M05) when reversing rotation direction during a tapping canned cycle (G84).

Value	Meaning
NO	It is not necessary.
YES	It is necessary.

Default value: YES

MINSOVR (P10) **MAXSOVR (P11)**

Indicate the minimum and maximum % applicable to the programmed spindle speed.

Possible values
Integer numbers between 0 and 255.

Default value: For MINSOVR (P10) = 50.
For MAXSOVR (P11) = 150.

The resulting speed will be limited to the value indicated by s.m.p. MAXVOLT1 (P37), MAXVOLT2 (P38), MAXVOLT3 (P39) or MAXVOLT4 (P40) corresponding to the selected gear.

SOVRSTEP (P12)

Indicates the incremental step of the programmed spindle speed every time the override keys at the operator panel are pressed.

Possible values
Integer numbers between 0 and 255.

Default value: 5

NPULSES (P13)

Indicates the number of pulses per revolution provided by the spindle encoder. 0 means that there is no spindle encoder.

It must be set when the drive's velocity command is analog or it is sent via CAN (DRIBUSLE = 0 or 1).

When the main spindle does not have an encoder (NPULSES=0), the CNC shows its theoretical rpm (affected by the %).

Possible values
Integer numbers between 0 and 65535.

Default value: 1000

When using a CAN servo system, if parameter NPULSES and parameters INPREV and OUTPREV of all the gears are set with a ·0· value, the CNC will assume the equivalent ones of the drive.

DIFFBACK (P14)

Indicates whether the feedback device uses differential signals (double ended) or not.

Value	Meaning
NO	It does NOT use differential signals.
YES	It uses differential signals.

Default value: YES

FBACKAL (P15)

Indicates whether the feedback alarm for this axis will be ON or OFF.

Value	Meaning
OFF	No feedback alarm desired, it is canceled.
ON	Feedback alarm is being used.

Default value: ON

AXISCHG (P16)

Indicates the counting direction. If correct, leave it as is, but to change it, select YES if it was set to NO and viceversa. If this parameter is changed, s.m.p. LOOPCHG (P26) must also be changed so the spindle does not "run away".

Possible values
NO / YES.

Default value: NO

DWELL (P17)

Indicates the dwell from the moment the "ENABLE" signal is activated until the velocity command is sent out.

Possible values
Integers between 0 and 65535 ms.

Default value: 0 (there is no dwell).

ACCTIME (P18)

This parameter is used when working with the spindle in closed loop and it indicates the acceleration time given to reach the maximum speed set by s.m.p. MAXVOLT1 (P37) thru MAXVOLT4 (P40) in each gear. This time is also valid for the deceleration stage.

Possible values
Integers between 0 and 65535 ms.

Default value: 0 (there is no control).

INPOSW (P19)

Indicates the width of the IN POSITION zone where the CNC considers the spindle to be in position when working in closed loop (M19).

Possible values
Between 0 and 99999.9999 degrees.

Default value: 0.01 degrees.

INPOTIME (P20)

Indicates the time period that the spindle must remain in the "IN POSITION" zone in order to consider it to be in position.

This prevents the CNC from considering the spindle to be in position and executing the next block on those machines where the spindle could just overshoot the "IN POSITION" zone.

Possible values
Integers between 0 and 65535 ms.

Default value: 0

MAXFLWE1 (P21)

Indicates the maximum following error allowed for the spindle when moving in closed loop (M19).

Possible values
Between 0 and 99999.9999 degrees.

Default value: 30 degrees.

MAXFLWE2 (P22)

Indicates the maximum following error allowed for the spindle when stopped in closed loop (M19).

Possible values
Between 0 and 99999.9999 degrees.

Default value: 0.1 degrees.

PROGAIN (P23)

The CNC takes this parameter into account when operating in closed loop (M19).

Indicates the value of the proportional gain. Its value represents the analog command in millivolts corresponding to a following error of 1 degree.

Velocity command (mV)
= Following error (degrees) x PROGAIN

Possible values
Integers between 0 and 65535 mV/degree.

Default value: 1000 mV/degree.

This value is taken for the first spindle gear and the CNC calculates the values for the rest of the gears.

Example:

s.m.p. MAXGEAR1 (P2) = 500 rev/min. The desired speed for a 1 degree of following error is S = 1000°/min (2.778 rev/rpm).

Command from the drive: 9.5V for 500 rpm

Velocity command corresponding to S = 1000 º/min. (2.778 rpm)

Velocity command = (9.5/500) x 2,778 = 52.778 mV

Therefore "PROGAIN" = 53

DERGAIN (P24)

The CNC takes this parameter into account when operating in closed loop (M19).

Indicates the value of the derivative gain. Its value represents the analog command (in millivolts) corresponding to a change in following error of 1mm (0.03937 inches) in 10 milliseconds.

This velocity command will be added to the one calculated for the proportional gain.

Command

$$= \left(\xi \cdot \text{PROGAIN} + \frac{\xi \cdot \text{DERGAIN}}{10 \cdot t}\right)$$

It is a good idea to also use the acc./dec. a.m.p. ACCTIME2 (P18) for this axis (with a value other than "0") if this gain is to be applied.

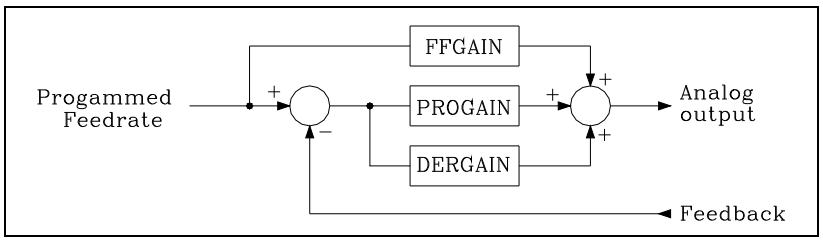
Possible values
Integer numbers between 0 and 65535.

Default value: 0 (derivative gain not applied)

FFGAIN (P25)

The CNC takes this parameter into account when operating in closed loop (M19).

Indicates the % of velocity command due to the programmed speed. The rest will depend upon the following error; both the proportional and derivative gains will be applied onto this following error.



Command

$$= \left(\xi \cdot \text{PROGAIN} + \frac{\xi \cdot \text{DERGAIN}}{10 \cdot t} + \frac{\text{FFGAIN} \times \text{Fprog} \times \text{MAXVOLT}}{100 \cdot \text{G00FEED}}\right)$$

The feed-forward gain improves the position loop minimizing the following error, but it should only be used when working with acceleration/deceleration.

Possible values
Integer numbers between 0 and 100.

Default value: 0 (feed-forward gain not applied)

Usually, a value between 40% and 80% is assigned depending mainly on the type of machine and its characteristics.

LOOPCHG (P26)

Indicates the sign of the velocity command. If correct, leave it as is, but to change it, select YES if it was set to NO and viceversa.

Possible values
NO / YES.

Default value: NO

MINANOUT (P27)

Indicates the minimum value for the spindle velocity command.

Possible values
It is given in D/A converter units and it admits integer values between 0 and 32767 which corresponds to an analog command of 10V.

Default value: 0

MINANOUT	Minimum velocity command
1	0.3 mV.
- - -	- - -
3277	1 V.
- - -	- - -
32767	10 V.

SERVOFF (P28)

Defines the velocity command offset value for the drive.

Possible values
It is given in D/A converter units and it admits integer values between 0 and ±32767 which corresponds to an analog command of 10V.

Default value: 0 (not applied)

SERVOFF	Command
-32767	-10 V.
- - -	- - -
-3277	-1 V.
- - -	- - -
1	0.3 mV.
- - -	- - -
3277	1 V.
- - -	- - -
32767	10 V.

LOSPDLIM (P29) UPSPDLIM (P30)

Indicate the upper and lower limits of the actual spindle speed so the CNC can "notify" the PLC (by means of the "REVOK" signal) that the actual spindle rpms are the same as the programmed ones.

Possible values
Integer numbers between 0 and 255.

Default value: For LOSPDLIM (P29) = 50%.

Default value: For UPSPDLIM (P30) = 150%.

DECINPUT (P31)

Indicates whether or not the spindle has a home switch to synchronize the spindle when working in M19.

Value	Meaning
NO	It has no home switch.
YES	It has a home switch.

Default value: YES

REFPULSE (P32)

Indicates the type of marker pulse Io to synchronize the spindle when working in M19.

Value	Meaning
+ sign	Positive pulse (5 V).
- sign	Negative pulse (0 V).

Default value: + sign

REFDIREC (P33)

Indicates the rotating direction when synchronizing the spindle during M19.

Value	Meaning
+ sign	Positive direction.
- sign	Negative direction.

Default value: + sign

REFEED1 (P34)

Indicates the spindle's positioning speed when in M19 and the synchronizing speed until it finds the home switch.

Possible values
Between 0.0001 degrees/min and 99999.9999 degrees/min.

Default value: 9000 degrees/min.

REFEED2 (P35)

Indicates the synchronizing speed of the spindle after hitting the home switch and until it finds the marker pulse.

Possible values
Between 0.0001 degrees/min and 99999.9999 degrees/min.

Default value: 360 degrees/min.

REFVALUE (P36)

Indicates the position value assigned to the reference point of the spindle (home or marker pulse).

Possible values
Within ±99999.9999 degrees.

Default value: 0

MAXVOLT 1 (P37) **MAXVOLT 2 (P38)** **MAXVOLT 3 (P39)** **MAXVOLT 4 (P40)**

Indicates the velocity command corresponding to the maximum speed of gears 1, 2, 3 and 4.

Possible values
Integer numbers between 0 mV and 9999 mV.

Default value: 9500 (9.5 V)

There is no need to set this parameter for an axis handled via CAN.

GAINUNIT (P41)

The CNC takes this parameter into account when operating in closed loop (M19).

Defines the units for s.m.p. PROGAIN (P23) and DERGAIN (P24).

Value	Meaning
0	millivolts/degree.
1	millivolts/0.01 degree

Default value: 0 (mV/degree)

This parameter is used when working with the spindle in closed loop.

A value of "1" will be assigned when the analog command corresponding to a following error of 1 degree is very small. This offers greater sensitivity for adjusting s.m.p. PROGAIN (P23) and DERGAIN (P24)

ACFGAIN (P42)

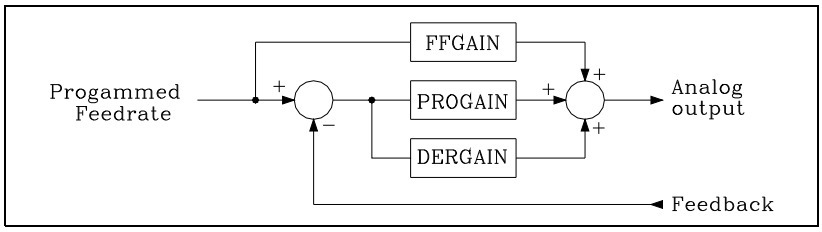
The CNC takes this parameter into account when operating in closed loop (M19).

Indicates whether or not the value assigned to a.m.p. DERGAIN (P24) is applied onto the variations of the programmed speed (AC-forward).

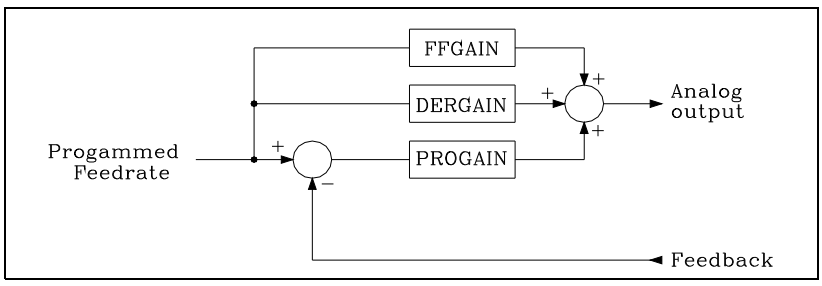
Value	Meaning
NO	It is applied on variations of following error (derivative gain).
YES	It is applied on the variations of the programmed speed that are due to acceleration/deceleration (AC-forward).

Default value: YES

ACFGAIN = NO



ACFGAIN = YES



M19TYPE (P43)

This parameter sets the type of spindle orient (M19) available.

It indicates whether the spindle must be homed when switching from open to closed loop or it is enough to home it once on power-up.

Value	Meaning
0	When switching from open loop to closed loop.
1	Once after power-up.

Default value: 0

DRIBUSID (P44)

It indicates the address of the digital drive (CAN) associated with the spindle. It corresponds with the value of the drive's rotary switch (address, device select).

Value	Meaning
0	Analog spindle.
1 - 8	Address of the digital drive.

Default value: 0

It is recommended (not necessary) that the Can addresses of the various axes and spindles be consecutive and start from number ·1· (the address of the CNC is always ·0·). For example, with 3 CAN axes and 1 CAN spindle, the values of this parameter must be 1, 2, 3 and 4.

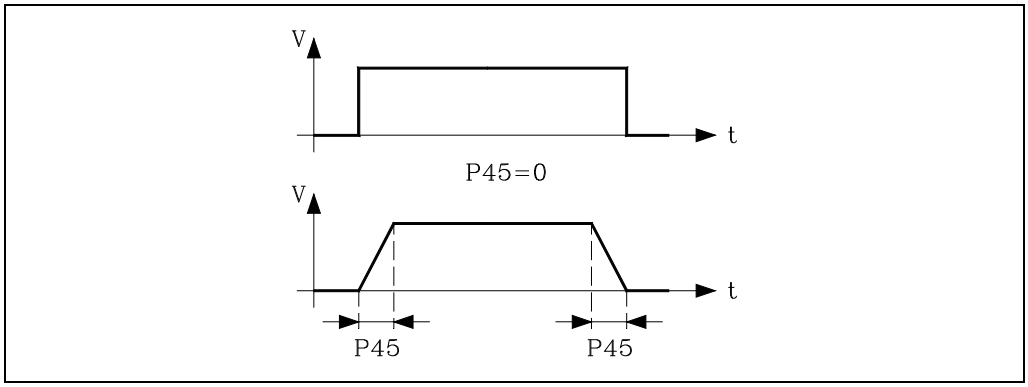
OPLACETI (P45)

When working in open loop (M3, M4) spindle command variations may be in a step or in a ramp.

This parameter indicates the duration of the ramp in milliseconds for the maximum "S". If OPLACETI=0, it will be in a step.

Possible values
Integers between 0 and 65535 ms.

Default value: 0 (in steps).



SMOTIME (P46)

Sometimes the axis does not respond as desired on particular movements (handwheel movements, etc.).

In these cases, the response of the spindle may be smoothed by applying a filter to the speed changes.

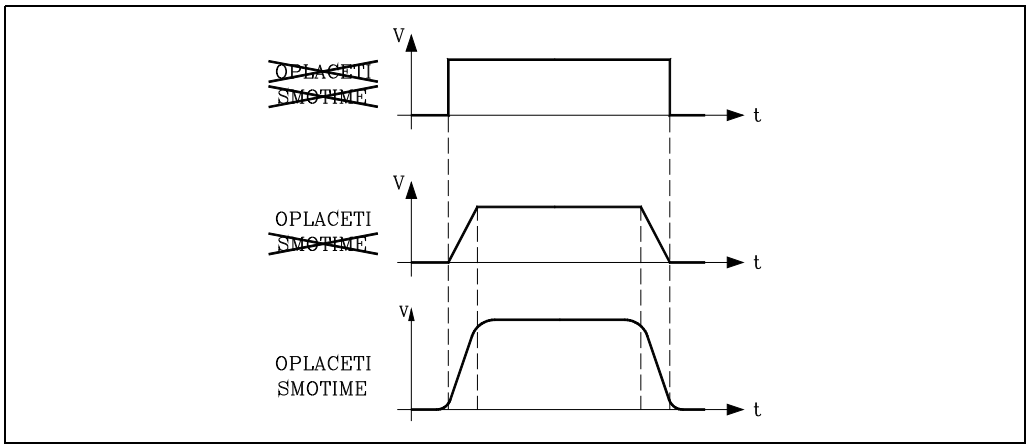
This filter is set by parameter SMOTIME that indicates the duration of the filter in milliseconds, value given by g.m.p. LOOPTIME (P72).

Possible values
Integers between 0 and 64 times the value assigned to g.m.p. LOOPTIME (P72). If LOOPTIME = 0 (4 ms), the maximum value that could be assigned to SMOTIME will be 64 x 4 = 256 ms.

Default value: 0 (not applied)

To obtain a better response, parameter SMOTIME of the axes interpolating with each other should be set with the same value.

The spindle's response can also be smoothened when working in open loop (M3, M4). In this case, s.m.p. OPLACETI (P45) and SOMTIME (P46) must be used.



ACCTIME2 (P47) **PROGAIN2 (P48)** **DERGAIN2 (P49)** **FFGAIN2 (P50)**

These parameters define the second set of gains and accelerations. They must be set like the parameters that define the first set.

First set	Second set
ACCTIME (P18)	ACCTIME2 (P47)
PROGAIN (P23)	PROGAIN2 (P48)
DERGAIN (P24)	DERGAIN2 (P49)
FFGAIN (P25)	FFGAIN2 (P50)

To select the second set of gains and accelerations, set g.m.p. ACTGAIN2 (P108) correctly or activate the CNC's general logic input ACTGAIN2 (M5013).

DRIBUSLE (P51)

The CNC considers this parameter when using a digital drive (CAN). Axis parameter DRIBUSID (P56) other than 0.

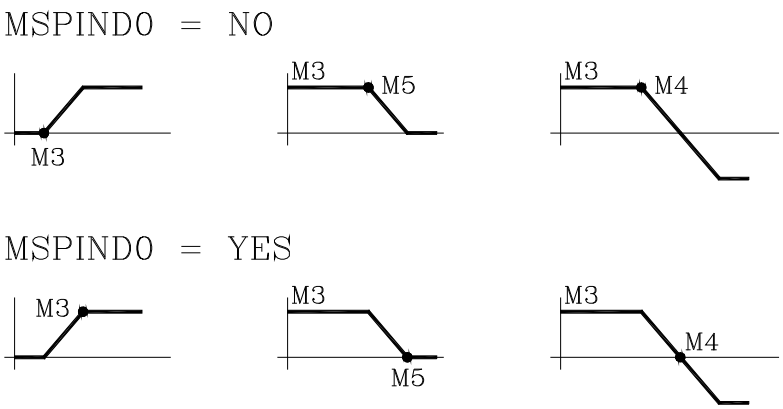
Even when the data exchange between the CNC and the drive is done via digital CAN bus, one must define whether the feedback is also handled via bus or through the corresponding connector for the axis or spindle.

Value	Meaning
0	The feedback is done via connector.
1	The feedback is done via CAN. First feedback (motor feedback).

DRIBUSLE = 0	The CNC controls the position loop. The axis feedback is done via connector. The command to the drive is sent out via CAN.
DRIBUSLE = 1	The CNC controls the position loop. The axis feedback is done via CAN. First feedback (motor feedback). The command to the drive is sent out via CAN.

MSPINDO (P52)

Indicates when functions M3, M4, M5 are to be sent out. While the spindle is accelerating and decelerating.



SYNPOSOF (P53) **SYNSPEOF (P54)**

Not being used.

ACCTIME3 (P55) **PROGAIN3 (P56)** **DERGAIN3 (P57)** **FFGAIN3 (P58)**

These parameters define the third set of gains and accelerations. They must be set like the parameters that define the first set.

First set	Second set	Third set
ACCTIME (P18)	ACCTIME2 (P47)	ACCTIME3 (P55)
PROGAIN (P23)	PROGAIN2 (P48)	PROGAIN3 (P56)
DERGAIN (P24)	DERGAIN2 (P49)	DERGAIN3 (P57)
FFGAIN (P25)	FFGAIN2 (P50)	FFGAIN3 (P58)

Possible values
The same as for the first gear.

Default value: For ACCTIME3 (P55) = 4000 ms .
For PROGAIN3 (P56) = 50 mV/degree.
For DERGAIN3 (P57) = 0.
For FFGAIN3 (P58) = 100.

When working with FFGAIN3 (P58) = 100, set the MAXGEAR and MAXVOLT parameters properly.

ACCTIME4 (P59) **SECACESP (P60)** **SYNCPOLA (P61)**

Not being used.

CONCLOOP (P62)

It indicates whether the spindle operates in closed positioning loop (as if it were an axis) or not.

Value	Meaning
NO	It operates in open loop.
YES	It operates in closed position loop (as if it were an axis).

Default value: NO

In order to operate in closed positioning loop, the spindle must have an encoder and a good servo system for the full speed range.

When working with M19, the first two sets of gains and accelerations are used regardless of the value given to this parameter.

When working in closed positioning loop (M3, M4, M5) the third set of gains and accelerations is used: ACCTIME3, PROGAIN3, DERGAIN3 and FFGAIN3.

SYNMAXSP (P63) M3M4SIM (P64)

Not being used.

SINMAGNI (P65)

It indicates the multiplying factor (x1, x4, x20, etc.) that the CNC must apply only to sinusoidal feedback signals of the spindle.

For square feedback signals, this parameter must be set to 0 and the CNC will always apply a multiplying factor of x4.

Possible values
Integer numbers between 0 and 255.

Default value: 0

Spindle feedback resolution is set by s.m.p. NPULSES (P13) and SINMAGNI (P65).

Example

We would like to obtain a 0.001º resolution by using a 3600 pulse/rev sinusoidal encoder.

We must calculate the multiplying factor "SINMAGNI" to be applied by the CNC to the pulses provided by the encoder in order to obtain the desired resolution.

SINMAGNI = degrees per turn / (number of pulses x resolution)

SINMAGNI = 360 / (3600 x 0,001) = 100

Therefore: NPULSES =3600 SINMAGNI=100

SLIMIT (P66)

Maximum safety limit for the spindle speed. This limit is activated from the PLC and is applied in all the work modes, including the PLC channel. When the spindle is controlled by the PLC by means of the PLCCNTL mark, this limit is ignored.

Possible values
Between 0 and 65535 rpm.

Default value: 0

This limit is activated using the mark SLIMITAC (M5059). When this limit is canceled, the CNC recovers the programmed speed.

This limit permits clearing the spindle speed temporarily via PLC, e.g. when opening the doors, etc.

ORDER (P67)

Filter order. The down ramp is dampened down; the larger the number the greater the drop.

Value	Filter type
[0 - 4]	Low passing filter
[0 - 4]	Notch filter (anti-resonance)
[0 - 30]	FAGOR filter

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

When applying a filter, it must be set with an order of ·3·. Before setting it to another value, consult with Fagor Automation's technical service.

If the filter has been designed wrong, it will not be applied.

The filters are not applied while moving with an electronic handwheel or a mechanical handwheel or while tracing. It is recommended not to activate these filters on machines carrying out movements against a hard stop.

Using a FAGOR filter does not admit values greater than 15.

When detecting that the FAGOR filter order is too high for the filter configuration (according to parameters FREQUEN and LOOPTIME), on power-up it will issue the message: "It is recommended to lower the order of the frequency filter".

It is recommended to start from low values (e.g.: ORDER=5) and go on increasing this value until that message is displayed.

When using filters, it is recommended to use low values of PLC parameter PLC CPUTIME (e.g.: 0, 1), to assign to the CPU the strictly necessary amount of power.

TYPE (P68)

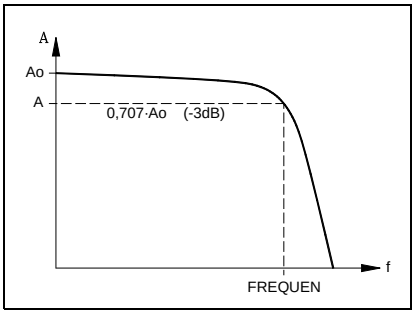
Filter type. Three types of filters may be used, namely "low passing", "anti-resonance (band rejection, notch filter)" and "FAGOR (low passing)". To obtain a good machining quality, all the axes and the spindle interpolating with each other should be defined with the same type of filter and with the same frequency. For the spindle, the filters are only applied in M19 and in rigid tapping where the spindle is interpolated with the Z axis.

Value	Meaning
0	"Low passing" filter.
1	"Anti-resonance" (notch) filter.
2	"FAGOR (low passing)" filter.

Default value: 0

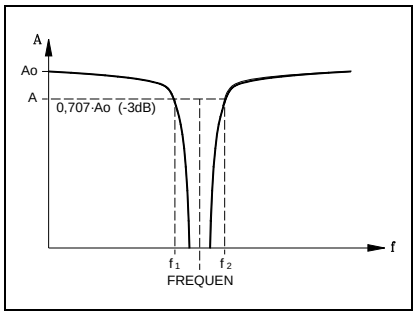
When defining anti-resonance filters, parameters NORBWIDTH and SHARE must also be set.

"Low passing" filter.



The "low passing" filter is used to limit the jerk by making the movements smoother although it has the drawback that it rounds the corners slightly.

Anti-resonance filter (notch filter).



The "anti-resonance" (notch) filter must be used when the machine has a resonance frequency to be eliminated.

Starting the CNC up while Fagor filters are active.

When starting the CNC up, if the Fagor filters are active on any of the axes and g.m.p IPOTIME (P73) =1, the CNC will display the following error message:

- Parameter TYPE=2 is incompatible with general parameter IPOTIME.

When starting the CNC up, if the Fagor filters are active on any of the axes and a.m.p SMOTIME (P58) is other than 0, the CNC will display the following error message:

- Parameter TYPE=2 is incompatible with general parameter SMOTIME.

After the start-up, if the value of that parameter is not changed, the CNC will cancel that parameter automatically.

FREQUEN (P69)

The meaning of this parameter depends on the type of filter being applied.

On "low passing" and "FAGOR" filters, it indicates the break point frequency or frequency where the amplitude drops 3 dB or it reaches 70% of the nominal amplitude.

-3dB = 20 log (A/Ao) ==> A = 0.707 Ao

For the "anti-resonance" (notch) filter, it indicates the mid frequency or frequency at which the resonance reaches its maximum value.

Possible values
Between 0 and 500.0 Hz.

Default value: 30

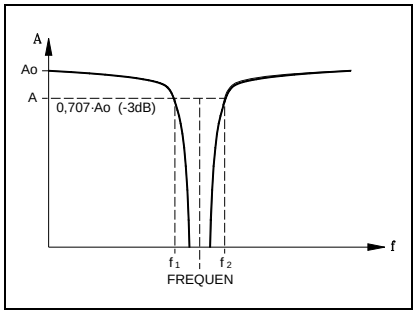
NORBWID (P70)

Standardized bandwidth.

This parameter is only taken into account for the "anti-resonance (notch)" filter type.

Possible values
between 0 and 100.0

Default value: 1



It is calculated with the following formula.

Points f1 and f2 correspond to the cutoff frequency or frequency at which its amplitude drops 3 dB or reaches 70% of the nominal amplitude.

$$NORBWID = \frac{FREQUEN}{(f_2 - f_1)}$$

SHARE (P71)

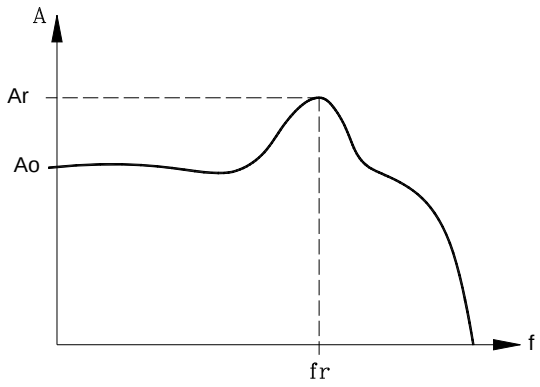
Signal percentage that passes through the filter. This value must be equivalent to the percentage overshooting of the resonance because it has to make up for it.

This parameter is only taken into account for the "anti-resonance (notch)" filter type.

Possible values
between 0 and 100

Default value: 100

Calculation example for a particular response of the machine.



SHARE=100(Ar-Ao)/Ao

INPREV1 (P72) **OUTPREV1 (P73)** **INPREV2 (P74)** **OUTPREV2 (P75)**
INPREV3 (P76) **OUTPREV3 (P77)** **INPREV4 (P78)** **OUTPREV4 (P79)**

When using a CAN servo system (only with DRIBUSLE = 0), these parameters set the gear ratios in each range.

Parameters INPREV1 trough INPUTRE4 indicate the input speed for each gear.

Parameters OUTPREV1 through OUTPREV4 indicate the output speed for each gear.

When using a CAN servo system, if parameter NPULSES and parameters INPREV and OUTPREV of all the gears are set with a ·0· value, the CNC will assume the equivalent ones of the drive.

JERKLIM (P80)

Square-sine (bell shape) ramp acceleration. This type of ramp is used to gain in smoothness. This spindle parameter becomes effective with RESET in machine parameters.

Value	Meaning
JERKLIM = 0	Linear acceleration ramp

Default value: 0

A JERKLIM value other than zero activates the square-sine ramp.

It comes in degrees/s^3, in other words, a parameter value of 20 means a jerk of 20000 degrees/s^3.

This parameter only affects the spindle acceleration in open loop (M3, M4, M5).

The parameter value so maximum acceleration (resulting from OPLACETI) is reached in half the acceleration time up to MAXGEAR1 is calculated as follows:

JERKLIM = 6000 · MAXGEAR1 / OPLACETI ^{2}

In this case, the spindle will take twice as long to reach the MAXGEAR1 speed than it would without jerk.

The JERKLIM value depends on the dynamics of the machine.

ACCTIMET (P81) PROGAINT (P82) DERGAINT (P83) FFGAINT (P84)

These parameters define the third set of gains and accelerations. They must be set like the parameters that define the first set.

First set	Second set	Third set
ACCTIME (P18)	ACCTIME2 (P47)	ACCTIMET (P81)
PROGAIN (P23)	PROGAIN2 (P48)	PROGAINT (P82)
DERGAIN (P24)	DERGAIN2 (P49)	DERGAINT (P83)
FFGAIN (P25)	FFGAIN2 (P50)	FFGAINT (P84)

To select the third set of gains and accelerations, set g.m.p. ACTGAINT (P185) correctly or activate the CNC's general logic input ACTGAINT (M5063).

THREAOVR (P85)

When beginning to machine long threads on large lathes, the part usually "bends". To prevent this, it is possible to change the spindle override in the first passes. This parameter affects canned cycles G86 and G87 while machining the thread.

Possible values	Meaning
0% - 30%	Maximum override increment.

Default value: 0 (the override cannot be varied while threading)

A value of 30 means that the override may be varied between 70% and 130%.

In spite of this, the limits set for the spindle with spindle machine parameters MINSOVR (P10) and MAXSOVR (P11), can never be exceeded. On the other hand, it will not be possible to vary the override in the last threading pass, it will be set to the value imposed in the previous threading pass.

Spindle machine parameter M19TYPE (P43) must be set to "1" in order for the override to work while threading.



In order not to damage the thread when varying the override, the feed-forward value of the axes must be close to 100% so as to work with a near-zero following error.

OPLDECTI (P86)

It indicates the duration of the spindle deceleration ramp in open loop.

Possible values
Integers between 0 and 65535 ms.

Default value: 0.

The s.m.p. OPLDECTI (P86) works on the first and second spindle.

If OPLDECTI (P86) = 0, s.m.p. OPLACETI (P45) will be applied both for acceleration and deceleration.

If s.m.p. JERKLIM (P80) other than 0 and s.m.p. OPLDECTI (P86) other than 0, parameter JERKLIM will be applied for deceleration. In this case, the deceleration dynamics will be of the square-sine type.

4.5 Drive parameters

This option is available when the CNC uses digital servo drive system; i.e. the drives are connected to the CNC via CAN.

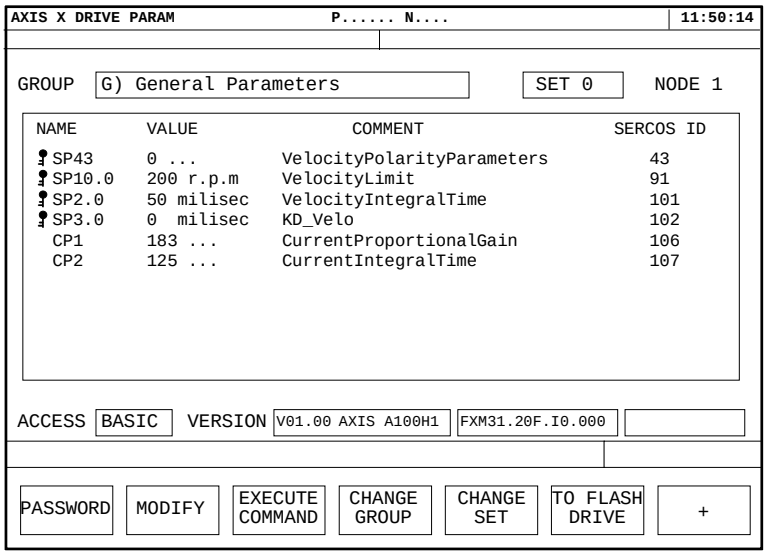
CAN interface

This option works fine for spindles with drive version SPD V7.01 or newer. It also works fine for axes with drive version ACSD V1.01 or newer.

This option displays the tables of the drive parameters that are stored in the hard disk (KeyCF) and the softkeys of the digital axes. Press one of those softkeys for editing the drive parameters for that particular axis.

When selecting the drive parameters at the CNC, it will display the ones stored in each drive and if any is modified, it is modified at the drive. The CNC does not have parameters of the drive although their copies may be stored in the hard disk (KeyCF).

When accessing the parameters of a drive, the CNC shows a screen that looks like this. Refer to the drive manual for further details on commands, variables, etc. displayed on the screen



- In the GROUP window, one must select the group of parameters or variables to be displayed. To change the group, press the [Change Group] softkey, select the new group with the [⬆] [⬇] keys and press [ENTER].
- In the SET window, one must select the set of parameters or variables to be displayed. To select another set, press the [Change set] softkey, select the new set with the [⬆] [⬇] keys and press [ENTER].
- The NODE window shows the node number identifying that drive in the CAN connection; i.e. the position of its rotary switch.

  In other words, the position of the Sercos switch. The main window shows the variables or parameters of the selected group and set indicating their Fagor name in each variable, its value, its meaning and its identifier. If the variable does not have a write permission, a key will appear before the Fagor name.

  This information is updated when selecting a new information (group or set), when modifying a variable or parameter or when pressing page/up page-down. It is not refreshed continuously.
- The ACCESS window shows the permitted access level. There are 3 access levels at the drive: basic level, OEM level and Fagor level. To change the level, press the [Password] softkey, key in the relevant code and press [ENTER].
- The VERSION window shows the software version installed at the drive, the name of the motor associated with the drive and the drive model.

Softkeys available in this mode

Password

Modifies the access level selected in the "Access" window.

In the case of CAN servo system, to access the drive parameters with an OEM access level, the password is defined in Utilities mode as OEMPSW.

Modify

To modify the variables that are not protected (those without a key icon).

After selecting the variable with the [⬆] [⬇] keys, pressing the "modify" softkey displays two windows. The first one shows the range of possible values and the second one the current value. Enter the new value and press [ENTER].

The drive assumes that value and refreshes the screen.

Execute command

Shows the list of commands that can be executed by the drive. Select one using the [⬆] [⬇] keys and press [ENTER].

Change group

Selects the group of parameters or variables to be displayed.

Change set

Selects the set number of the parameters or variables to be displayed.

To drive flash

The drive stores all its parameters in its flash memory and it then executes a soft-reset command. This command interrupts the communication, press [ENTER] to restore it.

Save

It makes a copy of the parameters of the drive's RAM memory into the CNC's hard disk (KeyCF) or to a peripheral device or PC through the serial line.

The parameters are storedwith the name of the axis they are associated with (for example, the X axis parameters). A file saved from the CNC via WinDNC may be loaded into the drive via DDSSETUP and vice versa.

Load

It copies into the drive's RAM memory the parameters saved in the CNC's hard disk (KeyCF) or in a peripheral device or in PC through the serial line.

The CNC copies the axis parameters that are being edited.

Drive Errors

It displays a window with the warnings and errors of the drive. If all of them do not fit on the screen, use [⬆] [⬇] keys.

Options

It shows a screen where one can select to display either all the parameters and variables or just the ones than can be modified.

Press the [Modify Option] softkey to change it and [ENTER] to validate it. This option is common to all the axes.

4.6 Serial line parameters

BAUDRATE (P0)

Indicates the communication speed, in baud, between the CNC and the peripherals.

It is given in baud and it is selected with the following code:

Value	Meaning	Value	Meaning
0	110 baud.	7	9.600 baud.
1	150 baud.	8	19.200 baud.
2	300 baud.	9	38.400 baud.
3	600 baud.	10	57.600 baud.
4	1.200 baud.	11	115.200 baud.
5	2.400 baud.	12	Reserved.
6	4.800 baud.		

Default value: 11 (115200 baud)

NBITSCHR (P1)

Indicates the number of data bits per transmitted character.

Value	Meaning
0	Uses the 7 least significant bits of an 8-bit character. It is used when transmitting ASCII characters (standard)
1	Uses all 8 bits of the transmitting character. Used when transmitting special characters whose codes are greater than 127.

Default value: 1

PARITY (P2)

Indicates the type of parity check used.

Value	Meaning
0	No parity.
1	Odd parity.
2	Even parity.

Default value: 0

STOPBITS (P3)

Indicates the number of stop bits at the end of each transmitted word.

Value	Meaning
0	1 STOP bit.
1	2 STOP bits.

Default value: 0

PROTOCOL (P4)

Indicates the type of communications protocol to be used.

Value	Meaning
0	Communications protocol for general device.
1	DNC communications protocol.
2	Communications protocol for Fagor floppy disk unit.

Default value: 1 (DNC)

PWONDNC (P5)

Indicates whether the DNC feature will be active on power-up or not.

Value	Meaning
NO	Not active on power-up.
YES	Active on power-up

Default value: NO

DNCDEBUG (P6)

Indicates whether the debugging feature for DNC communications is active or not.

It is advisable to use this safety feature in all DNC communications. It could be deactivated in the debugging process.

Value	Meaning
NO	Debug NOT active. Communication aborted.
YES	Debug active. Communication not aborted.

Default value: NO

ABORTCHR (P7)

Indicates the character used to abort communications with general peripheral device.

Value	Meaning
0	CAN
1	EOT

Default value: 0

EOLCHR (P8)

Indicates the character used to indicate "end of line" when communicating with general peripheral device.

Value	Meaning
0	LF
1	CR
2	LF-CR
3	CR-LF

Default value: 0

EOFCHR (P9)

Indicates the character used to indicate "end of text" (end of file) when communicating with a general peripheral device.

Value	Meaning
0	EOT.
1	ESC.
2	SUB
3	ETX

Default value: 0

XONXOFF (P10)

Indicates whether the XON-XOFF communications protocol is active or not when operating with a generic peripheral.

Value	Meaning
ON	It is active.
OFF	It is NOT active.

Default value: ON

4.7 Ethernet parameters

These parameters may be used to configure the CNC like any other node in the network, the DNC for Ethernet, the hard disk (KeyCF) and the Ethernet network. Doing that requires the Ethernet option.

Parameters	Configuration
Basic configuration: DIRIP (P24) NETMASK (P25) IPGATWAY (P26) (optional)	Configure the CNC like another node in the network. The KeyCF is accessible via FTP.
Basic configuration and also: CNHDPAS1 (P7)	Protect the access from the network to the KeyCF using a password.
Basic configuration and also: IPWDNC (P27)	Configure the DNC for Ethernet.
Basic configuration and also: IPSNFS (P28) DIRNFS (P29)	Configure the KeyCF.

If the CNC is configured like another node in the network, it may be accessed from any PC of the network knowing its IP. Only the CNC's KeyCF may be accessed; i.e. it is not possible to access programs in RAM nor read variables, tables, etc.

With the CNC configured in the network, the following is possible from any PC of the network:

- Access the part-program directory of the hard disk (KeyCF).
- Edit, modify, delete, rename, etc., the programs stored on the hard disk (KeyCF).
- Copy programs from the hard disk (KeyCF) to the PC and vice versa.

HDDIR (P0) **CNMODE (P1)**

Not being used.

CNID (P2)

CNC name when connecting it via FTP (only when allowed by the FTP client).

Possible values
It admits up to a maximum of 15 characters (without blank spaces).

Default value: FAGORCNC

CNGROUP (P3) **. . .** **CNHDDIR1 (P6)**

Not being used.

CNHDPAS1 (P7)

Password for accessing the KeyCF from the network.

Possible values
It admits up to a maximum of 15 characters (without blank spaces).

EXTNAME2 (P8) **. . .** **SERUNI2 (P21)**

Not being used.

DNCEACT (P22)

Not being used.

IPTYPE (P23)

Reserved. It must be set to "0".

DIRIP (P24)

CNC's IP address.

Possible values
Four numbers between 0 and 255 separated by dots.

Default value: 0.0.0.0 (the network is not activated)

NETMASK (P25)

Network mask.

Possible values
Four numbers between 0 and 255 separated by dots.

Default value: 0.0.0.0 (the network is not activated)

IPGATWAY(P26)

Gateway IP address.

Possible values
Four numbers between 0 and 255 separated by dots.

Default value: 0.0.0.0 (it has no gateway)

IPWDNC (P27)

WinDNC server's IP address.

The WinDNC server is the external device to connect with via DNC. This device may be a CNC, or a PC with WinDNC.

Defining it as 0.0.0.0 does not allow transferring from the CNC, but it is possible from the PC.

Possible values
Four numbers between 0 and 255 separated by dots.

Default value: 0.0.0.0

IPSNFS (P28) **DIRNFS (P29)**

Not being used.

MACID (P30)

Reserved. It must be set to "0".

ETHEINLE (P31)

Not being used.

CNC connection in an Ethernet network. Connection using Windows® 95 or 98.

Connection with a shared CNC without password

At the Web browser (e.g. IExplorer) or from the file explorer (only on Windows 98), write in the command line the CNC's IP address.

For example: `ftp://10.0.7.224`

Connection with a shared CNC with password

At the Web browser (e.g. IExplorer) or from the file explorer (only on Windows 98), write in the command line the user name, the password and the CNC's IP address. The user name is always "cnc".

For example: `ftp://cnc:password@10.0.17.62`

Assign a name to the IP address

The IP address may be assigned a name for easier identification. This operation is carried out at the PC and there are two different ways to do it.

- Editing the file "c:\windows\hosts". This file may be modified with any text editor.

  In the file, add a line containing the CNC'S IP address and the name to identify it with. For example:

  10.0.7.40 CNC_1

  10.1.6.25 MILL_MACH_01

  On the Web browser or from the file explorer (only on Windows 98), write the defined name in the command line.

  For example (CNC without password): `ftp://CNC_01`.

  For example (CNC with password): `ftp://cnc:password@MILL_MACH_01`

- Through the "Favorites" menu of the Web browser.

  In the Web explorer, write the IP address in the command line. After accessing the site, select the "Favorites" option on the menu > add to favorites and assign a name to that IP address. This way, it is possible to access the CNC by selecting the assigned name on the "Favorites" menu.

On the Iexplorer browser, it is called "Favorites". This name may vary depending on the Web browser being used.

CNC connection in an Ethernet network. Connection using Windows® 2000 or XP.

The easiest way to access the CNC's hard disk from a PC is configuring a new connection. At the file explorer, select My Network sites > Add network sites. It will show the Windows help to add network sites that permits configuring the connection step by step. Press the –OK– button to go on to the next step.

Follow the instructions shown on the screen to configure the connection; refer to the Windows® help for additional information.

Connection with a shared CNC without password

1. First, select the network site, in this case an ftp folder. Write "`ftp://`" followed by the CNC's IP address defined by machine parameter DIRIP (P24).

   For example: `ftp://10.0.17.62`

2. Define how the session is initiated, anonymously or not When the CNC is shared without password, the session is initiated anonymously.
3. Define the name to be associated with the new connection. This is the name that will appear on the PC's net directory. Just select it from the list to start the connection.

   For example: `FAGOR_CNC`

Connection with a shared CNC with password

1. First, select the network site, in this case an ftp folder. Write "`ftp://`" followed by the CNC's IP address defined by machine parameter DIRIP (P24).

   For example: `ftp://10.0.17.62`

2. Define the user name and how the session is initiated, anonymously or not. When the CNC is shared with password, the session is not initiated anonymously. The user must identify itself and it must be as "`cnc`" or "`CNC`".
3. Define the name to be associated with the new connection. This is the name that will appear on the PC's net directory. Just select it from the list to start the connection.

   For example: `FAGOR_CNC`

After the configuration is done and every time the connection is made, a window will open requesting the user name and password. As user name, select "`cnc`" or "`CNC`" and as password the one defined by machine parameter CNHDPAS1 (P7).

To make it easier, the –Save password– option may be selected in this window. This way, it will no longer request the password when connecting again and it will access the KeyCF directly.



Use the "save password" option with caution. Bear in mind that if you save the password, it will not be requested for the connection and, therefore, anybody is free to access the CNC from the PC.

4.8 PLC Parameters

WDGPRG (P0)

Indicates the Watchdog time-out period for the main PLC program.

Possible values
Integers between 0 and 65535 ms.

Default value: 0

WDGPER (P1)

Indicates the Watch-Dog time-out period for the periodic module of the PLC.

Possible values
Integers between 0 and 65535 ms.

Default value: 0

USER0 (P2) . . . USER23 (P25)

Parameters "USER0" through "USER23" do not mean anything to the CNC.

They could contain the type of information that the OEM may find necessary to customize this machine, such as: Information about the type of machine, PLC program version, etc.

This information can be accessed from the PLC program by means of the "CNCRD" high-level instruction.

Possible values
USER0(P2) - USER7(P9)
Integer numbers between 0 and 255.
USER0(P10) - USER7(P17)
Integer numbers between 0 and 65535.
USER0(P18) - USER7(P25)
Within ±99999.9999 mm or ±3937.00787 inches.

Default value: 0

CPUTIME (P26)

This parameter indicates the time the system CPU dedicates to the PLC.

Value	Meaning
0	1 ms every 8 samplings.
1	1 ms every 4 samplings.
2	1 ms every 2 samplings.
3	1 ms every sampling. With LOOPTIME = 4, 5 or 6
4	2 ms every sampling. With LOOPTIME = 4, 5 or 6
5	3 ms every sampling. With LOOPTIME 5 or 6.
6	4 ms every sampling. With LOOPTIME = 6
7	4 ms every sampling. With LOOPTIME = 6

Default value: 0

The sampling period is determined by the g.m.p. LOOPTIME (P72). Hence, for a sampling period of 4 msec. and a CPUTIME=0, the system CPU dedicates 1 millisecond every 8 samplings (thus, 32 milliseconds) to the PLC.

The Status window of the PLC statistics screen indicates the time the system CPU dedicates to the PLC. Refer to the operating manual.

Same as with sinewave feedback, number of axes and the user channel active, the PLC demands calculation time from the system CPU.

The more time the CPU dedicates to the PLC, the greater the sampling time will be, g.m.p. LOOPTIME (P72).

PLCMEM (P27)

Not being used.

SRR700 (P28) . . . **SRR739 (P67)**

Not being used.

SWR800 (P68) . . . **SWR819 (P87)**

Not being used.

OCANSPE (P88) **IOCAGEN (P89)**

Not being used.

IOCANID1 (P90) **IOCANID2 (P91)** **IOCANID3 (P92)** **IOCANID4 (P93)**

Not being used.

ICAN1 (P94) **OCAN1 (P95)** **ICAN2 (P96)** **OCAN2 (P97)**
CAN3 (P98) **OCAN3 (P99)** **ICAN4 (P100)** **OCAN4 (P101)**

Not being used.

NUICAN1 (P102) **NUOCAN1 (P103)** **NUICAN2 (P104)**
NUOCAN2 (P105) **NUICAN3 (P106)** **NUOCAN3 (P107)**
NUICAN4 (P108) **NUOCAN4 (P109)**

Not being used.

IANA5V (P130)

Not being used.

NUILO1 (P131) **NUOLO1 (P132)**

These PLC machine parameters may be used to redefine the numbering of the inputs/outputs of the local expansion module without having to modify the PLC program.

Value	Meaning
NUILO1	Numbering of the first input of the expansion module with inputs/outputs (I/O).
NUOLO1	Numbering of the first output of the expansion module with inputs/outputs (I/O).

The CPU of the 8035 CNC may have digital 16I/8O and optionally a single local digital expansion of 24I/16O.

The inputs/outputs that do not belong to the expansion module are numbered starting from I1 and O1 and cannot be set by parameters.

IMPORTANT: The numbering of both the first local input and the first local output of the expansion module must be a multiple of 8 plus 1 (1+ 8n).

If incoherent parameter settings are detected on power-up, an error message will be issued indicating it.

Inside the expansion module, the numbering of the rest of inputs/outputs will be sequential from the first one on.

The numbering for the inputs/outputs of the expansion module will be different depending on the values entered in parameters NUILO1 and NUOLO1.

To properly number the inputs/outputs of the expansion module follow the indications of the example.

Example:

Let's assume an 8035 CNC with 16i/8O and a local expansion module with 24I/16O. How should all the inputs/outputs of the expansion module be numbered?

▶ The inputs/outputs that do not belong to the expansion module (16I/8O) cannot be set by parameters; therefore, the first input is always numbered as I1 and the first output as O1.

▶ The inputs/outputs of the expansion module with the PLC parameters set to zero:

Parameters of the expansion module
NUILO1 = 0
NUOLO1 = 0

will be:

I/O of the expansion module	
I17 - I40	O9 - O24

The inputs of the expansion module are numbered sequentially after the last input that does not belong to the expansion module (I16+1 = I17). Follow the same procedure for the outputs.

▶ The inputs/outputs of the expansion module with the PLC parameters set to a value other than zero and multiple of (1+ 8n) where "n" is a natural number:

Parameters of the expansion module
NUILO1 = 65
NUOLO1 = 33

will be:

I/O of the expansion module	
I65 - I86	O33 - O48

The inputs of the expansion module are numbered sequentially after the value assigned to parameter NUILO1 (I65) chosen at will with the restriction (8n+1). Follow the same procedure for the outputs.

The values of the PLC machine parameters mentioned earlier should be multiple of 16 for best managing the inputs and outputs in time.

NUILO2 (P133) **NUOLO2 (P134)** **NUILO3 (P135)**
NUOLO3 (P136) **NUILO4 (P137)** **NUOLO4 (P138)**

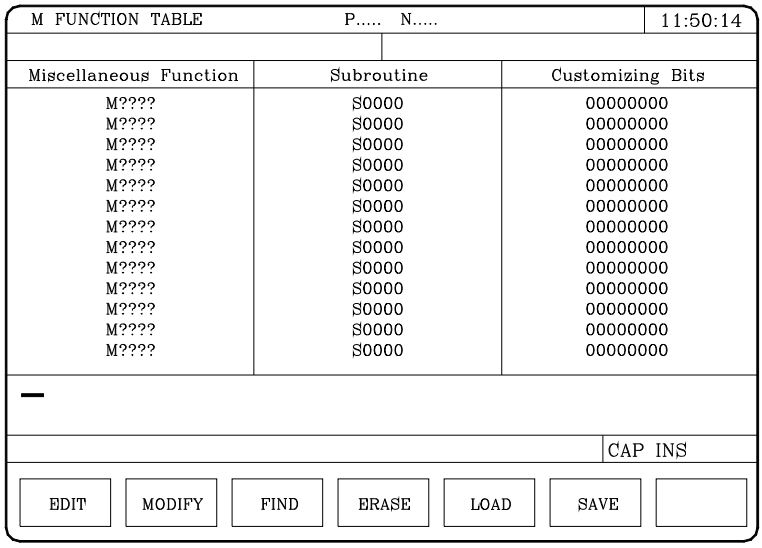
Not being used.

4.9 Tables

4.9.1 Miscellaneous (M) function table

The number of M functions in this table is determined by the g.m.p. NMISCFUN (P29), being possible to define up to 255 M functions.

It must borne in mind that functions: M00, M01, M02, M03, M04, M05, M06, M8, M9, M19, M30, M41, M42, M43 and M44, besides what is indicated in this table, have specific meanings when programming the CNC.



Each miscellaneous function will be called by its M number.

Possible values
Integer numbers between 0 and 9999. The table elements that are not defined will be displayed as M????.

A subroutine can be associated with each M function and it will be indicated by the letter S.

Possible values
Integer numbers between 0 and 9999. If 0 is assigned to this field, it means that the M function has no subroutine associated with it.

The third field consists of 8 customizing bits called bit 0 through bit 7:

*	*	*	*	*	*	*	*
7	6	5	4	3	2	1	0

bit 0

Indicates whether the CNC must wait or not for the AUXEND signal (M done) to consider it executed and go on to the next program block

Value	Meaning
0	The AUXEND signal is expected.
1	The AUXEND signal is NOT expected.

bit 1

Indicates whether the M function is executed before or after the movement block where it is programmed.

Value	Meaning
0	It is executed before the move.
1	It is executed after the move.

bit 2

Indicates whether the M function interrupts the block preparation or not.

Value	Meaning
0	It does NOT interrupt the block preparation.
1	It interrupts the block preparation.

bit 3

Indicates whether the M function is executed or not after the associated subroutine is executed.

Value	Meaning
0	It is executed after the associated subroutine.
1	ONLY the associated subroutine is executed.

bit 4

When bit "2" has been set to "1", it indicates whether block preparation is to be interrupted until the execution of the M function begins or until it ends (until the M-done signal is received).

Value	Meaning
0	It interrupts block preparation until the execution of the "M" function begins.
1	It interrupts block preparation until the "M-done" signal (AUXEND) is received.

bit 5 **bit 6** **bit 7**

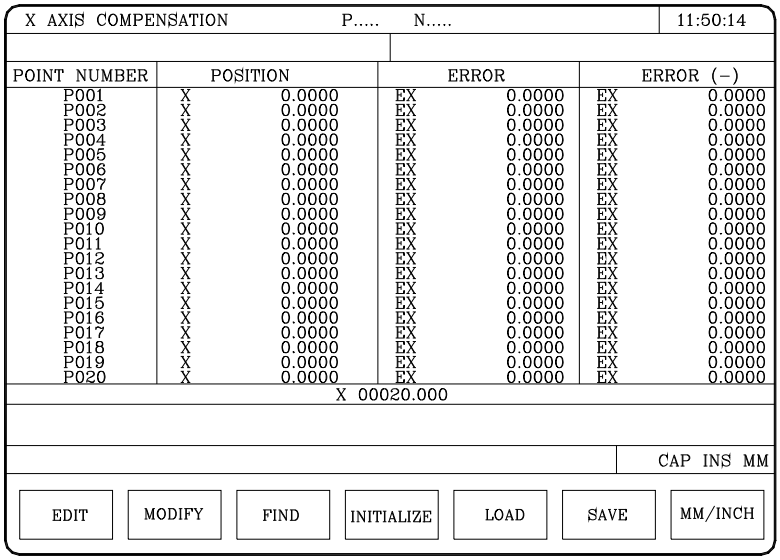
Not being used at this time.

When executing an M function which has not been defined in the M table, the programmed function will be executed at the beginning of the block and the CNC will "wait" for the "AUXEND" signal to continue the execution of the program.

4.9.2 Leadscrew error compensation table

The CNC provides a table for each one of the axes requiring leadscrew compensation. The CNC will provide a table for each one of the axes having leadscrew compensation. This type of compensation is selected by setting a.m.p. LSCRWCOM (P15).

The number of elements of the table must be set by a.m.p. NPOINTS (P16), being possible to define up to 255 points per axis. Different compensation values may be defined at each point for each moving direction.



Each parameter of the table represents a point of the profile to compensate. The following information is defined at each point:

- The position occupied by the point in the profile (position to compensate). This position is defined by its coordinate referred to machine reference zero.

Possible values
Within ±99999.9999 mm or ±3937.00787 inches.

- The error of the leadscrew at that point, when moving in the positive direction.

Possible values
Within ±99999.9999 mm or ±3937.00787 inches.

- The error of the leadscrew at that point, when moving in the negative direction.

Possible values
Within ±99999.9999 mm or ±3937.00787 inches.

For each axis position, define the amount of error to be compensated in both directions. If the amount of error in the negative direction is zero in all points, it assumes that the amount of error defined for the positive direction is valid for both directions.

Leadscrew error compensation on rotary axes

On rotary axes, although the display is limited between 0 and 360º, the internal count is accumulative. When using leadscrew error compensation, set positions 0° and 360°, first and last point of the table, with the same amount of error. This way, the CNC will apply the same compensation in all the revolutions.

Otherwise, the compensation will be limited to the indicated field.

Considerations and limitations

When defining the profile points in the table, the following requirements must be met:

- The axis points must be in sequential order starting from the most negative (least positive) point to be compensated.
- For those points outside the compensation zone, the CNC will apply the compensation value corresponding to the table point closest to them.
- The amount of error of the machine reference point may have any value.
- The error difference between two consecutive points must not be greater than the distance between them (maximum slope= 100%).

Bidirectional compensation of the leadscrew error is available from versions V7.11 (mill) and V8.11 (lathe) on.

When updated from a version that does not have bidirectional compensation, it keeps the error values in the positive direction and it sets a zero error in the negative direction for all the points.

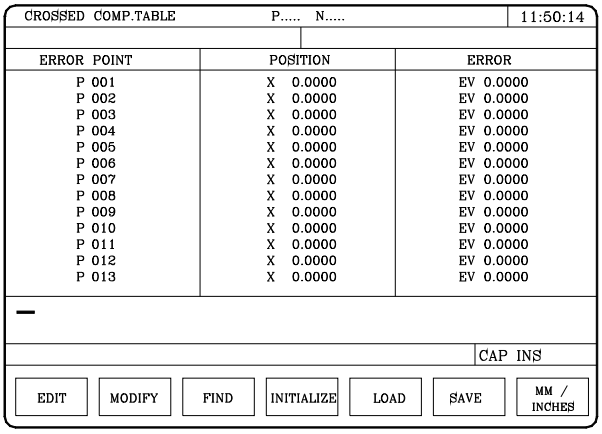
When changing to a version that does not have bidirectional compensation, it keeps the error values in the positive direction, but it loses the error values in the negative direction. Also, the amount of error for the machine reference point must be zero.

4.9.3 Cross compensation parameter table

To enable this table, set g.m.p.:

MOVAXIS (P32) COMPAXIS (P33) NPCROSS (P31)

Parameter MOVAXIS indicates the axis that moves and COMPAXIS the axis affected by the movement of the "movaxis" (to be compensated) and NPCROSS indicates the number of points in the table.



The table must indicate the amount of error to be compensated in specific positions of the moving axis.

The position is defined in home coordinates (referred to machine reference zero). Depending on g.m.p. TYPCROSS (P135), the CNC will take into account either the theoretical or real (actual) coordinates.

Possible values for the position and error fields:

Possible values
Within ±99999.9999 mm or ±3937.00787 inches.

When defining the profile points in the table, the following requirements must be met:

- The axis points must be in sequential order starting from the most negative (least positive) point to be compensated.
- For those points outside the compensation zone, the CNC will apply the compensation value corresponding to the table point closest to them.

When both leadscrew and cross compensations are applied on the same axis, the CNC will apply the sum of the two.

4.

MACHINE PARAMETERS
Tables

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

CONCEPTS

5

It is recommended to save the machine parameters as well as the PLC program and files into the hard disk (KeyCF) or in a peripheral or PC to avoid losing them.

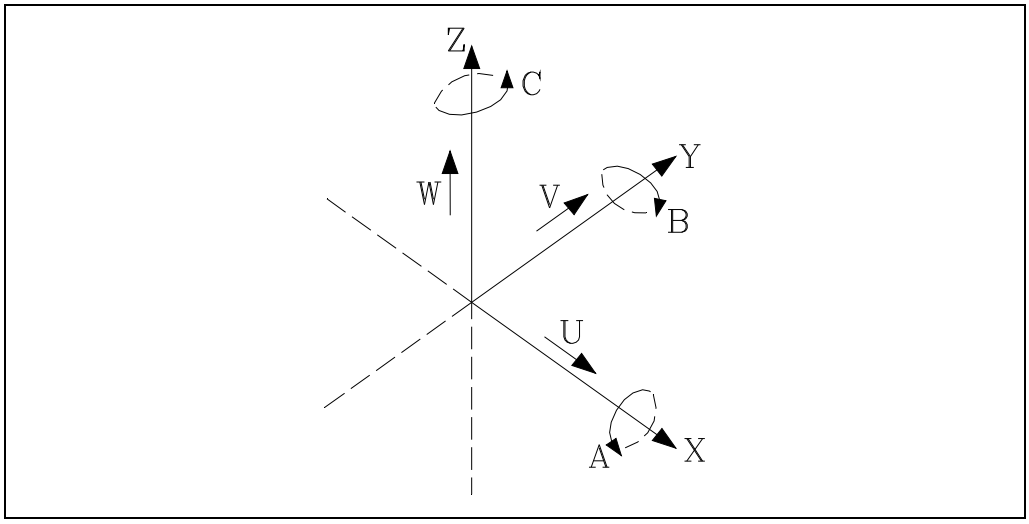
5.1 Axes and coordinate systems

Given that the purpose of the CNC is to control the movement and positioning of axes, it is necessary to determine the position of the point to be reached through its coordinates.

The CNC allows you to use absolute, relative or incremental coordinates throughout the same program.

Axis nomenclature

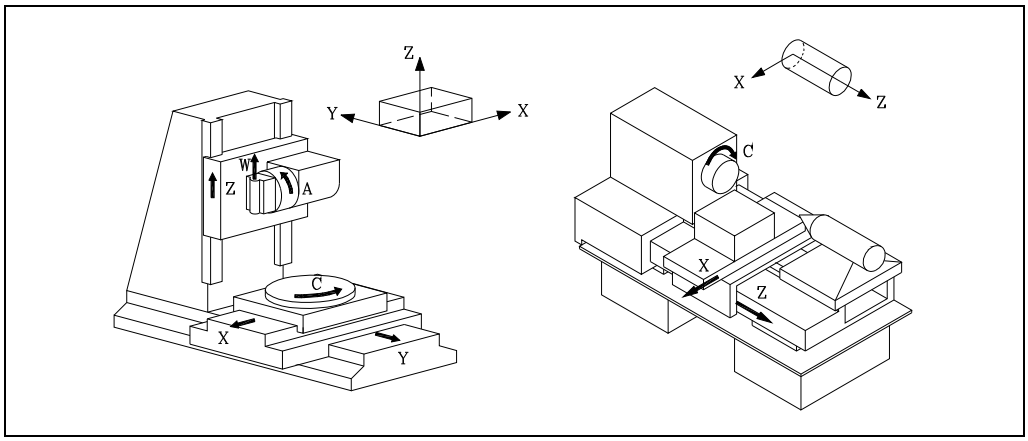
The axes are named according to DIN 66217.



Characteristics of the system of axes:

X and Y	main movements on the main work plane of the machine.
Z	parallel to the main axis of the machine, perpendicular to the main XY plane.
U, V, W	auxiliary axes parallel to X, Y, Z respectively.
A, B, C	Rotary axes on each axis X, Y, Z.

The figure below shows an example of the nomenclature of the axes on a milling-profiling machine with a tilted table.



Axis selection

Out of the 9 possible axes that there could be, the CNC lets the OEM select up to 3 of them at the mill model and 2 of them at the lathe model.

Moreover, all the axes should be suitably defined as linear, rotary, etc. through the axis machine parameters.

There is no limitation to the programming of the axes, and up to 3 axes may be interpolated at the same time.

Example of milling.

The machine has two regular linear axes: X, Y and Z, an analog spindle (S) and a handwheel.

Setting of g.m.p. AXIS1 (P0) through AXIS8 (P7).

AXIS1 (P0) = 1	X axis	associated with feedback X1 and output O1.
AXIS2 (P1) = 2	Y axis	associated with feedback X2 and output O2.
AXIS3 (P2) = 3	Z axis	associated with feedback X3 and output O3.
AXIS4(P3)=10	Spindle (S)	associated with feedback X5 (1-6) and output O5.
AXIS5 (P4) = 0		
AXIS6 (P5) = 0		
AXIS7 (P6) = 11	Handwheel	associated with feedback input X6 (1-6).
AXIS8 (P7) = 0		

The CNC activates a machine parameter table for each axis (X, Y, Z, U) and another one for the spindle (S).

a.m.p. AXISTYPE (P0) must be set as follows.

X axis	AXISTYPE (P0) = 0	Regular linear axis
Y axis	AXISTYPE (P0) = 0	Regular linear axis
Z axis	AXISTYPE (P0) = 0	Regular linear axis

s.m.p. SPDLTYPE (P0) must be set as follows:

Spindle	SPDLTYPE (P0) = 0	±10V spindle analog output.

Likewise, a.m.p DFORMAT (P1) and s.m.p. DOFORMAT (P1) must be properly set to indicate their display formats.

Example of lathe.

The machine has two regular linear axes: X and Z and an analog spindle (S).

Setting of g.m.p. AXIS1 (P0) through AXIS8 (P7).

AXIS1 (P0) = 1	X axis	associated with feedback X1 and output O1.
AXIS2 (P1) = 3	Z axis	associated with feedback X2 and output O2.
AXIS3 (P2) = 10	Spindle (S)	associated with feedback X3 and output O3.
AXIS4(P3)=0		
AXIS5 (P4) = 0		
AXIS6 (P5) = 0		
AXIS7 (P6) = 0		
AXIS8 (P7) = 0		

The CNC activates a machine parameter table for each axis (X, Z) and another one for the spindle (S).

a.m.p. AXISTYPE (P0) must be set as follows.

X axis	AXISTYPE (P0) = 0	Regular linear axis
Z axis	AXISTYPE (P0) = 0	Regular linear axis

s.m.p. SPDLTYPE (P0) must be set as follows:

Spindle	SPDLTYPE (P0) = 0	±10V spindle analog output.

Likewise, a.m.p DFORMAT (P1) and s.m.p. DOFORMAT (P1) must be properly set to indicate their display formats.

5.1.1 Rotary axes

With this CNC, it is possible to select the type of rotary axis by means of a.m.p. AXISTYPE(P0).

Normal rotary axis	AXISTYPE (P0) = 2
Positioning-only rotary axis	AXISTYPE (P0) = 3
Rotary Hirth axis	AXISTYPE (P0) = 4

By default, their position is always displayed between 0 and 360º (Rollover axis). If these limits are not to be set, modify a.m.p. ROLLOVER (P55).

ROLLOVER = YES	Rotary axis display between 0 and 360º
ROLLOVER = NO	No display limits.

Although the display is limited between 0 and 360º, the internal count is accumulative. Therefore, a.m.p. "LIMIT+(P5)" and "LIMIT-(P6)" should be set to limit the maximum number of turns in each direction.

When both parameters are set to "0", the axis can move indefinitely in either direction (rotary tables, indexers, etc.). See *"4.3 Axis parameters"* on page 116.

When using leadscrew error compensation, set positions 0° and 360°, first and last point of the table, with the same amount of error. This way, the CNC will apply the same compensation in all the revolutions. See *"5.5.7 Leadscrew error compensation"* on page 210.

Normal rotary axes

They can interpolate with linear axes. G00 and G01 movement.

- Absolute coordinate programming (G90).

  The sign indicates the turning direction and the end coordinate the position (between 0 and 359.9999).

- Incremental coordinate programming (G91).

  The sign indicates the turning direction. If the programmed movement exceeds 360º, the axis will turn more than once before positioning at the desired point.

Normal rotary axis **AXISTYPE=2**		
LIMIT+ = 8000 LIMIT- =-8000	ROLLOVER=YES	It counts between 0º and 360º. G90 The sign indicates the turning direction. G91 The sign indicates the turning direction.
	ROLLOVER=NO	It counts between 7999.9999º and -7999.9999º. G90 and G91 as linear axis.
LIMIT+ = 0 LIMIT- =0	ROLLOVER=YES	It counts between 0º and 360º. G90 The sign indicates the turning direction. G91 The sign indicates the turning direction.
	ROLLOVER=NO	There are 2 loops, one between 0º and 360º and the other between 0º and -360º It is possible to switch from one to the other. G90 and G91 as linear axis.
LIMIT+ = 350 LIMIT- =10	ROLLOVER=YES/NO	It can only move between 10º and 350º. With G90 and G91 like when LIMIT+=8000. An error message is issued if the target position is beyond the limits.

Positioning-only rotary axis

It cannot interpolate with linear axes. Movement always in G00, and they do not admit tool radius compensation (G41, G42).

- Absolute coordinate programming (G90).

  Always positive and in the shortest direction. End coordinate between 0 and 359.9999.

- Incremental coordinate programming (G91).

  The sign indicates the turning direction. If the programmed movement exceeds 360º, the axis will turn more than once before positioning at the desired point.

Positioning-only rotary axis **AXISTYPE=3**		
LIMIT+ = 8000 LIMIT- =-8000	ROLLOVER=YES	It counts between 0º and 360º. G90 does not admit negative values. Always via the shortest path. G91 The sign indicates the turning direction.
	ROLLOVER=NO	It counts between 7999.9999º and -7999.9999º. G90 and G91 as linear axis.
LIMIT+ = 0 LIMIT- =0	ROLLOVER=YES	It counts between 0º and 360º. G90 does not admit negative values. Always via the shortest path. G91 The sign indicates the turning direction.
	ROLLOVER=NO	There are 2 loops, one between 0º and 360º and the other between 0º and -360º It is possible to switch from one to the other. G90 and G91 as linear axis.
LIMIT+ = 350 LIMIT- =10	ROLLOVER=YES/NO	It can only move between 10º and 350º. With G90 and G91 like when LIMIT+=8000. An error message is issued if the target position is beyond the limits.

Rotary Hirth axis

It is a positioning-only axis which cannot take decimal coordinates. All positioning movements must be in whole degrees.

More than one Hirth axis may be used, but they can only move one at a time.

Rotary Hirth axis (whole degrees) **AXISTYPE=3**		
LIMIT+ = 8000 LIMIT- =-8000	ROLLOVER=YES	It counts between 0º and 360º. G90 does not admit negative values. G91 The sign indicates the turning direction.
	ROLLOVER=NO	It counts between 7999.9999º and -7999.9999º. G90 and G91 as linear axis.
LIMIT+ = 0 LIMIT- =0	ROLLOVER=YES	It counts between 0º and 360º. G90 does not admit negative values. Always via the shortest path. G91 The sign indicates the turning direction.
	ROLLOVER=NO	There are 2 loops, one between 0º and 360º and the other between 0º and -360º It is possible to switch from one to the other. G90 and G91 as linear axis.
LIMIT+ = 350 LIMIT- =10	ROLLOVER=YES/NO	It can only move between 10º and 350º. With G90 and G91 like when LIMIT+=8000. An error message is issued if the target position is beyond the limits.

5.1.2 Gantry axes

Gantry axes are any two axes that, due to the way the machine is built, must move together in synchronism. For example: bridge type mills.

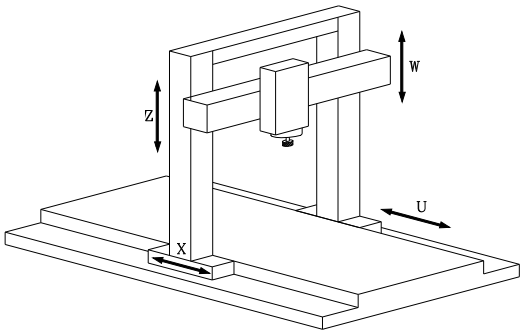
Only the movements of one of those axes must be programmed and it is called the main or master axis. The other axis is referred to as "slave axis".

In order to operate this way, it is necessary to have the a.m.p. GANTRY (P2) corresponding to both axes set as follows:

- Parameter "GANTRY" of the main axis set to "0".
- Parameter "GANTRY" of the slave axis must indicate which axis is its "master" (or main axis).

Also, a.m.p. MAXCOUPE (P45) of the slave axis must indicate the maximum allowed difference between the following errors of both axes.

Example of a bridge type milling machine with two Gantry axes (X-U, Z-W).



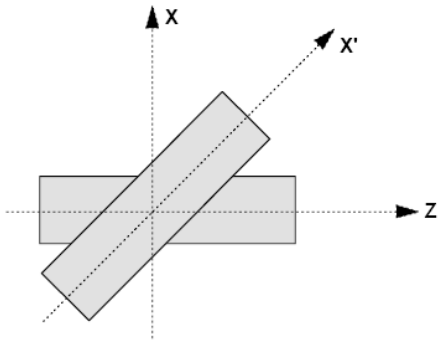
Machine parameters.

X axis	GANTRY = 0
U axis	GANTRY = 1
Z axis	GANTRY = 0
W axis	GANTRY = 3

5.1.3 Incline axis

With the angular transformation of an incline axis, it is possible to make movements along an axis that is not perpendicular to another.

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 for sturdiness reasons is not perpendicular to the Z axis.

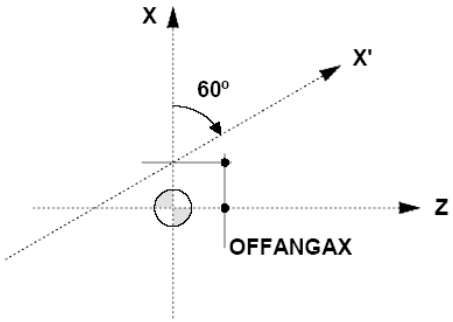


X Cartesian axis.
X' Angular axis.
Z Orthogonal axis.

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.

Configuring the incline axis

The incline axis is configured by the following general machine parameters.



ANGAXNA X
ORTAXNA Z
ANGANTR 60º

Configuring the axes

The parameter ANGAXNA configures the incline axis. The parameter ORTAXNA defines the axis perpendicular to the Cartesian axis associated with the incline axis.

The parameter OFFANGAX sets the distance between machine zero and the origin that defines the coordinate system of the incline axis. The axes defined in parameters "ANGAXNA" and "ORTAXNA"must exist and must be linear. Those axes may have Gantry axes associated with them.

Angle of the incline axis

The parameter ANGANTR defines the angle between the Cartesian axis and the angular axis it is associated with. The angle is positive when the angular axis has been rotated clockwise and negative if otherwise. If its value is 0º, there is no need to do an angular transformation.

Programming and movements

Coordinates display

If the incline axis is active, the coordinates displayed will be those of the Cartesian system; otherwise, it will display the coordinates of the real axes.

Movement programming

The inclined plane is activated from the part-program (function G46). Two kinds of movements may be executed.

- The movements are programmed in the Cartesian system and are transformed into movements on the real axes.
- Movement along the incline axis, but programming the coordinate in the Cartesian system. While this mode is active, in the motion block only the coordinate of the incline axis must be programmed.

Jog movements

PLC mark "MACHMOVE" determines how the manual movements with handwheels or with the keyboard will be carried out.

MACHMOVE = 0 Movements on the Cartesian axes.

MACHMOVE = 1 Movements on the incline axes of the machines.

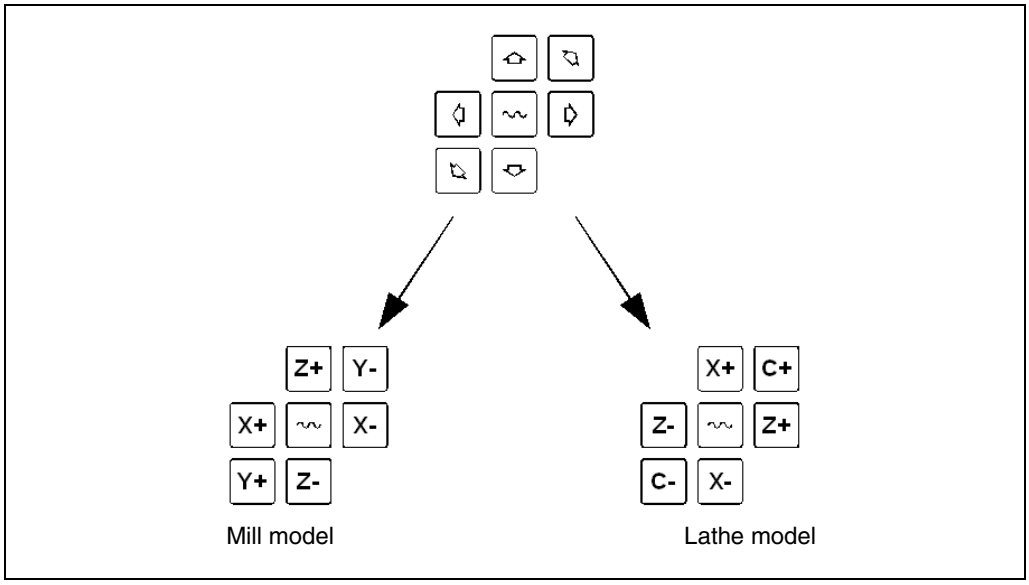
Home search

While searching home, the movements are carried out on the incline axes of the machine. The incline axis is deactivated when searching home on any axis that is part of the incline axis configuration.

5.2 Jog

5.2.1 Relationship between the axes and the JOG keys

The CNC has 3 pairs of keys to manually control the machine axes.



5.2.2 Path-jog mode

This mode may be used to act upon the jog keys of an axis to move both axes of the plane at the same time for chamfering (straight sections) and rounding (curved sections). The "path jog" mode acts when the switch is in the continuous or incremental jog positions.

The CNC assumes as "Path jog" the keys associated with the X axis.

Feature setting

This feature must be managed from the PLC.

To activate or cancle the "Path jog" work mode, act upon the logic CNC input "MASTRHND" M5054.

M5054 = 0 "Path JOG" function off.

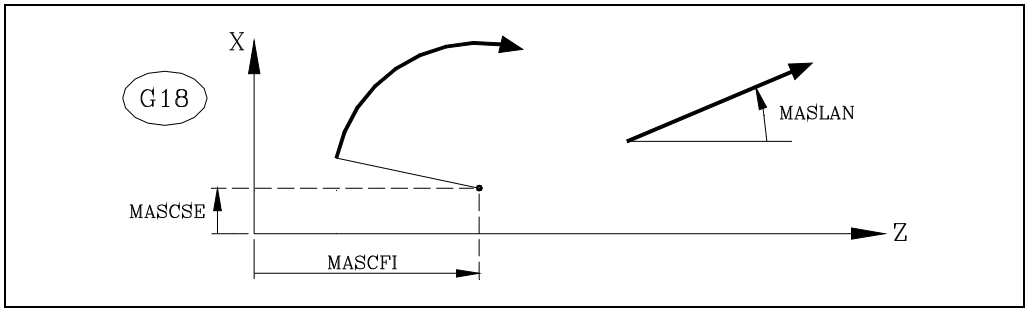
M5054 = 1 "Path JOG" function on.

To indicate the type of movement, use CNC logic input "HNLINARC" M5053.

M5053 = 0 Linear path.

M5053 = 1 Arc path.

For a linear path, indicate the path angle in the MASLAN variable (value in degrees between the linear path and the first axis of the plane). For an arc, indicate the arc center coordinates in the MASCFI, MASCSE variables (for the first and second axes of the main plane).



Variables MASLAN, MASCFI and MASCSE may be read and written from the CNC, DNC and PLC.

Operation of the "path jog" feature

The "path jog" mode is only available with the X axis keys. When pressing one of the keys associated with the X axis, the CNC behaves as follows:

Switch position	Path jog	Type of movement
Continuous jog	OFF	Only the axis and in the indicated direction
	ON	Both axes in the indicated direction and along the indicated path
Incremental jog	OFF	Only the axis, the selected distance and in the indicated direction
	ON	Both axes, the selected distance and in the indicated direction, but along the indicated path
Handwheel		It ignores the keys.

The rest of the jog keys always work in the same way, whether "path jog" is on or off. The rest of the keys move only the axis and in the indicated direction.

Considerations about the jog movements

This mode assumes as axis feedrate the one selected in jog mode and it will also be affected by the feedrate override switch. If F0 is selected, it assumes the one indicated by machine parameter "JOGFEED (P43)". This mode ignores the rapid jog key.

"PATH JOG" movements respect the travel limits and the work zones.

"Path jog" movements may be aborted in the following ways:

- By pressing the [STOP] key.
- By turning the JOG switch to one of the handwheel positions.
- By setting general logic input "MASTRHND (M5054)" = 0.
- By setting general logic input "\STOP (M5001)" = 0.

5.3 Movement with an electronic handwheel

Depending on their configuration, the available handwheels are:

- General handwheel.

  It can be used to jog any axis one by one.

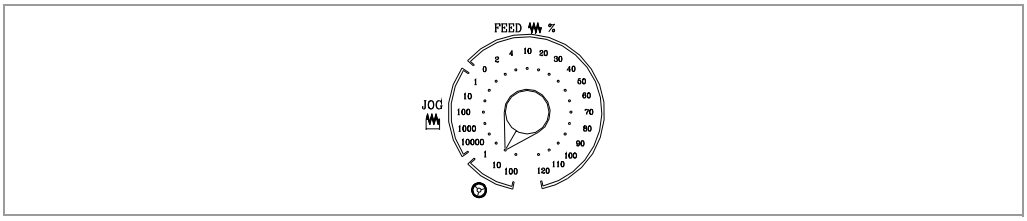
  Select the axis and turn the handwheel to move it.

- Individual handwheel.

  It replaces the mechanical handwheels.

  Up to 2 handwheels can be used (one per axis).

  It only moves the axis it is associated with.



To move any of them, turn the switch to any of the handwheel positions. Positions 1, 10 and 100 indicate the multiplying factor being applied besides the internal x4 to the feedback pulses supplied by the electronic handwheel.

For example, if the manufacturer has set a distance of 0.100 mm or 0.0100 inches per handwheel turn for switch position 1:

Switch position	Distance per turn
1	0.100 mm or 0.0100 inches
10	1.000 mm or 0.1000 inches
100	10.000 mm or 1.0000 inches

There are 3 operating modes with handwheels:

Standard handwheel:

- With the general handwheel, select the axis to be moved and turn the handwheel.
- With individual handwheels, turn the handwheel associated with the axis to be moved.

Path handwheel

- For chamfering and rounding corners.
- 2 axes are moved along a selected path (chamfer or rounding) by moving a single handwheel.
- This feature must be managed from the PLC.
- The general handwheel is assumed as the "path handwheel" or the individual handwheel associated with the X axis (Mill) or Z (lathe).

Feed handwheel mode

- To control the feedrate of the machine.
- This feature must be managed from the PLC.

Depending on the turning speed of the handwheel and the position of the selector switch, when requesting a movement at a faster feedrate than the maximum allowed.

- *With individual handwheels, the movement stops when stopping the handwheel. It does not move the indicated distance.*
- *With general handwheels, g.m.p. HDIFFBAC (P129) indicates whether the movement is stopped or it moves the indicated distance.*

5.3.1 Standard handwheel

General handwheel.

1. Select the axis to be jogged.

   Press one of the JOG keys of the axis to be jogged. The selected axis will be highlighted.

   When using a Fagor handwheel with an axis selector button, the axis may be selected as follows:

   - Push the button on the back of the handwheel. The CNC select the first axis and it highlights it.
   - When pressing the button again, the CNC selects the next axis and so on in a rotary fashion.
   - To deselect the axis, hold the button pressed for more than 2 seconds.

2. Jog the axis.

   Once the axis has been selected, it will move as the handwheel is being turned and in the direction indicated by it.

Individual handwheels.

Each axis will move as the corresponding handwheel is being turned according to the switch position and in the direction indicated by it.

Simultaneous handwheels.

The machine may have a general handwheel and up to 3 individual handwheels associated with each axis. The individual handwheels have priority over the general handwheel. So, if an individual handwheel is moving, the general handwheel will be ignored.

5.3.2 Path handwheel

With this feature, it is possible to jog two axes at the same time along a linear path (chamfer) or circular path (rounding) with a single handwheel.

The CNC assumes as the "path handwheel" the general handwheel or, when this one is missing, the one associated with the X axis (mill) or Z axis (lathe).

Feature setting

This feature must be managed from the PLC.

To activate or cancle the "Path jog" work mode, act upon the logic CNC input "MASTRHND" M5054.

M5054 = 0 "Path JOG" function off.

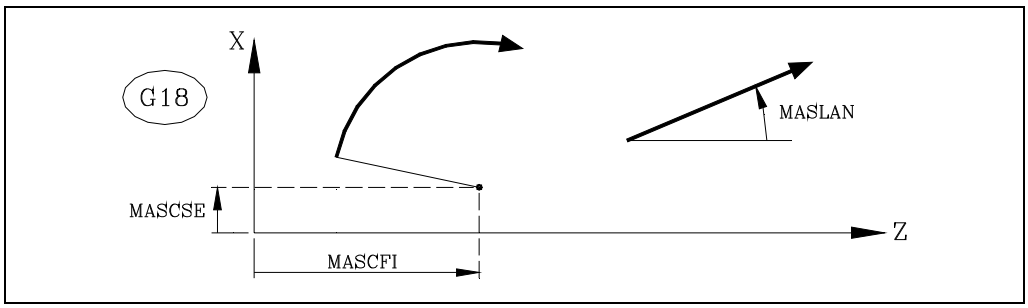
M5054 = 1 "Path JOG" function on.

To indicate the type of movement, use CNC logic input "HNLINARC" M5053.

M5053 = 0 Linear path.

M5053 = 1 Arc path.

For a linear path, indicate the path angle in the MASLAN variable (value in degrees between the linear path and the first axis of the plane). For an arc, indicate the arc center coordinates in the MASCFI, MASCSE variables (for the first and second axes of the main plane).



Variables MASLAN, MASCFI and MASCSE may be read and written from the CNC, DNC and PLC.

The next example uses the [O2] key to activate and deactivate the "path handwheel" mode and the [O3] key to indicate the type of movement.

```
DFU B29 R561 = CPL M5054
```

Activate or cancel the "path handwheel" mode.

```
DFU B31 R561 = CPL M5053
```

Select the type of movement; straight section or arc section.

Simultaneous handwheels

When selecting the path handwheel mode, the CNC behaves as follows:

- If there is a general handwheel, it will be the one working in path handwheel mode. The individual handwheels, if any, will remain associated with the corresponding axes.
- If there is no general handwheel, one of the individual handwheels starts working in "path handwheel" mode. The one associated witht the X axis if mill model or the one associated with the Z if lathe model

5.3.3 Feed handwheel mode

Usually, when making a part for the first time, the machine feedrate is controlled by means of the feedrate override switch.

From this version on, it is also possible to use the machine handwheels to control that feedrate. This way, the machining feedrate will depend on how fast the handwheel is turned. To do this, proceed as follows:

- Inhibit all the feedrate override switch positions from the PLC.
- Detect how far the handwheel is turned (reading of pulses received).
- Set the corresponding feedrate override from the PLC depending on the pulses received from the handwheel.

The following CNC variables return the number of pulses the handwheel has turned.

HANPF	shows the number of pulses of the 1st handwheel.
HANPS	shows the number of pulses of the 2nd handwheel.
HANPT	shows the number of pulses of the 3rd handwheel.
HANPFO	shows the number of pulses of the 4th handwheel.

PLC programming example.

The machine has a button to activate and deactivate this feature (feed handwheel) and the feedrate control is carried out with the second handwheel.

```
CY1
R101=0
```

Resets the register containing the previous handwheel reading.

```
END

PRG
DFU I71 = CPL M1000
```

Every time the button is pressed, mark M1000 is inverted.

```
M1000 = MSG1
```

If the feature is active, a message is displayed.

```
NOT M1000
= AND KEYDIS4 $FF800000 KEYDIS4
= JMP L101
```

If this feature is not active, it enables all the positions of the feedrate override switch and resumes program execution

```
DFU M2009
= CNCRD(HANPS,R100,M1)
= SBS R101 R100 R102
= MOV R100 R101
= MLS R102 3 R103
= OR KEYDIS4 $7FFFFF KEYDIS4
```

If this feature is activated and an upflank occurs at the clock mark M2009, it reads in R100 the handwheel pulses (HANPS), calculates in R102 the number of pulses received since the last reading, updates R101 for the next reading, calculates in R103 the value of the right feedrate and inhibits all the positions of the feedrate override switch (KEYDIS4).

```
CPS R103 LT 0 = SBS 0 R103 R103
CPS R103 GT 120 = MOV 120 R103
```

It adjusts the value of R103 (feedrate %). It ignores the handwheel turning direction (sign) and limits the value to 120%

```
DFU M2009
= CNCWR(R103,PLCFRO,M1)
```

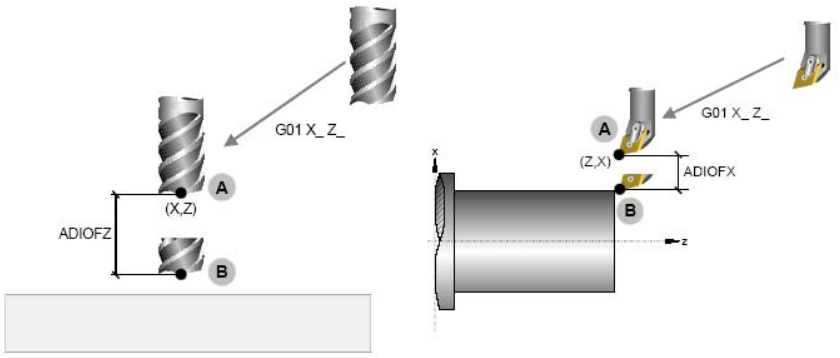
With the up flank at the clock mark M2009, set the calculated feedrate override (PLCFRO=R103)

```
L101
END
```

5.3.4 "Additive handwheel" mode

With manual intervention or additive handwheel, it is possible to move the axes manually while a program is being executed. To do this, once this option is activated, use the handwheel to make a movement to be added to the one resulting from the automatic execution. This movement will be applied as if it were another zero offset.

The general handwheel will be used as the additive handwheel. If there is no general handwheel, use the handwheel associated with the axis.



(A) Position of the tool while in execution.
(B) Position of the tool after a manual intervention.

The intervention with an additive handwheel is only possible in execution mode, even while the program is interrupted. However, it is not allowed in the tool inspection mode.

The additive handwheel may be enabled for a coordinate transformation G46 (inclined axis) where the handwheel movements are applied to the machining operation even if they are not shown on the graphics screen.

The offset caused by the additive handwheel stays active after disabling the handwheel and it is reset to zero after a home search. The offset stays active or is reset after an M02 or M30 and after an emergency or a reset depending on the setting of g.m.p. ADIMPG (P176).

Considerations

- The movement with the additive handwheel on the master axis is also applied to the slave axis when using axes that are gantry, slaved or synchronized by PLC.
- When testing the software limits during block preparation, it checks the theoretical coordinate ignoring the additional movement of the additive handwheel.
- The mirror image by PLC is not applied to the additive handwheel movement.

Configuring the additive handwheel

When enabling the additive handwheel, the following must be borne in mind.

- If the DWELL parameter of an axis has been set and it is not previously in motion, it activates the ENABLE mark of the axis and waits a time period indicated in DWELL to check whether its SERVOON has been activated or not.
- The acceleration applied to the additive handwheel movement is that of parameter. ACCTIME of the axis.
- On Gantry axes, the movement of the master axis using an additive handwheel is also applied to the slave axis.
- The mirror image by PLC is not applied to the additive handwheel movement.
- When testing the software limits during block preparation, it checks the theoretical coordinate ignoring the additional movement of the additive handwheel.

The additive handwheel is configured by machine parameters and is activated and deactivated by PLC.

Activating and deactivating the additive handwheel

The additive handwheel is activated and deactivated with the mark MANINT(X-C). The PLC sets one of these signals high to activate the additive handwheel on each axis. Only one additive handwheel may be enabled at a time. If there are more than one mark active, only the first one will be attended to.

Configuring the additive handwheel

The parameter ADIMPG enables the additive handwheel and makes it possible to configure its operation.

Handwheel resolution and maximum feedrate.

The resolution of the additive handwheel depends on the setting of parameter ADIMPG (P176). There are two options to set the resolution:

- The resolution of the handwheel is set by parameter ADIMPRES (P177) of the axis.
- The handwheel resolution is set with the switch of the operator panel. If the switch is not in the handwheel position, it assumes a x1 factor.

Maximum feedrate allowed, due to the additive handwheel, is limited by parameter ADIFEED (P84).

Coordinates display

Parameter DIPLCOF determines whether the CNC takes into consideration or not the additive zero offset when displaying the coordinates of the axes on the screen and when accessing the POS(X-C) and TPOS(X-C) variables.

5.4 feedback system

The various feedback inputs available at the CNC admit sinusoidal and squarewave differential signals from feedback systems. The following axis machine parameters indicate the type of feedback system and the resolution used for each axis.

- When using linear feedback devices.

PITCH (P7)	Pitch of the linear encoder being used.
NPULSES (P8)	= 0
DIFFBACK (P9)	Indicates whether the feedback device uses differential signals (double ended) or not.
SINMAGNI (P10)	Feedback multiplying factor applied by the CNC.
FBACKAL (P11)	Feedback alarm (only with differential signals).

- When using rotary encoders.

PITCH (P7)	On rotary axes, it sets the degrees per turn of the encoder. On linear axes, it sets the leadscrew pitch.
NPULSES (P8)	Number of pulses (lines) per encoder turn.
DIFFBACK (P9)	Indicates whether the feedback device uses differential signals (double ended) or not.
SINMAGNI (P10)	Feedback multiplying factor applied by the CNC.
FBACKAL (P11)	Feedback alarm (only with differential signals).

Next, the feedback counting speed (frequency) limitation is described as well as how to set these machine parameters for the axes.

5.4.1 Counting speed limitation

Sinusoidal signals

The maximum counting speed (frequency) for sinusoidal feedback is 250 kHz.

The maximum feedrate for each axis on linear systems will depend upon the selected resolution and the signal pitch (distance per pulse) in use, whereas on rotary encoders it will depend on the number of pulses per revolution.

Example 1:

When using a Fagor linear encoder, the signal pitch is 20 µm. Therefore, with a counting resolution of 1 µm, the maximum feedrate will be:

20 µm/pulse x 250.000 pulses/sec = 300 m/min.

When using Fagor linear encoders, the maximum feedrate is limited by their own characteristics to 60 m/min.

Example 2:

Using an indexer with a sinusoidal Fagor encoder of 3600 lines per turn, for a feedback resolution of 1 µm, the maximum axis feedrate will be:

(360 degrees/turn / 3600 pulses/turn) x 250.000 pulses/s. = 25.000 degrees/s.= 1.500.000 degrees /min

Since Fagor sinusoidal encoders admit a frequency of up to 200 kHz, the maximum feedrate will be:

(360 degrees/turn / 3600 pulses/turn) x 200.000 pulses/s. =

= 20,000 degrees/sec.= 1,200,000 degrees/min.

Squarewave signals

The maximum frequency (speed) for differential squarewave feedback is 425 kHz with a separation of 450 ns between A and B flanks which is equivalent to 90º ±20º.

The maximum feedrate for each axis will depend upon the selected resolution and the signal pitch (distance per pulse) in use.

When using Fagor linear encoders, the maximum feedrate is limited by their own characteristics to 60 m/min.

When using FAGOR rotary encoders, their intrinsic output frequency limit is (200Kz).

5.4.2 Resolution

The CNC provides a number of machine parameters for the axes and for the spindle in order to establish the counting resolution of each one of the axes and the spindle.

PITCH (P7)

Defines the pitch of the ballscrew or the linear encoder being used. When using a Fagor linear encoder, this parameter must be set with the pitch value of the feedback signals (20 µm or 100 µm).

When using a rotary axis, indicate the number of degrees per encoder revolution. E.g. if the encoder is mounted on the motor and the axis has a gear ratio of 1/10, parameter PITCH must be set with the value of 360/10 = 36.

NPULSES (P8)

Indicates the number or pulses/rev provided by the rotary encoder. When using a linear encoder, it must be set to ·0·. When using a gear reduction on the axis, the whole assembly must be taken into account when defining the number of pulses per turn.

SINMAGNI (P10)

Indicates the multiplying factor (x1, x4, x20, etc.) that the CNC must apply only to sinusoidal feedback signal.

For square feedback signals, this parameter must be set to 0 and the CNC will always apply a multiplying factor of x4.

The counting resolution for each axis will be defined by means of the combination of these parameters as shown in the following table:

	PITCH	NPULSES	SINMAGNI
Square signal encoder	Leadscrew pitch	Nr of pulses	0
Sinusoidal signal encoder	Leadscrew pitch	Nr of pulses	multiplying factor
Square signal linear encoder	linear encoder pitch	0	0
Sinusoidal signal linear encoder	linear encoder pitch	0	multiplying factor

Example 1:
Resolution in "mm" with squarewave encoder.

We would like to obtain a 2µm resolution by using a squarewave encoder mounted on 5 mm pitch leadscrew.

Since the CNC applies a x4 multiplying factor to squarewave signals, we would require an encoder that provides the following number of pulses (lines) per turn.

Nr of pulses = ballscrew pitch / (multiplying factor x Resolution)

Nr pulses = 5000 µm / (4 x 2 µm) = 625 pulses/turn

Therefore:

INCHES = 0 PITCH=5.0000 NPULSES = 625 SINMAGNI=0

Allthough the CNC accepts a maximum squarewave frequency of 400 kHz, when using Fagor squarewave rotary encoders their output frequency is limited to 200 kHz; thus, the maximum possible feedrate (F) will be:

Max. feedrate = (200.000 pulses/sec. / 625 pulses/turn) x 0.2 inch/turn

Max. feedrate = 1600 mm/s = 96 m/min.

Example 2:
Resolution in "mm" with sinusoidal signal encoder

We would like to obtain a 2µm resolution by using a 250-line sinusoidal encoder mounted on 5 mm-pitch ballscrew.

We must calculate the multiplying factor "SINMAGNI" to be applied by the CNC to the pulses provided by the encoder in order to obtain the desired resolution.

SINMAGNI = ballscrew pitch / (Nr pulses x Resolution)

SINMAGNI = 5000 µm / (250 x 2 µm) = 10

Therefore:

INCHES = 0 PITCH=5.0000 NPULSES = 250 SINMAGNI=10

Although the CNC accepts a maximum squarewave frequency of 250 kHz, when using Fagor sinusoidal rotary encoders, the output frequency is limited to 200 kHz; thus, the maximum possible feedrate (F) will be:

Max. feedrate = (200.000 pulses/sec. / 250 pulses/turn) x 0.2 inch/turn

Max. feedrate = 4.000 mm/s = 240 m/min.

Example 3:
Resolution in "mm" with squarewave linear encoder

Since the CNC applies a x4 multiplying factor to squarewave signals, we must select a linear encoder whose grading pitch is 4 times the desired resolution.

FAGOR linear encoders use a grading pitch of either 20 µm or 100 µm. Therefore, the resolution that can be obtained with them are: 5 µm (20/4) or 25 µm (100/4).

Therefore:

INCHES = 0 PITCH=0.0200 NPULSES = 0 SINMAGNI=0
PITCH=0.1000

The CNC's maximum squarewave feedback input frequency is 400 kHz which means that the maximum feedrate obtainable with a 20 µm pitch linear encoder is:

Max. Feed = 20 µm/pulse x 400.000 pulses/sec.

Max. feedrate = 8000 mm/s = 480 m/min.

When using Fagor linear encoders, the maximum feedrate is limited by their own characteristics to 60 m/min.

Example 4:
Resolution in "mm" with sinusoidal signal linear encoder

We have a sinusoidal linear encoder with a 20 µm pitch and we would like to obtain 1 µm resolution.

We must calculate the multiplying factor "SINMAGNI" to be applied by the CNC to the pulses provided by the linear encoder in order to obtain the desired resolution.

SINMAGNI = linear encoder pitch / resolution = 20 µm / 1 µm = 20

Therefore:

INCHES = 0 PITCH=0.0200 NPULSES = 0 SINMAGNI=20

The CNC's maximum sinusoidal feedback input frequency is 250 kHz which means that the maximum feedrate for this axis will be:

Max. Feed = 20 µm/pulse x 250.000 pulses/sec.

Max. feedrate = 5.000 mm/s = 300 m/min.

When using Fagor linear encoders, the maximum feedrate is limited by their own characteristics to 60 m/min.

Example 5:
Resolution in "inches" with squarewave encoder

Calculate the necessary squarewave encoder line count and parameter settings to obtain a 0.0001 inch counting resolution on a 4 pitch ballscrew (4 turns/inch = 0.25 inch/rev.).

Since the CNC applies a x4 multiplying factor to squarewave signals, we would require an encoder that provides the following number of pulses (lines) per turn.

Nr of pulses = ballscrew pitch / (multiplying factor x Resolution)

Nr pulses = 0.25 / (4 x 0.0001) = 625 pulses/turn

Therefore:

INCHES = 1 PITCH=0.25000 NPULSES = 625 SINMAGNI=0

Allthough the CNC accepts a maximum squarewave frequency of 400 kHz, when using Fagor squarewave rotary encoders their output frequency is limited to 200 kHz; thus, the maximum possible feedrate (F) will be:

Max. feedrate = (200.000 pulses/sec. / 625 pulses/turn) x 0.255 inch/turn

Max. feedrate = 80 inch/sec. = 4800 inch/min.

Example 6:
Resolution in "inches" with sinusoidal encoder

We would like to obtain a 0.0001 inch resolution by using a 250-line sinusoidal encoder mounted on a leadscrew with a 5 turns/inch pitch.

We must calculate the multiplying factor "SINMAGNI" to be applied by the CNC to the pulses provided by the encoder in order to obtain the desired resolution.

SINMAGNI = ballscrew pitch / (Nr pulses x Resolution)

SINMAGNI = 0.2 inch/turn / (250 x 0.0001) = 8

Therefore:

INCHES = 1 PITCH=0.20000 NPULSES = 250 SINMAGNI=8

Although the CNC accepts a maximum squarewave frequency of 250 kHz, when using Fagor sinusoidal rotary encoders, the output frequency is limited to 200 kHz; thus, the maximum possible feedrate (F) will be:

Max. feedrate = (200.000 pulses/sec. / 250 pulses/turn) x 0.2 inch/turn

Max. feedrate = 160 inch/sec. = 9.600 inch/min.

Example 7:
Resolution in "degrees" with squarewave encoder

We would like to obtain a 0.0005º resolution by using a squarewave encoder mounted on a x10 reduction gear.

Since the CNC applies a x4 multiplying factor to squarewave signals, we would require an encoder that provides the following number of pulses (lines) per turn.

Nr of pulses = º/turn / (multiplying factor x gear ratio x Resolution)

Nr of pulses = 360 / (4 x 10 x 0.0005) = 18,000 pulses/turn

Therefore:

INCHES = 0 PITCH=36.0000 NPULSES = 18000 SINMAGNI=0

Allthough the CNC accepts a maximum squarewave frequency of 400 kHz, when using Fagor squarewave rotary encoders their output frequency is limited to 200 kHz; thus, the maximum possible feedrate (F) will be:

Max. feedrate = (200,000 pulses/sec) / (18,000 pulses/turn)

Max. feedrate =11.111 turns/sec = 666.666 rpm

Example 8:
Resolution in "degrees" with sinusoidal encoder

We would like to obtain a 0.001º resolution by using a 3600 pulse/rev sinusoidal encoder.

We must calculate the multiplying factor "SINMAGNI" to be applied by the CNC to the pulses provided by the encoder in order to obtain the desired resolution.

SINMAGNI = degrees per turn / (Nr. of pulses x Resolution)

SINMAGNI = 360 / (3600 x 0.001) = 100

Therefore:

INCHES = 0 PITCH=360.0000 NPULSES = 3600 SINMAGNI=100

Although the CNC accepts a maximum squarewave frequency of 250 kHz, when using Fagor sinusoidal rotary encoders, the output frequency is limited to 200 kHz; thus, the maximum possible feedrate (F) will be:

Max. feedrate = (200,000 pulses/sec) / (3.600 pulses/turn)

Max. feedrate =55,5556 turns/sec = 3333,33 rpm

5.5 Axis adjustment

In order to be able to set the axes, their corresponding feedback devices must be previously connected to the CNC.

Before making this adjustment, position the axes near the middle of their travel and place the hard stops (monitored by the electrical cabinet) near these mid-travel points in order to prevent any possible damage to the machine.

The axis adjustment is carried out in two steps. First, the servo drive loop is adjusted and, then, the CNC loop.

Drive loop setting

1. Verify that the power output of the drives is OFF. Set all a.m.p. FBALTIME (P12) to a value other than "0"; for example, FBALTIME=1000.
2. Turn the CNC OFF.
3. Turn the drive power output ON.
4. Turn the CNC ON.
5. If the axis runs away, the CNC will issue the following error message for this axis. Turn the CNC off and swap the tacho wires at the drive.
6. Repeat steps 4 and 5 until the CNC stops issuing errors.

Loop setting of the CNC.

The axes are set one at a time.

1. Select the JOG operating mode at the CNC
2. Jog the axis to be adjusted.

   If the axis runs away, the CNC issues the corresponding following error message. In this case, the a.m.p. LOOPCHG (P26) must be changed.

   If the axis does not run away, but it does not move in the desired direction, Change both a.m.p. AXISCHG (P13) and LOOPCHG (P26).

5.5.1 Drive setting

Offset (drift) adjustment

This adjustment is made on one axis at a time:

- Select the JOG mode at the CNC and press the softkey sequence: [Display] [Following error]. The CNC shows the current following Error (axis lag) of the axes.
- Adjust the offset by turning the offset potentiometer at the drive (NOT AT THE CNC) until a "0" following error is obtained.

Maximum feedrate adjustment

The drives should be adjusted so they provide maximum axis feedrate when receiving a velocity command of 9.5 V.

Set each a.m.p. MAXVOLT (P37) = 9500 so the CNC outputs a maximum velocity command of 9.5 V.

The maximum axis feedrate, a.m.p. MAXFEED (P42), depends on the motor rpm as well as on the gear ratio and type of ballscrew being used.

Example for the X axis:

The maximum motor rpm is 3,000 and the ballscrew pitch is 5mm/rev. Thus:

Maximum rapid traverse feedrate (G00) = ballscrew rpm. x ballscrew pitch

"MAXFEED" (P42) = 3,000 rpm. x 5 mm/rev. = 15000 mm/min.

In order to adjust the drive, a.m.p. G00FEED (P38) should be set to the same value as a.m.p. MAXFEED (P42).

Also, a small CNC program must be executed that will move the axis back and forth a short distance in order to verify that the amount of following error in both directions is the same. One such program could be:

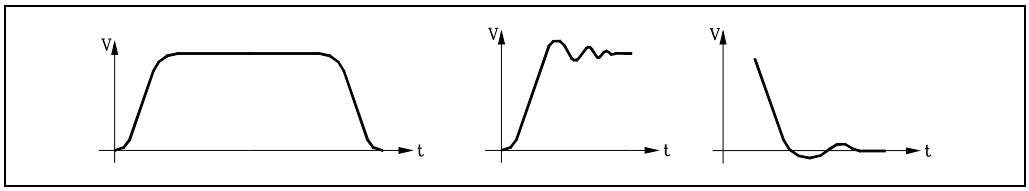
```
N10     G00 G90 X200
N20     X -200
        (RPT N10, N20)
```

While the axis is moving back and forth, measure the analog command provided by the CNC to the drive and adjust the feed potentiometer at the drive (NOT AT THE CNC) until reaching 9.5 V.

5.5.2 Gain setting

The various types of gains must be adjusted for each axis in order to optimize the system's performance for the programmed movements.

An oscilloscope is highly recommended to make this critical adjustment by monitoring the tacho signals. The illustration below shows the optimum shape for this signal (on the left) and the instabilities to be avoided during start-up and braking.



There are three gain types for each axis. They are adjusted by means of axis machine parameters and following the sequence indicated next.

Proportional gain

It defines the velocity command corresponding to a feedrate resulting in 1 mm of following error.

It is defined with a.m.p. PROGAIN (P23).

Feed-forward gain

It sets the percentage of velocity command due to the programmed feedrate.

To use it, acc/dec must be active ACCTIME (P18).

It is defined with a.m.p. FFGAIN (P25).

Derivative gain or AC-forward gain

The "derivative gain" sets the percentage of velocity command applied depending on the fluctuations of following error.

The "AC-forward gain" sets the percentage of velocity command proportional to the feedrate increments (acceleration and deceleration stages).

To use it, acc/dec must be active ACCTIME (P18).

It is defined with a.m.p. DERGAIN (P24) and ACFGAIN (P46).

If ACFGAIN = No	it applies derivative gain
If ACFGAIN = Yes	it applies AC-forward gain.

5.5.3 Proportional gain setting

In a "pure" proportional position loop, the velocity command of the CNC to control an axis is, at all times, proportional to the following error (axis lag) which is the difference between its theoretical and actual (real) position.

Velocity command = Proportional Gain x Following Error

a.m.p. PROGRAIN (P23) sets the value of the proportional gain. Expressed in millivolts/mm, it takes any integer between 0 and 65535.

Its value indicates the velocity command corresponding to a feedrate resulting in 1 millimeter (0.03937 inch) of following error.

Example:

The maximum feedrate for a particular axis (rapid traverse G00) is 15 m/min, but we would like to limit its maximum porgrammable machining feedrate (F) to 3 m/min with an axis lag of 1 mm at a feedrate of 1 m/min (gain of 1 in metric).

a.m.p. G00FEED (P38) must be set to 15,000 (15 m/min).

a.m.p. MAXVOLT (P37) must be set to 9500 and the servo drive adjusted so as to provide 15m/min with a velocity command of 9,5 V.

a.m.p. MAXFEED (P42) must be set to 3,000 (3 m/min).

Velocity command corresponding to F 1000 mm/min:

Velocity command = (F x 9.5V) / "G00FEED"

Velocity command = (1000 mm/min x 9.5V) / 15000 mm/min = 0.633V

Velocity command = 633 mV

Therefore, "PROGAIN" (P23) = 633

Considerations to bear in mind

When setting the proportional gain:

- The maximum amount of following error allowed by the CNC for the axis is the value indicated by a.m.p. MAXFLWE1 (P21). When exceeded, the CNC issues the corresponding following error message.
- The amount of following error decreases as the gain increases, but it tends to make the system unstable.
- In practice, most machines show an excellent behavior with a unitary gain (gain of 1, 1 mm of following error for a feedrate of 1m/minute).



Once the axes have been adjusted separately, the ones being interpolated together should be further adjusted so their following errors are as identical as possible.

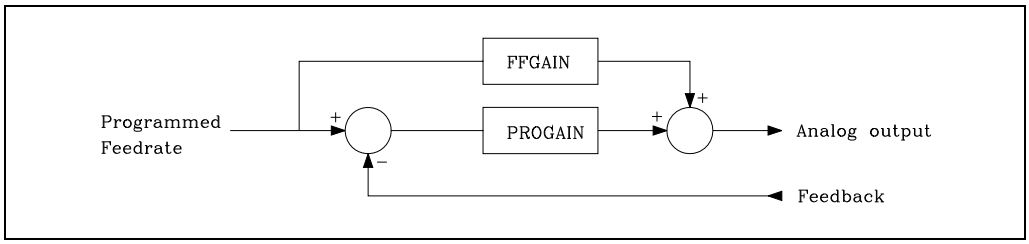
The more identical their following errors are, the more "round" the programmed circles will turn out.

5.5.4 Feed-forward gain setting

With the feed-forward gain, it is possible to reduce the following error without increasing the gain, thus keeping the system stable.

It sets the percentage of velocity command due to the programmed feedrate; the rest depends on the proportional and derivative/AC-forward gains.

This gain is only to be used when operating with acceleration/deceleration control.



For example, if a.m.p. FFGAIN (P25) has been set to "80", the axis velocity command will be:

- 80% of it will depend on the programmed feedrate (feed-forward gain).
- 20% of it will depend on the axis following error (proportional gain).

Setting the Feed-Forward gain involves a critical adjustment of a.m.p. MAXVOLT (P37).

1. Move the axis in G00 and at 10%.
2. Measure the actual analog command at the drive.
3. Set parameter MAXVOLT (P37) to a value 10 times the measured value.

   For example, If the measured command voltage was 0.945 V, then set this parameter to 9.45 V, in other words: P37=9450.

Next, set a.m.p. FFGAIN (P25) to the desired value.

As an example, the following values may be used:

For slow machining.

between 40 and 60%

For regular feed machining.

between 60 and 80%

Machines (laser, plasma).

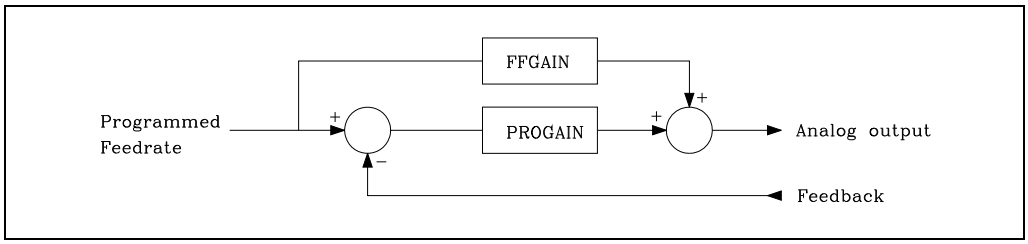
between 80 and 100%

5.5.5 Derivative (AC-forward) gain setting

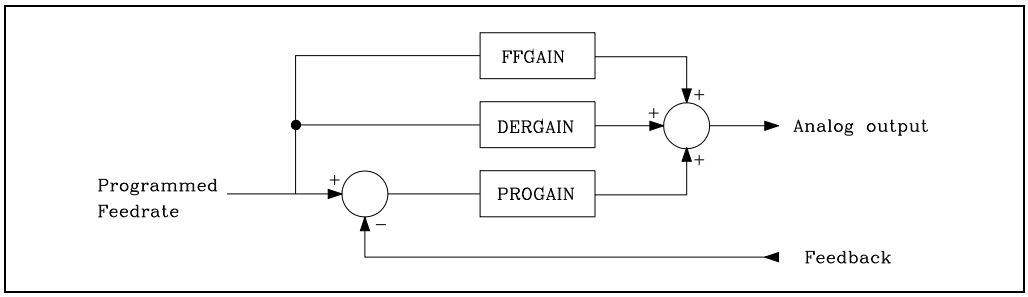
With the derivative gain, it is possible to reduce the following error during the acc./dec. stages.

Its value is given by a.m.p. DERGAIN (P24).

When this additional velocity command is due to fluctuations of following error, "ACFGAIN" (P46) = NO, it is called "derivative gain".



When it is due to variations of the programmed feedrate, "ACFGAIN" (P46) = YES, it is called AC-forward gain" since it is due to acc./dec.



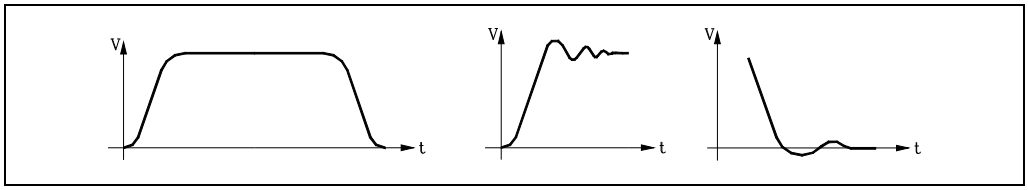
Best results are usually obtained when using it as AC-forward Gain, "ACFGAIN" (P46) = YES together with feed-forward gain.

This gain is only to be used when operating with acceleration/deceleration control.

A practical value between 2 to 3 times the Proportional Gain, "PROGAIN" (P23), may be used.

To perform a critical adjustment, proceed as follows:

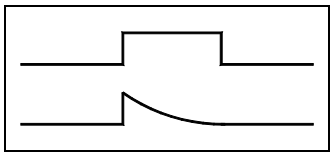
- Verify that there are no oscillations of following error, In other words, that it is not unstable.
- Check,with an oscilloscope, the tacho voltage or the command voltage at the drive (velocity command), verify that it is stable (left graph) and that there are no instabilities when starting up (center graph) or when braking (right graph).



5.5.6 Leadscrew backlash compensation.

On this CNC, the leadscrew backlash may be compensated for when reversing the direction of movement. Leadscrew backlash is defined with a.m.p. BACKLASH (P14).

Sometimes, an additional velocity command pulse may also be needed to recover the possible backlash when reversing the axis movement. The additional velocity command pulse may be either rectangular or exponential.



If the duration of the rectangular pulse is adjusted for low speed, it may be too high for high speed or not enough for low speed when adjusted for high speed. In these cases, it is recommended to use the exponential type that applies a strong pulse initially and decreases in time.

a.m.p. BACKNOUT (P29) sets the value of the additional velocity command and a.m.p. BACKTIME (P30) indicates the duration of this additional command pulse and general machine parameter ACTBAKAN (P145) indicates the type of backlash peak applied.

Cutting the compensation peak off.

Every time the axis movement is inverted, the CNC will apply to that axis the velocity command corresponding to the movement plus an additional velocity command (to make up for backlash). This additional command is eliminated (compensation peak cutoff) depending on the values of the following parameters:

G.m.p. BAKTIME (P30), g.m.p. ACTBAKAN (P145) and a.m.p. PEAKDISP (P98).

Hysteresis in the reversal movement compensation command

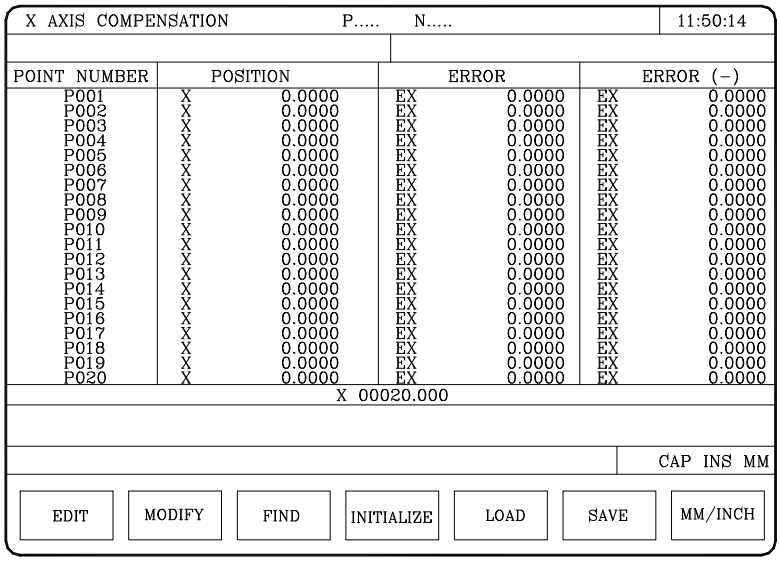
Axis machine parameter REVEHYST (P99) is used to be able to control when the compensation should really be applied after detecting a movement reversal and not applying it every time a reversal command is received.

This feature should only be applied in situations where the moving direction reversals are very small (e.g. ±1dµm) The purpose is to prevent the reversal compensation to be applied in these situations, because it can cause slight machining marks (ridges) on the part.

5.5.7 Leadscrew error compensation

The CNC provides a table for each one of the axes requiring leadscrew compensation. It is possible to define different compensation values for each moving direction. This type of compensation is activated by setting a.m.p. LSCRWCOM (P15)=ON for the desired axis.

The CNC enables one leadscrew error compensation table for each axis. The number of elements of the table is determined by the a.m.p. NPOINTS (P16), being possible to define up to 255 points per axis.



Each parameter of the table represents a point of the profile to compensate. The following information is defined at each point:

- The position occupied by the point in the profile (position to compensate). It is defined by its coordinate referred to machine zero. Possible values ±99999.9999 mm or ±3937.00787 inches.
- The amount of error of the axis in this point in the positive direction. Possible values ±99999.9999 mm or ±3937.00787 inches.
- The amount of error of the axis in this point in the negative direction. Possible values ±99999.9999 mm or ±3937.00787 inches.

For each axis position, define the amount of error to be compensated in both directions. If the amount of error in the negative direction is zero in all points, it assumes that the amount of error defined for the positive direction is valid for both directions.

Leadscrew error compensation on rotary axes

On rotary axes, although the display is limited between 0 and 360º, the internal count is accumulative. When using leadscrew error compensation, set positions 0° and 360°, first and last point of the table, with the same amount of error. This way, the CNC will apply the same compensation in all the revolutions.

Otherwise, the compensation will be limited to the indicated field.

Considerations and limitations

When defining the profile points in the table, the following requirements must be met:

- The axis points must be in sequential order starting from the most negative (least positive) point to be compensated.
- For those points outside the compensation zone, the CNC will apply the compensation value corresponding to the table point closest to them.
- The amount of error of the machine reference point may have any value.
- The error difference between two consecutive points must not be greater than the distance between them (maximum slope= 100%).

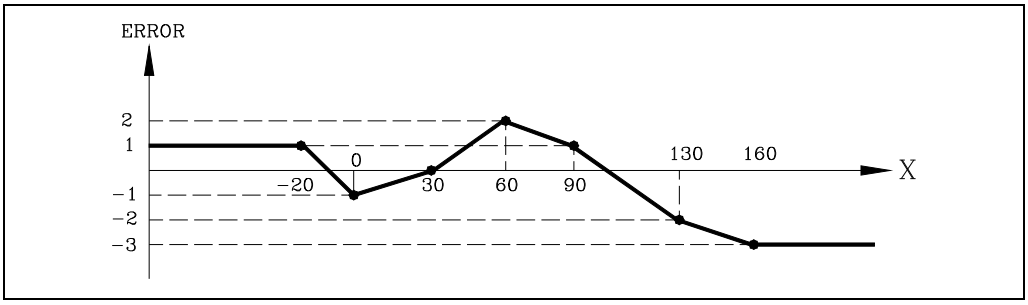
Bidirectional compensation of the leadscrew error is available from versions V7.11 (mill) and V8.11 (lathe) on.

When updated from a version that does not have bidirectional compensation, it keeps the error values in the positive direction and it sets a zero error in the negative direction for all the points.

When changing to a version that does not have bidirectional compensation, it keeps the error values in the positive direction, but it loses the error values in the negative direction. Also, the amount of error for the machine reference point must be zero.

Setting example:

The X axis ballscrew must be compensated in the positive direction for between X-20 and X160 according to the leadscrew error graph below:



Set a.m.p. LSCRWCOM (P15) = ON and NPOINTS (P16) = 7

Considering that the Machine Reference Point (physical location of the marker pulse) is located 30 mm from HOME (machine reference zero), at X30. The leadscrew error compensation parameters must be set as follows:

Point	Position		Positive error		Negative error	
P001	X	-20,000	EX	0,001	EX	0
P002	X	0,000	EX	-0,001	EX	0
P003	X	30,000	EX	0,000	EX	0
P004	X	60,000	EX	0,002	EX	0
P005	X	90,000	EX	0,001	EX	0
P006	X	130,000	EX	-0,002	EX	0
P007	X	160,000	EX	-0,003	EX	0

5.6 Reference systems

A CNC machine needs the following origin and reference points defined :

Machine zero

Machine's origin point. This is set by the manufacturer as the origin of the coordinate system of the machine.

Part zero

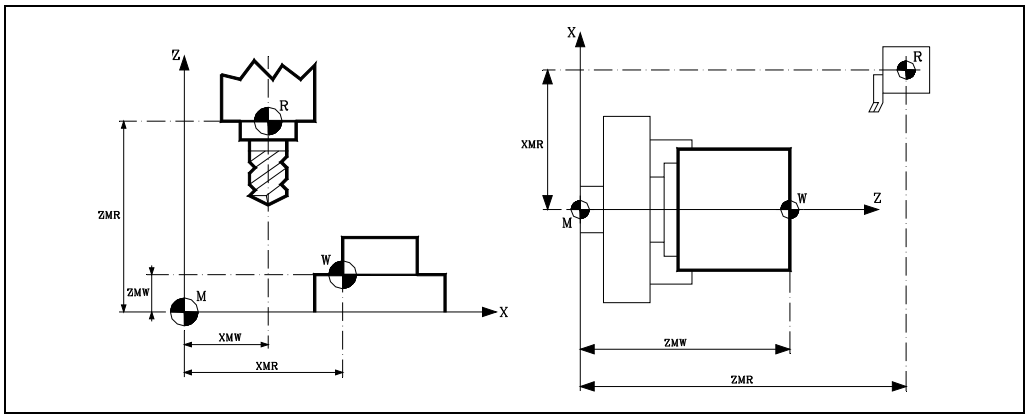
Part origin point. This is the origin point that is set for programming the measurements of the part. It can be freely selected by the programmer, and its machine reference zero can be set by the zero offset.

Reference point

This is a point on the machine established by the manufacturer (physical location of the marker pulse from the feedback device).

When the feedback system is semi-absolute (with distance-coded reference mark, Io), this point is only used when leadscrew error compensation must be applied on the axis.

When the feedback is a normal incremental system (without distance-coded reference mark, Io), besides using this point in the leadscrew error compensation, the system is synchronized at this point instead of having to move the axis all the way to the Machine Reference Zero (home).



M	Machine zero
W	Part zero
R	Machine reference point
XMW, YMW, ZMW, etc	Coordinates of part zero
XMR, YMR, ZMR, etc	Coordinates of machine reference point

5.6.1 Home search

With this CNC, home search may be performed in jog mode or by program. Home search may be carried out on one axis at a time or on several axes at the same time.

When this search (with or without distance-coded Io) is carried out in JOG mode, the active zero offset will be cancelled and the CNC will display the position values indicated by a.m.p. REFVALUE (P36). In all other cases, the active part zero will be maintained and the CNC will display the position values with respect to that part zero.

- On axes with no distance-coded feedback system:
  - The CNC moves all selected axes that have a home switch in the direction indicated by a.m.p. REFDIREC (P33).

    This movement will be carried out at the feedrate established by a.m.p. REFEED1 (P34) until the home switch is hit.

    Once all the axes have reached their respective home switches, the machine reference search (marker pulse) will be performed moving the selected axes one by one and in the selected sequence.

    This second movement will be carried out at the feedrate established by a.m.p. REFEED2 (P35) for each axis until the marker pulse of the feedback system is found.
  - If machine parameter I0TYPE (P52) =3, the home search procedure is the following:

    The CNC moves all selected axes that have a home switch in the direction indicated by a.m.p. REFDIREC (P33).

    This movement will be carried out at the feedrate established by a.m.p. REFEED1 (P34) until the home switch is hit.

    Once all the axes have reached their respective home switches, the axes move back one at a time in the selected order and at REFEED2 until the switch is released.

    Once it has released it, it will recognize the first reference mark found without changing either its moving direction or its feedrate.
- On axes with distance-coded feedback system:

  Home switches are no longer necessary since the axes may be homed anywhere along its travel. However, a.m.p. REFVALUE (P36) must be set when operating with leadscrew error compensation.

  The home search will be performed on one axis at a time and in the selected sequence.

  The axes will move a maximum of 20 mm or 100 mm in the direction set by a.m.p. REFDIREC (P33) at the feedrate set by a.m.p. REFEED2 (P35).

  If, during the home search, the home switch is pressed (if any), the CNC will reverse the homing direction.



If after the machine is all set up it is necessary to remove the feedback system, it may happen that when it is reinstalled, its marker pulse is no longer at the same physical location as it was before.

In that case, the distance (shift) between the previous marker pulse location and the current one must be assigned to a.m.p. REFSHIFT (P47) of the affected axis in order for the machine reference point (home) to remain the same.

This way, when searching home, the axis will move this additional distance, indicated by a.m.p. REFSHIFT (P47) value, after finding the new marker pulse of the feedback system. This movement is carried out at the feedrate indicated by a.m.p. REFEED2 (P35).

Gantry axes

Home search on Gantry axes may be carried out in JOG mode or by program. It will be carried out as follows:

- On axes with no distance-coded feedback system:
  - The CNC starts the movements of both axes in the direction indicated by a.m.p. REFDIREC (P33) of the main axis.

    These movements will be performed at the feedrate indicated by a.m.p. REFEED1 (P34) for the main axis until the home switch for this axis is hit.

    Then, the home search will start on both axes at the feedrate indicated by a.m.p. REFEED2 (P35) of the main axis.

    The CNC will wait until the marker pulse (home) of the slave axis is found and then, it will look for the marker pulse from the main axis.
  - If machine parameter I0TYPE=3, the home search procedure is the following:

    The CNC starts the movements of both axes in the direction indicated by a.m.p. REFDIREC (P33) of the main axis.

    These movements will be performed at the feedrate indicated by a.m.p. REFEED1 (P34) for the main axis until the home switch for this axis is hit.

    It then moves back at REFEED2 until the home switch is released.

    Once the switch has been released, the CNC will wait until the marker pulse (home) of the slaved axis is found and then, it will look for the marker pulse from the main axis. This is done without changing the direction or the speed of the movement.
- On axes with distance-coded feedback system:

  The CNC starts moving both axes in the direction indicated by a.m.p. REFDIREC (P33) for the main axis at the feedrate indicated by a.m.p. REFEED2 (P35) of the main axis.

  The CNC will wait until the marker pulse (home) of the slave axis is found and then, it will look for the marker pulse from the main axis.

If the difference obtained between both reference positions is not the same as the one indicated by a.m.p. REFVALUE (P36) for both axes, the CNC will correct the position of the slave axis. This will end the home search operation.

When this search is carried out in the JOG mode, the active zero offset will be cancelled and the CNC will display the position value indicated by a.m.p. REFVALUE (P36) for the main axis. In all other cases, the displayed position value will be referred to the zero offset (or part zero) active before the home search.



If the a.m.p. REFDIREC (P33) of the main axis has been set for a positive direction, the a.m.p. REFVALUE (P36) of the slave axis must be set to a value lower than that assigned to the main axis.

Likewise, if the a.m.p. REFDIREC (P33) of the main axis has been set for a negative direction, the a.m.p. REFVALUE (P36) of the slaved axis must be set to a value greater than that assigned to the main axis. They must never have the same value.

When encoders are used for feedback, the difference between the values assigned to a.m.p. REFVALUE (P36) of both axes must be smaller than the pitch of the ballscrew.

It is recommended that the distance between the marker pulses of both encoders be half the leadscrew pitch.

Gantry axes. Managing two home switches

Managing two home switches is only possible if axis machine parameter I0TYPE (P52) =3.

If both the master and the slave axes have a home switch (a.m.p. DECINPUT (P31) of the master and slave are YES), the home search will be carried out as follows:

The CNC starts the movements of both axes in the direction indicated by a.m.p. REFDIREC (P33) of the main axis.

This movement is carried out at the feedrate indicated by a.m.p. REFEED1 (P34) of the main axis. The axes move until one of them presses its home switch.

Then, the home search will start on the axis that pressed the home switch first at the feedrate indicated by a.m.p. REFEED2 (P35) of the main axis.

Once the first axis has been homed, its coordinate is initialized with a.m.p. REFVALUE (P36) and it starts homing the other axis.

Master and slave start moving together at a.m.p. REFEED1 (P34) of the main axis until detecting the home switch of the second axis.

It, then, starts homing the second axis at a.m.p. REFEED2 (p35) of the main axis and once detected, it initializes its coordinate.

After this, depending on the value of axis machine parameter DIFFCOMP (P96), it will compensate for the difference between the master and slave axes or it will leave uncompensated.

If the master axis is the first one to press the home switch and its a.m.p. REFSHIFT (P47) is other than zero, it does not start the second home search until executing the movement for the REFSHIFT (P47) on the master axis.

Special cases:

- If when starting the home search, either the master or the slave is pressing the home switch, the axes will move until releasing the home switch and it then homes that axis first.
- If when starting the home search both the master and the slave are pressing the home switch, it first homes the master axis.
- When commanding a simultaneous homing of the gantry axis and other axes; it first moves all the axes having a home switch until all the home switches are pressed (in the case of a gantry pair, one of the home switches will be pressed).

  After this, if I0TYPE (P52) =3, the axes move one by one to release the home switches and search home in the selected order.

Managing the alignment between master and slave using a PLC mark and a machine parameter

After homing both axes of the Ganty pair, if a.m.p. of the master DIFFCOMP (P96) = 1, it corrects the position difference of the slave so the coordinate difference between the master and the slave is zero.

Whether parameter DIFFCOMP = ·1· or = ·0·, the difference between the Gantry axes may be corrected at any time using the PLC marks SERVOaxisON and the DIFFCOMaxis where "axis" is the name or the logic number of the master axis. The theoretical difference between the master and the slave is corrected as follows:

- With the leading edge (up flank) of DIFFCOMaxis while SERVOaxisON = 1.
- With the leading edge (up flank) of SERVOaxisON while DIFFCOMaxis = 1.

  In this case, to correct the theoretical difference between master and slave, both the master and the slave axes must be set as Gantry axis or as DRO axis. Otherwise, the upflank of the SERVOaxisON mark corrects the following error of the slave axis.

Besides, the value of axis machine parameter MAXDIFF (P97) is taken into account when it is about correct the position difference.

If the position difference between master and slave is not compensated because the coordinate difference is greater than the value of a.m.p. MAXDIFF, PLC mark MAXDIFFaxis will be activated. In this case the PLC can issue a warning.

5.6.2 Setting on systems without distance-coded feedback

Machine reference point

The reference point must be adjusted on one axis at a time. The following procedure is recommended:

- Indicate in the a.m.p. REFPULSE (P32) the type of marker pulse Io being used for Home Search.
- Likewise, set a.m.p. REFDIREC (P33) to indicate the direction of the axis when searching Home.
- On the other hand, set a.m.p. REFEED1 (P34) that defines the approach feedrate of the axis until the home switch is pressed and a.m.p. REFEED2 (P35) that indicates the homing feedrate until the reference mark (marker pulse) is detected.
- The machine reference point will be set to "0". a.m.p. REFVALUE (P36).
- Once in the JOG mode and after positioning the axis in the right area, start homing the axis. When done, the CNC will assign a "0" value to this point.
- After moving the axis to the Machine Reference Zero or up to a known position (with respect to Machine Reference Zero), observe the position reading of the CNC for that point.

  This will be distance from the Machine Reference Zero to that point. Therefore, the value to be assigned to a.m.p. REFVALUE (P36), which defines the coordinate corresponding to the Machine Reference Point (physical location of the marker pulse).

  REFVALUE = Machine coordinate – CNC reading.

  Example:

  If the point whose known position is located 230 mm from Machine Reference Zero and the CNC reads -123.5 mm as the coordinate value for this point, the coordinate of the Machine Reference Point with respect to Machine Reference Zero will be:

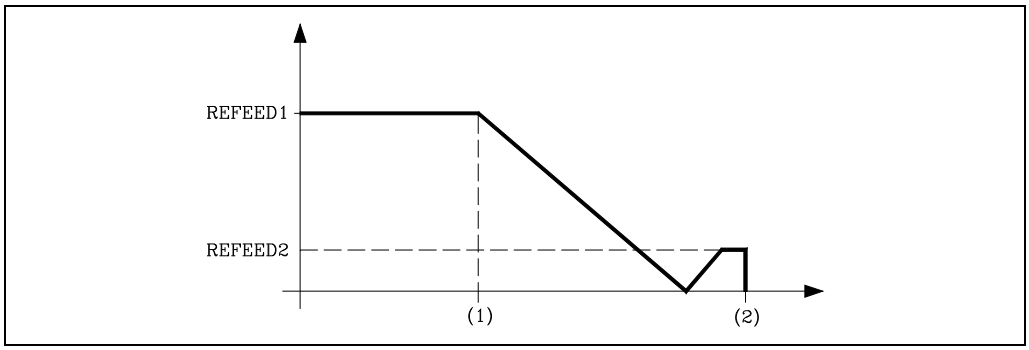
  "REFVALUE" = 230 - (-123.5) = 353.5 mm.
- After allocating this new value, press SHIFT + RESET or turn the CNC off and back on in order for the CNC to assume this new value.
- The axis must be homed again in order for it to assume its right reference values.

Considerations

If at the time when the home search is requested, the axis is sitting on the home switch, the axis will back up (in the direction opposite to the one indicated by "REFDIREC (P33) ") until it is off the switch and then, it will go on to search home.

If the axis is positioned beyond the software limits "LIMIT+" (P5) and "LIMIT-" (P6), it must be brought back into the work area (within those limits) and on the proper side for referencing (home searching).

Care must be taken when placing the home switch and when setting feedrates "REFEED1 (P34)" and "REFEED2 (P35)". The home switch (1) will be installed so the marker pulse (2) will be found in the zone corresponding to feedrate "REFEED2" (P35). If there is no room for it, reduce the value of "REFEED1 (P34)". For example, for rotary encoders whose consecutive marker pulses are very close to each other.



When the selected axis does not have a machine reference (home) switch (a.m.p. DECINPUT (P31) = NO), the CNC will move the spindle at the feedrate set by a.m.p. REFEED2 (P35) until the first marker pulse from the current position is found, thus ending the home search.

FAGOR linear encoders (scales) provide a negative marker (reference) pulse Io every 50 mm (about 2 inches) and the FAGOR rotary encoders provide one positive reference pulse per revolution.

Do not mistake the type of pulse provided by the feedback system with the value to be assigned to a.m.p. REFPULSE (P32). This parameter must indicate the type of active flank (leading or trailing edge), positive or negative of the reference mark (Io) used by the CNC.

5.6.3 Setting on systems with distance-coded feedback

Offset adjustment

The offset of the linear encoder must be adjusted on one axis at a time, preferably, following this procedure:

1. Set the following a.m.p:

   REFDIREC (P33) Homing direction.

   "REFEED2" (P35) Homing feedrate.
2. Verify that the value allocated to a.m.p. REFPULSE (P32) (type of marker pulse of the feedback system) is correct.

   To do this, set a.m.p. DECINPUT (P31) = NO and a.m.p. I0TYPE (P52) = 0. Then perform a home search.

   If assumed immediately, change a.m.p. REFPULSE (P32) and check again.
3. Set a.m.p. I0TYPE (P52) = 1 and ABSOFF (P53) = 0.
4. Once in JOG mode and after positioning the axis in the proper area, home the axis. The new position value displayed by the CNC is the distance from the current point to the origin of the linear encoder.
5. Perform several consecutive home searches and observe the CNC display during the whole process.

   The counting must be continuous. If it is not, if jerky, set a.m.p. I0TYPE (P52) = 2 and repeat steps 4 and 5.
6. Move the axis up to the Machine Reference Zero or up to a point whose position with respect to Machine Reference Zero is already known and observe the position value displayed by the CNC. This value is the distance from the current point to the origin of the linear encoder.
7. The value to be assigned to a.m.p. ABSOFF (P53) must be calculated with the following formula:

   ABSOFF (P53) = CNC reading - Machine coordinate.

   Example:

   If the point whose position is already known is located 230 mm from Machine Reference Zero and the CNC shows -423.5 mm as the position for this point, the linear encoder offset will be:

   ABSOFF (P53) = -423,5 - 230 = -653.5 mm.
8. After allocating this new value, press SHIFT + RESET or turn the CNC off and back on in order for the CNC to assume this new value.
9. Home the axis again in order for it to assume the new correct reference values.

Considerations

If the axis is positioned beyond the software limits "LIMIT+" (P5) and "LIMIT-" (P6), it must be brought back into the work area (within those limits) and on the proper side for referencing (home searching).

When using distance-coded linear encoders, home switches are no longer necessary.

However, home switches may be used as travel limits during home search.

If while homing, the home switch is pressed, the axis will reverse its movement and it will keep searching home in the opposite direction.

Distance-coded Fagor linear encoders have negative coded marker pulses (Io).

Do not mistake the type of pulse provided by the feedback system with the value to be assigned to a.m.p. REFPULSE (P32).

This parameter must indicate the type of active flank (leading or trailing edge), positive or negative of the reference mark (Io) used by the CNC.

If while homing an axis, its corresponding DECEL* signal is set high, the axis will reverse movement and the home search will be carried out in the opposite direction.

5.6.4 Axis travel limits (software limits)

Once all the axes have been referenced, their software limits must be measured and set.

This operation must be carried out one axis at a time and it could be done as follows:

- Move the axis in the positive direction towards the end of the axis travel stopping at a safe distance from the mechanical end-of-travel stop.
- Assign the coordinate shown by the CNC for that point to a.m.p. LIMIT+ (P5).
- Repeat these steps in the negative direction assigning the resulting coordinate to a.m.p. LIMIT- (P6).
- Once both travel limits have been set for all the axes, press SHIFT + RESET or turn the CNC OFF and back ON in order for these new values to be assumed by the CNC.

5.7 Unidirectional approach

The FAGOR 8055 CNC provides a number of machine parameters to help improve repeatability when positioning the axes in rapid (G00) by always approachingg the end point in the same direction.

"UNIDIR" (P39)

Indicates the direction of unidirectional approach.

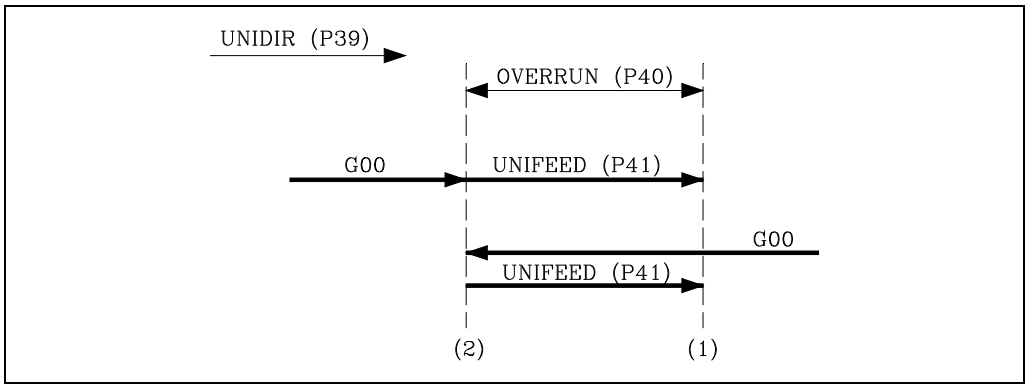
OVERRUN

Indicates the distance to be kept between the approach point and the programmed point. If this parameter is set to 0, the CNC will not perform the unidirectional approach.

"UNIFEED" (P41)

Indicates the feedrate to be used from the approach point to the programmed point.

The CNC will calculate the approach point (2) based on the programmed target point (1) and the a.m.p. UNIDIR (P39) and OVERRUN (P40).



The positioning will be carried out in two stages:

1. Rapid positioning (G00) up to the calculated approach point (2). If the axis is moving in the direction opposite to that indicated by "UNIDIR", it will overshoot the programmed point.
2. Positioning at feedrate UNIFEED (P41) from this point to the programmed point (1).

5.8 Auxiliary M, S, T function transfer

Every time a block is executed in the CNC, information is passed to the PLC about the M, S, and T functions which are active.

Auxiliary M function

The CNC uses logic outputs "MBCD1" thru "MBCD7" (R550 thru R556) to "tell" the PLC which M functions it must execute. One function per logic output.

It also activates the general logic output "MSTROBE" to "tell" the PLC to start executing them.

Every time the CNC detects an M function, it analyzes the M function table to find out when to pass it along to the PLC (either before or after the movement) and whether it must wait for the "AUXEND" signal or not before resuming program execution.

If the programmed function is not defined in that table, it will be executed at the beginning of the block and the CNC will wait for the "AUXEND" signal to resume program execution.

See *"9.1 Auxiliary M, S, T functions"* on page 318. See *"10.6 General logic outputs"* on page 353. See *"4.9 Tables"* on page 173.

Example 1:

Execution of a motion block containing 7 M functions 4 of which are executed before the axes move (M51, M52, M53, M54) and 3 afterwards (M61, M62, M63).

1. It sends out to the PLC the 4 M functions programmed to be executed before the move

   It sets logic outputs "MBCD1=51", "MBCD2=52" "MBCD3=53" "MBCD4=54" and it activates the general logic output "MSTROBE to "tell" the PLC to go ahead with their execution.

   Should any of them need the AUXEND activated, the CNC will "wait" for this signal to be activated before going on to executing the rest of the block.

   If none of them need the AUXEND signal activated, the CNC will maintain the "MSTROBE" signal activated for a period of time set by the general machine parameter "MINAENDW (P30)". This output stays active for the time indicated by g.m.p. MINAENDW (P30).
2. The programmed axis move will be executed.
3. It sends out to the PLC the 3 M functions programmed to be executed after the move.

   It sets logic outputs "MBCD1=61", "MBCD2=62", "MBCD3=63" and it activates the general logic output "MSTROBE to "tell" the PLC to go ahead with their execution.

   Should any of them need the AUXEND activated, the CNC will "wait" for this signal to be activated before going on to executing the rest of the block.

   If none of them need the AUXEND signal activated, the CNC will maintain the "MSTROBE" signal activated for a period of time set by the general machine parameter "MINAENDW (P30)". This output stays active for the time indicated by g.m.p. MINAENDW (P30).

Example 2:

Execution of a motionless block containing 7 M functions 4 of which are executed before the axes move (M51, M52, M53, M54) and 3 afterwards (M61, M62, M63).

1. It sends out to the PLC the 4 M functions programmed to be executed before the move

   It sets logic outputs "MBCD1=51", "MBCD2=52" "MBCD3=53" "MBCD4=54" and it activates the general logic output "MSTROBE to "tell" the PLC to go ahead with their execution.

   Should any of them need the AUXEND activated, the CNC will "wait" for this signal to be activated before going on to executing the rest of the block.

   If none of them need the AUXEND signal activated, the CNC will maintain the "MSTROBE" signal activated for a period of time set by the general machine parameter "MINAENDW (P30)". This output stays active for the time indicated by g.m.p. MINAENDW (P30).
2. It sends out to the PLC the 3 M functions programmed to be executed after the move.

   It sets logic outputs "MBCD1=61", "MBCD2=62", "MBCD3=63" and it activates the general logic output "MSTROBE to "tell" the PLC to go ahead with their execution.

Should any of them need the AUXEND activated, the CNC will "wait" for this signal to be activated before going on to executing the rest of the block.

If none of them need the AUXEND signal activated, the CNC will maintain the "MSTROBE" signal activated for a period of time set by the general machine parameter "MINAENDW (P30)". This output stays active for the time indicated by g.m.p. MINAENDW (P30).

S function

The CNC transfers the "S function" out to the PLC only when using the BCD-coded "S" output. s.m.p. SPDLTYPE (P0) set to other than "0".

The CNC sends the programmed "S" value via logic output "SBCD" (R557) and activates the general logic output "SSTROBE" to indicate to the PLC to go ahead with its execution.

This transmission is carried out at the beginning of the block execution and the CNC will wait for the "AUXEND" general input to be activated and then consider its execution completed.

T function

The CNC will indicate via the variable "TBCD" (R558) the T function which has been programmed in the block and activates the general logic output "TSTROBE" to tell the PLC to go ahead with its execution.

This transmission is made at the beginning of the block execution and the CNC will wait for the general input "AUXEND" to be activated to consider the execution completed.

Second T function

The CNC transfers the second T function to the PLC in the following cases:

- When having a machining center with non-random tool magazine. g.m.p. TOFFM06 (P28) = YES and RANDOMTC (P25) = NO
- When using a random tool magazine, g.m.p. RANDOMTC (P25) = YES and a special tool change takes place. See the chapter "Tool table" of the operating manual.

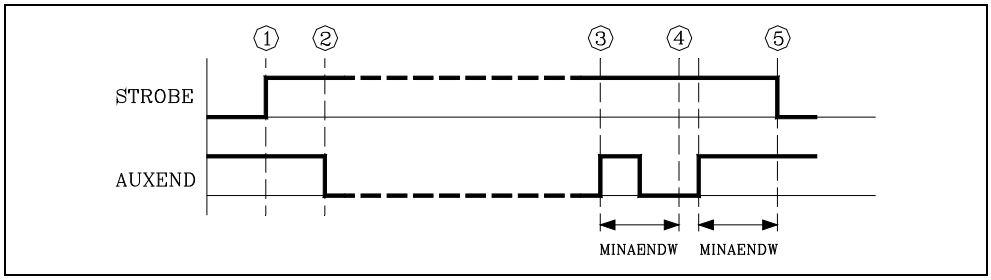
On executing the M06 function, a the CNC indicates the position of the magazine (empty pocket) where the tool being in the spindle must be placed.

This indication will be made by means of the variable "T2BCD" (R559) and by activating the general logic output "T2STROBE" to tell the PLC that it must execute this. The CNC will wait for the general input AUXEND to be activated to consider the execution completed.

It must be borne in mind that at the beginning of the execution of the block, the CNC can tell the PLC the execution of the M, S, T and T2 functions by activating their STROBE signals together and waiting for a single signal "AUXEND" for all of them.

5.8.1 Transferring M, S, T using the AUXEND signal

1. Once the block has been analyzed and after sending the corresponding values in the "MBCD1-7", "SBCD", "TBCD" and "T2BCD" variables, the CNC will tell the PLC by means of the general logic outputs "MSTROBE", "SSTROBE", "TSTROBE" and "T2STROBE" that the required auxiliary functions must be executed.



2. When the PLC detects the activation of one of the STROBE signals, it must deactivate the general CNC logic input "AUXEND" to tell the CNC that the execution of the corresponding function or functions has begun.
3. The PLC will execute all the auxiliary functions required, it being necessary to analyze the general CNC logic outputs:

   "MBCD1" through "MBCD7" and "MSTROBE"

   to execute the M functions.

   "SBCD" and "SSTROBE"

   to execute the S function

   "TBCD" and "TSTROBE"

   to execute the T function

   "T2BCD and "T2STROBE"

   to execute the second T function

   Once this has been executed the PLC must activate the general logic input "AUXEND" to indicate to the CNC that the processing of the required functions was completed.
4. Once the general input "AUXEND" is active, the CNC will require that this signal be kept active for a period of time greater than that defined by means of the g.m.p. MINAENDW (P30).

   This way, erroneous interpretations of this signal by the CNC due to an improper PLC program logic are avoided .
5. Once the period of time "MINAENDW (P30)" has elapsed with the general input "AUXEND" at a high logic level, the CNC will deactivate the general logic outputs "MSTROBE", "SSTROBE", "TSTROBE", "T2STROBE" to tell the PLC that the execution of the required auxiliary function or functions has been completed.

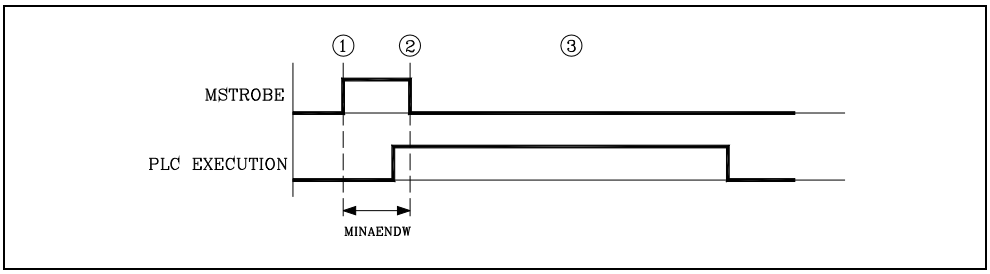
When executing 2 consective blocks which send information to the PLC and after finishing the execution of the first block, the CNC waits a MINAENDW period of time before starting to execute the second block.

This way, it assures that a MINAENDW delay takes place between the STROBE off (end of first block) and STROBE on (beginning of the second block).

It is advisable for the "MINAENDW (P30)" value to be equal to or greater than the duration of a PLC cycle, in order to ensure the detection of this signal by the PLC.

5.8.2 Transferring the auxiliary (miscellaneous) M functions without the AUXEND signal

1. Once the block has been analyzed and after passing the corresponding values in variables "MBCD1-7", the CNC will tell the PLC through the general logic output "MSTROBE" that the required auxiliary function or functions must be executed.



2. The CNC will keep the general logic output "MSTROBE" active during the time indicated by means of g.m.p. MINAENDW (P30).

   Once this period of time has elapsed the CNC will continue to execute the program.

   It is advisable for the "MINAENDW (P30)" value to be equal to or greater than the duration of a PLC cycle, in order to ensure the detection of this signal by the PLC.

3. When the PLC detects the activation of the general logic signal "MSTROBE" it will execute the required auxiliary "M" functions at the CNC logic outputs "MBCD1 thru 7".

5.9 Spindle

5.9.1 Spindle types

The setting of s.m.p. SPDLTYPE (P0) allows the following possibilities:

SPDLTYPE = 0 Spindle analog command output..

SPDLTYPE = 1, 2-digit BCD coded spindle command output S.

SPDLTYPE = 2, 8-digit BCD coded spindle command output S.

When using BCD (2 or 8 digits) coded command output, the spindle will operate in open loop and it will be controlled by means of functions M3, M4 and M5.

When using analog command output, the spindle can operate:

- In open loop, controlled by means of functions M3, M4 and M5.
- In closed loop, by means of function M19. This requires an encoder mounted on the spindle and s.m.p. NPULSES (P13) must be set to a value other than "0".
- Controlled via PLC. With this feature, the PLC may take control of the spindle for a certain period of time.

  A typical application of this feature is the control of the spindle oscillation during the spindle gear change.

Regardless of the type of spindle command being used, the CNC admits up to 4 spindle gears.

The spindle gear change may be made either manually or automatically by the CNC.

To change spindle gears, functions M41, M42, M43 and M44 are used to let the PLC know which one is to be selected.

5.9.2 Spindle speed (S) control

BCD output

When using BCD (2 or 8 digits) coded command output, the spindle will operate in open loop and it will be controlled by means of functions M3, M4 and M5.

To do that, set s.m.p. SPDLTYPE (P0) with the right value.

SPDLTYPE = 1, 2-digit BCD coded spindle command output S.

SPDLTYPE = 2, 8-digit BCD coded spindle command output S.

Whenever a new spindle speed is selected, the CNC will transfer the programmed S value into register "SBCD" (R557) and it will activate general logic output "SSTROBE" (M5533) to "tell" the PLC to go ahead with its execution.

This transmission is carried out at the beginning of the block execution and the CNC will wait for the "AUXEND" general input to be activated and then consider its execution completed.

If it uses 2-bit BCD code, the CNC will indicate the S value to the PLC by means of this register and according to the following conversion table:

Programmed S	S BCD
0	00
1	20
2	26
3	29
4	32
5	34
6	35
7	36
8	38
9	39
10-11	40
12	41
13	42
14-15	43
16-17	44
18-19	45
20-22	46
23-24	47
25-27	48
28-31	49
32-35	50
36-39	51
40-44	52
45-49	53

Programmed S	S BCD
50-55	54
56-62	55
63-70	56
71-79	57
80-89	58
90-99	59
100-111	60
112-124	61
125-139	62
140-159	63
160-179	64
180-199	65
200-223	66
224-249	67
250-279	68
280-314	69
315-354	70
355-399	71
400-449	72
450-499	73
500-559	74
560-629	75
630-709	76
710-799	77

Programmed S	S BCD
800-899	78
900-999	79
1000-1119	80
1120-1249	81
1250-1399	82
1400-1599	83
1600-1799	84
1800-1999	85
2000-2239	86
2240-2499	87
2500-2799	88
2800-3149	89
3150-3549	90
3550-3999	91
4000-4499	92
4500-4999	93
5000-5599	94
5600-6299	95
6300-7099	96
7100-7999	97
8000-8999	98
9000-9999	99

If a value over 9999 is programmed the CNC will tell the PLC the spindle speed corresponding to value 9999.

If S output in 8-digit BCD is used the CNC will indicate the programmed spindle speed to the PLC by means of this register. This value will be coded in BCD format (8 digits) in thousandths of a revolution per minute.

S 12345.678 0001 0010 0011 0100 0101 0110 0111 1000

Analog output

In order for the CNC to provide an analog output to control the spindle speed, it is necessary to set s.m.p. SPDLTYPE (P0) = 0.

The CNC will generate the analog output (within +10V.) corresponding to the programmed rotation speed or a unipolar velocity command if the s.m.p. POLARM3 (P7) and POLARM4 (P8) have been assigned the same value.

The Closed Loop mode of operation (with M19) is described later on in this manual.

PLC controlled spindle

With this feature, the PLC may take control of the spindle for a certain period of time.

To do that, follow these steps:

1. Have the PLC place the "S" value at CNC logic input "SANALOG" (R504). This "S" value corresponds to the velocity command value to be applied to the spindle drive.

   Also, set CNC logic input "PLCCNTL" (M5465) high to let the CNC know that from this moment on, the PLC is the one setting the velocity command for the spindle.
2. From this instant on, the CNC outputs the spindle velocity command indicated by the PLC at CNC logic input "SANALOG" (R504).

   If the PLC changes the value of the "SANALOG" input, the CNC will update the velocity command accordingly.
3. Once the operation has concluded, the CNC must recover the control of the spindle back from the PLC. To do this, CNC logic input "PLCCNTL" (M5465) must be set low again.

A typical application of this feature is the control of the spindle oscillation during the spindle gear change.

5.9.3 Spindle gear change

With this CNC, the machine can use a gear box for adjusting the best spindle speed and torque for the particular machining needs at any time.

The CNC admits up to 4 spindle gears that are determined by s.m.p. "MAXGEAR1 (P2)", "MAXGEAR2 (P3)", MAXGEAR3 (P4)" and "MAXGEAR4 (P5)". They indicate the maximum speed (in rpm) for each range.

The value assigned to "MAXGEAR1 (P2)" will be the one corresponding to the lowest gear and the one assigned to "MAXGEAR4 (P5)" will be the one corresponding to the highest gear.

When not using all 4 gears, use the lower parameters starting with MAXGEAR1 (P2). Set the unused gears with the same value as the highest of the ones used.

The auxiliary functions M41, M42, M43 and M44 are used to "tell" the PLC that spindle gear 1, 2, 3 or 4 must be selected.

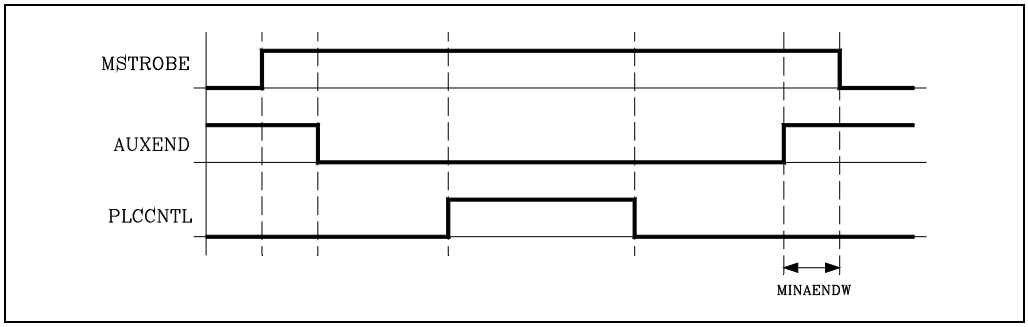
In turn, the PLC must "tell" the CNC the speed gear being selected. This will be indicated by means of the logic inputs for the spindle: "GEAR1 (M5458)", "GEAR2 (M5459)", "GEAR3 (M5460)" and "GEAR4 (M5461)".

Since to each "S" speed corresponds a spindle gear, before selecting a new "S" one must:

1. Analyze whether the new "S" involves a gear change.
2. If it does, execute the M function corresponding to the new gear (M41 thru M44) in order for the PLC to select it.
3. Wait for the PLC to select the new gear. Check spindle logic inputs "GEAR1" (M5458), "GEAR2" (M5459), "GEAR3" (M5460) and "GEAR4" (M5461).
4. Select the new speed "S".

To have the CNC perform all these operations automatically, set s.m.p. AUTOGEAR (P6) =YES to indicate that the gear change is to be generated by the CNC.

Automatic gear change controlled by the PLC



When the CNC detects a gear change, it sends out to the PLC the corresponding M code (M41 thru M44) via one of the logic outputs "MBCD1-7" (R550 thru R556).

It also activates general logic output "MSTROBE" (M5532) to "tell" the PLC to go ahead with the execution.

The PLC deactivates CNC general logic input "AUXEND" (M5016) to indicate to the CNC that it began processing the "M" function.

When requiring spindle oscillation control during a gear change, follow these steps:

1. Indicate, from the PLC, at CNC logic input "SANALOG" (R504) the value of the residual S command to be applied to the spindle drive.

   Also, set CNC logic input "PLCCNTL" (M5465) high to let the CNC know that from this moment on, the PLC is the one setting the velocity command for the spindle.

2. From this instant on, the CNC outputs the spindle velocity command indicated by the PLC at CNC logic input "SANALOG" (R504).

   If the PLC changes the value of the "SANALOG" input, the CNC will update the velocity command accordingly.

3. Once the operation has concluded, the CNC must recover the control of the spindle back from the PLC. To do this, CNC logic input "PLCCNTL" (M5465) must be set low again.

Once the requested gear change is completed, the PLC must set the corresponding CNC logic input "GEAR1" (M5458), "GEAR2" (M5459), "GEAR3" (M5460) or "GEAR4" (M5461) high.

Finally, the PLC will reactivate CNC general logic input "AUXEND" (M5016) indicating to the CNC that it has finished executing the auxiliary function.

Automatic gear change when working with M19

Every time M19 is programmed, it is recommended that the corresponding spindle gear be selected.

If no gear is already selected, the CNC proceeds as follows:

It converts the speed indicated in degrees per minute ats.m.p. REFEED1 (P34) into rpm.

It selects the spingle gear corresponding to those rpm.

The spindle gear cannot be changed when operating in M19. The gear must be selected beforehand.

5.9.4 Spindle in closed loop

One of the following conditions must be met in order for the spindle to operate in closed loop by means of "spindle orientation (M19)":

- It is an analog or digital drive with feedback to the CNC (via connector) and s.m.p NPULSES (P13) is other than 0.
- It is a digital drive (CAN) with feedback to the drive and s.m.p DRIBUSLE (P51) is other than 0.

Also, when switching from open to closed loop, either an "M19" or an "M19 S±5.5" must be executed.

The S±5.5 code indicates the spindle position, in degrees, from the spindle reference point (marker pulse).

When switching form open to closed loop, the CNC behaves as follows:

- **If the spindle has a home switch:**

  In this case, it is possible to detect the right reference mark among the possible ones (having the encoder at the motor) originated especially by different gear ratios.

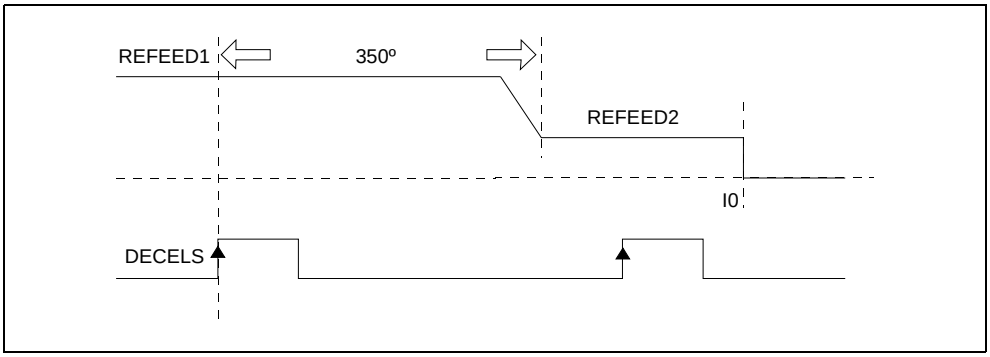
  In order for the drive to detect the right reference mark, the home switch detection must be accurate. This is achieved by having the spindle turn one more revolution once the home switch has been detected. The end portion of this last spindle revolution is carried out slowly.

From versions V15.11 (mill model) and V16.11 (lathe model) on, it is possible to apply a derivative gain or feed-forward while homing the spindle.

Spindle home search:

The spindle must have an encoder so it can be homed.

Once the home switch has been detected, the spindle will keep turning in the same direction and at the speed indicated by s.m.p. REFEED1 (P34). Before completing the next 350º, the spindle slows down to the speed indicated by s.m.p. REFEED2 (P35). From then on, the home switch is detected while turning at REFEED2 speed and keeps moving until detecting the reference mark.



Some points to consider:

- The first movement until the home switch is detected and the entire next process is carried out in the direction indicated by s.m.p. REFDIREC (P33).
- The home search may be started when the spindle is stopped or moving (M3 or M4). If the home search starts when the spindle is stopped or it has to change its turning direction, the transition from the starting speed S0 to the one indicated by s.m.p. REFEED1 will be made with a linear acceleration ramp.
- If the home search starts when it is stopped and the home switch is pressed, it will also turn one more revolution.
- This type of spindle home search may be carried out with a CAN drive. In order for it to work on CAN or analog spindles when there are several reference marks per revolution due to different gear ratios, the reference mark must be managed with the DECELS signal ignoring the actual (real) reference mark signal.

  For greater accuracy, if the average cycle of the PLC exceeds 8 ms, we recommend to manage the DECELS input at the PLC with a periodic cycle equal to or shorter than 8 ms. It is also recommended to manage the DECELS input from a local input.

When managing the DECELS input from a CAN remote input, use the commands IREMRD, OREMWR and MWR inside the periodic cycle to obtain proper synchronism.

```
PE **
IREMRD
NOT I200= DECELS
OREMWR
MWR
END
```

The delay of the DECELS signal detector can cause a low home search feedrate "REFEED1". In order for this not to affect successive positioning movements that are not related with home search, we recommend to use a positioning feedrate higher than "REFEED1" through the "M19FEED" variable and its associated PLC mark "PLCFM19".

- **If the spindle does not have a home switch:**

  If the spindle does not have a home switch, it searches the encoder marker pulse at the turning speed set by s.m.p. REFEED2 (P35). S.m.p. REFDIREC (P33) sets the spindle homing direction.

  Then, it positions the spindle at the programmed S±5.5 point. S.m.p. REFVALUE (P36) sets the position value assigned to the reference point of the spindle (home or marker pulse).

Calculating spindle resolution

The CNC assumes that one encoder revolution represents 360º. Therefore, the feedback (counting) resolution depends on the number of lines of the spindle encoder.

Resolution = 360° / (4 x number of pulses per revolution)

Hence, to obtain a resolution of 0.001º, a 90,000 line encoder is required and a 180,000 line encoder to obtain a resolution of 0.0005º.

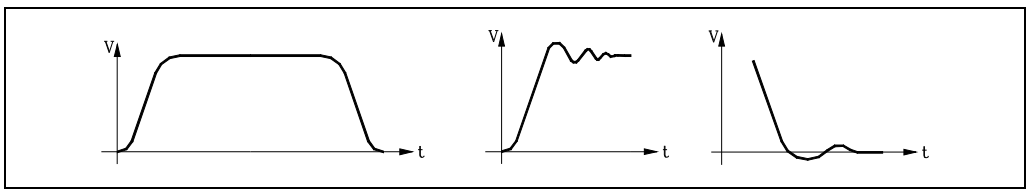
s.m.p. NPULSES (P13) must indicate the number of square pulses supplied by the spindle encoder.

In order to be able to use feedback alarm on the spindle encoder, "FBACKAL" (P15), the pulses provided by the encoder must be differential (double ended) squarewave "DIFFBACK (P14) = YES".

Gain setting

The various types of gains must be adjusted in order to optimize the system's performance for the programmed movements.

An oscilloscope is highly recommended to make this critical adjustment by monitoring the tacho signals. The illustration below shows the optimum shape for this signal (on the left) and the instabilities to be avoided during start-up and braking.



There are three types of gain. They are adjusted by means of machine parameters and following the sequence indicated next.

Proportional gain

It defines the velocity command corresponding to a feedrate resulting in 1º of following error.

It is defined with s.m.p. PROGAIN (P23).

Feed-forward gain

It sets the percentage of velocity command due to the programmed feedrate.

To use it, acc/dec must be active s.m.p. ACCTIME (P18).

It is defined with s.m.p. FFGAIN (P25).

Derivative gain or AC-forward gain.

The "derivative gain" sets the percentage of velocity command applied depending on the fluctuations of following error.

The "AC-forward gain" sets the percentage of velocity command proportional to the feedrate increments (acceleration and deceleration stages).

To use it, acc/dec must be active s.m.p. ACCTIME (P18).

It is defined with s.m.p. DERGAIN (P24) and ACFGAIN (P46).

If "ACFGAIN = No"	it applies derivative gain
If "ACFGAIN = Yes"	it applies AC-forward gain.

Proportional gain setting

In a "pure" proportional position loop, the velocity command provided by the CNC to control the spindle is, at all times, proportional to the following error (axis lag) which is the difference between its theoretical and actual (real) position.

Velocity command = Proportional Gain x Following Error

a.m.p. PROGRAIN (P23) sets the value of the proportional gain. Expressed in millivolts/degree, it takes any integer between 0 and 65535.

Its value indicates the velocity command corresponding to a feedrate resulting in 1º of following error.

This value is taken for the first spindle gear and the CNC calculates the values for the rest of the gears.

Example

The maximum speed for the 1st gear is 500 rpm and we would like to obtain 1º of following error at a speed of S = 1000 º/min. (2.778 rpm).

Command from the drive: 9.5V for 500 rpm

Velocity command corresponding to S = 1000 º/min. (2.778 rpm)

Analog velocity command = (S x 9,5 V) / "MAXGEAR1"

Analog command = (9.5 V / 500 rev/min) *2.778 rev/min =52.778 mV.

Therefore "PROGAIN" = 53

Bear in mind

When setting the proportional gain that:

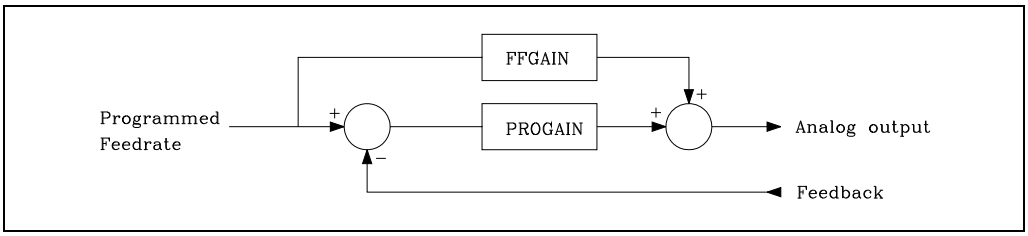
- The maximum amount of following error allowed by the CNC for the spindle is the value indicated by s.m.p. MAXFLWE1 (P21). When exceeded, the CNC issues the corresponding following error message.
- The amount of following error decreases as the gain increases, but it tends to make the system unstable.

Feed-forward gain setting

With the feed-forward gain, it is possible to reduce the following error without increasing the gain, thus keeping the system stable.

It sets the percentage of velocity command due to the programmed feedrate; the rest depends on the proportional and derivative/AC-forward gains.

This gain is only to be used when operating with acceleration/deceleration control.



For example, if s.m.p. FFGAIN (P25) has been set to "80", the spindle velocity command will be:

- 80% of it will depend on the programmed feedrate (feed-forward gain)
- 20% of it will depend on the spindle following error (proportional gain)

Setting the Feed-Forward gain involves a critical adjustment of s.m.p. MAXVOLT (P37).

1. Set the spindle at maximum speed and at 10%.
2. Measure the actual analog command at the drive.
3. Set parameter MAXVOLT (P37) to a value 10 times the measured value.

   For example, If the measured command voltage was 0.945 V, then set this parameter to 9.45 V, in other words: P37=9450.

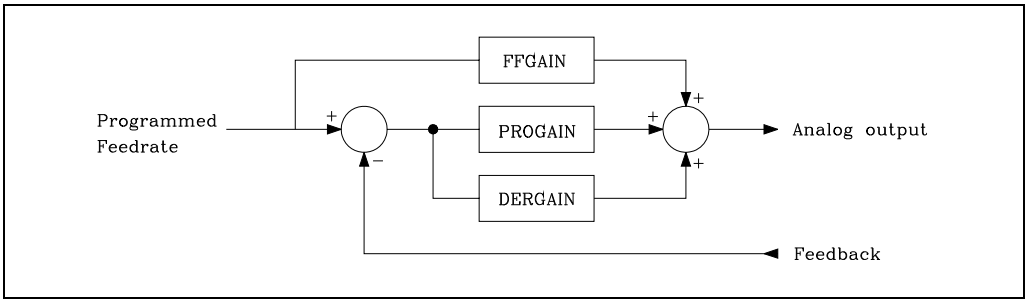
Next, set s.m.p. FFGAIN (P25) to the desired value.

Derivative (AC-forward) gain setting

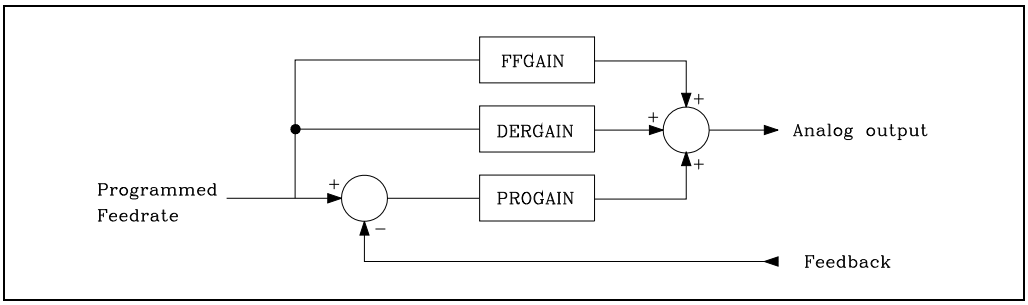
With the derivative gain, it is possible to reduce the following error during the acc./dec. stages.

Its value is given by s.m.p. DERGAIN (P24).

When this additional velocity command is due to fluctuations of following error, "ACFGAIN" (P46) = NO, it is called "derivative gain".



When it is due to variations of the programmed feedrate, "ACFGAIN" (P42) = YES, it is called AC-forward gain" since it is due to acc./dec.



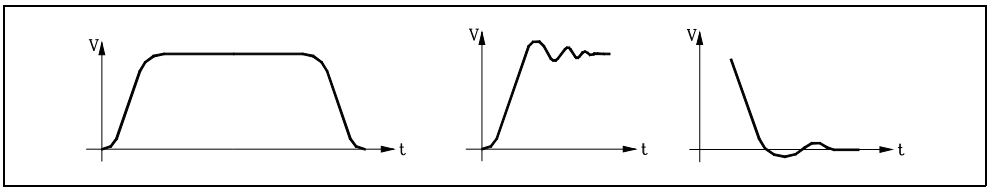
Best results are usually obtained when using it as AC-forward Gain, "ACFGAIN" (P42) = YES together with feed-forward gain.

This gain is only to be used when operating with acceleration/deceleration control.

A practical value between 2 to 3 times the Proportional Gain, "PROGAIN" (P23), may be used.

To perform a critical adjustment, proceed as follows:

- Verify that there are no oscillations of following error, In other words, that it is not unstable.
- Check,with an oscilloscope, the tacho voltage or the command voltage at the drive (velocity command), verify that it is stable (left graph) and that there are no instabilities when starting up (center graph) or when braking (right graph).



Machine reference point setting

To set the machine reference point proceed as follows:

- Indicate in the s.m.p. REFPULSE (P32) the type of marker pulse Io being used for Home Search.
- Likewise, set s.m.p. REFDIREC (P33) to indicate the direction of the axis when searching Home.
- On the other hand, set s.m.p. REFEED1 (P34) that defines the approach feedrate of the spindle until the home switch is pressed and s.m.p. REFEED2 (P35) that indicates the homing feedrate until the reference mark (marker pulse) is detected.
- The machine reference point will be set to "0". s.m.p. REFVALUE (P36).
- Once in JOG mode and after positioning the spindle in the right area, start homing the spindle. When done, the CNC will assign a "0" value to this point.
- After moving the spindle to the Machine Reference Zero or up to a known position (with respect to Machine Reference Zero), observe the position reading of the CNC for that point.

  This will be distance from the Machine Reference Zero to that point. Therefore, the value to be assigned to s.m.p. REFVALUE (P36), which defines the coordinate corresponding to the Machine Reference Point (physical location of the marker pulse).

  REFVALUE (P36) = Machine coordinate – CNC reading.

Example:

If the point whose known position is located at 12º mm from Machine Reference Zero and the CNC reads -123.5º as the coordinate value for this point, the coordinate of the Machine Reference Point with respect to Machine Reference Zero will be:

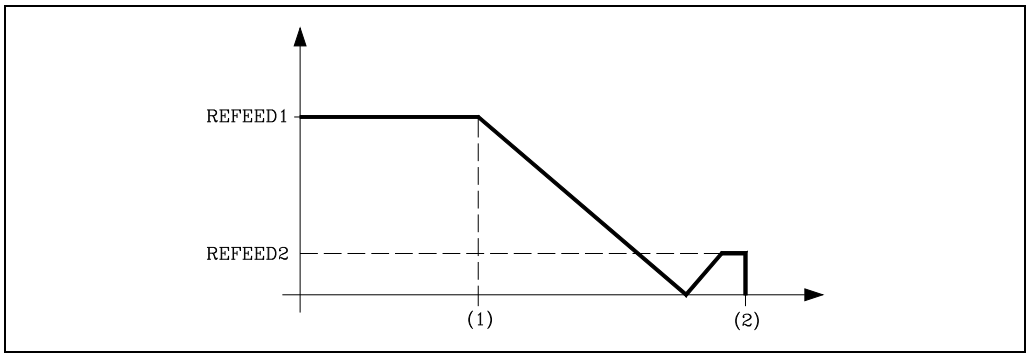
"REFVALUE" P36 = 12 - (-123.5) = 135.5º

- After allocating this new value, press SHIFT + RESET or turn the CNC off and back on in order for the CNC to assume this new value.
- The spindle must be homed again in order for it to assume its right reference values.

Considerations

If at the time when the home search is requested, the spindle is sitting on the home switch, the spindle will back up (in the direction opposite to the one indicated by "REFDIREC (P33) ") until it is off the switch and then, it will go on to search home.

Care must be taken when placing the home switch and when setting feedrates "REFEED1 (P34)" and "REFEED2 (P35)". The home switch (1) will be installed so the marker pulse (2) will be found in the zone corresponding to feedrate "REFEED2" (P35). If there is no room for it, reduce the value of "REFEED1 (P34)". For example, for encoders whose consecutive marker pulses are very close to each other.



When the spindle does not have a machine reference (home) switch (s.m.p. DECINPUT (P31) = NO), the CNC will move the spindle at the feedrate set by s.m.p. REFEED2 (P35) until the first marker pulse from the current position is found, thus ending the home search.

Fagor rotary encoders provide one positive reference pulse per revolution.

Do not mistake the type of pulse provided by the feedback system with the value to be assigned to s.m.p. REFPULSE (P32).

This parameter must indicate the type of active flank (leading or trailing edge), positive or negative of the reference mark (Io) used by the CNC.

5.10 Treatment of emergency signals

The CNC provides the following emergency signals:

/EMERGENCY STOP

Physical emergency input.

It is generated from the outside and corresponds to the physical emergency input.

This signal is active low (0 V).

/EMERGENCY OUTPUT

Physical emergency output.

It is generated internally when an error is detected at the CNC or at the PLC.

This signal is active low (0 V).

/EMERGEN (M5000)

Logic input of the CNC, generated by the PLC.

When the PLC activates this signal, the CNC stops the axis feed and the rotation of the spindle, and it displays the corresponding error message.

This signal is active low (0 V).

/ALARM (M5507)

Logic input of the PLC, generated by the CNC.

The CNC activates this signal to let the PLC "know" that an alarm or emergency condition has occurred.

This signal is active low (0 V).

CNC Treatment of emergency signals

The emergency inputs of the CNC are:

/EMERGEN (M5000)

Physical input coming from the PLC.

/EMERGENCY STOP

Physical input coming from the outside. Pin 10 of connector X2

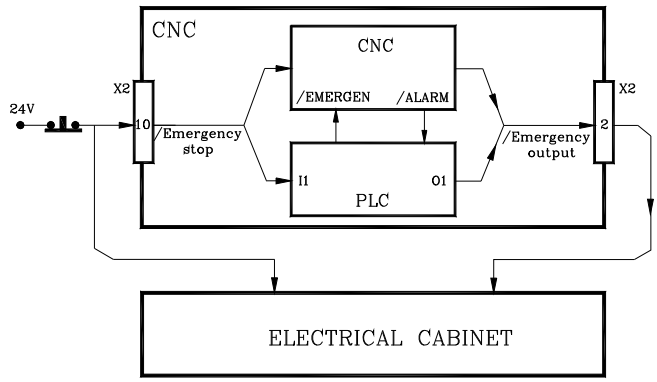
The emergency outputs of the CNC are:

/ALARM (M5507)

Physical output to the PLC.

/EMERGENCY OUTPUT

Physical output to the outside. Pin 2 of connector X2



There are to ways to cause an emergency at the CNC, by activating the physical input / EMERGENCY STOP or the general logic input "/EMERGEN" from the PLC.

Whenever any of these signals is activated, the CNC stops the axes feed and the spindle rotation and it displays the corresponding error message.

By the same token, when the CNC detects an internal malfunction or at an external device, it stops the axes feed and the spindle rotation displaying at the same time the corresponding error message.

In both cases, the CNC will activate the /EMERGENCY OUTPUT and /ALARM signals to indicate to the PLC and to the outside world that an emergency has occurred at the CNC.

Once the cause of the emergency has disappeared, the CNC will deactivate these signals to indicate to the PLC and to the outside world that everything is back to normal.

PLC Treatment of emergency signals

The emergency inputs of the PLC are:

/EMERGENCY STOP

Physical input coming from the outside.

/ALARM (M5507)

Physical input coming from the CNC.

The emergency outputs of the PLC are:

/EMERGENCY OUTPUT

Physical output to the outside.

/EMERGEN (M5000)

Physical output to the CNC.

There are two ways to "tell" the PLC that an emergency condition must be treated, by activating the physical input EMERGENCY STOP of the PLC (which is I1) or the general logic input "/ALARM" of the PLC which is mark M5507.

In both cases, the treatment of these signals will be up to the PLC programmer. The PLC program must have the necessary instructions to properly attend to these emergency inputs and act accordingly.

By the same token, the PLC program must have the necessary instructions to properly activate the emergency outputs when required.

These emergency signals are the physical output /EMERGENCY OUTPUT (output O1 of the PLC) and the general logic output /EMERGEN" which is mark M5000 of the PLC.

It must be borne in mind that every time a new PLC program cycle is initiated, the real inputs are updated with the physical inputs. Therefore, input I1 will have the value of the physical input / EMERGENCY STOP.

Also, before executing the PLC program cycle, the values of the M and R resources corresponding to the CNC logic outputs (internal variables) are updated as well as mark M5507 corresponding to the /ALARM signal.

After the execution of each cycle, the PLC updates the physical outputs with the values of the real outputs except the physical output /EMERGENCY OUTPUT which will be activated whenever the real output O1 or mark M5507 (/ALARM signal coming from the CNC) is active.

5.11 Digital CAN servo

CAN communication requires a drive version V7.02 or later.

G.m.p. CANSPEED (P169) may be used to set the CAN communication speed.

5.11.1 Communications channel

The data exchange between the CNC and the drives takes place in each position loop.

The more data to be transmitted, the more overloaded the transmission will be. These registers should be limited leaving only the ones absolutely necessary after the setup.

On the other hand, there is data that MUST be transmitted at each position loop (velocity commands, feedback, etc.) and other information that could be transmitted in several loops (monitoring, etc.). Since the CNC must know the priority for those transmissions, from now on, we will use the terms "cyclic channel" and "service channel" to refer to each of them.

Cyclic channel (fast channel)

Data transmitted at each position loop (velocity commands, feedback, etc.).

At every loop time, the CNC transmits to the drive through this channel the World Control (Speed Enable, Drive Enable, Homing Enable, bit handshake) and the velocity command. The drive transmits to the CNC the Word Status and the position value. The transmitted data depends on a.m.p. DRIBUSLE (P63).

The type of data to be transmitted (basically variables) must be indicated. The data to be sent to the drives must be placed in certain particular registers of the PLC and the data to be read from the drives is received in other registers of the PLC.

The registers to be used and the data to be transmitted (basically variable) are defined by machine parameters of the PLC. Use SRR700 (P28) through SRR739 (P67) parameters to transmit read-only variables. Use SWR800 (P68) through SWR819 (P87) parameters to transmit write variables.

The number of variables defined in this channel is limited depending on the number of axes, the sampling period and the transmission speed. A data overflow causes an error at the CNC.

Service channel (slow channel)

Data to be transmitted in several position loops (monitoring, etc.).

The service channel can only be accessed through a high-level block of a part-program, a PLC channel or a user channel.

Cyclic channel. Read-only variables for the CNC-PLC

The plc.m.p. SRR700 (P28) through SRR739 (P67) indicate which drive and what type of information will be placed in CNC registers R700 through R739.

P28=>R700 P29=>R701 P30=>R702 P31=>R703 etc.

These parameters are set in 1.5 format. The units digit identifies the drive (node) supplying the data and the decimals indicate the identifier number (see table below).

For example, "P32=1.00040" indicates that PLC register R704 contains the "VelocityFeedback" supplied by the drive located in bus node 1.

To identify the units of the variables, see the drive manual.

Read-only registers R700 through R739 are updated at the beginning of the PLC scan, unless the MRD instruction is used.

The type of information available and its associated identifiers are:

Type of information	Identifier
Class2Diagnostics (Warnings)	00012
Class3Diagnostics (OperationStatus)	00013
VelocityFeedback	00040
PositionFeedbackValue1	00051
TorqueFeedback	00084
CurrentFeedback	33079
FagorDiagnostics	33172
AnalogInputValue	33673
AuxiliaryAnalogInputValue	33674
DigitalInputsValues	33675
PowerFeedback	34468
PowerFeedbackPercentage	34469

The bits of identifier 33172 "FagorDiagnostics" contain the following information:

bits	Meaning	Id at the drive
0,1,2,3	GV25 ActualGearRatio	000255
4,5,6,7	GV21 ActualParameterSet	000254
8	SV4	000330
9	SV5	000331
10	SV3	000332
11	TV10 TGreaterEqualTx	000333
12	TV60 PGreaterEqualPx	000337

Cyclic channel. Write variables for the CNC-PLC

plc.m.p. SWR800 (P68) through SWR819 (P87) indicate which type of information has been put in registers R800 through R819 and which drive will be assigned that value.

P68=>R800 P69=>R801 P70=>R802 P71=>R803 etc.

These parameters are set in 1.5 format. The units digit identifies the drive (node) supplying the data and the decimals indicate the identifier number (see table below).

For example, "P70=2.34178" indicates that the value of PLC register R802 will be assigned to the "DigitalOutputsValues" of the drive located in bus node 2.

To identify the units of the variables, see the drive manual.

The type of information available and its associated identifiers are:

Type of information	Identifier
DA1Value	34176
DA2Value	34177
DigitalOutputsValues	34178
VelocityCommand	00036

The "VelocityCommand" variable can be modified for the axes that have been selected as DRO axes, by a.m.p. DROAXIS (P4) or via PLC by activating the logic CNC axis input "DRO1,2,3,...".

Service channel

The service channel can only be accessed through a high-level block of a part-program, a PLC channel or a user channel. All variables can be accessed except the string type appearing in the drive manual.

- Reading and writing from a part-program or from a user channel.

  Read: (P*** = SVARaxis **)

  Write: (SVARaxis** = P**)

  Example: (P110 = SVARX 40)
  It assigns to parameter P110 the value of the X axis variable with the identifier 40 which corresponds to "VelocityFeedback"

- Reading and writing from the PLC channel.

  Read: ... = CNCEX ((P*** = SVARaxis***), M1)

  Write: ... = CNCEX ((SVARaxis** = P***), M1)

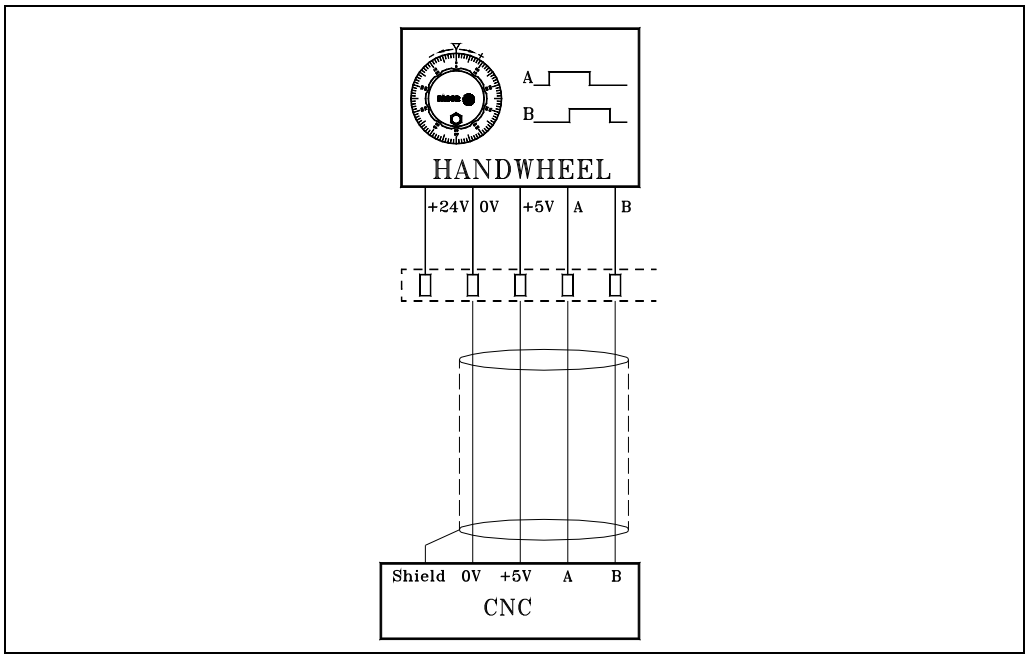
  Example: ... = CNCEX ((SVARX 100= P120),M1
  It assigns the value of parameter P120 to the X axis variable with identifier 100 (VelocityProportionalGain).

5.12 Fagor handwheels: HBA, HBE and LGB

Fagor handwheels HBA, HBE and LGB have:

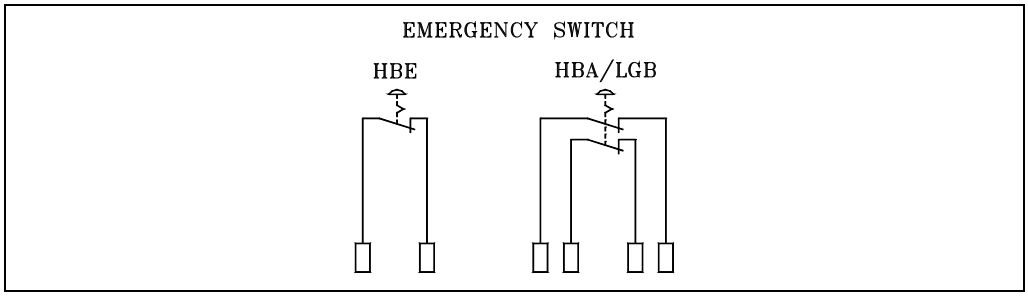
- a pulse generator (encoder).
- an emergency output.
- One or two enable buttons.
- An axis selector switch.
- A resolution selector switch.

The encoder signals must be taken to the specific connectors of the CNC.



In the example, the handwheel signals are taken to the feedback input (connector). Set the corresponding g.m.p. AXIS , for example: AXIS4(P3)=11.

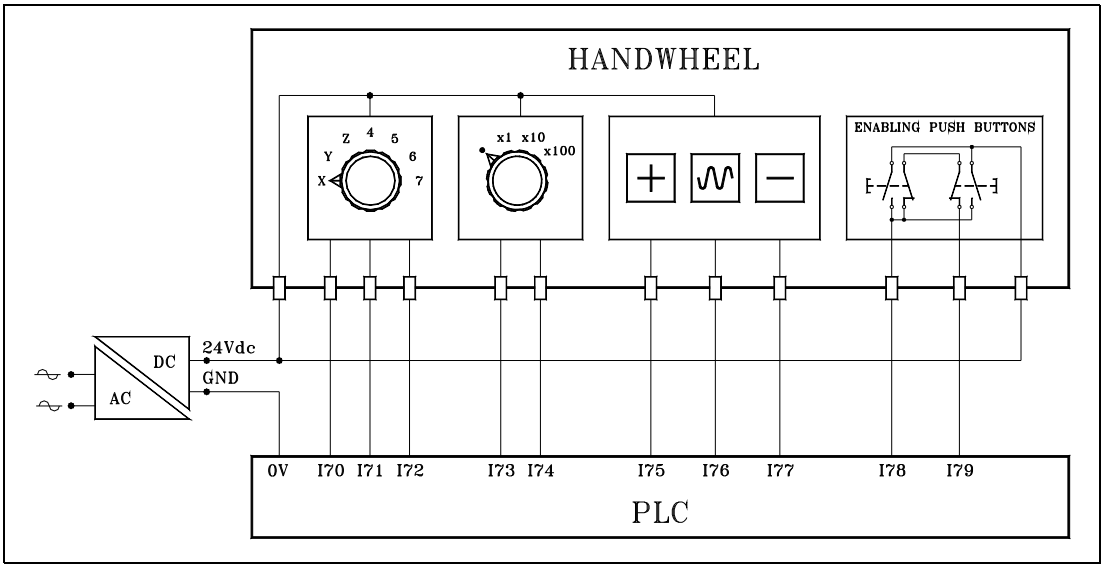
The emergency button must be used in the safety chain of the electrical cabinet.



The HBE handwheel has one contact and the HBA and LGB models have a dual safety contact.

The enable push button (or buttons), the axis selector and resolution selector switches are always handled by the PLC.

Connection example and PLC program for the HBA-072914 handwheel.



There are 2 ways to use the "Enable Push Button".

I78	Just press one of the buttons
I79	Both buttons must be pressed

The example uses input I79, making it necessary to push both buttons in order to use the handwheel.

Definition of symbols (mnemonics).

DEF	HDWON	M600	Handwheel jog
DEF	JOGON	M601	JOG
DEF	XSEL	M602	X axis selected
DEF	YSEL	M603	Y axis selected
DEF	ZSEL	M604	Z axis selected
DEF	4SEL	M605	4th axis selected
DEF	5SEL	M606	5th axis selected
DEF	6SEL	M607	6th axis selected
DEF	7SEL	M608	7th axis selected
PRG			
REA			

If the handwheel enable (I79) and the switch is at handwheel position (x1, x10 or x100)

I79 AND (I73 OR I74) = HDWON

	I73	I74
JOG	0	0
x1	0	1
x10	1	1
x100	1	0

To move the axes in JOG proceed as follows ....

- enable handwheel "I79" ...
- turn the switch to the (·) position: "NOT I73 AND NOT I74"
- position the CNC panel selector in the JOG area (not handwheel, not incremental) "SELECTOR > 7"

  I79 AND NOT I73 AND NOT I74 AND CPS SELECTOR GE 8

  = JOGON

Axis selection. Inputs I70, I71, I72.

NOT I70	AND	NOT I71	AND	NOT I72	= XSEL
NOT I70	AND	NOT I71	AND	I72	= YSEL
NOT I70	AND	I71	AND	I72	= ZSEL
NOT I70	AND	I71	AND	NOT I72	= 4SEL
I70	AND	I71	AND	NOT I72	= 5SEL
I70	AND	I71	AND	I72	= 6SEL
I70	AND	NOT I71	AND	I72	= 7SEL

	I70	I71	I72
XSEL	0	0	0
YSEL	0	0	1
ZSEL	0	1	1
4SEL	0	1	0
5SEL	1	1	0
6SEL	1	1	1
7SEL	1	0	1

If handwheel jog (HDWON), R60 must be ready to store what will be written into the HBEVAR variable. The "a, b, c" bits indicate the x1, x10, x100 factor for each axis and bit 30 (*) must be set to "1" in order for the CNC to read the handwheel pulses.

			C			B			A			W			V			U			Z			Y			X		
	*	^			c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a

() = MOV 0 R60 Delete its contents

Sets the bit (a) of the selected axis to "1". x1 multiplying factor.

HDWON AND XSEL	= MOV 1 R60
HDWON AND YSEL	= MOV 8 R60
HDWON AND ZSEL	= MOV $40 R60
HDWON AND 4SEL	= MOV $200 R60
HDWON AND 5SEL	= MOV $1000 R60
HDWON AND 6SEL	= MOV $8000 R60
HDWON AND 7SEL	= MOV $40000 R60

It then analyzes the multiplying factor indicated at the switch (x1, x10, x100).

I73 AND	I74	= RL1 R60 1 R60
I73 AND	NOT I74	= RL1 R60 2 R60

	I73	I74	c	b	a
x1	0	1	0	0	1
x10	1	1	0	1	0
x100	1	0	1	0	0

And finally, the bit 30 (*) of HBEVAR=1 is enabled, for the CNC to read the handwheel pulses.

( )= OR R60 $40000000 R60

When enabling the handwheel or changing the position of one of the switches, HBEVAR and its image register (R61) are updated (refreshed).

DFU HDWON OR CPS R60 NE R61	= MOV R60 R61
	= CNCWR(R61,HBEVAR,M201)

When disabling the handwheel, HBEVAR=0 and its image register (R61) are initialized.

DFD HDWON = MOV 0 R61 = CNCWR(R61,HBEVAR,M201)

If JOG movement (JOGON) and [+] key pressed: "I75", then axis movement in the positive direction.

JOGON AND I75 AND XSEL = AXIS+1

JOGON AND I75 AND YSEL = AXIS+2

JOGON AND I75 AND ZSEL = AXIS+3

JOGON AND I75 AND 4SEL = AXIS+4

JOGON AND I75 AND 5SEL = AXIS+5

JOGON AND I75 AND 6SEL = AXIS+6

JOGON AND I75 AND 7SEL = AXIS+7

If JOG movement (JOGON) and [-] key pressed: "I77", then axis movement in the negative direction.

JOGON AND I77 AND XSEL = AXIS-1

JOGON AND I77 AND YSEL = AXIS-2

JOGON AND I77 AND ZSEL = AXIS-3

JOGON AND I77 AND 4SEL = AXIS-4

JOGON AND I77 AND 5SEL = AXIS-5

JOGON AND I77 AND 6SEL = AXIS-6

JOGON AND I77 AND 7SEL = AXIS-7

If JOG movement (JOGON) and [Rapid] key pressed: "I76", axis movement in rapid.

JOGON AND I76 = MANRAPID

Safety. When releasing the "Enable Push Button", the STOP command is sent out to the CNC (100 ms pulse) to stop the possible movement active at the time (for example: 10 mm in incremental). Only if the JOG mode is selected and NOT MDI

DFD I79 = TG1 17 100

MANUAL AND NOT MDI AND T17 = NOT /STOP

END

In order to comply with the EN 61000 - 4 - 4 (IEC 1000 - 4 - 4) regulation on "immunity against rapid transients and blasts" use a 7x1x0.14 PVC shielded cable for the 5 V feedback cable.

5.13 Machine safety related functions

5.13.1 Maximum machining spindle speed

The following safety regulation forces to limit the spindle speed on lathes:

"A program will not be executed in machining mode unless the maximum spindle speed value for the part is entered as well as the proper maximum speed for the part holding fixture for the machine.

Should the operator forget to enter or validate these speeds in each program change, the execution in machining mode will not be possible.

It will not exceed the lower speed among the maximum by parameter, the maximum by program and the maximum entered manually.

There is a variable, MDISL, associated with the spindle speed limits to make this routine easier. This variable is read-write from the PLC and read-only from DNC and CNC.

Besides updated by the PLC, this variable can also be updated in the following cases:

- When programming G92 in MDI mode.
- When programming G92 in ISO code in MC or TC mode.
- In MC or TC mode, when a new speed limit is defined in the "SMAX" field.

The speed limits entered via CNC, PLC (PLCSL) and DNC (DNCSL) keep the same functionality and priority and are not affected by the MDISL variable; in other words, the CNC also limits the spindle speed with these variables.

Management via PLC

To comply the safety regulation, we recommend to manage from the PLC the variables associated with speed limit as shown in the following example. It applies the following restrictions:

- A new part-program cannot be executed without previously entering the spindle speed limit. Otherwise, an error message will be issued.

  When repeating the execution of the program, the speed limit needs not be entered, it must only be entered when executing the program for the first time.
- While executing a program, if a new limit is entered in MDI, it replaces the previous one.
- In independent MC or TC cycles it is not required to enter the SMAX because it is already defined in each cycle.
- If the program being executed has a G92 function, the program will only be valid if the value defined in G92 is smaller than the one programmed by MDI.
- When having two main spindles, the speed limit entered will be valid for both.

PLC programming example.

```
PRG
REA
()=CNCRD(OPMODA,R100,M1000)
```

Reading of the OPMODA variable.

```
B0R100 AND INCYCLE = M100
```

Indicator of program in execution.

```
;
DFU M100 = CNCRD(PRGN,R101,M1000) = CNCRD(MDISL,R102,M1000)
```

At the beginning of the execution, it reads the program being executed (CNCRD) and the speed limit set by MDISL.

```
;
M100 = CNCRD(PRGSL,R103,M1000)
```

While executing, it reads the speed limit set by the CNC.

```
M100 AND CPS R101 NE R201 = M101
```

If there is a new program in execution, it activates mark M101.

```
;
M100 AND CPS R101 EQ R201 = M102
```

If it is the same program, it activates mark M102.

```
;
M101 AND CPS R102 EQ 0 = ERR10
```

If there is a new program in execution (M101) and the speed has not been limited with MDISL (R102), it issues error 10. This error must be defined in the PLC messages.

```
;
M101 AND CPS R102 NE 0 = MOV R101 R201 = MOV R102 R202
```

If there is a new program in execution (M101) and the speed has been limited with MDISL (R102), it copies the program number and the speed limit.

```
;
M102 AND CPS R102 NE 0 = MOV R102 R202
```

If the same program is in execution (M102) and the speed is limited again with MDIS (R102), it copies the speed limit.

```
;
M100 AND CPS R202 LT R103 = CNCWR(R202,PLCSL,M1000)
```

If there is a program in execution (M100) and the speed limit with MDISL (R202) is smaller than the limit by CNC (R103), it applies the limit by PLC (value set by MDISL).

```
;
M100 AND CPS R202 GT R103 = CNCWR(R210,PLCSL,M1000)
```

If there is a program in execution (M100) and the speed limit with MDISL (R202) is greater than the limit by CNC (R103), it does not limit the speed by PLC (R210=0).

```
;
DFD M100 = CNCWR(R210,PLCSL,M1000) = CNCWR(R210,MDISL,M1000)
```

After the execution, it cancels the speed limit by PLC and initializes the MDISL variable.

```
;
END
```

5.13.2 Cycle start disabled when hardware errors occur.

If when pressing the [CYCLE-START] key, a hardware error is detected (Axes board error, CAN board error, etc.), the CNC does not allow executing or simulating the program. When a hardware error occurs, the corresponding message is displayed.

5.14 Tool magazine

5.14.1 Tool change via PLC

If the tool change process is interrupted, the values of the tool magazine table and active tool may not reflect the machine's reality.

In order to be able to update the tool table, the tool change may be resumed from the PLC using variables TOOL, NXTOOL, TOD, NXTOD and TMZT. This way, it is possible to resume the tool change from the PLC and redefine the tool table according to their positions using the TMZT variable.

TOOL	Number of the active tool.
TOD	Number of active tool offset.
NXTOOL	Number of next tool. Tool that is selected but is awaiting the execution of M06 to be active.
NXTOD	Number of the next tool's offset.

Variables TOOL, NXTOOL, TOD and NXTOD can only be written from the PLC while no block or part-program is being executed or simulated.

Redefine the tool and tool magazine tables.

To allocate a magazine position to the tool that is considered active by the CNC, but is actually, physically, in the tool magazine, proceed as follows:

1. Deactivate the tool that the CNC considers active; TOOL=0 and TOD=0.
2. Assign to the tool the relevant position using the TMZT variable.

Before trying to write in variables TOOL, NXTOOL, TOD and NXTOD check the OPMODA variable to make sure that no block or part-program is being executed or simulated. The next bits of the OPMODEA variable must be set to ·0·.

Bit 0	Program in execution.
Bit 1	Program in simulation.
Bit 2	Block in execution via MDI, JOG.
Bit 8	Block in execution via CNCEX1.

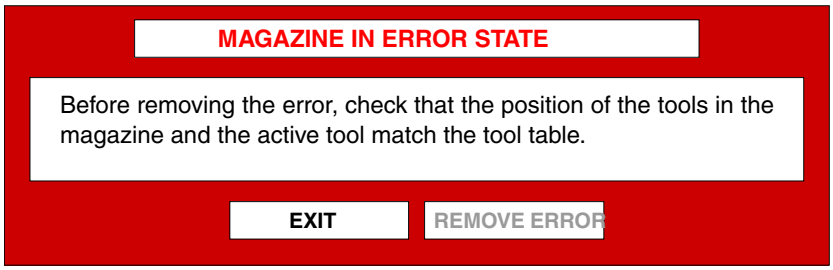
5.14.2 Tool change management

When doing a tool change, the following must be considered:

- The tool change is not validated until function T or M06 is completely and successfully executed. If it is not successfully executed, the magazine table will not be refreshed.
- If anything (a CNC error, PLC error, E-stop button pressed, CNC reset, etc.) happens while executing a T or M6 function, a mark (TMINEM) will be activated setting the CNC in an error state.
- When executing a tool change, O1 must be set =1 in the tool change operation of the PLC program. Otherwise, the CNC will issue the error message "tool magazine in error state".
- If an error is detected during a tool change, the CNC saves this error until it is canceled by a PLC mark (RESTMEM) or using the [REMOVE ERROR] option that appears in the error message.

  If this error is not cancelled as described earlier, it will stay in memory even if the CNC is powered down and back up several times. Even if the magazine error is memorized, the machine can keep on working.

  The CNC will only show this error if the user requests a new tool while the CNC's error state has not been solved. The magazine error only prevents a new tool change.



- During the error state, it is possible to execute any instruction in any mode (Jog, MDI) or even execute a program.

  Only the execution of any T or M6 will not be possible.

  This will be managed only if a tool magazine has been defined.

Example of a PLC program to manage the emergencies in the tool magazine:

```
;
TMINEM                ;Tool magazine manager in emergency state
= MSG100              ;Message to "check magazine and execute M98"
;
DFU TMINEM            ;Tool magazine manager in emergency state
= RES SETTMEM         ;Set the magazine manager in emergency state
;
M_SUBM06              ;Indicates tool changing subroutine. (M06) in
execution
AND NOT TMINEM        ;Tool magazine manager in emergency state
AND (NOT M_POTENCIA   ;Power-on and CNC-PLC OK
OR M_M06ERROR         ;An error occurs when executing M06
OR RESETOUT)          ;CNC reset
= SET SETTMEM         ;Set the magazine manager in emergency state
;
DFU SETTMEM           ;Set the magazine manager in emergency state
OR DFU TMINEM         ;Tool magazine manager in emergency state
= ERA M1007 1010      ;Initialize magazine management related marks
=RES M_SUBM06         ;Tool change subroutine. (M06) in execution
;
M98                   ;Confirm magazine checked with M98
AND TMINEM            ;Tool magazine manager in emergency state
= SET RESTMEM         ;Reset the emergency request to the manager
;
```

5.15 Gear ratio management on axes and spindle

The gear ratios on axes and spindle, depending on whether they are analog or CAN, are managed as follows:

CAN

If axis parameters INPREV (P87) and OUTPREV (P88) are 0, they will be assumed as if they were 1. There is no need to set a.m.p. PITCH (P7), except in the following case:

If axis parameters DRIBUSLE (P63) = 0 and a.m.p. INPREV (P87) and OUTPREV (P88) are 0, it will consider a.m.p. PITCH (P7) (it will not be ignored).

The gear ratio for an axis is set as follows:

- PITCHB (P86) = Leadscrew pitch.
- INPREV (P87) = Input revolutions.
- OUTPREV (P88) = Output revolutions.

Analog

The gear ratio for an axis is set as follows:

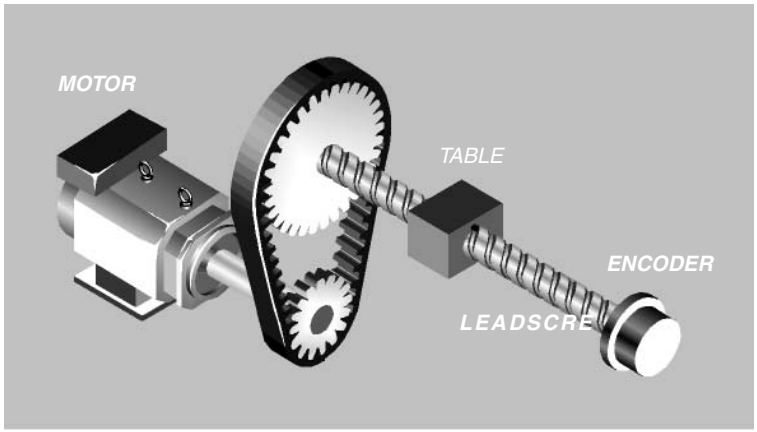
- If axis parameters PITCHB (P86), INPREV (P87) and OUTPREV (P88) are 0, the gear ratios are set as follows:
  - PITCH (P7) = Leadscrew pitch.
  - If there are gear ratios, PITCH (P7) = (leadscrew pitch x OUTPREV) / INPREV.
- If axis parameters PITCHB (P86), INPREV (P87) and OUTPREV (P88) are other than 0, the CNC will assume these values and will not issue an error.
- If the value of any of these parameters is other than 0, the CNC shows a message indicating that the parameters are wrong. In this case, in JOG mode or in execution, it displays an error and it will not be possible to move the machine.



In any configuration where the values of INPREV or OUTPREV are indivisible, the home signal will be generated from the home switch (DECEL) on.*

5.15.1 Example of axes: outside feedback without gear box

In this case, on linear axes, the encoder is mounted directly to the leadscrew and on rotary axes is mounted directly to the rotation center. If it is a rotary axis, the leadscrew pitch will be 360.



If the maximum feedrate for an axis is 20 m/min, with a leadscrew pitch of 20 and a 3-to-1 gear ratio between the motor and the leadscrew. The encoder is an HOP model (Vpp) and provides 18,000 pulses per turn If a linear encoder is a FAGOR GOX model with a glass/steel-tape grading pitch of 20µ and the actual TTL pitch is 4µ.

CAN axes

1. External feedback connected to the CNC

- A.m.p. DRIBUSLE (P63) = 0.

Parameters involved in the calculation of velocity command:

- A.m.p. G00FEED (P38) = Maximum feedrate of the axis = 20000.
- A.m.p. PITCHB (P86) = Leadscrew pitch = 20.
- Motor gear ratio:

  A.m.p. INPREV (P87) = Input revolutions =3.

  A.m.p. OUTPREV (P88) = Output revolutions = 1.
- NP121 (drive) = it automatically loads the value of a.m.p. INPREV (P87) of the CNC.
- NP122 (drive) = it automatically loads the value of a.m.p. OUTPREV (P88) of the CNC.
- NP123 (drive) = it automatically loads the value of a.m.p. PITCHB (P86) of the CNC.

Parameters involved in the calculation of position feedback.

With encoder:

- A.m.p. NPULSES (P8) = number of pulses per turn of the encoder = 18000.
- A.m.p. SINMAGNI (P10) = Multiplying factor if the encoder is sinusoidal = 1.
- A.m.p. EXTMULT (P57) = Multiplying factor of the feedback = 1.
- A.m.p. PITCHB (P86) = Leadscrew pitch = 20.

With linear encoder:

- A.m.p. PITCH (P7) = linear encoder pitch = 20.
- A.m.p. NPULSES (P8) = 0.
- A.m.p. SINMAGNI (P10) = Multiplying factor if the encoder is sinusoidal = 0.
- A.m.p. EXTMULT (P57) = Multiplying factor of the feedback = 20μ/4μ = 5.

Calculation of the velocity command for a G00FEED feedrate:

Velocity command = (G00FEED x INPREV) / (PITCHB x OUTPREV) = (20000 x 3) / (20 x 1) = 3000 rpm.

Calculation of the resulting resolution:

TTL encoder: Resolution = PITCHB / (4 x NPULSES)

Sinusoidal encoder: Resolution = PITCHB / (SINMAGNI x NPULSES)

TTL linear encoder: Resolution = PITCH / 4

Sinusoidal linear encoder:Resolution = PITCH / SINMAGNI

Analog axes

1. External feedback connected to the CNC

- A.m.p. DRIBUSLE (P63) = 0.

Parameters involved in the calculation of velocity command:

- A.m.p. G00FEED (P38) = Maximum feedrate of the axis = 20000.

Parameters involved in the calculation of position feedback.

With encoder:

- A.m.p. NPULSES (P8) = number of pulses per turn of the encoder = 18000.
- A.m.p. SINMAGNI (P10) = Multiplying factor if the encoder is sinusoidal = 1.
- A.m.p. EXTMULT (P57) = Multiplying factor of the feedback = 1.
- A.m.p. PITCHB (P86) = Leadscrew pitch = 20.
- Motor gear ratio:

  A.m.p. INPREV (P87) = Input revolutions =3.

  A.m.p. OUTPREV (P88) = Output revolutions = 1.

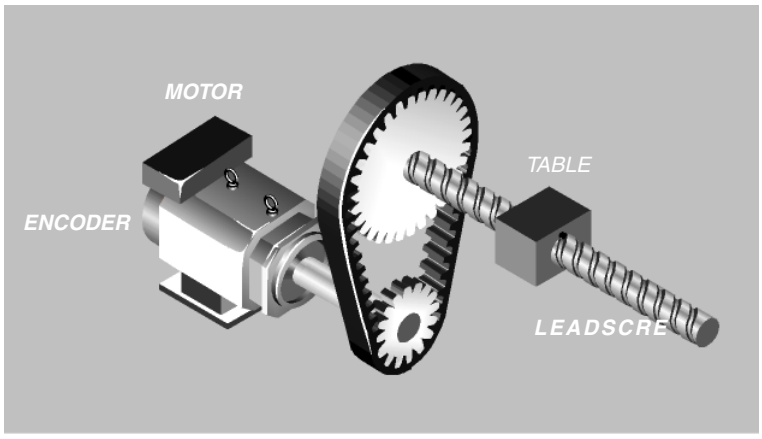
With linear encoder:

- A.m.p. PITCH (P7) = linear encoder pitch = 20.
- A.m.p. NPULSES (P8) = 0.
- A.m.p. SINMAGNI (P10) = Multiplying factor if the encoder is sinusoidal = 0.
- A.m.p. EXTMULT (P57) = Multiplying factor of the feedback = 20μ/4μ = 5.

Calculation of the motor speed with a velocity command of MAXVOLT for a G00FEED feedrate:

Motor speed = (G00FEED x INPREV) / (PITCHB x OUTPREV) = (20000 x 3) / (20 x 1) = 3000 rpm.

5.15.2 Example of axes: encoder at the motor



If the maximum feedrate for an axis is 20 m/min, with a leadscrew pitch of 20 and a 3-to-1 gear ratio between the motor and the leadscrew. The motor encoder provides 2500 pulses per turn.

CAN axes

- A.m.p. DRIBUSLE (P63) = 1.

Parameters involved in the calculation of velocity command:

- A.m.p. G00FEED (P38) = Maximum feedrate of the axis = 20000.
- A.m.p. PITCHB (P86) = Leadscrew pitch = 20.
- Motor gear ratio:
  - A.m.p. INPREV (P87) = Input revolutions =3.
  - A.m.p. OUTPREV (P88) = Output revolutions = 1.
- NP121 (drive) = it automatically loads the value of a.m.p. INPREV (P87) of the CNC.
- NP122 (drive) = it automatically loads the value of a.m.p. OUTPREV (P88) of the CNC.
- NP123 (drive) = it automatically loads the value of a.m.p. PITCHB (P86) of the CNC.

Calculation of the maximum speed of the motor for a G00FEED feedrate:

Maximum motor speed = (G00FEED x INPREV) / (PITCHB x OUTPREV)

= (20000 x 3) / (20 x 1) = 3000 rpm.

Analog axes

Parameters involved in the calculation of velocity command:

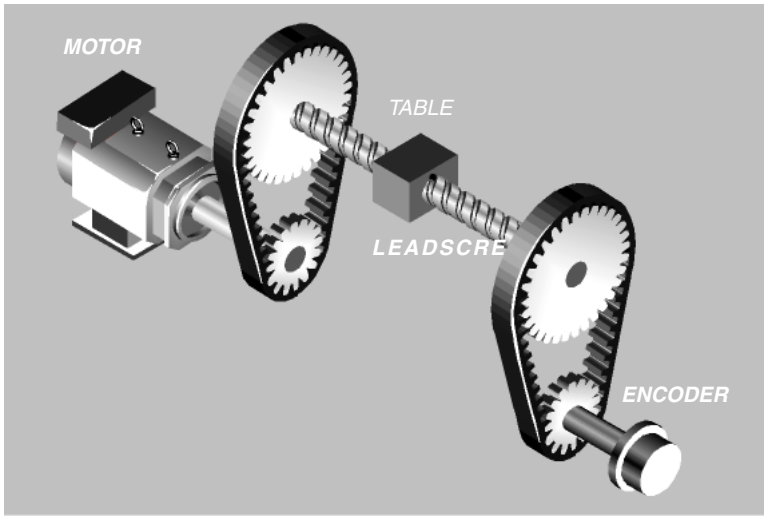
- A.m.p. G00FEED (P38) = Maximum feedrate of the axis = 20000.

Parameters involved in the calculation of position feedback.

- A.m.p. NPULSES (P8) = number of pulses per turn of the encoder = 2500.
- A.m.p. PITCHB (P86) = Leadscrew pitch = 20.
- Motor gear ratio:
  - A.m.p. INPREV (P87) = Input revolutions =3.
  - A.m.p. OUTPREV (P88) = Output revolutions = 1.

5.15.3 Example of axes: outside feedback with gear box

In this case, on linear axes, the encoder is mounted to the leadscrew through a gear box and on rotary axes is mounted to the rotation center through a gear box.



If the maximum feedrate for an axis is 20 m/min, with a leadscrew pitch of 20 and a 3-to-1 gear ratio between the motor and the leadscrew. The encoder is an HOP model (Vpp) and provides 18,000 pulses per turn and a 2-to-3 gear ratio.

CAN axes

1. External feedback connected to the CNC

- A.m.p. DRIBUSLE (P63) = 0.

Parameters involved in the calculation of velocity command:

- A.m.p. G00FEED (P38) = Maximum feedrate of the axis = 20000.
- A.m.p. PITCHB (P86) = Leadscrew pitch = 20.
- Motor gear ratio:
  - A.m.p. INPREV (P87) = Input revolutions =3.
  - A.m.p. OUTPREV (P88) = Output revolutions = 1.
- NP121 (drive) = it automatically loads the value of a.m.p. INPREV (P87) of the CNC.
- NP122 (drive) = it automatically loads the value of a.m.p. OUTPREV (P88) of the CNC.
- NP123 (drive) = it automatically loads the value of a.m.p. PITCHB (P86) of the CNC.

Parameters involved in the calculation of position feedback.

- A.m.p. NPULSES (P8) = number of pulses per turn of the encoder
  = 18000 / (3 / 2) = 12000. (Only integers are allowed).
- A.m.p. SINMAGNI (P10) = Multiplying factor if the encoder is sinusoidal = 1.
- A.m.p. EXTMULT (P57) = Multiplying factor of the feedback = 1.
- A.m.p. PITCHB (P86) = Leadscrew pitch = 20.

Analog axes

1. External feedback connected to the CNC

- A.m.p. DRIBUSLE (P63) = 0.

Parameters involved in the calculation of velocity command:

- A.m.p. G00FEED (P38) = Maximum feedrate of the axis = 20000.

Parameters involved in the calculation of position feedback.

- A.m.p. NPULSES (P8) = number of pulses per turn of the encoder = 18000.
- A.m.p. SINMAGNI (P10) = Multiplying factor if the encoder is sinusoidal = 1.
- A.m.p. EXTMULT (P57) = Multiplying factor of the feedback = 1.
- A.m.p. PITCHB (P86) = Leadscrew pitch = 20.
- Motor gear ratio:

  A.m.p. INPREV (P87) = Input revolutions =3.

  A.m.p. OUTPREV (P88) = Output revolutions = 1.

5.15.4 Example of spindle: external encoder without gear box

A spindle with 4 gears. The maximum speeds and gear ratios for each range (gear) are:

Gear 1: maximum speed 1000 rpm, gear ratio 4:1.

Gear 2: maximum speed 2000 rpm, gear ratio 2:1.

Gear 3: maximum speed 3000 rpm, gear ratio 4:3.

Gear 4: maximum speed 3500 rpm, gear ratio 1:1.

The encoder is an HOP model (Vpp) and provides 18,000 pulses per turn

CAN spindle

1. External encoder connected to the CNC

- A.m.p. DRIBUSLE (P63) = 0.

Parameters involved in the calculation of velocity command:

- S.m.p. MAXGEAR1 (P2) = maximum rpm of the first gear = 1000.
- S.m.p. MAXGEAR2 (P3) = maximum rpm of the second gear = 2000.
- S.m.p. MAXGEAR3 (P4) = maximum rpm of the third gear = 3000.
- S.m.p. MAXGEAR4 (P5) = maximum rpm of the fourth gear = 3500.
- A.m.p. INPREV1 (P72) = Input revolutions of the first gear =4.
- A.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- A.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
- A.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- A.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- A.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- A.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- A.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.

Parameters involved in the calculation of position feedback.

- A.m.p. NPULSES (P13) = number of pulses per turn of the encoder = 18000.
- A.m.p. SINMAGNI (P65) = Multiplying factor if the encoder is sinusoidal = 1.

Analog spindle

Parameters involved in the calculation of velocity command:

- S.m.p. MAXGEAR1 (P2) = maximum rpm of the first gear = 1000.
- S.m.p. MAXGEAR2 (P3) = maximum rpm of the second gear = 2000.
- S.m.p. MAXGEAR3 (P4) = maximum rpm of the third gear = 3000.
- S.m.p. MAXGEAR4 (P5) = maximum rpm of the fourth gear = 3500.
- A.m.p. INPREV1 (P72) = Input revolutions of the first gear =1.
- A.m.p. INPREV2 (P74) = Input revolutions of the second gear =1.
- A.m.p. INPREV3 (P76) = Input revolutions of the third gear =1.
- A.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- A.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- A.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- A.m.p. OUTPREV3 (P77) = output revolutions of the third gear =1.
- A.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.
- S.m.p. MAXVOLT1 (P37) = maximum velocity command for the first gear = 9500.
- S.m.p. MAXVOLT2 (P38) = maximum velocity command for the second gear = 9500.
- S.m.p. MAXVOLT3 (P39) = maximum velocity command for the third gear = 9500.
- S.m.p. MAXVOLT4 (P40) = maximum velocity command for the fourth gear.
  = 9500 x 3500 rpm / 4000 rpm = 8312.

Parameters involved in the calculation of position feedback.

- A.m.p. NPULSES (P13) = number of pulses per turn of the encoder = 18000.
- A.m.p. SINMAGNI (P65) = Multiplying factor if the encoder is sinusoidal = 1.

Calculation of motor speed for the MAXVOLT of each range (gear):

Motor speed = MAXGEAR x INPREV / OUTPREV

Motor speed with MAXVOLT1 = 1000 x 4 / 1 = 4000 rpm.

Motor speed with MAXVOLT2 = 2000 x 2 / 1 = 4000 rpm.

Motor speed with MAXVOLT3 = 3000 x 4 / 3 = 4000 rpm.

Motor speed with MAXVOLT4 = 3500 x 1 / 1 = 3500 rpm.

5.15.5 Example of spindle: encoder at the motor

A spindle with 4 gears. The maximum speeds and gear ratios for each range (gear) are:

Gear 1: maximum speed 1000 rpm, gear ratio 4:1.

Gear 2: maximum speed 2000 rpm, gear ratio 2:1.

Gear 3: maximum speed 3000 rpm, gear ratio 4:3.

Gear 4: maximum speed 3500 rpm, gear ratio 1:1.

The encoder is an HOP model (Vpp) and provides 18,000 pulses per turn

CAN spindle

- A.m.p. DRIBUSLE (P63) = 1.

Parameters involved in the calculation of velocity command:

- S.m.p. MAXGEAR1 (P2) = maximum rpm of the first gear = 1000.
- S.m.p. MAXGEAR2 (P3) = maximum rpm of the second gear = 2000.
- S.m.p. MAXGEAR3 (P4) = maximum rpm of the third gear = 3000.
- S.m.p. MAXGEAR4 (P5) = maximum rpm of the fourth gear = 3500.
- A.m.p. INPREV1 (P72) = Input revolutions of the first gear =4.
- A.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- A.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
- A.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- A.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- A.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- A.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- A.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.

Parameters involved in the calculation of position feedback.

- A.m.p. INPREV1 (P72) = Input revolutions of the first gear =4.
- A.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- A.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
- A.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- A.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- A.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- A.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- A.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.

Analog spindle

Parameters involved in the calculation of velocity command:

- S.m.p. MAXGEAR1 (P2) = maximum rpm of the first gear = 1000.
- S.m.p. MAXGEAR2 (P3) = maximum rpm of the second gear = 2000.
- S.m.p. MAXGEAR3 (P4) = maximum rpm of the third gear = 3000.
- S.m.p. MAXGEAR4 (P5) = maximum rpm of the fourth gear = 3500.
- S.m.p. MAXVOLT1 (P37) = maximum velocity command for the first gear = 9500.
- S.m.p. MAXVOLT2 (P38) = maximum velocity command for the second gear = 9500.
- S.m.p. MAXVOLT3 (P39) = maximum velocity command for the third gear = 9500.
- S.m.p. MAXVOLT4 (P40) = maximum velocity command for the fourth gear.
  = 9500 x 3500 rpm / 4000 rpm = 8312.

Parameters involved in the calculation of position feedback.

- A.m.p. NPULSES (P13) = number of pulses per turn of the encoder = 18000.
- A.m.p. SINMAGNI (P65) = Multiplying factor if the encoder is sinusoidal = 1.
- A.m.p. INPREV1 (P72) = Input revolutions of the first gear =4.
- A.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- A.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
- A.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- A.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- A.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- A.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- A.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.

Calculation of motor speed for the MAXVOLT of each range (gear):

Motor speed = MAXGEAR x INPREV / OUTPREV

Motor speed with MAXVOLT1 = 1000 x 4 / 1 = 4000 rpm.

Motor speed with MAXVOLT2 = 2000 x 2 / 1 = 4000 rpm.

Motor speed with MAXVOLT3 = 3000 x 4 / 3 = 4000 rpm.

Motor speed with MAXVOLT4 = 3500 x 1 / 1 = 3500 rpm.

5.15.6 Example of spindle: external encoder with gear box

A spindle with 4 gears. The maximum speeds and gear ratios for each range (gear) are:

Gear 1: maximum speed 1000 rpm, gear ratio 4:1.

Gear 2: maximum speed 2000 rpm, gear ratio 2:1.

Gear 3: maximum speed 3000 rpm, gear ratio 4:3.

Gear 4: maximum speed 3500 rpm, gear ratio 1:1.

The encoder is an HOP model (Vpp) and provides 18,000 pulses per turn and a 2-to-3 gear ratio.

CAN spindle

1. External encoder connected to the CNC

- A.m.p. DRIBUSLE (P63) = 0.

Parameters involved in the calculation of velocity command:

- S.m.p. MAXGEAR1 (P2) = maximum rpm of the first gear = 1000.
- S.m.p. MAXGEAR2 (P3) = maximum rpm of the second gear = 2000.
- S.m.p. MAXGEAR3 (P4) = maximum rpm of the third gear = 3000.
- S.m.p. MAXGEAR4 (P5) = maximum rpm of the fourth gear = 3500.
- A.m.p. INPREV1 (P72) = Input revolutions of the first gear =4.
- A.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- A.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
- A.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- A.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- A.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- A.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- A.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.

Parameters involved in the calculation of position feedback.

- S.m.p. NPULSES (P13) = number of pulses per turn of the encoder
  = 18000 / (3 / 2) = 12000. (Only integers are allowed).
- S.m.p. SINMAGNI (P65) = Multiplying factor if the encoder is sinusoidal = 1.

Analog spindle

Parameters involved in the calculation of velocity command:

- S.m.p. MAXGEAR1 (P2) = maximum rpm of the first gear = 1000.
- S.m.p. MAXGEAR2 (P3) = maximum rpm of the second gear = 2000.
- S.m.p. MAXGEAR3 (P4) = maximum rpm of the third gear = 3000.
- S.m.p. MAXGEAR4 (P5) = maximum rpm of the fourth gear = 3500.
- S.m.p. MAXVOLT1 (P37) = maximum velocity command for the first gear = 9500.
- S.m.p. MAXVOLT2 (P38) = maximum velocity command for the second gear = 9500.
- S.m.p. MAXVOLT3 (P39) = maximum velocity command for the third gear = 9500.
- S.m.p. MAXVOLT4 (P40) = maximum velocity command for the fourth gear.
  = 9500 x 3500 rpm / 4000 rpm = 8312.

Parameters involved in the calculation of position feedback.

- S.m.p. NPULSES (P13) = number of pulses per turn of the encoder = 18000.
- S.m.p. SINMAGNI (P65) = Multiplying factor if the encoder is sinusoidal = 1.
- A.m.p. INPREV1 (P72) = Input revolutions of the first gear =2.
- A.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- A.m.p. INPREV3 (P76) = Input revolutions of the third gear =2.
- A.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =2.
- A.m.p. OUTPREV1 (P73) = output revolutions of the first gear =3.
- A.m.p. OUTPREV2 (P75) = output revolutions of the second gear =3.
- A.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- A.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =3.

Calculation of motor speed for the MAXVOLT of each range (gear):

Motor speed = MAXGEAR x INPREV / OUTPREV

Motor speed with MAXVOLT1 = 1000 x 4 / 1 = 4000 rpm.

Motor speed with MAXVOLT2 = 2000 x 2 / 1 = 4000 rpm.

Motor speed with MAXVOLT3 = 3000 x 4 / 3 = 4000 rpm.

Motor speed with MAXVOLT4 = 3500 x 1 / 1 = 3500 rpm.

5.

CONCEPTS

Gear ratio management on axes and spindle

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

PLC RESOURCES

6

6.1 Inputs

They 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 the input number which is desired to reference, for example I1, I25, I102, etc.

The PLC may control 512 inputs although when communicating with the outside world it can only access the physical ones.

Local physical inputs are the ones corresponding to the central unit.

6.2 Outputs

They 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 the output number which is desired to reference, for example O1, O25, O102, etc.

The PLC may control 512 outputs although when communicating with the outside world it can only access the physical ones.

Local physical outputs are the ones corresponding to the central unit.

Output O1 coincides with the emergency output of the CNC (connector); thus, it must be kept high (logic level 1).

6.3 Marks

These are elements capable of memorizing in one bit (as if they were an internal relay) information defined by the user, their value being inalterable even when the power supply to the system is turned off.

This will be programmed by the letter M followed by the number of the mark which it is wished to reference, for example, M1, M25, M102, etc.

The PLC controls the following marks:

User marks	M1 - M2000
Arithmetic flag marks	M2003
Clock marks	M2009 - M2024
Fixed status marks	M2046 and M2047
Marks associated with messages	M4000 - M4127
Marks associated with errors	M4500 - M4563
Screen marks	M4700 - M4955
CNC communication marks	M5000 - M5957

Marks M1 thru M2047 have image values unlike the remainder of the marks, and so the PLC will always work with their real values.

The arithmetic flag mark available at the PLC is:

M2003 Is the Zero flag and is set to 1 (high logic level) when the result of an AND, OR, XOR operation is 0.

The clock marks M2009 to M2024, make up internal clocks of different periods which can be used by the user.

The following table shows the available marks and the average period of each one.

M2009	100 ms.
M2010	200 ms.
M2011	400 ms.
M2012	800 ms.
M2013	1.6 s.
M2014	3,2 s.

M2015	6,4 s.
M2016	12,8 s.
M2017	1 s.
M2018	2 s.
M2019	4 s.
M2020	8 s.

M2021	16 s.
M2022	32 s.
M2023	64 s.
M2024	128 s.

The fixed status marks available at the PLC are:

M2046 Always has a value of 0.

M2047 Always has a value of 1.

The PLC allows, by means of the activation of a series of message marks, the PLC message corresponding to the PLC message table to be displayed on the CNC screen. They can be named by means of the mark M4000 - M4254 or by means of their associated mnemonic MSG1 - MSG255:

M4000	M4001	M4002	--------	M4253	M4254
MSG1	MSG2	MSG3	--------	MSG254	MSG255

Likewise, 128 error marks are available which allow the error corresponding to the PLC error table to be displayed on the CNC screen as well as to interrupt the execution of the CNC program, stopping axis feed and spindle rotation. Activating any of these marks does not activate the external CNC emergency output.

They can be named by means of mark M4500-M4627 or by means of their associated mnemonic ERR1 - ERR128:

M4500	M4501	M4502	--------	M4626	M4627
ERR1	ERR2	ERR3	--------	ERR127	ERR128

Because the PLC program is not interrupted by these marks, it is advised to make it possible to change their status via accessible external inputs; otherwise, the CNC will keep receiving the same error at every PLC scan (cycle) thus preventing access to any PLC mode.

CNC user screens 0 through 255 may be activated by actvating each mark M4700 through M4955 They can be named by means of mark M4700-M4955 or by means of their associated mnemonic PIC0 - PIC255:

M4700	M4701	M4702	--------	M4954	M4955
PIC0	PIC1	PIC2	--------	PIC254	PIC255

The PLC has marks M5000 through M5957 to exchange information with the CNC, all of which have associated mnemonics. See chapter *"10 Logic CNC inputs and outputs"*.

6.4 Registers

These are elements which store a numerical value in 32 bits, their value remaining unalterable even when the power supply to the system is cut off.

They do not have image values and are represented by the letter R, followed by the register number it is desired to reference, for example R1, R25, R102, etc.

The PLC has the following registers:

User registers	R1-R499
Registers for communication with the CNC	R500-R559

The PLC will consider each value stored in each register as an integer with a sign, and can be within ±2147483647.

It is also possible to make reference to a BIT of the REGISTER by putting the letter B and the bit number (0/31) in front of the selected register. For example:

B7R155 Refers to bit 7 of register 155.

The PLC considers bit 0 as being the one with least significance and bit 31 as being the one with most significance.

The value stored in a register can be treated as being decimal, hexadecimal (preceded by “$”), binary (preceded by “B”) or in BCD. Example:

decimal	156
Hexadecimal	$9C
Binary	B0000 0000 0000 0000 0000 0000 1001 1100

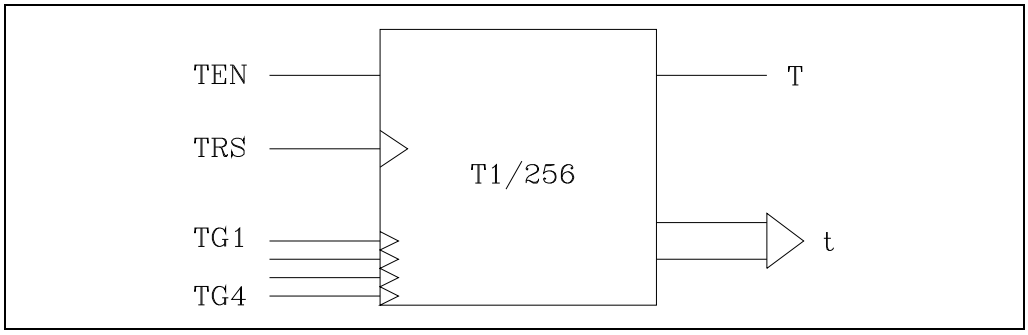
6.5 Timers

These are elements capable of maintaining their output at a determined logic level during a preset time (time constant), after which the output changes status.

They do not have image values and are represented by the letter T, followed by the number of the timer it is required to reference, for example, T1, T25, T102, etc.

The time constant is stored in a 32-bit variable, and so its value can be between 0 and 4294967295 milliseconds, which is equivalent to 1193 hours (almost 50 days).

The PLC has 256 timers, each of which has T status output and TEN, TRS, TG1, TG2, TG3 and TG4 inputs. It is also possible to consult at any moment the time which has elapsed from the moment it was activated.

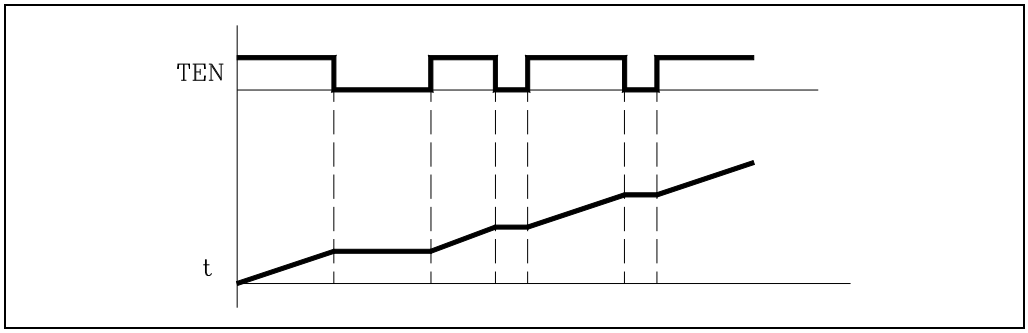


Enable input (TEN)

This input allows the timing of the timer to be stopped. It is referred to by the letter TEN followed by the number of the timer which is wished to reference, for example TEN 1, TEN 25, TEN 102, etc.

So that the time elapses within the timer this input must be at level “1”. By default and every time a timer is activated the PLC will assign this input a logic level of “1”.

If once the timer has been activated, TEN = 0, the PLC interrupts the timing and TEN must be set to "1" to resume timing.



Example:

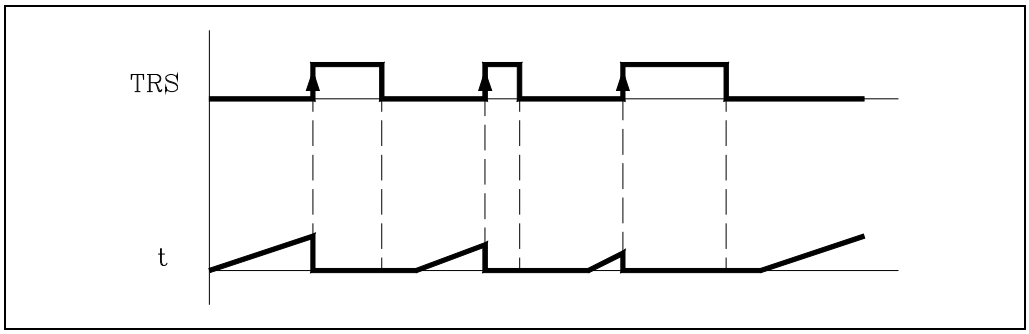
I2 = TEN 10 Input I2 controls the Enable input of timer T10.

Reset input (TRS)

This input allows the timer to be initialized, by assigning the value 0 to its T status and by canceling its count (it initializes this to 0). It is referred to by the letters TRS followed by the timer number it is wished to reference, for example TRS 1, TRS 25, TRS 102, etc.

This initialization of the timer will be made when a transition of logic level from “0” to “1” (leading edge) is produced. By default and every time a timer is activated the PLC will assign this input a logic level of “0”.

If, once the timer is activated, a leading edge is produced at the TRS input, the PLC initializes the timer, assigning value 0 to its T status and cancelling the count (it initializes this to 0). The timer is deactivated and its trigger input must be activated to turn the timer back on.



Example:

I3 = TRS 10 Input I3 controls the Reset input of timer T10.

Trigger input (TG1, TG2, TG3, TG4)

These inputs allow the timer to be activated, and it begins to time. They are referred to by the letters TG1, TG2, TG3, TG4 followed by the number of the timer it is required to reference and the value which is required to start the count with (time constant).

For example TG1 1 100, TG2 25 224, TG3 102 0, TG4 200 500, etc.

The time constant value is defined in thousandths of a second, and it is possible to indicate this by means of a numerical value or by assigning it the internal value of an R register.

TG1 20 100	Activates timer T20 by means of trigger input TG1 and with a time constant of 100 milliseconds.
TG2 22 R200	Activates timer T22 by means of trigger input TG2 and with a time constant which will be defined (in thousandths of a second) by the value of Register R200 when the instruction is executed.

Inputs TG1, TG2, TG3 and TG4 are used to activate the timer in four different operating modes:

- TG1 input in MONOSTABLE mode
- TG2 input in DELAYED CONNECTION mode
- TG3 input in DELAYED DISCONNECTION mode
- TG4 input in SIGNAL LIMITING mode

This activation of the timer is made when a logic level transition of any of these inputs is produced, either from “0” to “1” or from “1” to “0” (leading or trailing edge) depending on the chosen input. By default and every time the timer is initialized by means of the reset input (TRS), the PLC will assign logic level “0” to these inputs.

The operating mode of each of these trigger inputs is explained individually.

Status output (T)

This output indicates the logic status of the timer. It is referred to by the letter "T" followed by the timer number. For example: T1, T25, T102, etc.

The logic status of the timer depends on the operating mode selected by means of the trigger inputs TG1, TG2, TG3 and TG4, and so the activation or deactivation of this signal is explained in each of the PLC operating modes.

Elapsed time (T)

This output indicates the time elapsed in the timer since the moment it was activated. It is referred to by the letter "T" followed by the timer number. For example: T1, T25, T102, etc.

Although when written as T123 it coincides with the status output, both are different and they are also used in different types of instruction.

In binary type instructions, function T123 makes reference to the logic status of the timer.

T123 = M100 Assigns mark to M100 the status (0/1) of Timer 123.

In arithmetic and comparison functions T123 makes reference to the time elapsed in the timer from the moment it was activated.

```
I2 = MOV T123 R200
```

Transfers the time of T123 to register R200.

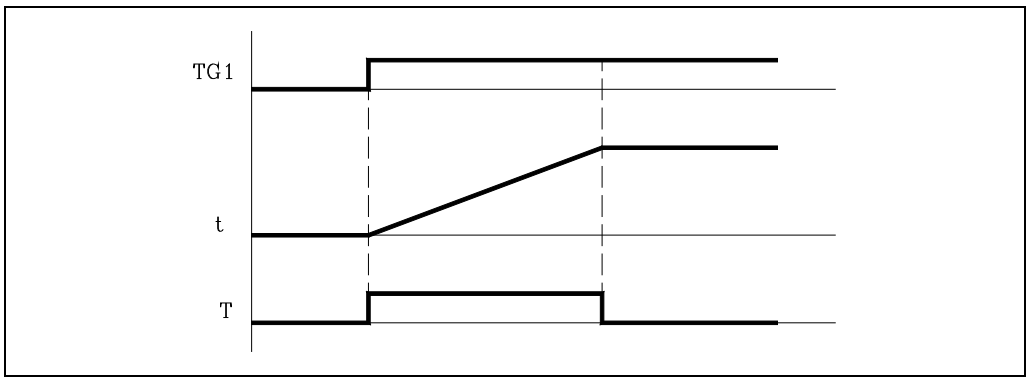
```
CPS T123 GT 1000 = M100
```

Compares the time elapsed at T123 is greater than 1000. If so, it activates mark M100.

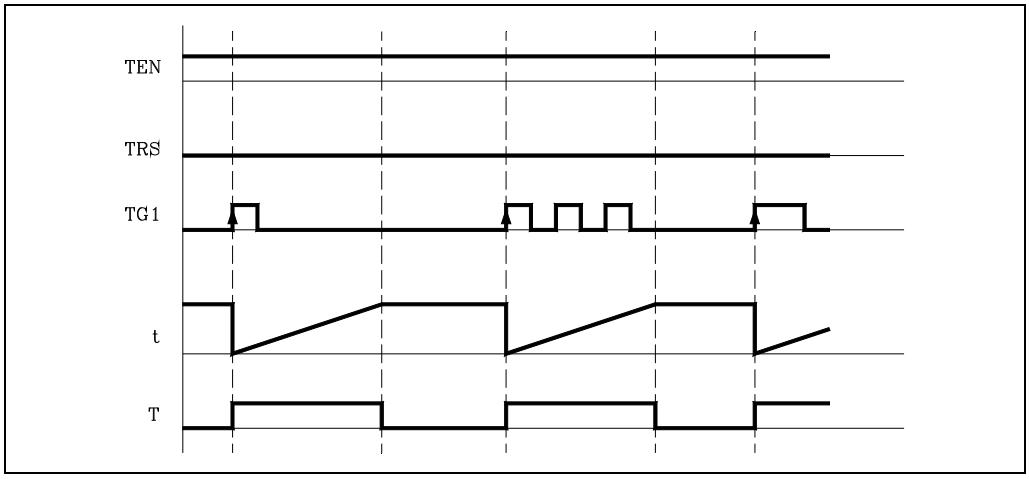
The PLC has a 32-bit variable to store the time of each timer.

6.5.1 Monostable mode. TG1 input

In this operational mode the timer status is kept at the high logic level (T=1) from the moment the TG1 input is activated until the time indicated by the time constant elapses.



If the timer is initialized with values TEN=1 and TRS=0, the timer will be activated when a leading edge is produced at input TG1. At that moment, the timer status output (T) changes states (T=1) and the timing starts from "0".



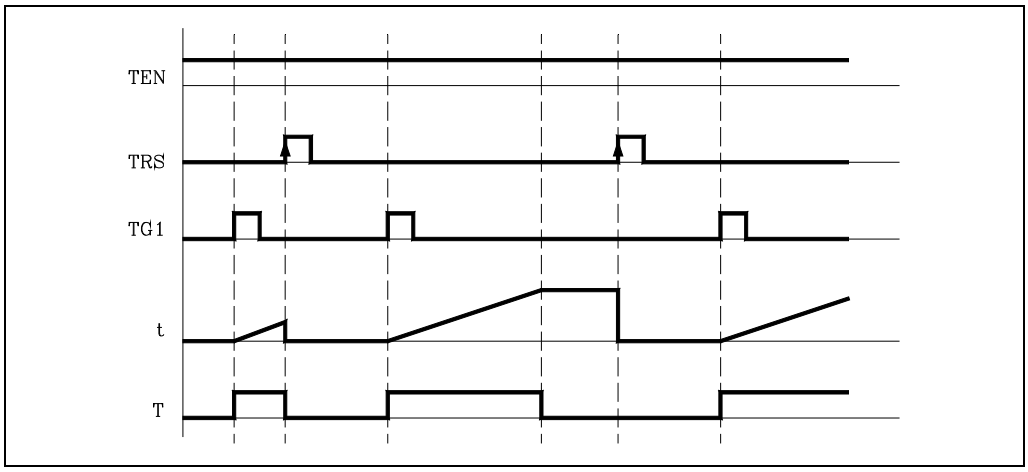
Once the time specified by the time constant has elapsed, timing will be considered as having finished. The timer status output (T) changes status (T=0) and the elapsed time will be maintained with the time value of the timer (T).

Any changes at the TG1 input (up or down-flank) while timing, has no effect.

If, once the timing is complete it is required to activate the timer again, another leading edge must be produced at the TG1 input.

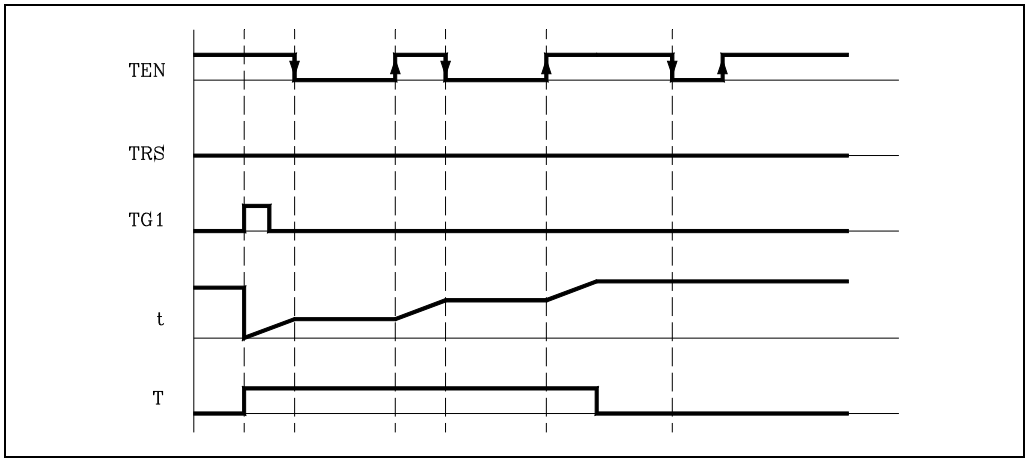
Operation of the TRS input in this mode

If a leading edge is produced at the TRS input at any moment during timing or afterwards, the PLC initializes the timer, assigning the value 0 to its T status and cancelling its count (it initializes this to 0). Due to the fact that the timer is initialized, it will be necessary to activate its trigger input to activate it again.



Operation of the TEN input in this mode

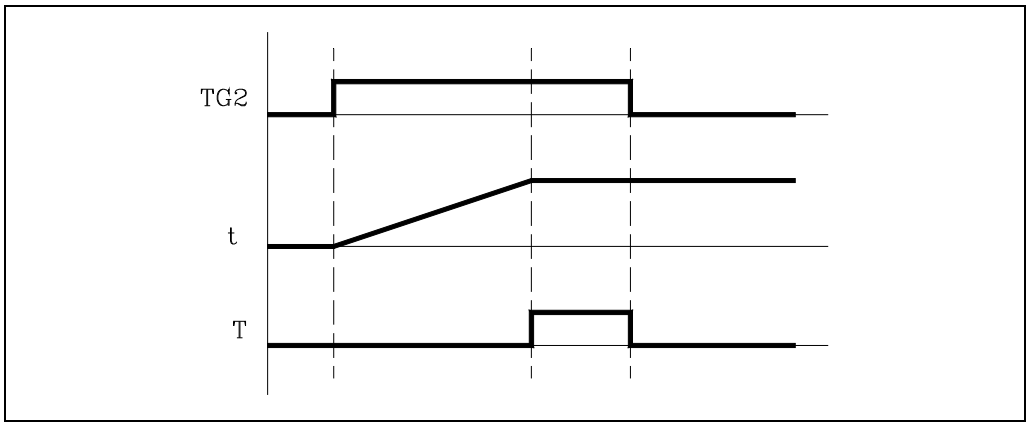
If once the timer has been activated, TEN = 0, the PLC interrupts the timing and TEN must be set to "1" to resume timing.



6.5.2 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".

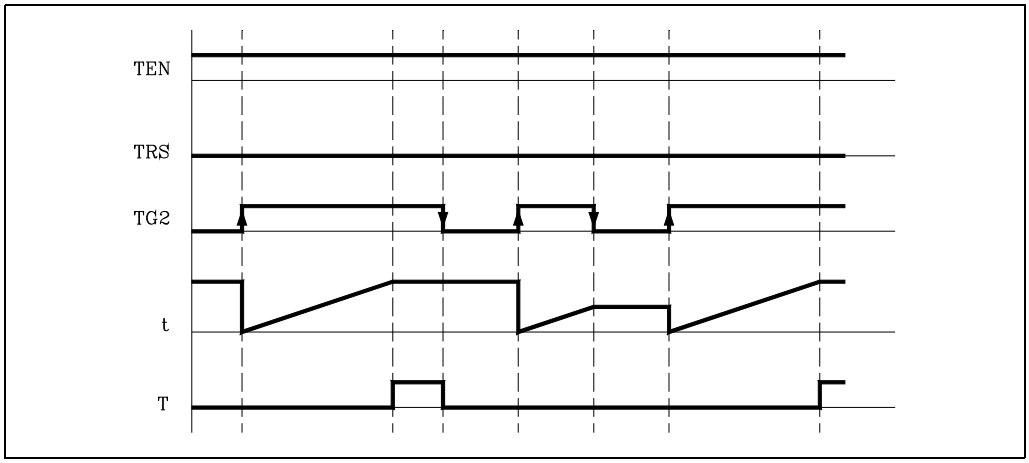
The time delay is set by the time constant.



If the timer is initialized with values TEN=1 and TRS=0, the timer will be activated when a leading edge is produced at TG2 input. At that instant, the timing "t" begins from "0".

Once the time specified by the time constant has elapsed the timing operation will be considered as having completed and the timer status output (T=1) will be activated and will remain in this status until the trailing edge is produced in the trigger input TG2.

The elapsed time will remain as a timer time value (T) once timing has been completed.

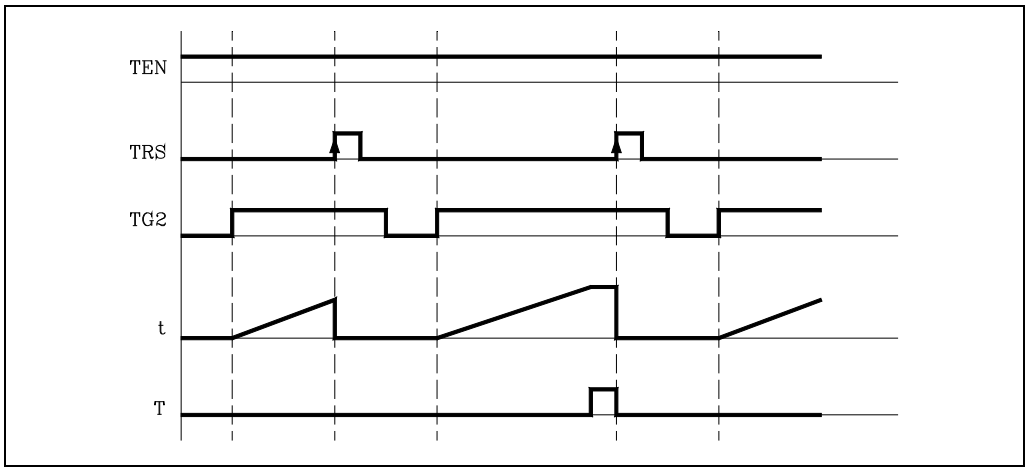


If, once the timing has finished, it is required to activate the timer again, another leading edge must be produced in the TG2 input.

If the trailing edge of the trigger input TG2 is produced before the time specified by the time constant has elapsed, the PLC will consider that the timing operation has concluded, maintaining the time count it had at that moment as the timer time (T).

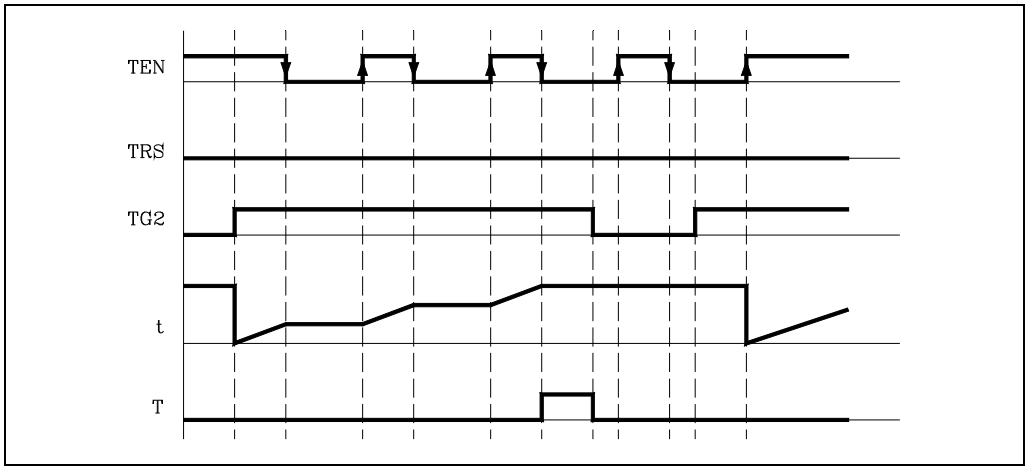
Operation of the TRS input in this mode

If a leading edge is produced at the TRS input at any moment during timing or afterwards, the PLC initializes the timer, assigning the value 0 to its T status and cancelling its count (it initializes this to 0). Due to the fact that the timer is initialized, it will be necessary to activate its trigger input to activate it again.



Operation of the TEN input in this mode

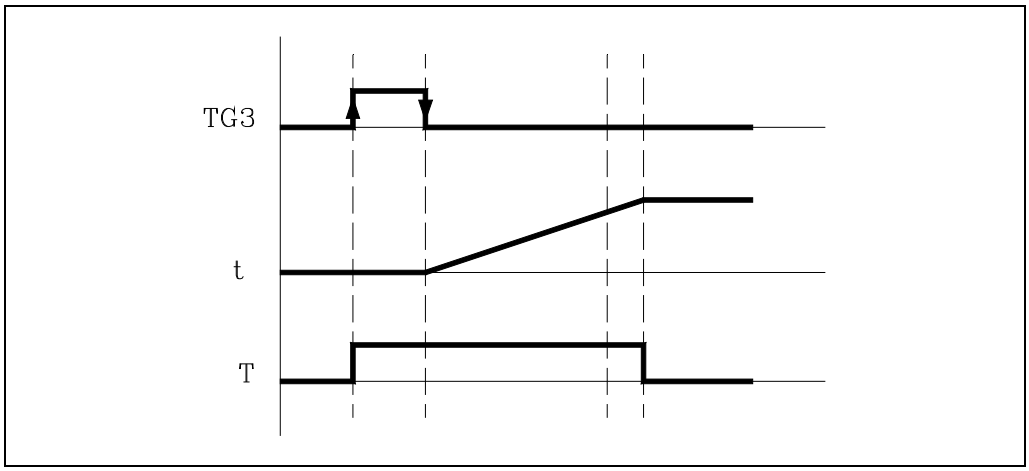
If once the timer has been activated, TEN = 0, the PLC interrupts the timing and TEN must be set to "1" to resume timing.



6.5.3 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.

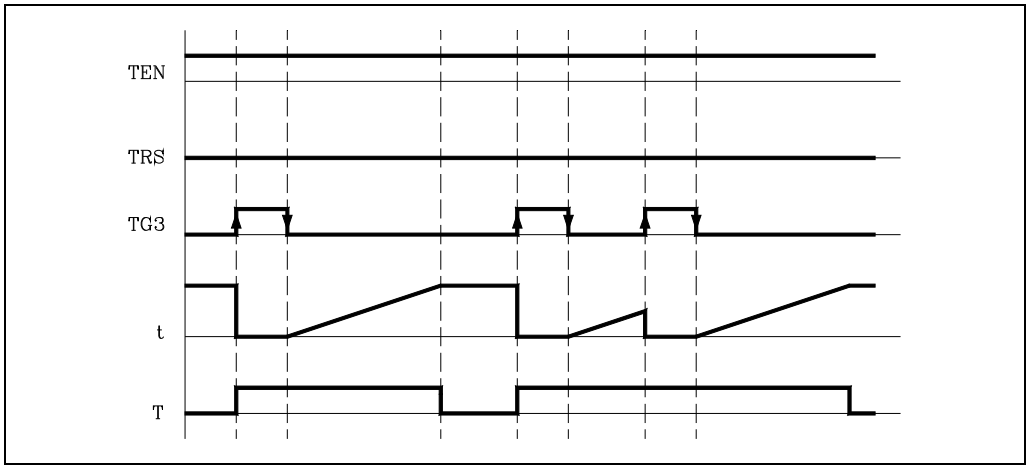


If the timer is initialized with values TEN=1 and TRS=0, the timer will be activated when a leading edge is produced at the TG3 input. At that moment, the timer status output will have a value of T=1.

The timer waits for a down-flank at input TG3 to start the "t" timing from "0".

Once the time specified by the time constant has elapsed the timing operation will be considered as having completed and the timer status output will be deactivated (T=0).

The elapsed time will remain as a timer time value (T) once timing has been completed.

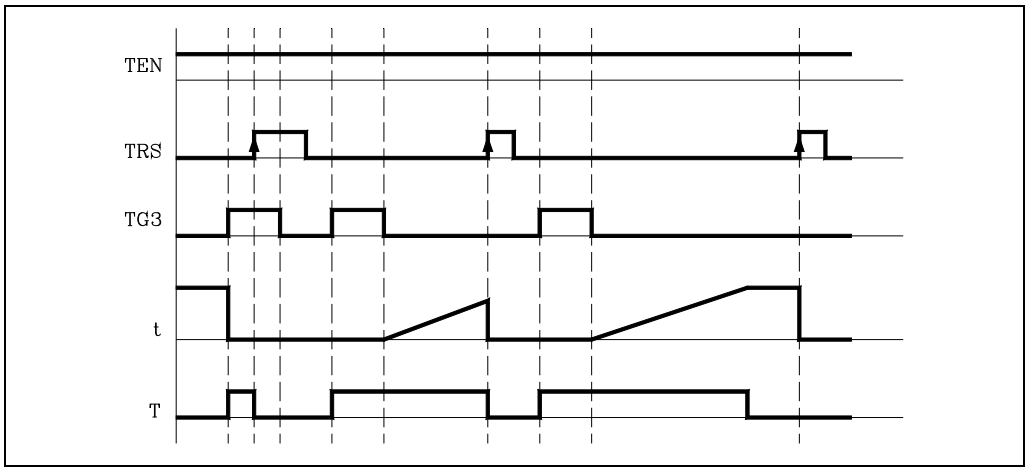


If, once the timing has finished, it is required to activate the timer again, another leading edge must be produced at the TG3 input.

If another leading edge of the trigger input TG3 is produced before the time specified by the time constant has elapsed, the PLC will consider that the timer has been activated again, maintaining its status (T=1) and initializing timing at 0.

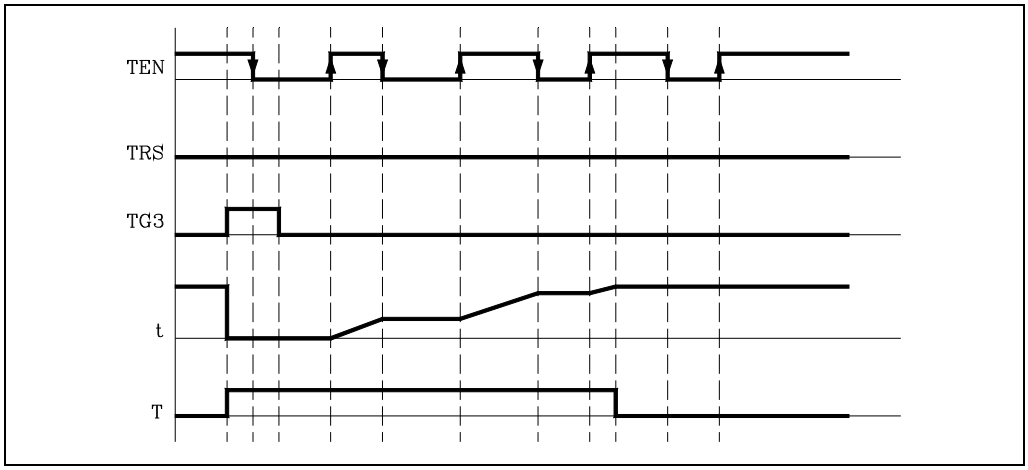
Operation of the TRS input in this mode

If a leading edge is produced at the TRS input at any moment during timing or afterwards, the PLC initializes the timer, assigning the value 0 to its T status and cancelling its count (it initializes this to 0). Due to the fact that the timer is initialized, it will be necessary to activate its trigger input to activate it again.



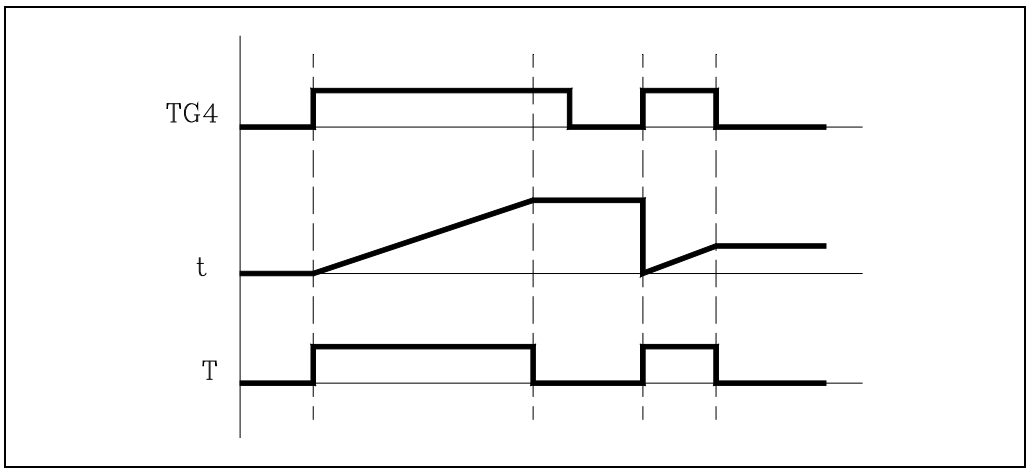
Operation of the TEN input in this mode

If once the timer has been activated, TEN = 0, the PLC interrupts the timing and TEN must be set to "1" to resume timing.

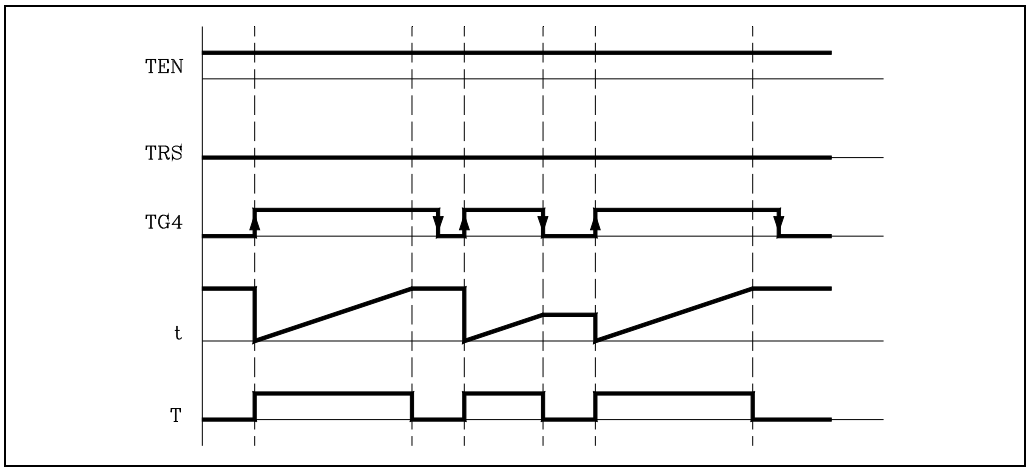


6.5.4 Signal limiting mode. TG4 Input

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.



If the timer is initialized with values TEN=1 and TRS=0, the timer will be activated when a leading edge is produced at the TG4 input. At that moment, the timer status output (T) changes states (T=1) and the timing starts from "0".



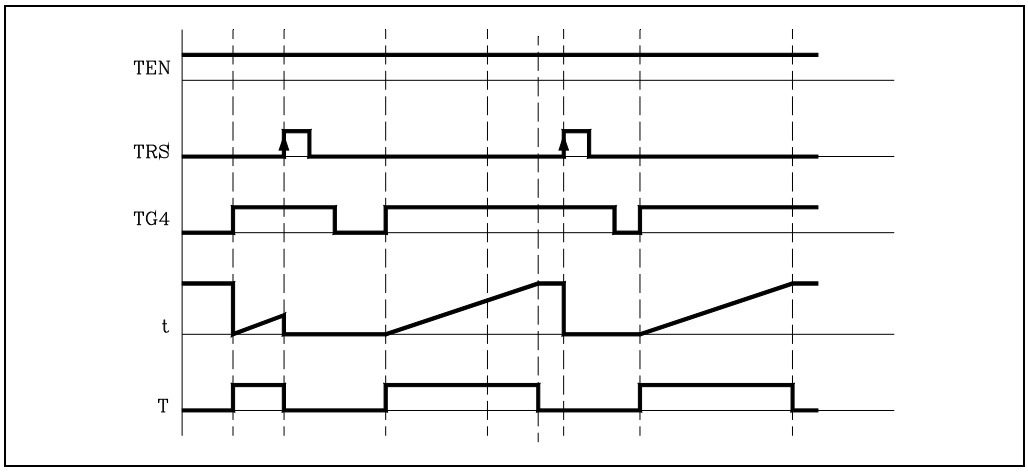
Once the time specified by the time constant has elapsed, timing will be considered as having finished. The time status output (T) changes status (T=0) and the elapsed time will be kept as a timer time value (T).

If, before the time specified by the time constant has elapsed, a trailing edge is produced in the trigger input TG4, the PLC will consider that the timing operation has concluded it will deactivate the status output (T=0) and maintain the value it has at that moment as the timer time value (T).

If, once the timing has concluded, it is required to activate the timer again, another leading edge must be produced at the TG4 input.

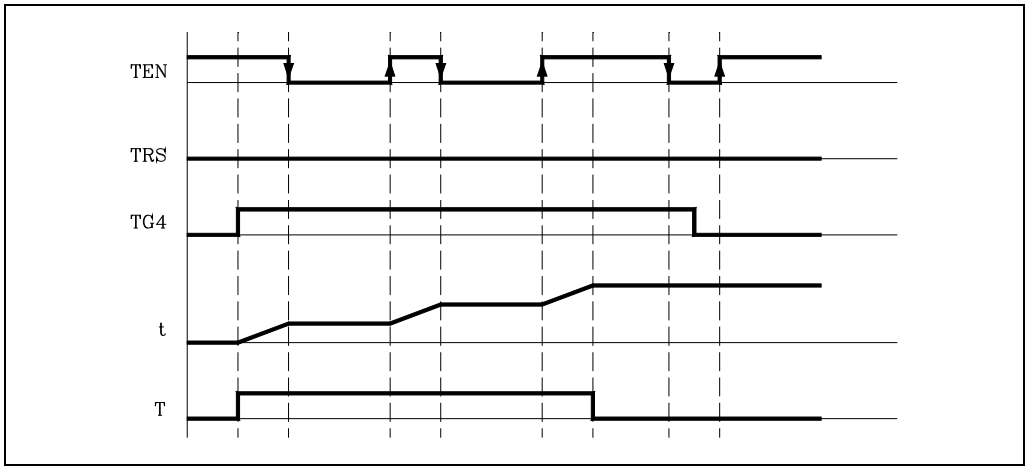
Operation of the TRS input in this mode

If a leading edge is produced at the TRS input at any moment during timing or afterwards, the PLC initializes the timer, assigning the value 0 to its T status and cancelling its count (it initializes this to 0). Due to the fact that the timer is initialized, it will be necessary to activate its trigger input to activate it again.



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.

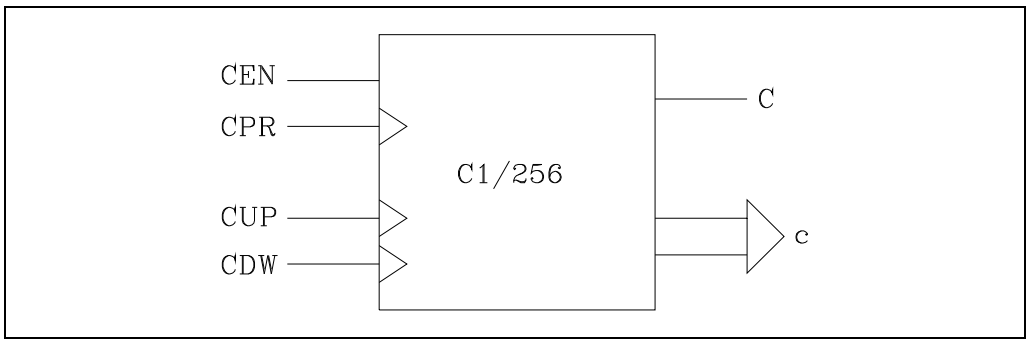


6.6 Counters

They are elements capable of counting up or down a preset amount of events. They do not have image values and are represented by the letter C, followed by the counter number which it is required to reference, for example C1, C25, C102, etc.

The counter's count is stored in a 32-bit variable. Consequently, its value will be in the ±2147483647 range.

The PLC has 256 counter, each of which has the C status output and CUP, CDW, CEN and CPR inputs. It is also possible to consult the count value at any time.



Feedback input (CUP)

This input allows the counter count to be increased in a unit every time a leading edge is produced in it. It is referred to by the letters CUP followed by the counter number, for example: CUP 1, CUP 25, CUP 102, etc.

Example:

I2 = CUP 10 — Every time a leading edge is produced at input I2 the counter count C10 will be increased.

Count-down input (CDW)

This input allows the counter count to be decreased in a unit every time a leading edge is produced in it. It is referred to by the letters CDW followed by the counter number, for example CDW 1, CDW 25, CDW 102, etc.

Example:

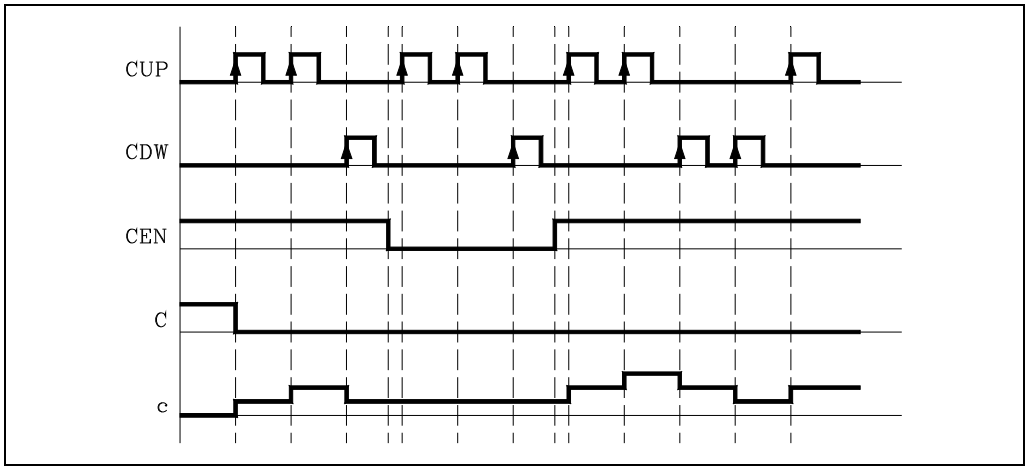
I3 = CDW 20 — Every time a leading edge is produced at input I3 the counter count C20 will be decreased.

Enable input (CEN)

This input allows the internal counter count to be stopped. It is referred to by the letters CEN followed by the counter number, for example: CEN 1, CEN 25, CEN 102, etc.

In order to be able to modify the internal count by means of the inputs CUP and CDW this input must be at logic level “1”. By default and every time a counter is activated the PLC will assign this input a logic level of “1”.

If CEN = 0 is selected the PLC stops the counter count, ignoring the inputs CUP and CDW until this input allows it (CEN = 1).



Example:

I10 = CEN 12 Input I10 controls the enable input of counter C12.

Preset input (CPR)

This input allows the counter to be preset with the desired value. It is referred to by the letters CPR followed by the number of the counter which is required to reference and the value to be assigned to the counter count.

For example CPR 1 100, CPR 25 224, CPR 102 0, CPR 200 500, etc.

The value of the count can be indicated by means of a numerical value or by assigning to it the internal value of an R register.

CPR 20 100 Presets the C20 counter to a value of 100.

CPR 22 R200 Presets the C22 counter with the value of the Register R200 when the instruction is executed.

The counter is preset with the indicated value with an up-flank at the CPR input.

Status output (C)

This output indicates the logic status 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.

Count value (C)

This output indicates the value of the internal counter count. It is referred to by the letter "C" followed by the counter number, for example: C1, C25, C102, etc.

Although when written C123 it coincides with the status output, both are different and, are used in different types of instructions.

In binary type instructions function C123 makes reference to the counter's logic status.

C123 = M100 Assigns mark to M100 the status (0/1) of counter 123.

In arithmetic and function comparison instructions C123 makes reference to the internal counter count.

```
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.

6.6.1 Operating mode of a counter

If the CEN counter input is initialized (CEN=1), the counter allows its count to be increased and decreased by means of the CUP and CDW inputs.

Operation of CUP and CDW inputs

Every time a leading edge is produced at the CUP input the counter increases its count by one count.

Every time a leading edge is produced at the CDW input the counter decreases its count by one count.

Operation of the CPR input

If a leading edge is produced at the CPR input the internal count value will take the new value assigned.

Operation of the CEN input

If CEN = 0 is selected the counter ignores both up-count (CUP) and down-count (CDW) inputs, it being necessary to assign CEN = 1 for the counter to take notice of these inputs.

INTRODUCTION TO THE PLC

7

It is recommended to save the PLC program and files into the hard disk (KeyCF) or in a peripheral or PC to avoid losing them.

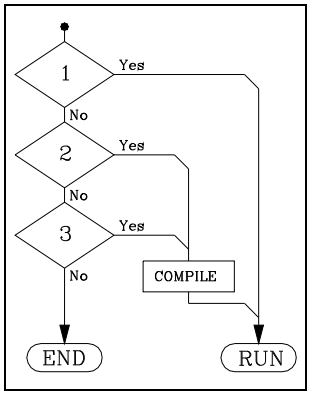
The PLC program (PLC_PRG) may be edited at the front panel or copied from the hard disk (KeyCF) or from a peripheral device or PC.

The PLC program (PLC_PRG) is stored in the internal CNC memory with the part-programs and it is displayed in the program directory (utilities) together with the part-programs.

Before executing the PLC_PRG program, it must be compiled. Once it is done compiling, the CNC requests whether the PLC should be started or not.

To make the operator life easier and avoid new compilations, the source code generated at each compilation is stored in memory.

After power-up, the CNC acts as follows:



1. If there is an executable program stored in memory, it executes it (RUN).
2. If there is no executable program, but there is a PLC_PRG in memory, it compiles it (COMPILE) and executes it (RUN).
3. If there is no PLC_PRG in memory, it looks for it in the hard disk (KeyCF).

   If it is there, it compiles it (COMPILE) and executes it (RUN). If it is not there, it does nothing. Later on, when accessing the Jog mode, Execution mode, etc. the CNC will issue the corresponding error message.

Once the program has been compiled, it is not necessary to keep the source program (PLC_PRG) in memory because the PLC always executes the executable program.

The PLC has 512 inputs and 512 outputs. Some of them, depending on the CNC configuration, communicate with external devices.

There is an exchange of information between the CNC and the PLC which is done automatically and the system has a series of commands which allow the following to be done quickly and simply:

The control of Logic CNC inputs and outputs by means of an exchange of information between both systems.

- The transfer from the CNC to the PLC of M, S and T auxiliary functions.
- To display a screen previously defined by the user, as well as generating messages and errors in the CNC.
- Read and modify internal CNC variables from the PLC.
- Access all PLC resources from any part-program.
- Monitor PLC resources on the CNC screen.
- Access to all PLC variables from a computer, via DNC and through the RS 232 C serial line.

7.1 PLC resources

Inputs (I)

They are elements that provide information to the PLC on the signals they receive from the outside world. They are represented by the letter I and there are 512 inputs available.

Outputs (O)

They are elements that let the PLC activate or deactivate the various devices of the electrical cabinet. These are represented by the letter O and there are 512 outputs available.

Marks (M)

These are elements capable of memorizing in one bit (as if it were an internal relay) the status of the different internal variables of the CNC (information of the logic outputs received in the communication between the CNC and the PLC of the CNC) and the status of the different variables of the PLC, whether these are internal or established by the user. They are represented by the letter M, and there are 2000 user marks and other special marks.

Registers (R)

These are elements which allow a numerical value to be stored in 32 bits or facilitate CNC-PLC communication with the Logic CNC inputs and outputs. They are represented by the letter R and there are 256 user registers and other special registers.

Timers (T)

These are elements which, once activated, alter the status of their output for a specific time (time constant). They are represented by the letter T, and there are 256 timers.

Counters (C)

They are elements capable of counting up or down a preset amount of events. They are represented by the letter C and there are 256 counters.

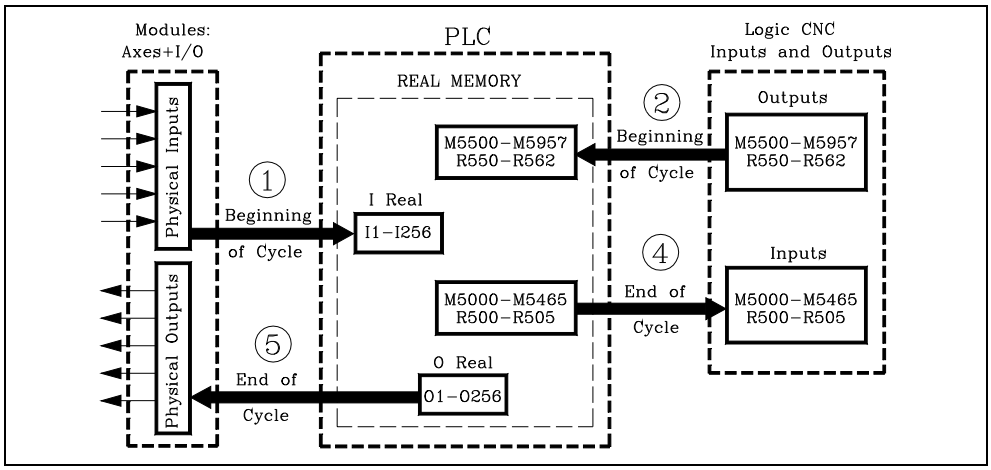
7.2 PLC program execution

The PLC executes the user program cyclically. In other words, once it executes the complete program, it restarts running this program from the first instruction.

This cyclic processing of the program is done as follows:

1. At the beginning of the cycle, PLC's "I" resources are assigned the current values of the physical inputs (connectors).

   For example, if the physical input I10 is at 24V, the PLC sets the I10 resource to "1".



2. It allocates the current values of the logic CNC outputs (CNCREADY, START, FHOUT, etc.) to PLC resources M5500 thru M5957 and R550 thru R562 .
3. It runs the program cycle.

   The following sections indicate how the PLC program is structured and which are its execution modules. See *"7.4 Modular structure of the program"* on page 291.
4. After executing the cycle, it updates the Logic CNC inputs (/EMERGEN, /STOP, /FEEDHOL, etc.) with the current values of PLC resources M5000 thru M5465 and R500 thru R505.
5. It assigns the current values of the PLC's "O" resources to the physical outputs (connectors).

   For example, if the "O5" resource is at "1", the PLC sets physical output O5 (connector) to 24V.
6. The cycle ends and is ready for the next scan.

Bear in mind that all the actions of the program executed by the PLC alter the status of its resources.

Example: I10 AND I20 = O5

When this condition is met [resource I10 is "1" and I20 is also "1"], the PLC sets resource "O5" to "1". If this condition is not met, the PLC sets resource "O5" to "0".

Therefore, the status of a resource may change during the execution of the PLC program.

Example, assuming that the initial status of resource M100 is "0":

M100 AND I7 = O3

Resource M100 = "0"

I10 = M100

M100 takes the value of resource I10

M100 AND I8 = M101

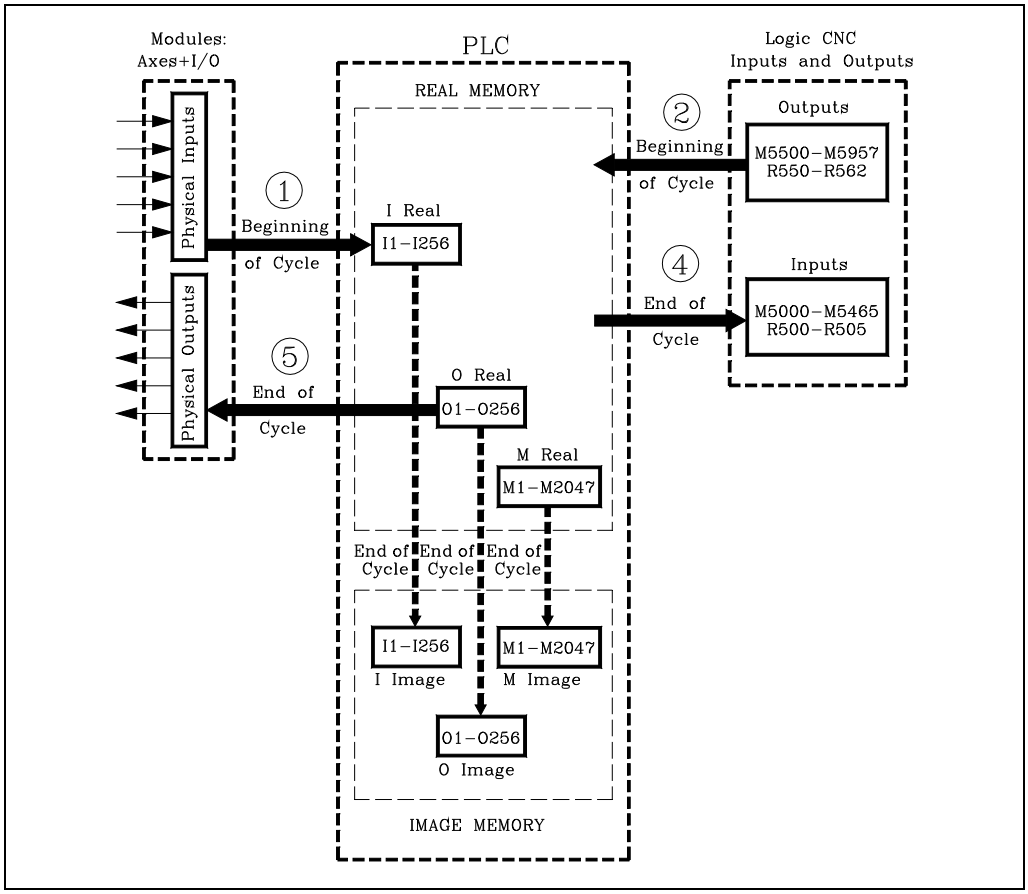
The value of M100 depends on the previous instruction.

This type of problems may be prevented by careful programming or by using "Image" resource values (instead of "Real" values).

The PLC has 2 memories to store the status of the registers, the real memory and the image memory.

All the steps described so far work with the real memory. Saying "value of a particular resource" is the same as saying "real value of a particular resource"

The image memory contains a copy of the values (status) that the resources had at the end of the previous cycle. The PLC makes this copy at the end of the cycle. The resources having an image value are: I1 thru I512, O1 thru O512 and M1 thru M2047



The following example shows how the PLC acts when working with real or image values.

PLC program	() = M1	Assigns the value of “1” to mark M1.
	M1 = M2	Assigns the value of M1 to M2.
	M2 = M3	Assigns the value of M2 to M3.
	M3 = O5	Assigns the value of M3 to output O5.

		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

As can be observed, the system is faster when operating with real resource values.

Operating with image values permits analyzing the same resource along the whole program with the same value regardless of its current (instantaneous) real value.

Operating with real values

In the first scan, when execution the instruction M1 = M2, M1 has a real value of “1” set by the previous instruction.

The same is true for instructions M2=M3 and M3=O5

That is why real values are used, output O1 takes the value of “1” in the first scan.

Operating with image values

The first cycle (scan) sets the real value of M1=1; but its image value will not be set to "1" until the end of the cycle.

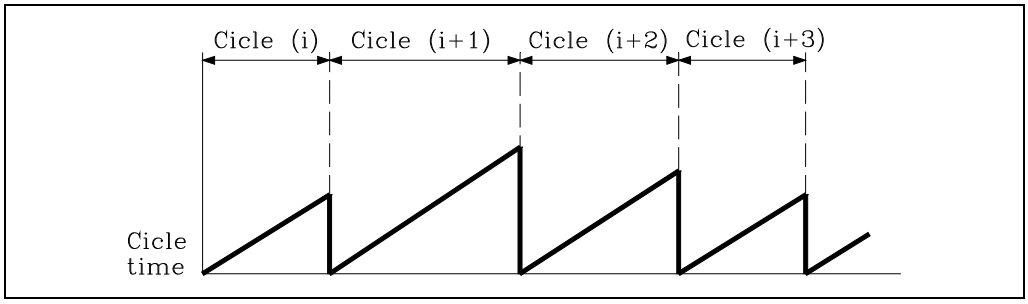
In the 2nd cycle (scan), the image value of M1 is "1" and the real value of M2 is set to "1"; but the image value of M2 will not be set to "1" until the end of the cycle.

In the 3rd cycle (scan), the image value of M2 is "1" and the real value of M3 is set to "1"; but the image value of M3 will not be set to "1" until the end of the cycle.

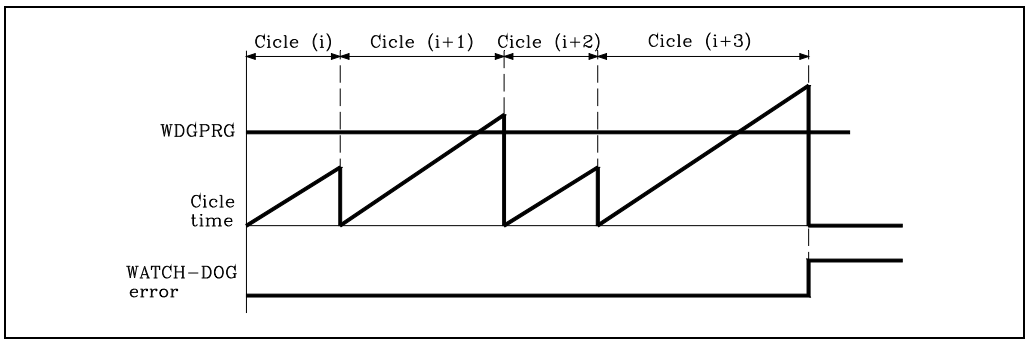
In the 4th cycle (scan), the image value of M3 is "1" and the real value of O5 is set to "1".

7.3 Loop time

The time the PLC requires to execute the program is called cycle time and can vary in the successive cycles of a same program, as the conditions under which they are executed are not the same.



plc.m.p WDGPRG (P0) sets the maximum cycle execution time This is called WATCH-DOG time and if a cycle is executed which lasts longer than 1.5 times this time, or two cycles are executed, one after the other, taking longer than this time period, the CNC will display the WATCH-DOG error of the main module.



This way, the execution of cycles that, due to their duration, disturb the operation of the machine can be prevented and the PLC can be prevented from executing a cycle which has no end due to a programming error.

7.4 Modular structure of the program

The program to be executed by the PLC consists of a series of modules which are appropriately defined by means of directing instructions.

The modules which can make up the program are:

- Main module (PRG)
- Periodic execution module (PE)
- First cycle module (CY1)

Each module must begin with the directing instruction which defines it (PRG, PE, CY1) and end with the directing instruction END.

Should the main program contain the main module only it is not necessary to place the instructions PRG and END.

7.4.1 First cycle module (CY1)

This module is optional and will only be executed when the PLC is turned on. It is used to initialize the different resources and variables with their initial values, before proceeding to execute the rest of the program.

This module operates by default with the real values of resources I, O, M.

It is not necessary for this to be at the beginning of the program, but must always be preceded by the instruction CY1.

7.4.2 Main module (PRG)

This module contains the user program. It will be executed cyclically and will be given the task of analyzing and modifying CNC inputs and outputs. Its execution time will be limited by the value of plc.m.p. WDGPRG (P0)

This module operates by default with the image values of resources I, O, M.

There can only be one main program and this must be preceded by the instruction PRG, it is not necessary to define it if it starts on the first line.

7.4.3 Periodic execution module (PE t)

This module is optional and will be executed every period of time t indicated in the directing instruction defining the module.

This module may be used to process certain critical inputs and outputs which cannot be checked or updated properly in the body of the main program due to its extended execution time.

Another application for this module is for those cases where specific tasks need not be evaluated at every PLC program cycle. Those tasks would be programmed in the periodic module and they would be executed with the frequency established by the execution time assigned to this module (for example: if t= 30,000; every 30 seconds).

A “t” value between 1 and 65535 milliseconds may be programmed.

The execution time of this module will be limited by the value of plc.m.p. WDGPER (P1)

This module operates by default with the real values of resources I, O, M.

Example:

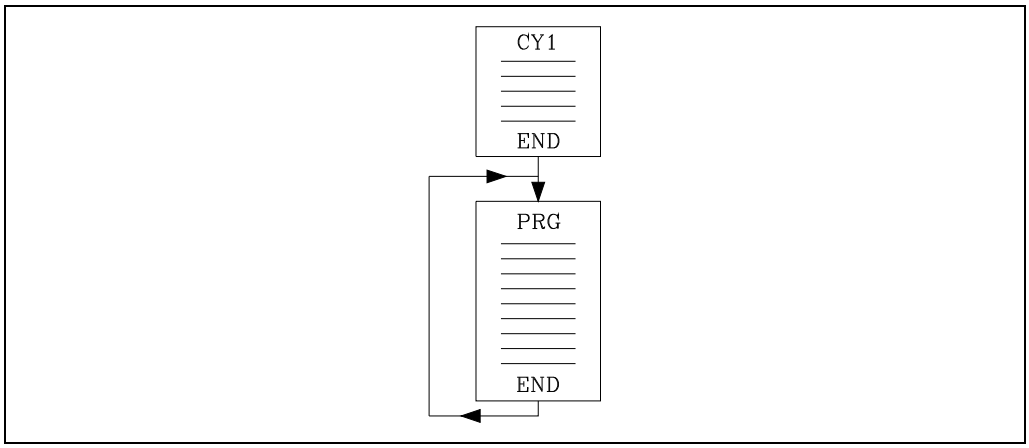
PE 10 Defines the beginning of the periodic module PE which will be executed every 10 milliseconds.

If this module is being executed with real values and acts on a physical output, this is updated at the end of the execution of the periodic module.

7.4.4 Priority of execution of the PLC modules

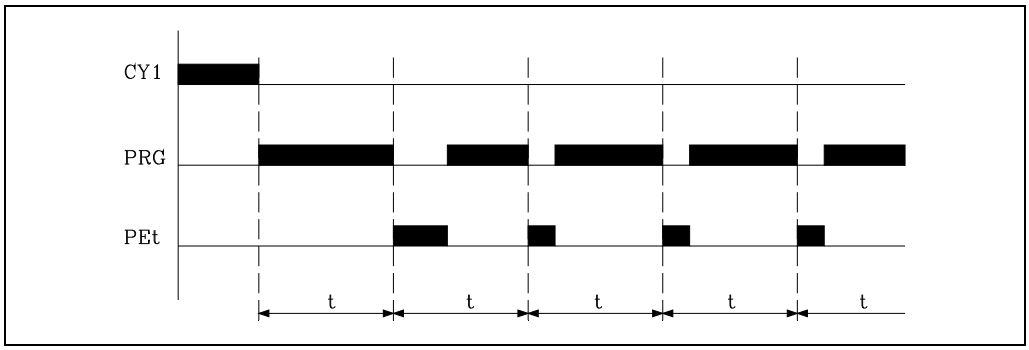
Every time the PLC program is started (command RUN) the first module to be executed is the first cycle module (CY1). Once execution has been completed, it will continue with the main module (PRG).

The main module will be executed cyclically until the execution of the PLC has stopped (command STOP).



The periodic module will be executed every time the time indicated in the directing instruction “PE t” elapses. This count starts when the execution of the main module (the first time) begins.

Every time this module is executed, the execution of the main module is interrupted, and its execution resumes when the execution of the periodic module finishes.



7.

INTRODUCTION TO THE PLC

Modular structure of the program

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

PLC PROGRAMMING

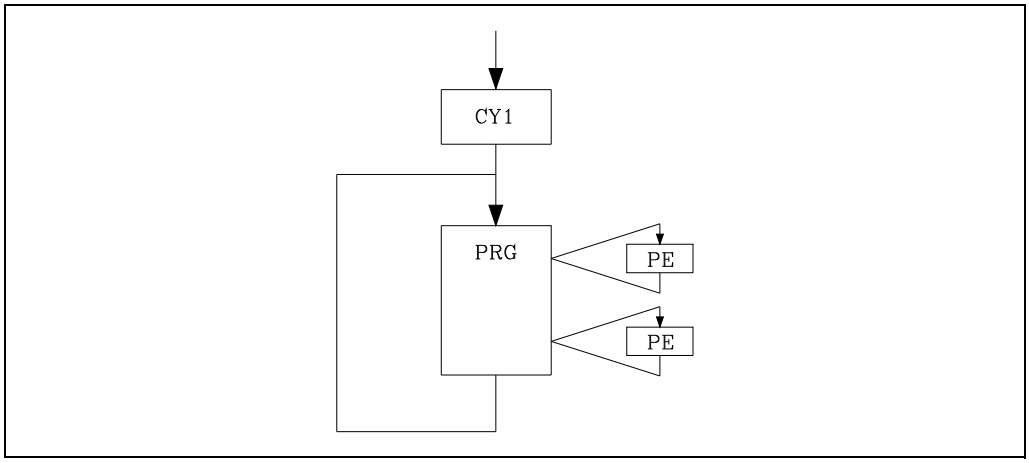
8

The PLC program is structured by modules and it may consist of:

- Main module (PRG).
- Periodic execution module (PE).
- First cycle module (CY1).

Every time the PLC program starts running, the CNC will execute first, if it has been defined, the First Cycle module (CY1). Then it will execute the Main Program module (PRG) continuously until the PLC program is stopped.

The periodic execution modules (PE) will be executed every so often with the frequency established for each of them. This time period starts counting from the time the CY1 cycle is ended. The execution of a periodic module temporarily interrupts the execution of the main module.



When defining the PLC program, both the processing of the main module (PRG) and the periodic modules (PE) must be taken into consideration.

The main module (PRG) will be processed cyclically. See *"7.2 PLC program execution"* on page 287.

The periodic module is optional and it is executed every so often as indicated by the directing instruction defining the module.

It is used to process certain critical inputs and outputs which cannot be properly evaluated within the main module because the cycle scan time for the main module would be too long for these resources to be checked and reacted upon.

It does not modify the status of the PLC resources. Therefore, the main module will resume execution as if the periodic module had not been executed at all.

The periodic module is processed as follows:

1. The PLC takes into account the current values, as just before executing the PE module, of the local physical inputs (connectors of the central unit).
2. Executes the periodic module.
3. It assigns the current values of the PLC's "O" resources to the local physical outputs (connectors of the central unit).
4. It ends the execution of the Periodic Module and resumes the execution of the main module.

8.1 Module structure

The modules which make up the PLC program (main module "PRG", periodic modules "PE" and first cycle module "CY1") consist of a series of instructions which, depending on their functionality, can be divided into:

- Directing instructions.
- Executable 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 allow inquiries to be made on and/or alterations to the status of PLC resources and consist of:

Logic expressions (Boolean 0/1)	I28 AND I30
Action instructions.	= O25

Logic expressions consist of:

Consulting instructions	I28, O25
Operators.	AND

All comments must begin with ";". ";". Lines beginning with a ";" are considered comments and are not executed.

Programming example:

```
PRG                ; Directing instruction
; Example          Comment.
I100 = M102        ; Executable proposition.
I28 AND I30        ; Logic expression.
= O25              ; Action instruction.
I32 \              ; Consulting instruction (1st part of the expression)
AND I36            ; Consulting instruction (1st part of the expression).
= M300             ; Action instruction.
END                ; Directing instruction.
```

See *"Summary of PLC commands"* on page 437.

Empty lines are not allowed, they must contain at least one comment.

8.2 Directing instructions

These provide the PLC with information on the type of module and the way it must be executed.

The directing instructions available at the PLC are:

PRG, PEt, CY1

Define the module type.

PRG	Main module.
CY1	First cycle module.
PE	Periodic module. It is executed every "t" milliseconds.

For example: PE 100 is executed every 100 ms.

END

Indicates the end of the module. If this is not defined, the PLC understands that this module ends in the last block of the program.

Example of programming using the directing instruction END:

CY1	Beginning of module CY1.
——-	
END	End of module CY1.
PRG	Beginning of module PRG.
——-	
END	End of module PRG.
PE 100	Beginning of module PE.
——-	
END	End of module PE.

Example of programming without using the directing instruction END:

CY1	Beginning of module CY1.
——-	
PRG	Beginning of module PRG.
——-	
PE 100	Beginning of module PE.
——-	
——	End of modules CY1, PRG and PE.

L

Label. Used to identify a program line, and is only used when references or program jumps are made.

It will be represented with the letter L followed by 3 figures (1-256), it not being necessary to follow any order and numbers out of sequence are permitted.

If there are 2 or more labels with the same number in a single program, the PLC will show the corresponding error when compiling it.

DEF

Symbol definition. Allows a symbol to be associated with any PLC variable, it being possible to reference this variable throughout the program by means of the variable name or by means of the associated symbol.

Example:

DEF EMERG I1

Assigns the EMERG symbol to input 11, so any reference throughout the program to EMERG will be interpreted by the PLC as a reference to I1.

It is also possible to associate a symbol to any number which can be given in decimal, with or without a sign, or hexadecimal format preceded with the "$" sign.

This option, among other applications, makes programming and later understanding of the PLC program much easier when trying to control the CNC by simulating its keyboard from the PLC program.

Example:

DEF HELP $FFF2

Assigns the “HELP” symbol to the code for the HELP key.

() = MOV HELP R101

Assigns the code corresponding to the “HELP” key to register R101.

CNCWR (R101, KEY, M101)

Indicates to the CNC that the key whose code is stored in register R101 and corresponds to the HELP key.

The PLC allows up to 400 symbol definitions which must always be programmed at the beginning of the program, before any other instruction, be this directing or executing.

A symbol will be made up with up to 8 characters, and must not coincide with any of the words reserved for instructions, nor be formed by the characters space” “, equal “=”, open and close parentheses “( )”, comma "," and semicolon “;”.

Duplicate symbols cannot be defined; but several symbols may be assigned to the same resource.

Example:

DEF EMRGOUT O1

DEF SALEMRG O1

The symbols associated to specialized marks and register (M> 2047 y R > =500) are pre-defined in the PLC and, therefore, it is not necessary to define them, nevertheless and if required, the PLC allows a different symbol to be assigned to them.

REA, IMA

Indicate to the PLC that the consultations defined below will be made on the real (REA) or image (IMA) values of I, O, M resources.

Counters, timers and registers do not have image values, so their real values will always be evaluated.

Action instructions (=O32) will always update the real values of PLC resources.

Example:

IMA

Consultations will evaluate image values.

I1 AND I2 = 01

REA

Consultations will evaluate real values.

IMA I3 AND REA M4 = 02

Evaluates the image of I3 and the real of M4.

IMA I5 REA = O3

Evaluates the image of I5 and the next ones in real.

IRD

Updates the real values of the local inputs after reading the relevant physical inputs.

Care must be taken when using this instruction since the current real values of the inputs will be lost.

OWR

Updates the local physical outputs with the current real values of the corresponding O resources.

MRD

Updates the values of resources M5000/5957 and R500/559 with the values of the logic outputs of the CNC.

Care must be taken when using this instruction since the current values of those resources will be lost. After executing this instruction, the new values will match those of the logic outputs of the CNC (internal variables).

MWR

Updates the logic inputs of the CNC (internal variables) with the current real values of resources M5000/5957 and R500/559.

TRACE

This instruction is used when working with the Logic Analyzer in order to capture data during the execution of the PLC cycle.

It must be born in mind that the logic analyzer performs a data capture at the beginning of each cycle (PRG and PE) after reading the physical inputs and updating the marks corresponding to the CNC logic outputs and just before starting the program execution.

Use this instruction to carry out another data capture while executing the PLC cycle.

Example of how to use the "TRACE" instruction:

```
PRG
----------
TRACE       Data capture.
----------
TRACE       Data capture.
----------
TRACE       Data capture.
----------
END
PE 5
----------
TRACE       Data capture.
----------
END
```

The data capture in the execution of the trace in this program takes place:

- At the beginning of each PRG cycle.
- Every time the periodic cycle is executed (every 5 milliseconds).
- 3 times while executing the PRG module.
- Once while executing the PE module.

This way, by means of the "TRACE" instruction the data capture can be done any time, especially at those program points considered more critical.

This instruction must only be used when debugging the PLC program and it should be avoided once the PLC program is fully debugged.

8.3 Consulting instructions

They may be used to check the status of PLC resources as well as the marks and registers for CNC-PLC communication. They are divided into:

- Simple consulting Instructions.
- Flank detection consulting Instructions.
- Comparison consulting Instructions.

All the consulting instructions allow the previous operand NOT, which reverses the result of the preceding consultation.

Example:

NOT I1 This Consultation will return a "0" if input I1 is at 1; and a "1" when input I1 is at 0.

Simple

They test the status of the resources and they return their logic state.

I	1/512	Inputs
O	1/512	Outputs
M	1/5957	Marks
T	1/256	Timers
C	1/256	Counters
B	0/31 R 1/499	Register Bits

Example:

I12 It will return a 1 if input 12 is active and a 0 if otherwise.

Flank detection

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. There are two types of instructions:

DFU

It detects an up-flank (0-to-1 change) at the indicated resource. It returns a "1" if it happened.

DFD

It detects an down-flank (0-to-1 change) at the indicated resource. It returns a "1" if it happened.

The programming format of the different combinations is:

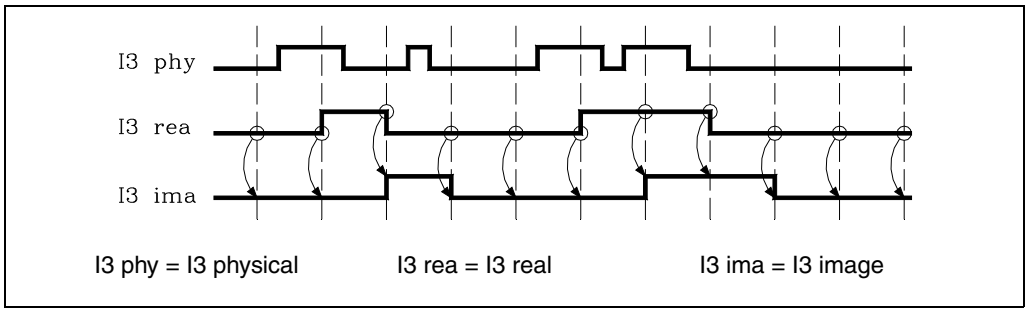
DFU (Up flank detection.)	I 1/512
DFD (Down flank detection)	O 1/512 M 1/5957

The consulting instructions to detect the flanks of marks M4000 thru M4127, M4500 thru M4563, M4700 thru M4955 and M5000 thru M5957 will be executed with their real values even when working with image values since these marks have no image values.

Considering that these instructions can evaluate real and image values, the following points must be taken into account:

The PLC updates the real values of the inputs at the beginning of the cycle, taking the values of the physical inputs.

The image values of the inputs, outputs and marks are updated after executing the program cycle.



Examples:

DFU I23 DFU B3R120 DFU AUXEND

Comparison

CPS

Used to compare two operands, checking whether the first one is greater than (GT), greater than or equal to (GE), equal to (EQ), not equal to (NE), smaller than or equal to (LE) or less than (LT) the second one.

The following may be used as operands: Timers (internal count), Counters (internal count), Registers, CNC-PLC communication registers and numbers (#) within ±2147483647 or between 0 and $FFFFFFFF.

The programming format of the different combinations is:

CPS	T 1/256	GT	T 1/256
	C 1/256	GE	C 1/256
	R 1/559	EQ	R 1/559
	#	NE	#
		LE	
		LT	

If the required condition is met, the consulting instruction returns a logic value "1" and a "0" if otherwise.

Programming examples:

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 on the counter T2 is EQUAL to the value of 100, timer T5 will be activated working as a monostable and with a time constant of 2 seconds.

8.4 Operators and symbols

Used to group and operate with different consulting instructions.

The available operators are:	NOT	AND	OR	XOR
The available symbols are:	(	)		

The operators are associated from left to right and the priorities ordered from the highest to the lowest are:

NOT AND XOR OR

With the "(" and ")" symbols, it is possible to clarify and select the order in which the logic expression is evaluated.

Example: (I2 OR I3) AND (I4 OR (NOT I5 AND I6)) = O7

NOT

Reverses 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.

( )

Open and close parenthesis.

They help clarify and select the order the logic expression is evaluated.

Example: (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", i.e.

( ) = O2

Output O2 will always be high (=1).

8.5 Action instruction.

The action instructions, depending on the result obtained in the logic expression may be used to alter the status of the PLC resources and CNC-PLC communication marks.

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 Action Instructions allow a previous NOT, which reverses the result of the expression for that action.

Example:

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.

Action instructions are divided into:

- Assignment Binary Action Instructions.
- Conditioned binary actions instructions.
- Sequence breaking action instructions.
- Arithmetic action instructions.
- Logic action instructions.
- Specific action instructions.

Action instructions can alter the status of all the PLC resources except that of 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.

8.5.1 Binary assignment instructions

They assign the value obtained from evaluating the logic expression (0/1) to the indicated resource.

= I	1/512	Inputs
= O	1/512	Outputs
= M	1/5957	Marks
= TEN	1/256	Timer enable
= TRS	1/256	Timer reset
= TGn	1/256 n/R	Timer trigger input
= CUP	1/256	Counter count up
= CDW	1/256	Counter count down
= CEN	1/256	Counter enable
= CPR	1/256 n/R	Counter preset
= B	0/31 R 1/499	Register Bits

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

It assigns to Mark M111 the value obtained in the evaluation of the Logic Expression (I2 OR I3) AND (I4 OR (NOT I5 AND I6)) .

8.5.2 Conditional binary action instructions

There are 3 instructions: SET, RES and CPL, that are used to change the status of the indicated resource.

Their programming format is:

= SET	I 1/512
= RES	O 1/512
= CPL	M 1/5957 B 0/31 R 1/559

= SET

If expression “1”, it assigns a “1” to the resource.

If the result of evaluating the logic expression is a “1”, it assigns a “1” to the indicated resource. If the result is "0", it does not change the resource.

Example: 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 assigns a “0” to the resource.

If the result of evaluating the logic expression is a “0”, it assigns a “1” to the indicated resource. If the result is "0", it does not change the resource.

Example: 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.

When the result of the logic expression is a "0", the PLC sets "M65=0" and does not change M55.

= CPL

If expression = 1, it complements the resource.

If the result of evaluating 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.

Example: DFU I8 OR DFD M22 = CPL B12R35

Every time an Up Flank (leading edge) is detected at input I8 or a down flank (trailing edge) in mark M22 the PLC will complement the status of bit 12 of register R35.

8.5.3 Sequence breaking action instructions

These actions interrupt the sequence of a program, resuming it somewhere else in the program.

That area must be identified with a label (L 1/256).

A subroutine is any part of the program that starts with a label (L1/256) and end with the directing instruction END.

= JMP

Unconditional Jump.

If the result of evaluating the logic expression is a “1”, it causes jump to the indicated label. If the result is a "0", it goes on to the next program line.

Example:

I8 = JMP L12	If I8 = 1 it goes on to L12
M14 AND B7R120 = O8	If I8=1 it is not executed
CPS T2 EQ 2000 = O12	If I8=1 it is not executed
L12	
(I12 AND I23) OR M54 = O6	

= CAL

Call to a subroutine.

If the result obtained in the evaluation of the logic expression is a “1” this action will execute the indicated subroutine.

Once the subroutine execution is over, the PLC will continue at the action instruction or executable instruction programmed after CAL.

If the result obtained in the evaluation of the logic expression is a “0” this action will be ignored by the PLC without executing the subroutine.

Example: 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 subroutine.

If the result obtained in the evaluation of the logic expression is a “1” this action will be treated by the PLC as if it involved the directing instruction END. 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.

If END is not programmed as end of subroutine, the PLC will continue executing until the end of the module (END) or the end of the program and it will finish the execution of the subroutine at that point.

It is advisable to place the subroutines after the END of the main program since if these are placed at the beginning, the PLC will start to execute them and will interpret the END of the subroutine as the END of the module, and it will consider that this has finished because no call was made to the subroutine.

8.5.4 Arithmetic action instructions

= MOV

It is used to move data from one PLC resource to another.

The programming format is:

	Origin	Destination	Source code	Destination code	Number of bits to transmit
MOV	I 1/512	I 1/512	(Bin)	0(Bin)	32
	O 1/512	O 1/512	1(BCD)	1(BCD)	28
	M 1/5957	M 1/5957			24
	T 1/256	R 1/559			20
	C 1/256				16
	R 1/559				12
	#				8
					4

The source and destination codes indicate the original and destination format (binary or BCD) of the data. 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	from Binary to Binary in 32 bits
MOV	O21	R100	0012	from Binary to Binary in 12 bits
MOV	C22	O23	0108	from Binary to BCD in 8 bits
MOV	T10	M112	1020	from 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 bits
999	with 12 bits

9999	with 16 bits
99999	with 20 bits
999999	with 24 bits

9999999	with 28 bits
99999999	with 32 bits

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.

Example: 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.

= NGU

Complements the bits of a register.

It changes the state of each one of the 32 bits of a register.

Example: 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

Register sign change.

Example: 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**

They may be used to carry out arithmetic operations such as addition (ADS), subtraction (SBS), multiplication (MLS), division (DVS) and module or remainder of a division (MDS).

Its programming format is:

ADS	R1/559	R1/559	R1/559
SBS	#	#	
MLS			
DVS			
MDS			

The operands may be: Registers, CNC-PLC communication registers and numbers (#) within ±2147483647 or between 0 and $FFFFFFFF

The result of the operation may be stored in a register or in CNC-PLC communication register

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
() = DVS	R100	1000	R115	R115 = 1234 : 1000 = 1
() = MDS	8765	1000	R116	R116 = 8765 MOD 1000= 765

If a division by "0" is performed in the DVS operation, the CNC stops the execution of the PLC program and it displays the corresponding error message.

8.5.5 Logic action instructions

= AND **= OR** **= XOR**

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	R1/559	R1/559	R1/559
OR	#	#	
XOR			

Registers (R1/559) or numbers expressed in decimal, hexadecimal or binary format can be defined as first or second operand.

The destination register indicates where the result of the operation will be deposited and will be defined by means of a register (R1/559).

The mark M2003 is called zero flag and indicates whether the result of an AND, OR, XOR, operation equals zero, in which case it follows that M2003=1.

Examples with: R200 = B1001 0010

R201 = B0100 0101

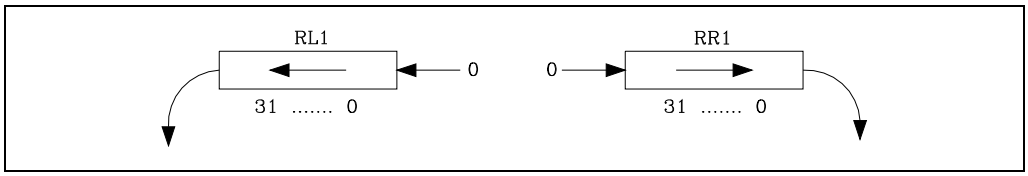
() = AND	R200	R201	R202	R202=B0	M2003=1
() = OR	R200	R201	R203	R203=B11010111	M2003=0
() = XOR	R200	R201	R204	R204=B11010111	M2003=0
() = AND	B1111	R201	R205	R205=B00000101	M2003=0
() = OR	R200	B1111	R206	R206=B10011111	M2003=0
() = XOR	B1010	B1110	R207	R207=B00000100	M2003=0

= RR **= RL**

Used to rotate registers clockwise (RR) or counterclockwise (RL). There are two types of rotations: type 1 (RR1 or RL1) and type 2 (RR2 or RL2).

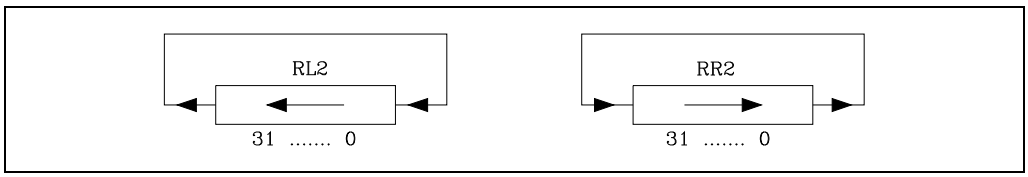
Rotation type 1 (RL1 or RR1):

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.



Its programming format is:

	Origin	Nr of repetitions	Destination
RR1	R1/559	R1/559	R1/559
RR2		0/31	
RL1			
RL2			

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.

Examples:

RR1 R100 1 R200

1 type 1 rotation to the right of the contents of R100 leaving the result in R200.

RL2 R102 4 R101

4 type-2 rotations to the left of the contents of R102 leaving the result in R101.

() = RL2 R17 4 R20

R17 =	0011	0000	1100	1100	0100	0110	1101	0100
R20 =	0000	1100	1100	0100	0110	1101	0100	0011

8.5.6 Specific action instructions

= ERA

Used to delete a group of resources. Indicate the first and last resource to be erased.

Its programming format is:

ERA	I	1/512	1/512
	O	1/512	1/512
	M	1/5957	1/5957
	T	1/256	1/256
	C	1/256	1/256
	R	1/559	1/559

The marks can be M1/2047, M4000/4127, M4500/4563, M4700/4955 or M5000/5957 and registers R1/559

When deleting a group of I, O, M or R, the PLC sets them to “0”.

If a group of timers is erased this is the equivalent of Resetting them and if a group of counters is erased this is similar to making a preset with a value 0 for them.

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).

Examples:

I12 = ERA O5 12

If input I12 has a value of "1" the PLC will set to 0 outputs O5 thru O12.

I23 = ERA C15 18

If input "I23 =1", the PLC presets counters C15 through C18 (both included) to "0".

= CNCRD **= CNCWR**

Access to the internal 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 reads the contents of the register into the variable.

The internal CNC variables are described in the chapter on "CNC-PLC communication".

The mark is set to "1" at the beginning of the operation and it keeps its value until the end of the operation.

When requesting information on a nonexistent variable (for example the position of an nonexistent axis), it will show the relevant error message.

Examples:

CNCRD (FEED, R150, M200)

Loads into register R150 the feedrate value selected at the CNC by means of function G94.

CNCWR (R92, TIMER, M200)

It resets the clock enabled by the PLC with the value contained in register R92.

= PAR

It analyzes the type of parity of a register.

Its programming format is:

PAR	R1/559	M1/5957

If the register being checked has an EVEN parity, this instruction will set the indicated mark to “1” and if its parity is ODD, it will set it to “0”.

Example:

I15 = PAR R123 M222

If I15 = 1 the PLC checks the parity of register R123 and sets M222 = 1 if it is EVEN or M222 = 0 if it is ODD.

8.

PLC PROGRAMMING
Action instruction.

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

CNC-PLC COMMUNICATION

9

With the data exchange between the CNC and the PLC, it is possible to:

- The control of logic inputs and outputs from the CNC by means of an exchange of information between both systems, which is done periodically and by means of specific PLC Marks and Registers.
- The transfer from the CNC to the PLC of M, S and T auxiliary functions.
- Display screens which have been defined previously by the user, as well as generating messages and errors in the CNC, by means of specific 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.
- Access to all PLC variables from a computer, via DNC through the RS 232 C serial line.

9.1 Auxiliary M, S, T functions

MBCD1 (R550) **MBCD2 (R551)** **MBCD3 (R552)** **MBCD4 (R553)**
MBCD5 (R554) **MBCD6 (R555)** **MBCD7 (R556)** **MBCDP1 (R565)**
MBCDP2 (R566) **MBCDP3 (R567)** **MBCDP4 (R568)** **MBCDP5 (R569)**
MBCDP6 (R570) **MBCDP7 (R571)**

MBCD* resgisters correspond to the main channel whereas MBCDP registers are for the PLC channel.

The CNC tells the PLC by means of these 32 bit registers, the miscellaneous M functions programmed in the block being executed.

If there are less than 7 miscellaneous M functions in each block, the CNC will send the information in the lower-numbered registers, assigning the value $FFFFFFFF to those which are left free.

This way, if a block contains functions M100, M120 and M135, the CNC will transfer the following information to the PLC:

MBCD1 (R550) = $100
MBCD2 (R551) = $120
MBCD3 (R552) = $135
MBCD4 (R553) = $FFFFFFFF.
MBCD5 (R554) = $FFFFFFFF.
MBCD6 (R555) = $FFFFFFFF.
MBCD7 (R556) = $FFFFFFFF.

To know whether a particular M function is programmed in the execution block, use one of the following methods:

1. Check all MBCD registers one by one until the specific “M” function is found or until one of them contains the $FFFFFFFF value.
2. Use the “MBCD*” format which permits checking all MBCD registers at the same time.

```
Example:
   CPS MBCD* EQ $30 = ...
```

It returns a “1” if it detects an M30, and a “0” if otherwise.

The miscellaneous M functions can be executed at the beginning or end of the block, according to how these are set in the miscellaneous M function table.

Besides, this table will indicate whether the CNC must wait, or not, for the general logic input AUXEND to consider the execution of the corresponding M as having been completed

SBCD (R557)

This register will be used when using a spindle operating with BCD coded S signal. s.m.p. SPDLTYPE (P0).

The auxiliary S function will always be executed at the beginning of the block and the CNC will wait for the general logic input AUXEND to be activated to consider the execution completed.

If S output in 2-digit BCD is used the CNC will tell the PLC, by means of this register the selected spindle speed according to the following conversion table:

Programmed S	S BCD	Programmed S	S BCD	Programmed S	S BCD
0	00	50-55	54	800-899	78
1	20	56-62	55	900-999	79
2	26	63-70	56	1000-1119	80
3	29	71-79	57	1120-1249	81
4	32	80-89	58	1250-1399	82
5	34	90-99	59	1400-1599	83
6	35	100-111	60	1600-1799	84
7	36	112-124	61	1800-1999	85
8	38	125-139	62	2000-2239	86
9	39	140-159	63	2240-2499	87
10-11	40	160-179	64	2500-2799	88
12	41	180-199	65	2800-3149	89
13	42	200-223	66	3150-3549	90
14-15	43	224-249	67	3550-3999	91
16-17	44	250-279	68	4000-4499	92
18-19	45	280-314	69	4500-4999	93
20-22	46	315-354	70	5000-5599	94
23-24	47	355-399	71	5600-6299	95
25-27	48	400-449	72	6300-7099	96
28-31	49	450-499	73	7100-7999	97
32-35	50	500-559	74	8000-8999	98
36-39	51	560-629	75	9000-9999	99
40-44	52	630-709	76		
45-49	53	710-799	77		

If a value over 9999 is programmed the CNC will tell the PLC the spindle speed corresponding to value 9999.

If S output in 8-digit BCD is used the CNC will indicate the programmed spindle speed to the PLC by means of this register.

This value will be coded in BCD format (8 digits) in thousandths of a revolution per minute.

S 12345.678 = 0001 0010 0011 0100 0101 0110 0111 1000

If no S has been programmed in the block, the CNC will assign a value of $FFFFFFFF to this register.

TBCD (R558)

The CNC tells the PLC by means of this 32-bit register, the pocket number in the magazine where the selected tool is.

If the g.m.p. RANDOMTC (P25) has been set so it is not a random magazine, the magazine pocket position coincides with the tool number.

This will be coded in BCD format (8 digits).

T 123 = 0000 0000 0000 0000 0000 0001 0010 0011

If no T has been programmed in the block, the CNC will assign a value of $FFFFFFFF to this register.

The T function will always be executed at the beginning of the block and the CNC will wait for the general logic input AUXEND to be activated to consider the execution completed.

T2BCD (R559)

This register is used when a special tool change has been made (family code >=200) or with machining centers with a non-random tool magazine (general machine parameter RANDOMTC (P25).

The CNC tells the PLC by means of the 32 bit register, the position of the magazine (empty pocket) in which the tool which was on the spindle must be deposited.

This will be coded in BCD code (8 digits). If a second T function is not required the CNC will assign a value $FFFFFFFF to the register.

The second T function will be sent together with M06 and the CNC will wait for the general logic input AUXEND to the activated to consider the execution completed.

9.2 Auxiliary M, S, T function transfer

Every time a block is executed in the CNC, information is passed to the PLC about the M, S, and T functions which are active.

M function

The CNC analyzes the M functions programmed in the block and in accordance with how these are defined, will send these to the PLC before and/or after the movement.

To do this, it uses variables “MBCD1” to “MBCD7” (R550 to R556) and activates the general logic output “MSTROBE” to indicate to the PLC that it must execute them.

Depending on how these functions are defined on the table, the CNC must wait, or not, for the general input “AUXEND” to be activated to consider the execution completed.

S function

If an S function has been programmed and the spindle has BCD input, the CNC will send this value to the variable “SBCD” (R557) and will activate the general logic output “SSTROBE” to indicate to the PLC that it must be executed.

This transmission is made at the beginning of the block execution and the CNC will wait for the general input "AUXEND" to be activated to consider the execution completed.

T function

The CNC will indicate via the variable “TBCD” (R558) the T function which has been programmed in the block and activates the general logic output “TSTROBE” to tell the PLC that it must execute it.

This transmission is made at the beginning of the block execution and the CNC will wait for the general input “AUXEND” to be activated to consider the execution completed.

Second T function

If this involves changing a special tool or a machining center with non-random tool magazine, the CNC will indicate, on executing the M06 function, the position of the magazine (empty pocket) in which the tool which was on the spindle must be deposited.

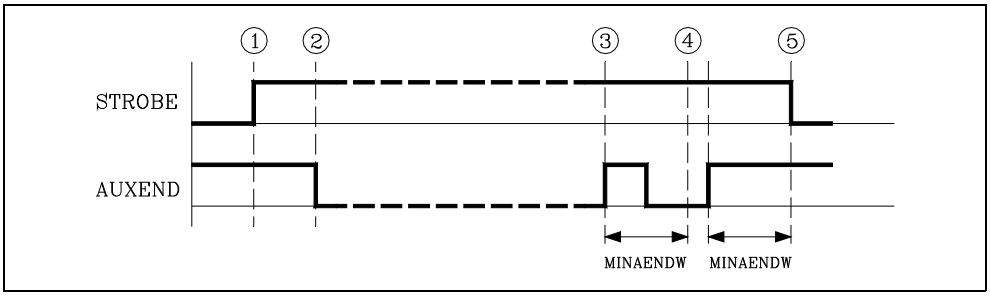
This indication will be made by means of the variable “T2BCD” (R559) and by activating the general logic output “T2STROBE” to tell the PLC that it must execute it. The CNC will wait for the general input AUXEND to be activated to consider the execution completed.



It must be borne in mind that at the beginning of the execution of the block, the CNC can tell the PLC the execution of the M, S, T and T2 functions by activating their STROBE signals together and waiting for a single “AUXEND” signal for all of them.

9.2.1 Transferring M, S, T using the AUXEND signal

1. Once the block has been analyzed and after sending the corresponding values in the variables "MBCD1-7", "SBCD", "TBCD" and "T2BCD", the CNC will tell the PLC by means of the general logic outputs "MSTROBE", "SSTROBE", "TSTROBE" and "T2STROBE" that the required auxiliary functions must be executed.



2. When the PLC detects that one of the STROBE signals is active, it must deactivate the general logic input "AUXEND" to tell the CNC that the execution of the corresponding function or functions is starting.
3. The PLC will execute all the auxiliary functions required, it being necessary to analyze the "MSTROBE", "SSTROBE", "TSTROBE", "T2STROBE" general logic outputs and the "MBCD1-7", "SBCD", "TBCD" and "T2BCD" variables in order to do this.

   Once this has been executed the PLC must activate the general logic input “AUXEND” to indicate to the CNC that the processing of the required functions was completed.
4. Once the general "AUXEND" input is activated, the CNC will require that this signal be kept active for a time period greater than the value given to the g.m.p. "MINAENDW" (P30).

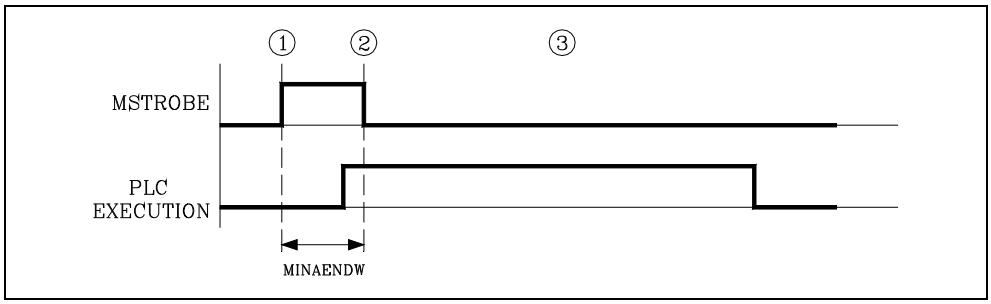
   This way, erroneous interpretations of this signal by the CNC due to an improper PLC program logic are avoided .
5. Once the period of time "MINAENDW" has elapsed with the general input "AUXEND" at a high logic level, the CNC will deactivate the general logic outputs "MSTROBE", "SSTROBE", "TSTROBE", "T2STROBE" to tell the PLC that the execution of the required auxiliary function or functions has been completed.



When the block being executed has several auxiliary functions (M, S, T), the CNC waits a time period set by g.m.p. MINAENDW (P30) between two consecutive transfers.

9.2.2 Transferring the auxiliary (miscellaneous) M functions without the AUXEND signal

1. Once the block has been analyzed and after passing the corresponding values in variables “MBCD1-7”, the CNC will tell the PLC through the general logic output “MSTROBE” that the required auxiliary function or functions must be executed.



2. The CNC will keep the general logic output "MSTROBE" active during the time indicated by means of g.m.p. MINAENDW (P30).

   Once this period of time has elapsed the CNC will continue to execute the program.

   It is advisable for the “MINAENDW” value to be equal to or greater than the duration of a PLC cycle, in order to ensure the detection of this signal by the PLC.

3. When the PLC detects the activation of the general logic signal “MSTROBE” it will execute the required miscellaneous “M” functions in the “MBCD1-7” variables.

9.3 Displaying messages, errors and screens

The PLC has a series of marks that allow messages and errors to be displayed in the CNC, as well as displaying screens which have been defined previously by the user.

Displaying messages

The PLC has 255 marks, with their corresponding mnemonic for displaying messages in the CNC.

M4000	MSG001	M4100	MSG101	M4252	MSG253
M4001	MSG002	M4101	MSG102	M4253	MSG254
M4002	MSG003	M4102	MSG103	M4254	MSG255
...	...	...	...		
...	...	...	...		

If one of these marks is activated (high logic level), the CNC will display the selected message number and its associated text on the PLC message display window (upper right-hand part).

The CNC allows a text to be associated to each PLC message (PLC message editing mode).

If the PLC activates 2 or more messages, the CNC will always display the message with the highest priority, this being understood as being the message with the lowest number. In this way, MSG1 will have the highest priority and MSG255 the lowest priority.

In this same message display window, the CNC can show the character + (plus sign), which indicates that there are more messages activated by the PLC, and these can be displayed if the active message page option is accessed in the PLC operating mode.

A message can be erased by deactivating it from the PLC program (low logic level) or from the CNC keyboard, after selecting it on the active messages page.

Nevertheless and depending on the program, the PLC may reactivate this message in the following cycle.

```
Example:
   DFU I10 = MSG1
   I10 = MSG2
```

1. Input I10 changes from 0 to 1.

   Messages MSG1 and MSG2 are activated.
2. The user deletes the messages using the keyboard.
3. In the next PLC cycle, since I10 is kept at “1”, MSG2 is activated again.

Displaying errors

The PLC has 128 marks, with their corresponding mnemonic, for displaying errors at the CNC.

M4500	ERR001	M4530	ERR031	M4625	ERR126
M4501	ERR002	M4531	ERR032	M4626	ERR127
M4502	ERR003	M4532	ERR033	M4627	ERR128
. . .	. . .	. . .	. . .		
. . .	. . .	. . .	. . .		

When one of these marks is activated (they are set high), they interrupt CNC part-program execution. It also displays the selected error message and its associated text in the middle of the screen.

The CNC allows a text to be associated to each PLC error (PLC error editing mode).

It is recommended to change the state of these marks by means of accessible external inputs since the PLC will not stop and the CNC will receive the error message in each new PLC cycle scan; thus preventing access to any of the PLC modes.

Displaying screens (pages)

The PLC has 256 marks with their corresponding mnemonic, for displaying screens (pages) at the CNC.

M4700	PIC000	M4900	PIC200	M4953	PIC253
M4701	PIC001	M4901	PIC201	M4954	PIC254
M4702	PIC002	M4902	PIC202	M4955	PIC255
. . .	. . .	. . .	. . .		
. . .	. . .	. . .	. . .		

If one of these marks is activated (high logic level), the CNC will display the character * (asterisk) on the PLC message display window (upper right-hand part) indicating that at least one of the 256 screens (pages) defined by the user in the graphic editor mode is activated.

The selected screens (pages) will be displayed, one by one, if the active page (screen) option is accessed in the PLC operating mode.

A page can be deactivated from the PLC program (by placing the corresponding mark at the low logic level) or, from the CNC keyboard, after selecting it in the active page mode.

9.4 Access to the PLC from the CNC

The CNC is provided with an operating mode in which it can:

- Monitor the user PLC program.
- Monitor PLC resources.
- Modify PLC resources.
- Execute PLC commands (compile, execute, etc.).
- Etc.

Likewise, the CNC allows access to all PLC variables of any part program and is provided with several high level language instructions for this purpose, which allow Inputs, Outputs, Marks, Registers, Timers and Counters to be read or modified.

9.5 Access to the PLC from a PC, via DNC

The CNC allows the PLC to communicate with a computer via DNC through the RS232C serial line.

In this way a computer can access the PLC carrying out:

- Transfer and reception of the user PLC program.
- Monitoring of the user PLC program.
- Monitoring of PLC resources.
- Consultation or modification of PLC resources.
- Execution of PLC commands (compile, execute, etc.).
- Etc.

The DNC manual can be applied for from the Commercial Department of Fagor Automation.

9.

CNC-PLC COMMUNICATION

Access to the PLC from a PC, via DNC

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

LOGIC CNC INPUTS AND OUTPUTS

10

Physical inputs and outputs are the names given to the set of inputs and outputs of the CNC system which, being controlled by the PLC, communicate with the outside through CNC connectors.

The CNC also has a series of logic inputs and outputs for the internal exchange of information with PLC marks and registers. This type of marks do not have images on the PLC.

Each of these CNC logic inputs and outputs may be referred to with the corresponding PLC resource or with their associated mnemonic. Mnemonics which begin with "/" indicate that the signal is active low (0 V). For example:

M5000	/EMERGEN	M5104	MIRROR1
M5016	AUXEND	M5507	/ALARM

All the mnemonics refer to their associated variable, it being necessary to use the NOT operator to refer to its negation, for example:

NOT M5000	NOT /EMERGEN
NOT M5016	NOT AUXEND

CNC logic inputs and outputs can be grouped in:

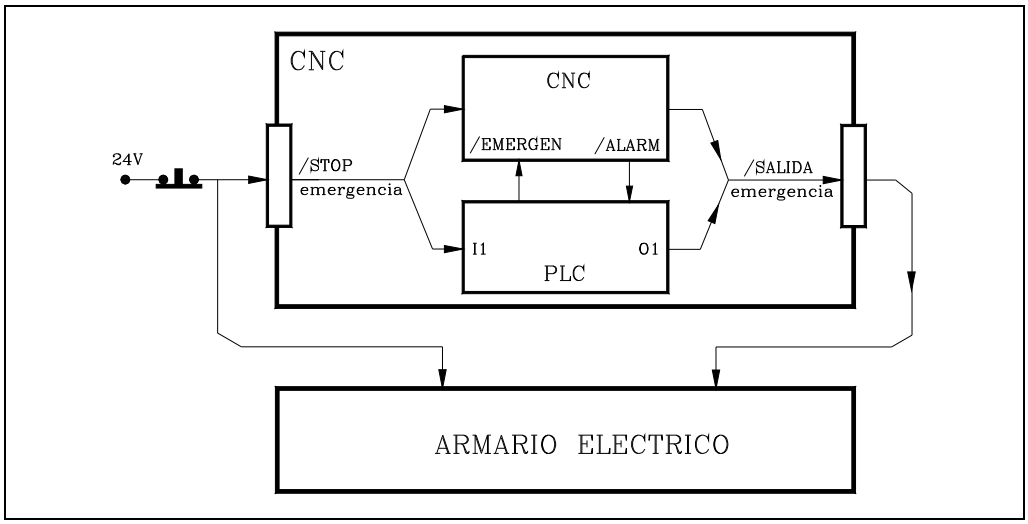
- General logic inputs.
- Axis logic inputs.
- Spindle logic inputs.
- Key inhibiting logic inputs.
- Logic inputs of the PLC channel
- General logic outputs.
- Axis logic outputs.
- Spindle logic outputs.
- Logic outputs of key status
- Logic inputs of the PLC channel

10.1 General logic inputs

These inputs must always be defined in the PLC program.
/EMERGEN (M5000) /STOP (M5001)
/FEEDHOL (M5002) /XFERINH (M5003)

/EMERGEN (M5000)

There are to ways to cause an emergency at the CNC, by activating the physical input /Emergency stop (pin 10 of connector X2) or the general logic input "/EMERGEN" from the PLC.



When the PLC sets the "/EMERGEN" input low (0V), the CNC stops the axes and the spindle and it displays the corresponding error message.

Also, the CNC activates the "/EMERGENCY OUTPUT" and "/ALARM" signals to let the outside world and the PLC know that an emergency has occurred at the CNC.

The CNC does not allow executing programs and it aborts any attempt to move the axes or the spindle while the "/EMERGEN" input is low (0V).

When the PLC brings the "/EMERGEN" input back high (24V), the CNC deactivates the "/EMERGENCY OUTPUT" and "/ALARM" signals to let the outside world and the PLC know that there is no longer an emergency at the CNC.

Example

I-EMERG AND (rest of conditions) = /EMERGEN

If the external emergency input is activated or any other emergency occurs, the general logic input /EMERGEN of the CNC. When there is no emergency, this signal must remain high.

/STOP (M5001)

When the PLC sets this signal low, the CNC stops the part program, and maintains spindle rotation.

In order to continue executing the program, as well as setting this signal at a high logic level, the general logic input CYSTART must be activated.

The treatment which this /STOP signal receives is similar to that given to the STOP key on the CNC front panel keeping all the keys enabled even when the /STOP signal is at low logic level (0) .

Example

( ) = /STOP

There is always permission to execute the part program.

/FEEDHOL (M5002)

When the PLC sets this signal low, the CNC stops the axes (maintaining spindle rotation). When the signal returns to the high logic level, the movement of the axes continues.

If the /FEEDHOL signal is activated (0V) in a block without motion, the CNC will continue the execution of the program until detecting a block with motion.

Example

() = /FEEDHOL

There is always permission to move the axes.

/XFERINH (M5003)

If the PLC sets this signal low, the CNC prevents the following block from starting, but finishes the one it is executing. When the signal returns to high logic level, the CNC continues to execute the program.

Example

( ) = /XFERINH

There is always permission to execute the next block.

CYSTART (M5007)

If the START key is pressed on the front panel of the CNC, this is indicated to the PLC by means of the general logic output START.

If the PLC program considers that there is nothing to prevent the part program form being executed, the CYSTART signal must be set at a high logic level, thus beginning the execution of the program.

The CNC will indicate by means of the general logic output INCYCLE that the program is being executed. As of that moment the CYSTART can return to low logic level.

Example

START AND (rest of conditions) = CYSTART

When the cycle START key is pressed, the CNC activates the general logic output START. The PLC must check that the rest of the conditions (hydraulic, safety devices, etc.) are met before setting the general input CYSTART high in order to start executing the program

SBLOCK (M5008)

When the PLC sets this signal high, the CNC changes to the single block execution mode.

The treatment this signal receives is similar to that given to the "single block" softkey.

MANRAPID (M5009)

If the PLC sets this signal at a high logic level, the CNC selects rapid feed for all the movements executed in JOG mode.

When the signal returns to a low logic level, the movements executed in JOG mode are made at the previously-selected feedrate.

The treatment which this signal receives is similar to that given to the rapid feedrate key on the control panel.

The EXRAPID (M5057) signal is similar, but for movements in execution mode.

OVRCAN (M5010)

If the PLC sets this signal at a high logic level, the CNC selects 100% feedrate override, irrespective of whether this is selected by the PLC, DNC, program or by the front panel switch.

While the OVERCAN signal is activated (logic 1), the CNC will apply in each mode 100 % of the feedrate corresponding to that mode.

LATCHM (M5011)

This allows the type of JOG key operation to be selected in JOG mode.

If the PLC sets this signal low, the axes will only move while the corresponding JOG key is pressed.

If the PLC sets this signal at a high logic level, the axes will move from the moment the corresponding JOG key is pressed until the STOP key or other JOG key is pressed. In this case, the movement will be transferred to that indicated by the new key.

ACTGAIN2 (M5013)

The axes and the spindle can have 3 sets of gains and accelerations.

By default, the first set is always assumed. The one indicated by the a.m.p. and s.m.p.: ACCTIME (P18), PROGAIN (P23), DERGAIN (P24) and FFGAIN (P25).

g.m.p. ACTGAIN2 (P108) indicates with which functions or in which mode the second set is applied, the one set by a.m.p. ACCTIME2 (P59), PROGAIN2 (P60), DERGAIN2 (P61) and FFGAIN2 (P62) or s.m.p. ACCTIME2 (P47), PROGAIN2 (P48), DERGAIN2 (P49) and FFGAIN2 (P50).

The gains and accelerations can also be changed from the PLC regardless of the active operating mode or function. To do this, use general input ACTGAIN2 (M5013).

ACTGAIN2 (M5013) = 0	The CNC assumes the first set.
ACTGAIN2 (M5013) = 1	The CNC assumes the second set.



The change of gains and accelerations is always made at the beginning of the block.

When working in round corner (G5), the change does not take place until G07 is programmed.

RESETIN (M5015)

This signal will be treated by the CNC when the JOG mode is selected and there is no movement of the axes or when a program to be executed is selected and it is not running.

When there is a rising edge (leading edge) of this signal (change from low to high) the CNC assumes the initial machining conditions selected by the machine parameter.

The CNC will indicate by means of the general logic output RESETOUT that this function has been selected.

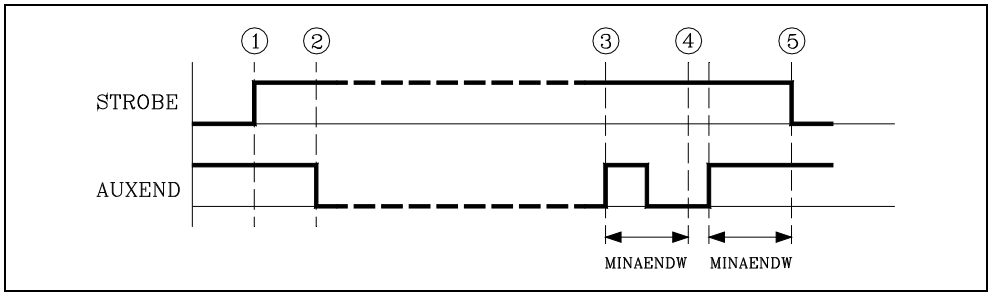
The treatment received by this signal is similar to that given to the RESET key on the front panel.

AUXEND (M5016)

This signal is used in the execution of auxiliary functions M, S and T, to tell the CNC that the PLC is executing them.

It operates in the following way:

1. Once the block has been analyzed and after sending the corresponding values in the variables "MBCD1-7", "SBCD", "TBCD" and "T2BCD", the CNC will tell the PLC by means of the general logic outputs "MSTROBE", "SSTROBE", "TSTROBE" and "T2STROBE" that the required auxiliary functions must be executed.



2. When the PLC detects that one of the STROBE signals is active, it must deactivate the general logic input "AUXEND" to tell the CNC that the execution of the corresponding function or functions is starting.
3. The PLC will execute all the auxiliary functions required, it being necessary to analyze the "MSTROBE", "SSTROBE", "TSTROBE", "T2STROBE" general logic outputs and the "MBCD1-7", "SBCD", "TBCD" and "T2BCD" variables in order to do this.

   Once this has been executed the PLC must activate the general logic input "AUXEND" to indicate to the CNC that the processing of the required functions was completed.
4. Once the general "AUXEND" input is activated, the CNC will require that this signal be kept active for a time period greater than the value given to the g.m.p. "MINAENDW" (P30).

   This way, erroneous interpretations of this signal by the CNC due to an improper PLC program logic are avoided .
5. Once the period of time MINAENDW has elapsed with the general input "AUXEND" at a high logic level, the CNC will deactivate the general logic outputs "MSTROBE", "SSTROBE", "TSTROBE", "T2STROBE" to tell the PLC that the execution of the required auxiliary function or functions has been completed.

TIMERON (M5017)

The CNC is provided with a timer which can be enabled and disabled. By means of this logic CNC input, it will be enabled (timing) when the PLC sets the signal TIMERON at a high logic level.

This general purpose timer can be accessed by means of the internal variable TIMER.

TREJECT (M5018)

The PLC sets this signal at a high logic level in order to tell the CNC to reject the tool in use, even though it may not have come to the end of its service life. An important application is to replace the tool when the PLC detects that it is broken.

PANELOFF (M5019)

The PLC sets this signal high to tell the CNC that the keyboard is deactivated.

It is recommended to change the state of this mark by means of an accessible external input since the PLC will not stop and the CNC will receive the error message in each new PLC cycle scan; thus preventing access to any of the PLC modes.

PLCABORT (M5022)

The PLC sets this signal high to indicate to the CNC that it must stop the PLC axes. It also cancels the rest of the movement and the possible blocks that might have been sent from the PLC.

Once this process is ended, the CNC automatically deactivates this signals.

On power-up, the CNC sets this mark to "0".

PLCREADY (M5023)

This mark indicates the PLC status.

PLCREADY = 0 PLC stopped.

PLCREADY = 1 PLC running.

If this mark is set to 0. The PLC program will stop.

This mark MUST be set to 1 so the CNC allows the spindle and/or the axes to be moved. Otherwise, it will issue the corresponding error message.

INT1 (M5024) **INT2 (M5025)** **INT3 (M5026)** **INT4 (M5027)**

The PLC sets one of these signals to logic state "1" to "tell" the CNC to interrupt the execution of the currently running program and jump to execute the interruption subroutine whose number is indicated in the general machine parameter "INT1SUB" (P35), "INT2SUB" (P36), "INT3SUB" (P37) or "INT4SUB" (P38) respectively.

All these inputs have the same priority and are active by level (not by flank or edge). Only the first one being detected high ("1") will be attended to.

The status of these signals "INT1", "INT2", "INT3", "INT4" are not stored; therefore, it is recommended to activate these marks at the PLC by means of an instruction of the "=SET" type. These marks will be deactivated automatically when starting the execution of the corresponding subroutine.

An interruption subroutine cannot, in turn, be interrupted.

BLKSKIP1 (M5028)

The PLC sets this signal at a high logic level to tell the CNC that the block skip condition "/ or /1" is met, therefore, the blocks which have this block skip condition will not be executed.

BLKSKIP2 (M5029)

The PLC sets this signal at a high logic level to tell the CNC that the block skip condition "/ or /2" is met, therefore, the blocks which have this block skip condition will not be executed.

BLKSKIP3 (M5030)

The PLC sets this signal at a high logic level to tell the CNC that the block skip condition "/ or /3" is met, therefore, the blocks which have this block skip condition will not be executed.

M01STOP (M5031)

The PLC sets this signal at a high logic level to tell the CNC to stop the execution of the part program when the auxiliary (miscellaneous) M01 function is executed.

RETRACE (M5051)

The CNC takes into account this input when retracing is available, g.m.p. RETRACAC other than zero. The retracing function can also be activated while function G51 (look-ahead) is active.

If while executing a part-program the PLC sets this signal high, retracing is activated. The CNC interrupts program execution and starts executing backwards what has been done so far.

When the PLC sets this signal back low, retracing is canceled. The CNC starts executing forward what was done backwards and it will go on to execute the part of the program that was not machined.

Retracing executes backwards the current block plus up to 75 blocks that were already executed.

The retracing function ends in the following cases:

- When the previous 75 blocks are retraced.
- When retraced all the way to the beginning of the program.
- When retracing back to block G51 (turn look-ahead on).
- When finding a block that contains an M function (only if RETRACAC = 1).
- When finding a block that contains an S or a T function.
- When finding a high-level language block.

In all cases, the CNC activates the RETRAEND (M5522) signal to let the PLC know that all possible blocks have been executed.

While the retracing function is active, neither tool inspection nor MDI operations are possible.

The Retracing function cannot be activated while a canned cycle is active.



When working with G51 active, it must be borne in mind that several blocks may go by from the moment the RETRACE mark until the machine begins retracing. Also, the look-ahead calculations will be different when moving forth and when moving back and, consequently both tool paths may be slightly different.

ACTLIM2 (M5052)

The PLC sets this signal high to "tell" to the CNC to activate the second travel limits set by means of variables LIMPL(X-C) and LIMMI(X-C).

The second travel limit of each axis will be taken into account if the first one has been set using a.m.p. LIMIT+ (P5) and LIMIT- (P6).

HNLINARC (M5053)

This signal is used when either the "path handwheel" or "path jog" work mode has been selected using general input "MASTRHND (M5054)". It allows selecting the type of movement.

M5053 = 0	Linear path.
M5053 = 1	Arc path.

For a linear path, the path angle must be indicated by the MASLAN variable and for an arc, the center coordinates must be indicated by the MASCFI and MASCSE variables

Variables MASLAN, MASCFI and MASCSE may be read and written from the CNC, DNC and PLC.

MASTRHND (M5054)

The PLC sets this signal high to "tell" the CNC to activate the "path handwheel" or the "path jog" mode.

M5054 = 0	Normal work mode with handwheels or jog keys.
M5054 = 1	"Path handwheel" or "path jog" function on.

EXRAPID (M5057)

The CNC only takes this signal into account if parameter RAPIDEN has been set to ·1· or ·2·.

If the PLC sets this signal high, the programmed movements are executed as follows.

"RAPIDEN" = 1	When the mark is activated, the programmed movements are executed in rapid. There is no need to press the "rapid" key.
RAPIDEN = 2	When the mark is activated, the "rapid" key is enabled. The key must be pressed to make movements in rapid; in other words, both the key and the mark must be active.

When the signal is set back low, the movements are executed at the programmed feedrate.

The treatment which this signal receives is similar to that given to the rapid feedrate key on the control panel.

The MANRAPID (M5009) signal is similar, but for movements in jog mode.

FLIMITAC (M5058)

When the PLC sets this signal high, it limits the feedrate of each axis to the value set by its a.m.p. "FLIMIT (P75)". When this limit is canceled, the CNC recovers the programmed feedrate.

SLIMITAC (M5059)

When the PLC sets this signal high, it limits the spindle speed to the value set by its s.m.p. "SLIMIT (P66)". When this limit is canceled, the CNC recovers the programmed turning speed.

When the spindle is controlled by the PLC by means of the PLCCNTL mark, this limit is ignored.

BLOABOR (M5060)

When the PLC sets this mark high, the CNC ends the movement in progress and starts executing the next block. If the interrupted block had M functions to be executed after the block, they will be executed before going on to the next block.

This mark only affects the execution in automatic mode and the simulation with motion.

This mark does not stay active after the execution. Once executed, the CNC deactivates it. Likewise, if they are activated in a block that does not accept them, they will also be deactivated; they do not stay active for the next block.

These marks affect the following functions.

- It affect motion blocks G0, G1, G2, G3.
- They affect the dwell programmed with G4.
- It affects the look-ahead. In this type of programs with very small blocks, it is not possible to stop at the same block where the "BLOABOR" mark is detected. In these cases, it will be canceled at the block where the axis is fully decelerated.

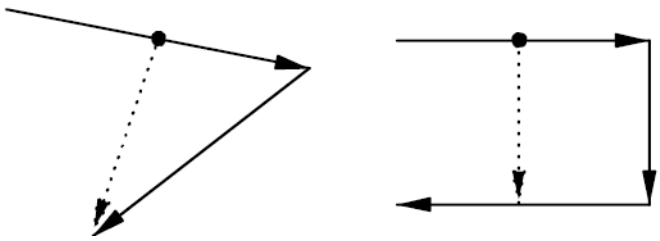
These marks do not affect the following functions.

- It does not affect motionless blocs, which will be executed.
- It does not affect the M functions that are executed after the block. These functions are always executed, even if the movement of the block is interrupted.
- It does affect not affect threading blocks G33. Nor does it affect the regular tapping or rigid tapping cycles regardless of the value of parameter STOPTAP.
- It does not affect spindle positioning blocks M19. If the spindle positioning is in a block containing axis movement, it aborts the movement of the axes, but it completes the positioning of the spindle.
- It does not interrupt the execution of function G74 (machine reference zero or home search).

Considerations for the execution

These marks do not affect block preparation. When canceling the execution of a block, the next movement is carried out up to the prepared target coordinates; no preparation is done.

On the other hand, only the programmed axes are involved in the next movement. The rest of the axes are ignored, even if there is a real difference in position because the previous block has been aborted.



Path 1 Path 2

The solid lines represent the programmed paths and the dashed lines the real paths, after activating the BLOABOR mark.

If a block is aborted and then the RETRACE function is activated, the retraced path (backwards) will not be the same as the one traveled forward. The two paths will not coincide either when aborting a block while the RETRACE function is active.

ACTGAINT (M5063)

The axes and the spindle can have 3 sets of gains and accelerations.

By default, the first set is always assumed. The one indicated by the a.m.p. and s.m.p.: ACCTIME (P18), PROGAIN (P23), DERGAIN (P24) and FFGAIN (P25).

G.m.p ACTGAINT indicates with which functions or in which work mode the third set is applied, the one indicated by a.m.p. ACCTIMET (P92), PROGAINT (P93), DERGAINT (P94) y FFGAINT (P95), and s.m.p. ACCTIMET (P81), PROGAINT (P82), DERGAINT (P83) and FFGAINT (P84).

The gains and accelerations can also be changed from the PLC regardless of the active operating mode or function. To do this, use general input ACTGAINT (M5063).

ACTGAINT (M5063) = 1 The CNC assumes the third set.



The change of gains and accelerations is always made at the beginning of the block.

When working in round corner (G5), the change does not take place until G07 is programmed.

SKIPCYCL (M5064)

In drilling, tapping and rigid tapping cycles of the mill model CNC, it is now possible to withdraw the tool to the starting plane and, once there, stop the spindle.

Once the retraction has been completed, the user will have the choice to finish the hole, go to the next one or go into tool inspection.

General logic input SKIPCYCL (M5064) is used to go on to the next hole once the retraction has been completed.

RETRACYC (M5065)

In drilling, tapping and rigid tapping cycles of the mill model CNC, it is now possible to withdraw the tool to the starting plane and, once there, stop the spindle.

Once the retraction has been completed, the user will have the choice to finish the hole, go to the next one or go into tool inspection.

The PLC activates this mark and the CNC cancels it automatically once the Z axis stopped and before beginning to retract.

SETTMEM (M5066)

PLC mark used by the OEM to activate an error during a tool change. When this mark is activated, the CNC activates the TMINEM mark.

RESTMEM (5067)

PLC mark that allows canceling the CNC's error state. This mark is activated when the user confirms that the tool magazine has been inspected and that everything is OK to resume work.

10.2 Axis logic inputs.

There are several groups of logic inputs (LIMIT, DECEL, etc.) which refer to the possible axes of the machine by means of digits 1 through 3 (LIMIT+2, DECEL1,etc.) or using the axis name (LIMIT+X, DECELZ, etc.).

The marks of the axes that do not exist in the machine parameters assume the value of the M2045 mark, that's always set to 0.

When monitoring the PLC program, it shows the edited marks, either with a letter or with a number. However, the resource windows created from monitoring will replace the marks with axis names by the marks with axis numbers. For example:

SERVOXON by SERVO1ON

SERVOZON by SERVO2ON if there is no Y axis, but there are X and Z axes.

The use of mnemonics with the axis name is available from versions V9.0x and V10.0x on. If PLC programs older than this version have defined these marks as symbols, when compiling the program will issue an error on this line.

Example: DEF ENABLEX M333

Mnemonics using numbers 1 through 3.

These signals are numbered as the logic order of the axes; it is not related to the values assigned to g.m.p. AXIS1 (P0) through AXIS8 (P7).

For example, if the CNC controls the X, Z and Y axes, the order will be: X, Y, Z and, therefore:

LIMIT+1, LIMIT-1, DECEL1, etc.	for the X axis:
LIMIT+2, LIMIT-2, DECEL2, etc.	for the Y axis:
LIMIT+3, LIMIT-3, DECEL3, etc.	for the Z axis:

Mnemonics using the axis name.

The mnemonics of the signals refer to the axis name.

Mnemonics with axis names offer the advantage that if an axis is eliminated, the PLC program will still be consistent with the rest of the axes.

LIMIT+1 (M5100) | LIMIT-1 (M5101)
LIMIT+2 (M5150) | LIMIT-2 (M5151)
LIMIT+3 (M5200) | LIMIT-3 (M5201)

The PLC sets these signals at a high logic level in order to tell the CNC that the corresponding axis has overrun the end of its range of movement in the positive (+) or negative (-) direction indicated by the limit switch.

In this case, the CNC stops axis feed and spindle rotation and displays the corresponding error on screen.

In manual (JOG) operating mode the axis which has overrun its range of travel can be moved in the correct direction in order to place it within the correct range of travel.

DECEL1 (M5102) **DECEL2 (M5152)** **DECEL3 (M5202)**

These signals are used by the CNC when machine reference search is made.

If the PLC sets one of these signals high, this indicates to the CNC that the machine reference search switch of the corresponding axis has been pressed.

When this signal is activated in the machine reference search mode, the CNC decelerates the axis, changing the rapid approach feedrate indicated by the a.m.p. "REFEED1", with the slow feedrate indicated by the a.m.p. "REFEED2". After decelerating it accepts the following reference signal from the corresponding axis feedback system as being valid.

INHIBIT1 (M5103) **INHIBIT2 (M5153)** **INHIBIT3 (M5203)**

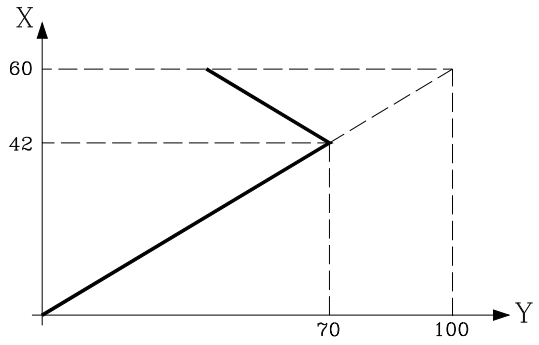
The PLC sets one of these signals at a high logic level in order to tell the CNC to prevent any movement of the corresponding axis. This movement will continue when the PLC sets this signal at the low logic level once more.

If the inhibited axis is moving together with other axes, all these stop moving until the signal returns to the low logic level.

MIRROR1 (M5104) **MIRROR2 (M5154)** **MIRROR3 (M5204)**

If the PLC sets one of these signals at a high logic level, the CNC applies mirror image to the movement of the corresponding axis.

It must be borne in mind that if this signal is activated during a programmed movement, the CNC will only apply mirror image to the movement, not to the final coordinate.




```
N00 G01 X0 Y0 F1000
N10 G01 X70 Y42
N20 G01 X100 Y60
N30 M30
```

If, when executing the programmed movement in block N20 the signal corresponding to the X axis "MIRROR1" is active, the CNC will apply mirror image to the remaining movement in X.

This way, the new end of travel point will be X40 Y60.

By means of the activation of these signals, symmetrical parts can be executed by using a single program, for example, soles of shoes.

In order to obtain the same effect as functions G11, G12, G13 and G14, it is necessary for the corresponding axis or axes to be positioned at part zero when these signals are activated.

SWITCH1 (M5105) **SWITCH2 (M5155)** **SWITCH3 (M5205)**

When having 2 axes controlled by the same servo drive, this mark may be used to toggle the velocity commands between the two axes.

DRO1 (M5106) **DRO2 (M5156)** **DRO3 (M5206)**

These inputs, together with the corresponding "SERVOON" inputs make it possible to operate with the axes as DRO.

In order for the axis to work in DRO mode, its "DRO" input must be high and its corresponding "SERVOON" input must be low.

When an axis works as a DRO, the positioning loop is open and its following error is ignored while in motion

If the DRO signal is brought back low, the axis will no longer behave as a DRO and the CNC will take as position value its current position assigning a 0 value to the following error.

SERVO1ON (M5107) **SERVO2ON (M5157)** **SERVO3ON (M5207)**

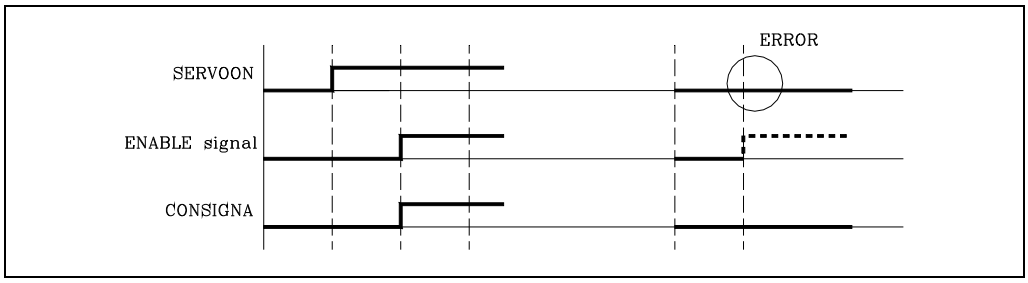
When one of these signals is set high, the CNC closes the positioning loop of the corresponding axis.

If set low, the CNC does not close the position loop of the axis. Any position deviation is stored as following error, thus when the signal gets back high, the axis moves to return to position

These signals are controlled by the PLC and when the positioning loop is to be closed, they will be processed by the CNC according to the value given to machine parameter "DWELL" (P17) for the axes.

DWELL = 0

When a.m.p. DWELL (P17) for the axis to be moved is set to 0, the CNC will check the status of the SERVOON signal at the time when the ENABLE must be output.



If the SERVOON signal is high, the CNC allows the movement of this axis by activating the ENABLE signal and providing the required velocity command output.

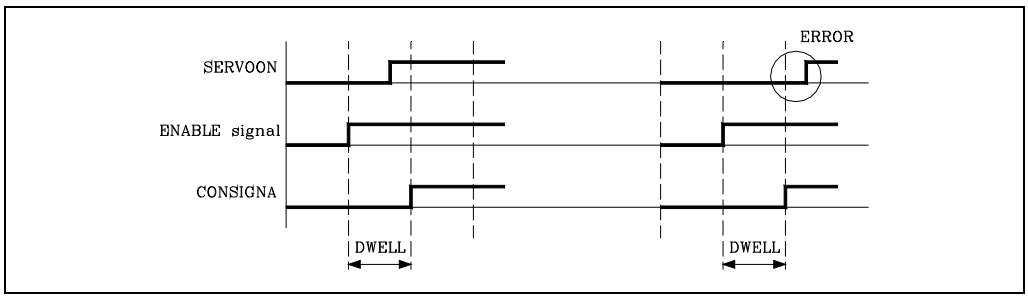
On the other hand, if the SERVOON signal is low or if it changes during the movement of the axes, the CNC stops the axes feed and the spindle rotation displaying the corresponding error message.

DWELL<>0

When a.m.p. DWELL (P17) for the axis to be moved is set to other than "0", the CNC will check the status of the SERVOON signal at the time when the ENABLE must be output.

When this signal (SERVOON) is high, the CNC allows the movement of the axis by activating the ENABLE signal and providing the required velocity command output.

On the other hand, if the SERVOON signal is low, the CNC activates the ENABLE signal and after "waiting" for a time period indicated in DWELL, it checks again the status of the SERVOON signal. If it is high, the required spindle velocity command will be output. If low, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.



Also, if the SERVOON signal changes states during the movement of the axis, the CNC stops the axes feed and the spindle rotation displaying the corresponding error message.

AXIS+1 (M5108) | AXIS-1 (M5109)
AXIS+2 (M5158) | AXIS-2 (M5159)
AXIS+3 (M5208) | AXIS-3 (M5209)

The CNC uses these signals when working in the manual (JOG) operating mode.

If the PLC sets one of these signals high, the CNC will move the corresponding axis in the direction indicated, positive (+) or negative (-). This movement will be performed at the feedrate override % currently selected.

The treatment which these signals receive is similar to that given to the JOG keys of the operator panel.

SPENA1 (M5110) | DRENA1 (M5111)
SPENA2 (M5160) | DRENA2 (M5161)
SPENA3 (M5210) | DRENA3 (M5211)

The CNC uses these signals when communicating with the drive via CAN. Every time the PLC sets one of these signals high, the CNC lets the corresponding drive know about it.

These signals correspond to the "speed enable" and "drive enable" signals of the drive. The drive manual describes how the two signals work, however, remember the following:

- Both signals must be initialized low when powering up the PLC.
- For normal drive operation, both signals must be set high.
- A down flank (trailing edge) of the DRENA signal (drive enable) turns off the power circuit of the drive and the motor loses its torque. In this situation, the motor is no longer governed and it will stop when its kinetic energy runs out. (Stop by friction).
- A trailing edge of the SPENA signal (speed enable) switches the "internal velocity reference" to "0" rpm and brakes the motor while maintaining its torque. Once the motor has stopped, the drive's power circuit is turned off and the motor has no torque.

ELIMINA1 (M5113) **ELIMINA2 (M5163)** **ELIMINA3 (M5213)**

If the PLC sets one these signals high, the CNC does not display the corresponding axis but keeps controlling it. Same as when setting a.m.p. DFORMAT (P1) =3.

The ELIMINA mark can be activated and deactivated at any time and it also cancels the feedback alarms which the machine parameter does not do.

SMOTOF1 (M5114) **SMOTOF2 (M5154)** **SMOTOF3 (M5214)**

The SMOTIME (P58) filter set for each axis with parameter P58 can be canceled from the PLC.

This SMOTIME filter will be activated or deactivated at the beginning of the block. If one of these logic inputs is activated or deactivated while the CNC is overlapping blocks being executed in round corner, it will be ignored until that operation is finished.

LIM1OFF (M5115) **LIM2OFF (M5165)** **LIM3OFF (M5215)**

The PLC sets one of these signals high so that the CNC ignores the software limits of the corresponding axis.

MANINT1 (M5116) **MANINT2 (M5166)** **MANINT3 (M5216)**

The PLC sets one of these signals high to activate the additive handwheel on each axis. Only one additive handwheel may be enabled at a time. If there are more than one mark active, only the first one will be attended to.

When a program is in execution and the mark associated with an axis is activated, it calculates the movement to be applied to that axis according to the resolution of the handwheel.

DIFFCOM1 (M5117) **DIFFCOM2 (M5167)** **DIFFCOM3 (M5217)**

Depending on the logic level of these signals, it corrects the theoretical difference between the master and the slave of a Gantry pair, after homing both axes of that pair.

The theoretical difference between the master and the slave is corrected as follows:

- With the leading edge (up flank) of DIFFCOMaxis while SERVOaxisON = 1.
- With the leading edge (up flank) of SERVOaxisON while DIFFCOMaxis = 1.

  In this case, to correct the theoretical difference between master and slave, both the master and the slave axes must be set as Gantry axis or as DRO axis. Otherwise, the upflank of the SERVOaxisON mark corrects the following error of the slave axis.

10.3 Spindle logic inputs.

LIMIT+S (M5450) I LIMIT-S (M5451) main spindle

The CNC uses this signal while searching home when the spindle changes to working in closed loop (M19). The CNC only considers the signals for the currently selected spindle.

The PLC sets one of the signals high to tell the CNC that the spindle has overrun its range of travel in the positive (+) or negative (-) direction.

In this case, the CNC stops axis feed and spindle rotation and displays the corresponding error on screen.

DECELS (M5452) Main spindle

The CNC uses this signal while searching home when the spindle changes to working in closed loop (M19). The CNC only considers the signals for the currently selected spindle.

The PLC sets this signal high to indicate to the CNC that the reference search switch is pressed.

When this signal is activated in the reference search mode the CNC decelerates the spindle, changing the rapid approach speed indicated by the s.m.p. REFEED1 (P34), with the slow feedrate indicated by the s.m.p. REFEED2 (P35). After decelerating, it accepts the following reference signal from the spindle feedback systems as being valid.

SPDLEINH (M5453) Main spindle

The CNC considers these 2 signals at all times so both spindles can be controlled by the PLC.

When the PLC sets this signal high, the CNC outputs a zero velocity command for the spindle.

SPDLEREV (M5454) Main spindle

The CNC considers these 2 signals at all times so both spindles can be controlled by the PLC.

When the PLC sets this signal high, the CNC reverses the programmed spindle turning direction.

If while being this signal high, a block containing an M3 or M4 is executed, the spindle will start turning in the opposite direction.

SMOTOFS (M5455) Main spindle

The SMOTIME (P46) filter set for the spindle with parameter P46 can be canceled from the PLC.

This SMOTIME filter will be activated or deactivated at the beginning of the block. If one of these logic inputs is activated or deactivated while the CNC is overlapping blocks being executed in round corner, it will be ignored until that operation is finished.

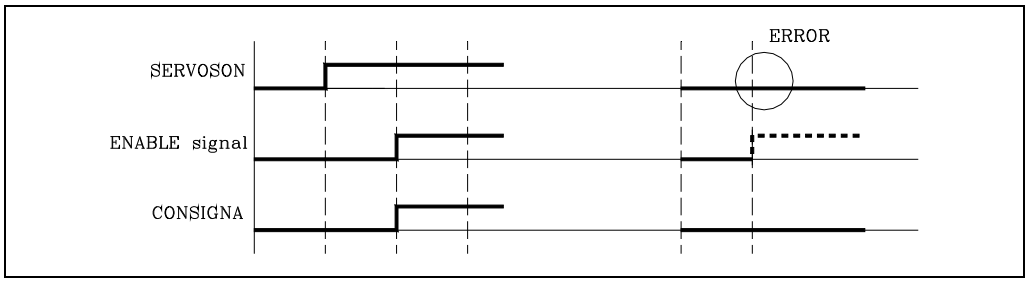
SERVOSON (M5457) Main spindle

These signals are controlled by the PLC and the CNC will process them only when the spindle is working in closed loop (M19). Its treatment depends on the value assigned to s.m.p. DWELL (P17).

DWELL = 0

If s.m.p. DWELL (P17) has been set to "0", the CNC will check the status of the SERVOSON signal at the time when the ENABLE signal is to be output.

If the SERVOSON signal is high, the CNC will allow the spindle to rotate by activating the ENABLE signal and providing the required velocity command output.



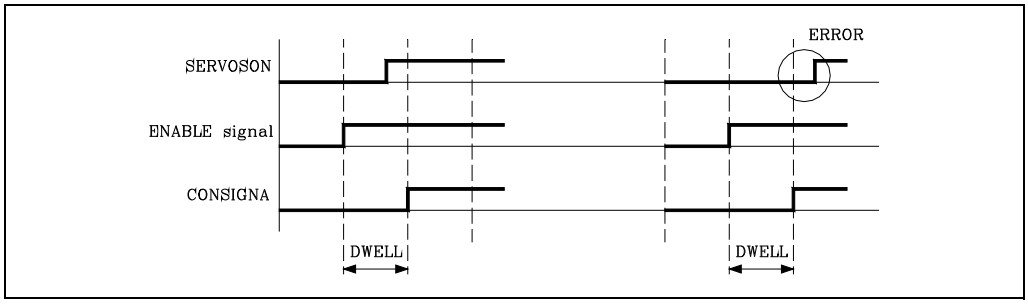
On the other hand, if the SERVOSON signal is low or if it changes to low during the rotation of the spindle, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.

DWELL<>0

If s.m.p. DWELL (P17) has been set to other than "0", the CNC will check the status of the SERVOSON signal at the time when the ENABLE signal is to be output.

If the SERVOSON signal is high, the CNC will allow the spindle to rotate by activating the ENABLE signal and providing the required velocity command output.

On the other hand, if the SERVOSON signal is low, the CNC will activate the ENABLE signal and, after waiting for a time period indicated by the value given to "DWELL", the CNC checks the SERVOSON signal again. If it is high, the required spindle velocity command will be output. If low, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.



Also, if it changes to low during the rotation of the spindle, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.

GEAR1 (M5458) GEAR2 (M5459) GEAR3 (M5460) GEAR4 (M5461) Main spindle

The PLC uses these signals to indicate to the CNC which spindle gear is currently selected (high logic level). The CNC only considers the signals for the currently selected spindle.

When any of the miscellaneous functions M41, M42, M43 or M44 is programmed, the CNC will "tell" the PLC so it selects the desired gear even if it is already selected.

When working with automatic gear changer, the CNC will check the currently selected gear (GEAR1... GEAR4) and if it does not match the selected speed, the CNC will let the PLC know using the relevant auxiliary function (M41, M42, M43 or M44) to select it.

Once the PLC selects the proper gear, it indicates it to the CNC by means of the logic input corresponding to the spindle (GEAR1 ...). GEAR4).

The spindle gear change depends on the setting of functions M41 through M44 in the M function table:

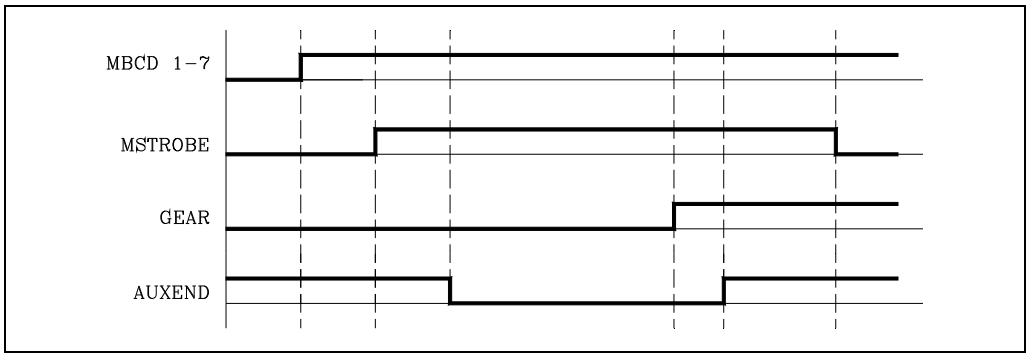
The M41, M42, M43 or M44 function uses the "AUXEND" signal:

The CNC indicates to the PLC the selected gear M41, M42, M43 or M44 in one of the registers "MBCD1" through "MBCD7" and it activates the "MSTROBE" signal to let the PLC "know" that it must execute it.

When the PLC detects the activation of the "MSTROBE" signal it must deactivate the general logic input "AUXEND" to "tell" the CNC that the execution of the gear change has started.

Once executed this function, the PLC will inform the CNC that the new gear has been selected by means of the logic input corresponding to the spindle ("GEAR1".... "GEAR4").

The PLC, then, activates the logic input "AUXEND" to "tell" the CNC that the execution of the gear change has been completed.



Once the "AUXEND" input is activated, the CNC will require that this signal be kept active for a time period greater than the value given to the g.m.p. "MINAENDW" (P30).

This way, erroneous interpretations of this signal by the CNC due to an improper PLC program logic are avoided .

Once the "MINAENDW" time has elapsed with the "AUXEND" general input kept high, the CNC will check whether the new spindle gear has been selected by verifying that the corresponding input (GEAR1... GEAR4) is set high.

If it is, it will cancel the general logic output "MSTROBE" to "tell" the PLC that the gear change has finished and if the corresponding input (GEAR1... GEAR4) is not selected, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.

If the M41, M42, M43 or M44 function does not use the "AUXEND" signal:

- The CNC indicates to the PLC the selected gear M41, M42, M43 or M44 in one of the registers "MBCD1" through "MBCD7" and it activates the "MSTROBE" signal to let the PLC "know" that it must execute it.
- The CNC will keep the output "MSTROBE" active during the time indicated by means of g.m.p. MINAENDW (P30).
- After this time, the CNC will check whether the new spindle gear has been physically selected by verifying that the corresponding GEAR input (GEAR1... GEAR4) is set high.
- If it is not selected, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.

SPENAS (M5462) I DRENAS (M5463) Main spindle

The CNC uses these signals when communicating with the drive via CAN. Every time the PLC sets one of these signals high, the CNC lets the corresponding drive know about it.

These signals correspond to the "speed enable" and "drive enable" signals of the drive. The drive manual describes how the two signals work, however, remember the following:

- Both signals must be initialized low when powering up the PLC.
- For normal drive operation, both signals must be set high.
- A down flank (trailing edge) of the DRENA signal (drive enable) turns off the power circuit of the drive and the motor loses its torque. In this situation, the motor is no longer governed and it will stop when its kinetic energy runs out. (Stop by friction).
- A trailing edge of the SPENA signal (speed enable) switches the "internal velocity reference" to "0" rpm and brakes the motor while maintaining its torque. Once the motor has stopped, the drive's power circuit is turned off and the motor has no torque.

MPLCFM19 (M5464) | M19FEED (R505) Main spindle

The CNC only considers the signals for the currently selected spindle.

The PLC uses the "PLCM19" signal to indicate to the CNC the positioning and rapid synchronized speed value to assume when operating in closed loop (M19).

When this input is low, the CNC assumes the value set by s.m.p. "REFEED1" (P34)

When this input is high, the CNC assumes the value set by the spindle input register "M19FEED" (R505).

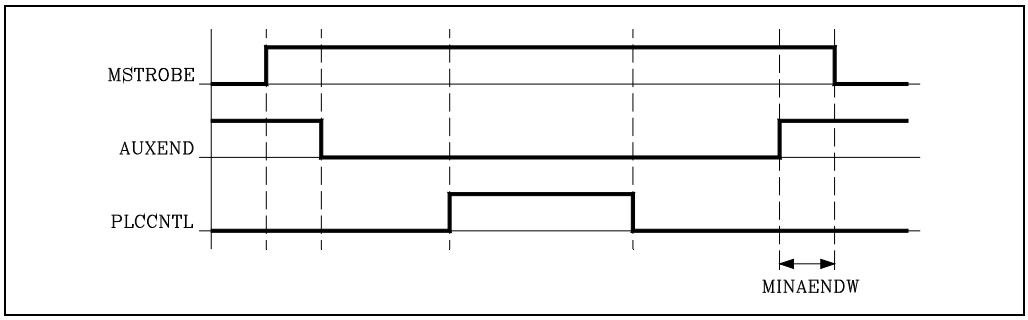
The "M19FEED" value is given in 0.0001º/min.

PLCCNTL (M5465) Main spindle

The CNC considers these 2 signals at all times so both spindles can be controlled by the PLC. This is used to tell the CNC that the spindle is controlled directly by the PLC (high logic level).

It is used, for example, for oscillating the spindle during a gear change or for changing tools.

The following example shows how a new spindle speed is selected involving a gear change.



After analyzing the block and detecting the gear change the CNC indicates this to the PLC in one of the "MBCD1-7" registers (M41 to M44) and will activate the general logic output "MSTROBE" to tell the PLC that it must execute it.

The PLC will deactivate the logic input AUXEND to tell the CNC that the treatment of the auxiliary function is starting.

After calculating the value corresponding to the residual velocity command S for the gear change, the PLC will indicate this to the CNC by means of the register "SANALOG", afterwards setting the signal "PLCCNTL" at a high logic level.

At this time the CNC will output the velocity command indicated in the register SANALOG.

Once the requested gear change has been made, the new active speed will be indicated to the CNC (spindle logic inputs GEAR1 to GEAR4).

In order to give the control of the spindle back to the CNC, the signal "PLCCNTL" must be set low.

Finally, the PLC will activate the logic input AUXEND once more to tell the CNC that the execution of the auxiliary function has been completed.

SANALOG (R504) Main spindle

The CNC considers these 2 signals at all times so both spindles can be controlled by the PLC. The PLC will indicate by means of this 32 bit register the spindle velocity command which the CNC must send out when it is controlled by the PLC.

SANALOG=32767 corresponds to an analog command of 10 V.

(10/32767) 0.305185 millivolts of analog command correspond to SANALOG=1.

This way, for 4V of velocity command, the following must be programmed:

SANALOG = (4x32767)/10 = 13107

For -4V of analog command, the following must be programmed:

SANALOG = (-4x32767)/10 = -13107

ELIMIS (M5456) Main spindle

If the PLC sets this signal high, the CNC does not display the corresponding spindle but keeps controlling it. Same as when setting a.m.p. DFORMAT (P1) =4.

This mark can be activated and deactivated at any time and it also cancels the feedback alarms which the machine parameter does not do.

10.4 Key inhibiting logic inputs.

KEYDIS1 (R500) **KEYDIS2 (R501)** **KEYDIS3 (R502)**
KEYDIS4 (R503)

The PLC can individually inhibit the operation of the panel keys, setting the corresponding bit of one of these 32-bit registers high.

Register	Bit	Inhibited key	Register	Bit	Inhibited key
KEYDIS1	0	F	KEYDIS2	0	B
KEYDIS1	1	L	KEYDIS2	1	H
KEYDIS1	2	Q	KEYDIS2	2	N
KEYDIS1	3	W	KEYDIS2	3	S
KEYDIS1	4	[SHIFT]	KEYDIS2	4	Y
KEYDIS1	5	9	KEYDIS2	5	[RESET]
KEYDIS1	6	6	KEYDIS2	6	[ESC]
KEYDIS1	7	3	KEYDIS2	7	[MAIN MENU]
KEYDIS1	8	E	KEYDIS2	8	A
KEYDIS1	9	K	KEYDIS2	9	G
KEYDIS1	10	P	KEYDIS2	10	M
KEYDIS1	11	V	KEYDIS2	11	R
KEYDIS1	12	[CAPS]	KEYDIS2	12	X
KEYDIS1	13	8	KEYDIS2	13	[ENTER]
KEYDIS1	14	5	KEYDIS2	14	[HELP]
KEYDIS1	15	2	KEYDIS2	15	
KEYDIS1	16	D	KEYDIS2	16	.
KEYDIS1	17	J	KEYDIS2	17	0
KEYDIS1	18	O	KEYDIS2	18	-
KEYDIS1	19	U	KEYDIS2	19	+
KEYDIS1	20	[SP]	KEYDIS2	20	
KEYDIS1	21	7	KEYDIS2	21	
KEYDIS1	22	4	KEYDIS2	22	
KEYDIS1	23	1	KEYDIS2	23	
KEYDIS1	24	C	KEYDIS2	24	[PG DW]
KEYDIS1	25	I	KEYDIS2	25	[PG UP]
KEYDIS1	26	Ñ	KEYDIS2	26	[⬆]
KEYDIS1	27	T	KEYDIS2	27	[⬇]
KEYDIS1	28	Z	KEYDIS2	28	[➡]
KEYDIS1	29	=	KEYDIS2	29	[⬅]
KEYDIS1	30	/	KEYDIS2	30	[CL]
KEYDIS1	31	*	KEYDIS2	31	[INS]

The keys inhibited by register KEYDIS3 (R502) depends on the CNC model available (M or T model).

Register	Bit	Inhibited key (·M· model)	Register	Bit	Inhibited key (·T· model)
KEYDIS3	0	F1	KEYDIS3	0	F1
KEYDIS3	1	F2	KEYDIS3	1	F2
KEYDIS3	2	F3	KEYDIS3	2	F3
KEYDIS3	3	F4	KEYDIS3	3	F4
KEYDIS3	4	F5	KEYDIS3	4	F5
KEYDIS3	5	F6	KEYDIS3	5	F6
KEYDIS3	6	F7	KEYDIS3	6	F7
KEYDIS3	7		KEYDIS3	7	
KEYDIS3	8	X+	KEYDIS3	8	3rd axis +
KEYDIS3	9	Y+	KEYDIS3	9	
KEYDIS3	10	Z+	KEYDIS3	10	X+
KEYDIS3	11	4+	KEYDIS3	11	
KEYDIS3	12	5+	KEYDIS3	12	4th axis +
KEYDIS3	13	Spdl override +	KEYDIS3	13	Spdl override +
KEYDIS3	14	Spdl CW	KEYDIS3	14	Spdl CW
KEYDIS3	15	START	KEYDIS3	15	START
KEYDIS3	16		KEYDIS3	16	
KEYDIS3	17		KEYDIS3	17	Z-
KEYDIS3	18	Rapid feedrate	KEYDIS3	18	Rapid feedrate
KEYDIS3	19		KEYDIS3	19	Z+
KEYDIS3	20		KEYDIS3	20	
KEYDIS3	21		KEYDIS3	21	
KEYDIS3	22	Spdl stop	KEYDIS3	22	Spdl stop
KEYDIS3	23		KEYDIS3	23	
KEYDIS3	24	X-	KEYDIS3	24	3rd axis -
KEYDIS3	25	Y-	KEYDIS3	25	
KEYDIS3	26	Z-	KEYDIS3	26	X-
KEYDIS3	27	4-	KEYDIS3	27	
KEYDIS3	28	5-	KEYDIS3	28	4th axis -
KEYDIS3	29	Spdl override -	KEYDIS3	29	Spdl override -
KEYDIS3	30	Spdl CCW	KEYDIS3	30	Spdl CCW
KEYDIS3	31	STOP	KEYDIS3	31	STOP

- Bit ·14· (spdl CW) corresponds to the key for starting the spindle clockwise.
- Bit ·30· (spdl CCW) corresponds to the key for starting the spindle counterclockwise.
- Bit ·22· (spdl stop) corresponds to the key for stopping the spindle.

Register KEYDIS4 disables the positions of the feedrate override switch (feedrate % selector).

Register	Bit	Inhibited key
KEYDIS4	0	Handwheel x100
KEYDIS4	1	Handwheel x10
KEYDIS4	2	Handwheel x1
KEYDIS4	3	Jog 10000
KEYDIS4	4	Jog 1000
KEYDIS4	5	Jog 100
KEYDIS4	6	Jog 10
KEYDIS4	7	Jog 1
KEYDIS4	8	Feedrate override 0%
KEYDIS4	9	Feedrate override 2%
KEYDIS4	10	Feedrate override 4%
KEYDIS4	11	Feedrate override 10%
KEYDIS4	12	Feedrate override 20%
KEYDIS4	13	Feedrate override 30%
KEYDIS4	14	Feedrate override 40%
KEYDIS4	15	Feedrate override 50%

Register	Bit	Inhibited key
KEYDIS4	16	Feedrate override 60%
KEYDIS4	17	Feedrate override 70%
KEYDIS4	18	Feedrate override 80%
KEYDIS4	19	Feedrate override 90%
KEYDIS4	20	Feedrate override 100%
KEYDIS4	21	Feedrate override 110%
KEYDIS4	22	Feedrate override 120%
KEYDIS4	23	
KEYDIS4	24	
KEYDIS4	25	
KEYDIS4	26	
KEYDIS4	27	
KEYDIS4	28	
KEYDIS4	29	
KEYDIS4	30	
KEYDIS4	31	

Should one of the inhibited positions of the feedrate override switch be selected, the CNC will take the value corresponding to the nearest uninhibited position below it. If all of them are inhibited, the lowest will be taken (0%).

For example, if only positions 110% and 120% of the switch are allowed and position 50% is selected, the CNC will take a value of 0%.

10.5 Logic inputs of the PLC channel

To govern the axes managed by PLC.

/FEEDHOP (M5004)

It is similar to general logic input /FEEDHOL (M5002), but for the PLC channel.

When the PLC sets this signal low, the CNC stops the axes (maintaining spindle rotation). When the signal returns to the high logic level, the movement of the PLC axes continues.

This input must always be defined in the PLC program.

/XFERINP (M5005)

It is similar to general logic input /XFERINH (M5003), but for the PLC channel.

If the PLC sets this signal low, the CNC prevents the following block from starting, but finishes the one it is executing. When the signal returns to high logic level, the CNC continues to execute the program.

This input must always be defined in the PLC program.

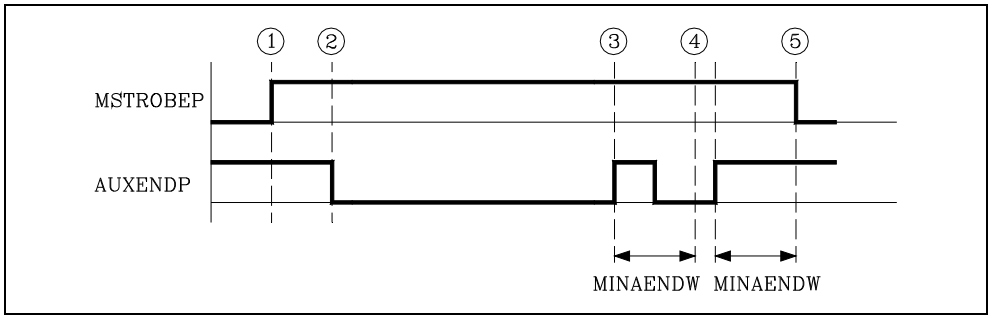
AUXENDP (M5006)

It is similar to general logic input AUXEND (M5016), but for the PLC channel.

This signal is used in the execution of auxiliary M functions to tell the CNC that the PLC is executing them.

It operates in the following way:

1. Once the block has been analyzed and after passing the corresponding values in variables "MBCD1-7", the CNC will tell the PLC through the general logic output "MSTROBEP" that the required auxiliary function or functions must be executed.



2. When the PLC detects the activation of the "MSTROBEP" signal it must deactivate the general logic input "AUXENDP" to "tell" the CNC that the execution of the function has started.
3. The PLC will execute all the required auxiliary functions analyzing general logic output "MSTROBEP" and variables "MBCDP1" through "MBCDP7" (R565 through R571).

   Once this has been executed the PLC must activate the general logic input "AUXENDP" to indicate to the CNC that the processing of the required functions was completed.
4. Once the general "AUXENDP" input is activated, the CNC will require that this signal be kept active for a time period greater than the value given to the g.m.p. "MINAENDW" (P30).

   This way, erroneous interpretations of this signal by the CNC due to an improper PLC program logic are avoided .
5. Once the period of time MINAENDW has elapsed with the general input "AUXENDP" at a high logic level, the CNC will deactivate the general logic output "MSTROBEP" to tell the PLC that the execution of the required auxiliary function or functions has been completed.

BLOABORP (M5061)

It is similar to general logic input BLOABOR (M5060), but for the PLC channel.

When the PLC sets this mark high, the CNC ends the movement in progress and starts executing the next block. If the interrupted block had M functions to be executed after the block, they will be executed before going on to the next block.

This mark only affects the execution in automatic mode and the simulation with motion.

This mark does not stay active after the execution. Once executed, the CNC deactivates it. Likewise, if they are activated in a block that does not accept them, they will also be deactivated; they do not stay active for the next block.

These marks affect the following functions.

- It affect motion blocks G0, G1, G2, G3.
- They affect the dwell programmed with G4.
- It affects the look-ahead. In this type of programs with very small blocks, it is not possible to stop at the same block where the "BLOABOR" mark is detected. In these cases, it will be canceled at the block where the axis is fully decelerated.

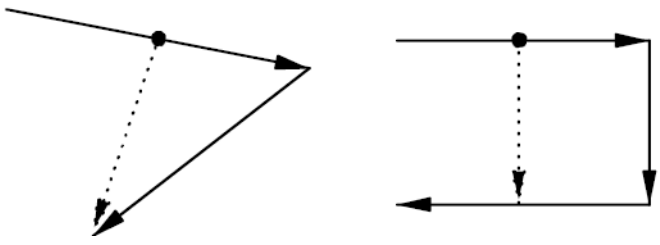
These marks do not affect the following functions.

- It does not affect motionless blocs, which will be executed.
- It does not affect the M functions that are executed after the block. These functions are always executed, even if the movement of the block is interrupted.
- It does affect not affect threading blocks G33. Nor does it affect the regular tapping or rigid tapping cycles regardless of the value of parameter STOPTAP.
- It does not affect spindle positioning blocks M19. If the spindle positioning is in a block containing axis movement, it aborts the movement of the axes, but it completes the positioning of the spindle.

Considerations for the execution

These marks do not affect block preparation. When canceling the execution of a block, the next movement is carried out up to the prepared target coordinates; no preparation is done.

On the other hand, only the programmed axes are involved in the next movement. The rest of the axes are ignored, even if there is a real difference in position because the previous block has been aborted.



The solid lines represent the programmed paths and the dashed lines the real paths, after activating the BLOABORP mark.

If a block is aborted and then the RETRACE function is activated, the retraced path (backwards) will not be the same as the one traveled forward. The two paths will not coincide either when aborting a block while the RETRACE function is active.

10.6 General logic outputs

CNCREADY (M5500)

The CNC activates and maintains this signal high if the autotest which the CNC makes when it is powered up has not detected any problem. Should any hardware error be detected (RAM, over-temperature, etc.) this signal is set low.

Example

CNCREADY AND (rest of conditions) = O1

The emergency output O1 of the PLC must be normally high. Should any problem come up on CNC power-up (CNCREADY), emergency output O1 must be set low (0V).

START (M5501)

The CNC sets this signal high in order to tell the PLC that the START key on the front panel has been pressed.

If the PLC program considers that there is nothing to prevent the part program from starting, it must set the general logic input CYSTART at a high logic level, thereby starting the execution of the program.

When the CNC detects an up flank (logic level change from low to high) at the CYSTART signal, it reset the START signal to low.

Example

START AND (rest of conditions) = CYSTART

When the cycle START key is pressed, the CNC activates the general logic output START. The PLC must check that the rest of the conditions (hydraulic, safety devices, etc.) are met before setting the general input CYSTART high in order to start executing the program

FHOUT (M5502)

The CNC sets this signal high in order to tell the PLC that the execution of the program is stopped due to one of the following causes:

- Because the CONTROL PANEL STOP key has been pressed.
- Because the general logic input /STOP has been set low, even though later it has returned high.
- Because the general logic input /FEEDHOL is low.

RESETOUT (M5503)

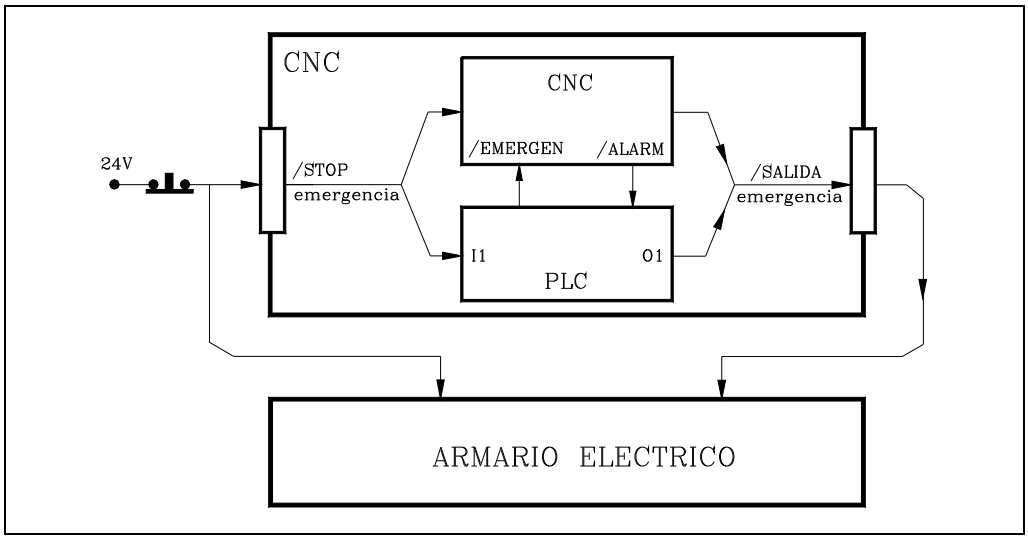
The CNC sets this signal high for 100 milliseconds, in order to tell the PLC that it is under initial conditions because the Reset key on the front panel has been pressed or because the general logic input RESETIN has been activated.

LOPEN (M5506)

The CNC sets this signal high in order to tell the PLC that the positioning loop of the axes is open since an error has occurred.

/ALARM (M5507)

The CNC sets this signal low in order to tell the PLC that an alarm or emergency condition has been detected. This signal will be set high once again, once the message from the CNC has been eliminated and the cause of the alarm has disappeared.



Likewise, while this signal is low, the CNC keeps the emergency output (pin 2 of connector X2) active (low).

Example

/ALARM AND (other conditions) = O1

The emergency output O1 of the PLC must be normally high. If an alarm or an emergency is detected at the CNC, the emergency output O1 must be set low (0V).

MANUAL (M5508)

The CNC sets this signal high to tell the PLC that the JOG (Manual) operating mode is selected.

AUTOMAT (M5509)

The CNC sets this signal high to tell the PLC that the automatic operating mode is selected.

MDI (M5510)

The CNC sets this signal high to tell the PLC that the MDI mode (manual data input) is selected in one of the operating modes (JOG, automatic, etc).

SBOUT (M5511)

The CNC sets this signal high to tell the PLC that the single block execution mode is selected.

INCYCLE (M5515)

The CNC sets this signal high while executing a block or moving an axis.

Once the execution of the program has been requested by the PLC to the CNC by means of the logic input CYSTART, the latter will indicate that it is being executed by setting the INCYCLE signal high.

This signal is maintained high until the CNC finishes the part program or when this is stopped by means of the STOP key on the CONTROL PANEL or the general logic input /STOP.

If the CNC is in the single block execution mode, the INCYCLE signal is set low as soon as the block execution is concluded.

If the CNC is in JOG mode, the INCYCLE signal is set low as soon as the position indicated has been reached.

If the CNC is in JOG mode and the axes are being jogged, the "INCYCLE" signal goes high while any of the jog keys are pressed.

RAPID (M5516)

The CNC sets this signal high to tell the PLC that a rapid positioning (G00) is being executed.

TAPPING (M5517)

The CNC sets this signal high to tell the PLC that a tapping canned cycle is being executed (G84).

THREAD (M5518)

The CNC sets this signal high to tell the PLC that a threading block is being executed (G33).

PROBE (M5519)

The CNC sets this signal high to tell the PLC that a probing movement is being executed (G75/G76).

ZERO (M5520)

The CNC sets this signal high to tell the PLC that a machine reference search is being executed (G74).

RIGID (M5521)

This output is only available on the mill model. The CNC set this signal high to indicate to the PLC that a RIGID TAPPING operation (G84) is being performed.

RETRAEND (M5522)

The CNC sets this signal high to indicate to the "PLC" that while retracing is active all the possible blocks have been retraced.

For further information, see general input "RETRACE (M5051)".

CSS (M5523)

This output is only available on the lathe model. The CNC sets this signal high to tell the PLC that the constant cutting speed function is selected (G96).

SELECT0 (M5524) **SELECT1 (M5525)** **SELECT2 (M5526)**
SELECT3 (M5527) **SELECT4 (M5528)** **SELECT5 (M5529)**
SELECT6 (M5530) **SELECT7 (M5531)** **SELECTOR (R564)**

The CNC uses these signals to indicate to the PLC the position selected at each keyboard switch.

SELECTOR indicates the position currently selected.

SELECT indicates the value applied by the CNC.

Usually, the two values coincide, except when a position has been selected which has been disabled with the KEYDIS4 input (R503). If while being the 60% and 120% inhibited, the 100% position is selected, SELECTOR will show the selected position (100%) and SELECT will show the value being applied (50%).

	SELECTOR								
	bit (7) SELECT7 ...	bit (6)	bit (5)	bit (4)	bit (3)	bit (2)	bit (1)	bit (0) ... SELECT0	Hex.
Handwheel x100	1	1	1	1	0	0	0	0	F0
Handwheel x10	1	1	1	1	0	0	0	1	F1
Handwheel x1	1	1	1	1	0	0	1	0	F2
JOG 10000	1	1	1	1	0	0	1	1	F3
JOG 1000	1	1	1	1	0	1	0	0	F4
JOG 100	1	1	1	1	0	1	0	1	F5
JOG 10	1	1	1	1	0	1	1	0	F6
JOG 1	1	1	1	1	0	1	1	1	F7
Feedrate override 0%	0	0	0	0	1	0	0	0	08

	SELECTOR								Hex.
	bit (7) SELECT7 ...	bit (6)	bit (5)	bit (4)	bit (3)	bit (2)	bit (1)	bit (0) ... SELECT0	
Feedrate override 2%	0	0	0	1	1	0	0	0	18
Feedrate override 4%	0	0	1	0	1	0	0	0	28
Feedrate override10%	0	0	1	1	1	0	0	0	38
Feedrate override 20%	0	1	0	0	1	0	0	0	48
Feedrate override 30%	0	1	0	1	1	0	0	0	58
Feedrate override 40%	0	1	1	0	1	0	0	0	68
Feedrate override 50%	0	1	1	1	1	0	0	0	78
Feedrate override 60%	1	0	0	0	1	0	0	0	88
Feedrate override 70%	1	0	0	1	1	0	0	0	98
Feedrate override 80%	1	0	1	0	1	0	0	0	A8
Feedrate override 90%	1	0	1	1	1	0	0	0	B8
Feedrate override 100%	1	1	0	0	1	0	0	0	C8
Feedrate override 110%	1	1	0	1	1	0	0	0	D8
Feedrate override 120%	1	1	1	0	1	0	0	0	E8

MSTROBE (M5532)

The CNC sets this signal high to tell the PLC that it must execute the auxiliary M function or functions which are indicated in registers "MBCD1" to "MBCD7" (R550 to R556).

SSTROBE (M5533)

This signal will be used when operating a spindle with BCD coded S signal. s.m.p. SPDLTYPE (P0).

The CNC sets this signal high to tell the PLC that it must execute the auxiliary S function which is indicated in the register "SBCD" (R557).

TSTROBE (M5534)

The CNC sets this signal high to tell the PLC that it must execute the auxiliary S function which is indicated in the register "TBCD" (R558).

In this register the CNC will tell the PLC the position of the magazine where the selected tool is.

If the g.m.p. RANDOMTC (P25) has been set so it is not a random magazine, the magazine pocket position coincides with the tool number.

T2STROBE (M5535)

This register is used when a special tool change has been made, family code or 200 or with machining centers with a non-random tool magazine (g.m.p. RANDOMTC (P25).

The CNC sets this signal high to tell the PLC that it must execute a second auxiliary T function indicated in the register "T2BCD" (R559).

In this register the CNC indicates to the PLC the position of the magazine in which the tool which was on the spindle must be placed.

ADVINPOS (M5537)

It is used on punch presses that have an eccentric cam as a punching system.

The CNC sets this signal high a specific time period before the axes reach position. This time is set by g.m.p. ANTIME (P69).

This reduces idle time, thus increasing the number of punches per minute.

INTEREND (M5538) I INPOS (M5539)

The CNC uses these two signals to let the PLC "know" that the theoretical interpolation between axes has been completed (INTEREND) and that all the axes involved in the interpolation are in position (INPOS).

The CNC sets the "INTEREND" signal high when the interpolation is ended being low while in execution.

When the CNC verifies that all the axes have been within the dead band (in position zone INPOSW P19) for a time period indicated in the a.m.p INPOTIME (P20), it will consider that all of them are in position and it will inform the PLC by setting the logic output "INPOS" high.

The logic output "INTEREND" can be used when it is required to activate mechanisms before the axes reach their position.

DM00 (M5547)

The CNC sets this signal high to tell the PLC that the auxiliary function M00 (program stop) is programmed in the block being executed.

DM01 (M5546)

The CNC sets this signal high to tell the PLC that the auxiliary function M01 (conditional stop) is programmed in the block being executed.

DM02 (M5545)

The CNC sets this signal high to tell the PLC that the auxiliary function M02 (program end) is programmed in the block being executed.

DM03 (M5544)

The CNC sets this signal high to tell the PLC that the spindle is turning clockwise or that the auxiliary function M03 is programmed in the block being executed.

DM04 (M5543)

The CNC sets this signal high to tell the PLC that the spindle is turning counter-clockwise or that the auxiliary function M04 is programmed in the block being executed.

DM05 (M5542)

The CNC sets this signal high to tell the PLC that the spindle is stopped or that the auxiliary function M05 is programmed in the block being executed.

DM06 (M5541)

The CNC sets this signal high to tell the PLC that the spindle is stopped or that the auxiliary function M06 is programmed in the block being executed (tool change).

DM08 (M5540)

The CNC sets this signal high to tell the PLC that the coolant output is activated or that the auxiliary function M08 is programmed in the block being executed.

DM09 (M5555)

The CNC sets this signal high to tell the PLC that the coolant output is deactivated or that the auxiliary function M09 is programmed in the block being executed.

DM19 (M5554)

The CNC sets this signal high to tell the PLC that it is working with spindle orientation or that the auxiliary function M19 is programmed in the block being executed.

DM30 (M5553)

The CNC sets this signal high to tell the PLC that the auxiliary function M30 (program end) is programmed in the block being executed.

DM41 (M5552)

The CNC sets this signal high to tell the PLC that the first spindle gear is selected or that the auxiliary function M41 is programmed in the block being executed.

DM42 (M5551)

The CNC sets this signal high to tell the PLC that the second spindle gear is selected or that the auxiliary function M42 is programmed in the block being executed.

DM43 (M5550)

The CNC sets this signal high to tell the PLC that the third spindle gear is selected or that the auxiliary function M43 is programmed in the block being executed.

DM44 (M5549)

The CNC sets this signal high to tell the PLC that the fourth spindle gear is selected or that the auxiliary function M44 is programmed in the block being executed.

RETRACT (M5567)

In drilling, tapping and rigid tapping cycles of the mill model CNC, it is now possible to withdraw the tool to the starting plane and, once there, stop the spindle.

Once the retraction has been completed, the user will have the choice to finish the hole, go to the next one or go into tool inspection.

General logic output RETRACT (M5567) is activated at the end of the stop and canceled when the retraction in drilling or mill type threading has ended.

In the retraction of axes in the lathe model, general logic output RETRACT (M5567) is activated when pressing [STOP] and the CNC begins the retraction (withdrawal). This mark stays active until the retraction distances set in G233 are reached.

TMINEM (M5569)

Mark that is activated when the CNC detects an error during a tool change. This mark stays in memory until it is canceled by the RESTMEM mark or using the [REMOVE ERROR] option that appears in the error message.

10.7 Logic outputs of the axes

There are several groups of logic inputs (ENABLE, DIR, etc.) which refer to the possible axes of the machine by means of digits 1 through 3 (ENABLE2, DIR1,etc.) or using the axis name (ENABLEX, DIRZ, etc.).

The marks of the axes that do not exist in the machine parameters assume the value of the M2045 mark, that's always set to 0.

When monitoring the PLC program, it shows the edited marks, either with a letter or with a number. However, the resource windows created from monitoring will replace the marks with axis names by the marks with axis numbers. For example:

ENABLEX with ENABLE1

ENABLEZ by ENABLE2 if there is no Y axis, but there are X and Z axes.

The use of mnemonics with the axis name is available from versions V9.0x and V10.0x on. If PLC programs older than this version have defined these marks as symbols, when compiling the program will issue an error on this line.

Example: DEF ENABLEX M333

Mnemonics using numbers 1 through 3.

These signals are numbered as the logic order of the axes; it is not related to the values assigned to g.m.p. AXIS1 (P0) through AXIS8 (P7).

For example, if the CNC controls the X, Z and Y axes, the order will be: X, Y, Z and, therefore:

ENABLE1, DIR1, REFPOIN1, INPOS1	for the X axis:
ENABLE2, DIR2, REFPOIN2, INPOS2	for the Y axis:
ENABLE3, DIR3, REFPOIN3, INPOS3	for the Z axis:

Mnemonics using the axis name.

The mnemonics of the signals refer to the axis name.

Mnemonics with axis names offer the advantage that if an axis is eliminated, the PLC program will still be consistent with the rest of the axes.

ENABLE1 (M5600) **ENABLE2 (M5650)** **ENABLE3 (M5700)**

The CNC sets these signals at a high logic level to tell the PLC to allow the corresponding axis to move.

DIR1 (M5601) **DIR2 (M5651)** **DIR3 (M5701)**

The CNC uses these signals to tell the PLC in which direction the axes move.

If the signal is high this indicates that the corresponding axis moves in a negative direction.

If the signal is low this indicates that the corresponding axis moves in a positive direction.

REFPOIN1 (M5602) **REFPOIN2 (M5652)** **REFPOIN3 (M5702)**

The CNC sets these signals high to tell the PLC that the machine reference search has been made already. The CNC forces a home search on an axis by setting its mark low.

The marks are set low in the following instances:

- On CNC power-up.
- After executing the sequence [SHIFT] [RESET].
- When the feedback is direct through the axes board and a feedback alarm occurs.
- When modifying certain machine parameters; for example, number of axes.

In all these cases, a home search must be carried out so the signal is set back high.

DRSTAF1 (M5603) | DRSTAS1 (M5604)
DRSTAF2 (M5653) | DRSTAS2 (M5654)
DRSTAF3 (M5703) | DRSTAS3 (M5704)

The CNC uses these signals when communicating with the drive via CAN and they indicate the drive status.

	DRSTAF*	DRSTAS*
Starting the drive up. Actuating the main switch of the electrical cabinet supplies 24 Vdc to the drive. The drive runs an internal test. If correct, it activates the output "System OK".	0	0
Drive ready. From that moment on, the power supply must be turned on. When there is power at the drive bus, it is ready to output torque.	0	1
Bus with voltage. To do that, activate the "drive enable" and "speed enable" inputs.	1	0
Motor with torque. Once the "drive enable" and "speed enable" are activated, the drive is running properly.	1	1

When an internal error occurs at the drive, the DRSTAF* and DRSTAS* signals are set low (logic level low).

MAXDIFF1 (M5605) **MAXDIFF2 (M5655)** **MAXDIFF3 (M5705)**

These marks are activated if the position difference between master and slave is not compensated because the coordinate difference is greater than the value of a.m.p. MAXDIFF (P97). This can happen after homing the two axes of a Gantry pair.

This way, the PLC can issue a warning that the position difference between the master and the slave has not been compensated.

ANT1 (M5606) **ANT2 (M5656)** **ANT3 (M5706)**

These signals are related to a.m.p. MINMOVE (P54).

If the axis move is smaller than the value indicated by this a.m.p. MINMOVE (P54), the corresponding axis logic output "ANT1 thru "ANT7" goes high.

INPOS1 (M5607) **INPOS2 (M5657)** **INPOS3 (M5707)**

The CNC sets these signals high to tell the PLC that the corresponding axis is in position.

There is also the general logic output INPOS in which the CNC indicates to the PLC that all the axes have reached their position.

10.8 Spindle logic outputs.

ENABLES (M5950) Main spindle

This signal is used when working with the spindle in closed loop (M19). The CNC only considers the signals for the currently selected spindle.

The CNC sets this signal high to tell the PLC to allow the spindle to move.

DIRS (M5951) Main spindle

This signal is used when working with the spindle in closed loop (M19). The CNC only considers the signals for the currently selected spindle.

The CNC uses this signal to tell the PLC in which direction the spindle is moving.

If the signal is at a high logic level, this indicates that the spindle moves in a negative direction.

If the signal is low, this indicates that the spindle moves in a positive direction.

REFPOINS (M5952) Main spindle

This signal is used when working with the spindle in closed loop (M19). The CNC only considers the signals for the currently selected spindle.

The CNC sets this signal high to tell the PLC that the spindle reference point search has already been made.

This is set low when the CNC is powered up, after executing the [SHIFT] [RESET] sequence or a feedback alarm occurs due to loss of count, and every time a change is made from closed loop (M19) to open loop.

DRSTAFS (M5953) I DRSTASS (M5954) Main spindle

The CNC uses these signals when communicating with the drive via CAN and they indicate the drive status.

	DRSTAF*	DRSTAS*
Actuating the main switch of the electrical cabinet supplies 24 Vdc to the drive. The drive runs an internal test. If correct, it activates the output "System OK".	0	0
From that moment on, the power supply must be turned on. When there is power at the drive bus, it is ready to output torque.	0	1
To do that, activate the "drive enable" and "speed enable" inputs.	1	0
Once the "drive enable" and "speed enable" are activated, the drive is running properly.	1	1

When an internal error occurs at the drive, the DRSTAF* and DRSTAS* signals are set low (logic level low).

REVOK (M5956) Main spindle

The CNC only considers the signals for the currently selected spindle.

When working with M03 and M04 the CNC sets this signal high to tell the PLC that the real spindle revolutions correspond to those programmed.

The CNC will activate this signal every time the real revolutions are within the range defined by s.m.p. "LOSPDLIM" and "UPSPDLIM".

When working with the spindle in closed loop (M19), the CNC sets this signal high if the spindle is stopped.

INPOSS (M5957) Main spindle

This signal is used when working with the spindle in closed loop (M19). The CNC only considers the signals for the currently selected spindle.

The CNC sets this signal high to tell the PLC that the spindle is in position.

10.9 Logic outputs of key status

KEYBD1 (R560) **KEYBD2 (R561)** **KEYBD3 (R562)**
KEYBD4 (R563)

These registers indicate whether or not one of the keys on the keyboard or on the operator panel is pressed.

When one of these keys is pressed, the corresponding bit will be set high and it will return low when the key is released.

Register	Bit	keystroke code
KEYBD1	0	F
KEYBD1	1	L
KEYBD1	2	Q
KEYBD1	3	W
KEYBD1	4	[SHIFT]
KEYBD1	5	9
KEYBD1	6	6
KEYBD1	7	3
KEYBD1	8	E
KEYBD1	9	K
KEYBD1	10	P
KEYBD1	11	V
KEYBD1	12	[CAPS]
KEYBD1	13	8
KEYBD1	14	5
KEYBD1	15	2
KEYBD1	16	D
KEYBD1	17	J
KEYBD1	18	O
KEYBD1	19	U
KEYBD1	20	[SP]
KEYBD1	21	7
KEYBD1	22	4
KEYBD1	23	1
KEYBD1	24	C
KEYBD1	25	I
KEYBD1	26	Ñ
KEYBD1	27	T
KEYBD1	28	Z
KEYBD1	29	=
KEYBD1	30	/
KEYBD1	31	*

Register	Bit	keystroke code
KEYBD2	0	B
KEYBD2	1	H
KEYBD2	2	N
KEYBD2	3	S
KEYBD2	4	Y
KEYBD2	5	[RESET]
KEYBD2	6	[ESC]
KEYBD2	7	[MAIN MENU]
KEYBD2	8	A
KEYBD2	9	G
KEYBD2	10	M
KEYBD2	11	R
KEYBD2	12	X
KEYBD2	13	[ENTER]
KEYBD2	14	[HELP]
KEYBD2	15	
KEYBD2	16	.
KEYBD2	17	0
KEYBD2	18	-
KEYBD2	19	+
KEYBD2	20	
KEYBD2	21	
KEYBD2	22	
KEYBD2	23	
KEYBD2	24	[PG DW]
KEYBD2	25	[PG UP]
KEYBD2	26	[⬆]
KEYBD2	27	[⬇]
KEYBD2	28	[➡]
KEYBD2	29	[⬅]
KEYBD2	30	[CL]
KEYBD2	31	[INS]

The values returned by register KEYBD (R562) depend on the CNC model available (M or T model).

Register	Bit	keystroke code (·M· model)	Register	Bit	keystroke code (·T· model)
KEYBD3	0	F1	KEYBD3	0	F1
KEYBD3	1	F2	KEYBD3	1	F2
KEYBD3	2	F3	KEYBD3	2	F3
KEYBD3	3	F4	KEYBD3	3	F4
KEYBD3	4	F5	KEYBD3	4	F5
KEYBD3	5	F6	KEYBD3	5	F6
KEYBD3	6	F7	KEYBD3	6	F7
KEYBD3	7		KEYBD3	7	
KEYBD3	8	X+	KEYBD3	8	3rd axis +
KEYBD3	9	Y+	KEYBD3	9	
KEYBD3	10	Z+	KEYBD3	10	X+
KEYBD3	11	4+	KEYBD3	11	
KEYBD3	12	5+	KEYBD3	12	4th axis +
KEYBD3	13	Spdl override +	KEYBD3	13	Spdl override +
KEYBD3	14	Spdl CW	KEYBD3	14	Spdl CW
KEYBD3	15	START	KEYBD3	15	START
KEYBD3	16		KEYBD3	16	
KEYBD3	17		KEYBD3	17	Z-
KEYBD3	18	Rapid feedrate	KEYBD3	18	Rapid feedrate
KEYBD3	19		KEYBD3	19	Z+
KEYBD3	20		KEYBD3	20	
KEYBD3	21		KEYBD3	21	
KEYBD3	22	Spdl stop	KEYBD3	22	Spdl stop
KEYBD3	23		KEYBD3	23	
KEYBD3	24	X-	KEYBD3	24	3rd axis -
KEYBD3	25	Y-	KEYBD3	25	
KEYBD3	26	Z-	KEYBD3	26	X-
KEYBD3	27	4-	KEYBD3	27	
KEYBD3	28	5-	KEYBD3	28	4th axis -
KEYBD3	29	Spdl override -	KEYBD3	29	Spdl override -
KEYBD3	30	Spdl CCW	KEYBD3	30	Spdl CCW
KEYBD3	31	STOP	KEYBD3	31	STOP

Register KEYDIS4 refers to the positions of the feedrate override switch (feedrate % selector).

Register	Bit	Key pressed
KEYBD4	0	Handwheel x100
KEYBD4	1	Handwheel x10
KEYBD4	2	Handwheel x1
KEYBD4	3	Jog 10000
KEYBD4	4	Jog 1000
KEYBD4	5	Jog 100
KEYBD4	6	Jog 10
KEYBD4	7	Jog 1
KEYBD4	8	Feedrate override 0%
KEYBD4	9	Feedrate override 2%
KEYBD4	10	Feedrate override 4%
KEYBD4	11	Feedrate override 10%
KEYBD4	12	Feedrate override 20%
KEYBD4	13	Feedrate override 30%
KEYBD4	14	Feedrate override 40%
KEYBD4	15	Feedrate override 50%

Register	Bit	Key pressed
KEYBD4	16	Feedrate override 60%
KEYBD4	17	Feedrate override 70%
KEYBD4	18	Feedrate override 80%
KEYBD4	19	Feedrate override 90%
KEYBD4	20	Feedrate override 100%
KEYBD4	21	Feedrate override 110%
KEYBD4	22	Feedrate override 120%
KEYBD4	23	
KEYBD4	24	
KEYBD4	25	
KEYBD4	26	
KEYBD4	27	
KEYBD4	28	
KEYBD4	29	
KEYBD4	30	
KEYBD4	31	

10.

LOGIC CNC INPUTS AND OUTPUTS

Logic outputs of key status

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

ACCESS TO INTERNAL CNC VARIABLES

11

The CNC has a number of internal variables that may be accessed from the user program, from the PLC program or via DNC. Depending on how they are used, these variables may be read-only or read-write.

Reading and writing variables from the PLC

The PLC provides two instructions (actions) which permit to read or modify the various internal variables of the CNC from the PLC.

Reading variables. –CNCRD– command

The CNCRD command allows reading the CNC's internal variables. Its programming format is:

```
CNCRD (Variable, Register, Mark)
```

This PLC action loads the contents of the indicated variable into the selected register. If this instruction has been executed properly, the PLC will assign a value of "0" to the indicated "error detection" mark and "1" if otherwise.

```
CNCRD (FEED, R150, M200)
```
It loads the value of the feedrate selected at the CNC when working in G94 into the PLC register R150.

When requesting information about a nonexisting variable (i.e. the position value of a nonexisting axis), this instruction will not alter the contents of the register and it will set the selected error mark indicating that the variable does not exist.

Writing variables. –CNCWR– command

The CNCWR command allows writing the CNC's internal variables. Its programming format is:

```
CNCWR (Register, Variable, Mark)
```

This PLC action loads the contents of the indicated register into the selected variable. If this instruction has been executed properly, the PLC will assign a value of "0" to the indicated "error detection" mark and "1" if otherwise.

```
CNCWR (R92, TIMER, M200)
```
It resets the clock enabled by the PLC with the value contained in register R92.

When trying to modify the contents of a nonexisting variable or assign an improper value to it, the selected "error mark" will be set to "1" which will indicate that this instruction is incorrect.

When performing an improper reading or writing request, the PLC will continue the execution of the program unless interrupted by the programmer after having analyzed the "error" mark defined in the instruction.

Identifying the variables in the PLC commands

These variables are accessed from the PLC using high level commands. Each one of these variables is referred to by its mnemonic that must be written in upper-case (capital) letters.

- Mnemonics ending in *(X-C)* indicate a set of 9 elements formed by the corresponding root followed by X, Y, Z, U, V, W, A, B and C.

ORG(X-C) ->	ORGX	ORGY	ORGZ
	ORGU	ORGV	ORGW
	ORGA	ORGB	ORGC

- Mnemonics ending in *n* indicate that the variables are grouped in tables. To access an element of any of these tables, indicate the field of the desired table using the relevant mnemonic followed by the desired element.

TORn ->	TOR1	TOR3	TOR11

  These variables can also be referred to by its corresponding mnemonic and a register that indicates the element number of that table.

TORn ->	TOR R1	TOR R23

```
CNCRD (TOR R222, R100, M102)
```

It assigns the radius value of the offset indicated by Register R222 to register R100

11.1 Variables associated with tools.

These variables are associated with the tool offset table, tool table and tool magazine table, so the values which are assigned to or read from these fields will comply with the formats established for these tables.

Tool table for mill model CNC

The radius (R), length (L) and wear offset (I, K) values of the tool are given in the units set by g.m.p. INCHES.

If, INCHES = 0, in ten-thousandths of a millimeter (±999999999).

If INCHES = 1, in hundred-thousandths of an inch (±393700787).

If rotary axis, in ten-thousandths of a degree (±999999999).

Tool table for lathe model CNC

The length (X, Z), radius (R) and wear offset (I, K) values of the tool are given in the units set by g.m.p. INCHES.

If, INCHES = 0, in ten-thousandths of a millimeter (±999999999).

If INCHES = 1, in hundred-thousandths of an inch (±393700787).

If rotary axis, in ten-thousandths of a degree (±999999999).

The location (tool shape) code (F) will be an integer between 0 and 9.

Tool table at the mill model CNC

The tool offset number is an integer between 0 and 255. The maximum number of tool offsets is limited by g.m.p. NTOFFSET.

The family code is a number between 0 and 255.

0 to 199 if it is a normal tool.

200 to 255 if it is a special tool.

The nominal life is given either in minutes or in operations (0··65535).

The real (actual) life is given either in hundredths of a minute (0··9999999) or in operations (0··999999).

Tool table at the lathe model CNC

The tool offset number is an integer between 0 and 255. The maximum number of tool offsets is limited by g.m.p. NTOFFSET.

The family code is a number between 0 and 255.

0 to 199 if it is a normal tool.

200 to 255 if it is a special tool.

The nominal life is given either in minutes or in operations (0··65535).

The real (actual) life is given either in hundredths of a minute (0··9999999) or in operations (0··999999).

The cutter angle is given in ten-thousandths of a degree (0··359999).

The cutter width is given in the units set by g.m.p. INCHES.

If, INCHES = 0, in ten-thousandths of a millimeter (±999999999).

If INCHES = 1, in hundred-thousandths of an inch (±393700787).

If rotary axis, in ten-thousandths of a degree (±999999999).

The cutting angle is given in ten-thousandths of a degree (0··359999).

Tool magazine table

Each magazine position is represented as follows:

1..255	Tool number.
0	The magazine position is empty.
-1	The magazine position has been canceled.

The tool position in the magazine is represented as follows:

1..255	Position number.
0	The tool is in the spindle.
-1	Tool not found.
-2	The tool is in the change position.

Read-only variables

Variables TOOL, NXTOOL, TOD and NXTOD can only be written from the PLC while no block or part-program is being executed or simulated.

TOOL

Returns the number of the active tool.

```
CNCRD(TOOL,R100,M100)
```
Assigns the number of the active tool to register R100.

TOD

Returns the number of the active tool offset.

NXTOOL

Returns the next tool number, which is selected but is awaiting the execution of M06 to be active.

NXTOD

Returns the number of the tool offset corresponding to the next tool, which is selected but is awaiting the execution of M06 to be active.

TMZPn

Returns the position occupied in the tool magazine by the indicated tool (n).

Read-and-write variables

TLFDn

This variable allows the tool offset number of the indicated tool (n) to be read or modified in the tool table.

TLFFn

This variable allows the family code of the indicated tool (n) to be read or modified in the tool table.

TLFNn

This variable allows the value assigned as the nominal life of the indicated tool (n) to be read or modified in the tool table.

TLFRn

This variable allows the value corresponding to the real life of the indicated tool (n) to be read or modified in the tool table.

TMZTn

This variable allows the contents of the indicated position (n) to be read or modified in the tool magazine table.

HTOR

The HTOR variable indicates the tool radius being used by the CNC to do the calculations.

Being a variable that can be read and written by the CNC and read-only from the PLC and DNC, its value may be different from the one assigned in the table (TOR).

On power-up, after a T function, after a RESET or after an M30 function, it assumes the value of the table (TOR).

Application example

To machine a profile with a residual stock of 0.5 mm running 0.1mm-passes with a tool whose radius is 10 mm.

Assign to the tool radius the value of:

10.5 mm in the table and execute the profile.

10,4 mm in the table and execute the profile.

10,3 mm in the table and execute the profile.

10,2 mm in the table and execute the profile.

10,1 mm in the table and execute the profile.

10,0 mm in the table and execute the profile.

However, if while machining, the program is interrupted or a reset occurs, the table assumes the radius value assigned in that instant (e.g.: 10.2 mm). Its value has changed.

To avoid this, instead of modifying the tool radius in the (TOR) table, use the variable (HTOR) to change the tool radius value used by the CNC to calculate.

Now, if the program is interrupted, the tool radius value initially assigned in the (TOR) table will be correct because it has not changed.

Read-and-write variables of the mill model CNC

TORn

This variable allows the value assigned to the radius of the indicated tool offset (n) in the tool offset table to be read or modified.

```
CNCRD(TOR3,R100,M102);
```
Assigns the R value of tool offset 3 to register R100.

```
CNCWR(R101,TOR3,M101)
```
Assigns the value indicated in R101 to the radius of tool offset 3.

TOLn

This variable allows the value assigned to the length of the indicated tool offset (n) to be read or modified in the tool offset table.

TOIn

This variable allows the value assigned to the wear in radius (I) of the indicated tool offset (n) to be read or modified in the tool offset table.

TOKn

This variable allows the value assigned to the wear in length (K) of the indicated tool offset (n) to be read or modified in the tool offset table.

Read-and-write variables of the lathe model

TOXn

This variable allows reading or modifying the length value along the X axis assigned to the indicated tool offset (n).

```
CNCRD (TOX3, R100, M102)
```
Loads R100 with the length value along X of the tool offset 3.
```
CNCWR (R101, TOX3, M101)
```
Assigns the value indicated in R101 to the length along X of tool offset 3.

TOZn

This variable allows reading or modifying the length value along the Z axis assigned to the indicated tool offset (n).

TOFn

This variable allows reading or modifying the location code (F) of the indicated tool offset (n).

TORn

This variable allows reading or modifying the radius R value of the indicated tool offset (n).

TOIn

This variable allows reading or modifying the length wear value (I) along the X axis of the indicated tool offset (n).

TOKn

This variable allows reading or modifying the length wear value (K) along the Z axis of the indicated tool offset (n).

NOSEAn:

This variable allows reading or modifying the cutter angle assigned to the indicated tool (n) in the tool table.

NOSEWn

This variable allows reading or modifying the cutter width assigned to the indicated tool (n) in the tool table.

CUTAn

This variable allows reading or modifying the cutting angle assigned to the indicated tool (n) in the tool table.

11.2 Variables associated with zero offsets.

These variables are associated with the zero offset table, due to which the values that will be assigned to or read from these fields will comply with the formats established for this table.

The possible zero offsets in addition to the additive offset indicated by the PLC, are G54, G55, G56, G57, G58 and G59.

The values are given in the units set by g.m.p. INCHES.

If, INCHES = 0, in ten-thousandths of a millimeter (±999999999).

If INCHES = 1, in hundred-thousandths of an inch (±393700787).

If rotary axis, in ten-thousandths of a degree (±999999999).

Although there are variables which refer to each axis, the CNC only allows those referring to the axes selected at the CNC. Thus, if the CNC controls the X and Z axes, it only allows the variables ORGX and ORGZ in the case of ORG(X-C).

Read-only variables

ORG(X-C)

Returns the value of the active zero offset in the selected axis. The value of the additive offset indicated by the PLC or by the additive handwheel is not included in this value.

ADIOF(X-C)

It returns the value of the zero offset generated by the additive handwheel in the selected axis.

Read-and-write variables

ORG(X-C)n

This variable allows the value of the selected axis to be read or modified in the table corresponding to the indicated zero offset (n).

```
CNCRD(ORGX 55,R100,M102)
```
Loads register R100 with the X value of G55 in the zero offset table.

```
CNCWR (R101, TOX3, M101)
```
Assigns the value indicated in R101 to the Y value of G54 in the zero offset table.

PLCOF(X-C)

This variable allows the value of the selected axis to be read or modified in the table of additive offsets indicated by the PLC.

Accessing any of the PLCOF(X-C) variables interrupts block preparation and the CNC waits for that command to be executed before resuming block preparation.

11.3 Variables associated with machine parameters

These variables associated with machine parameters are read-only variables.

Refer to the installation and start-up manual to know the format of the values returned. The values of 1/0 correspond to the parameters that are set as YES/NO, +/- or ON/OFF.

Values regarding position and feedrate values will be given in the units set by g.m.p. INCHES.

If, INCHES = 0, in ten-thousandths of a millimeter (±999999999).

If INCHES = 1, in hundred-thousandths of an inch (±393700787).

If rotary axis, in ten-thousandths of a degree (±999999999).

OEM programs or subroutines

These variables may be read and written when executed inside an OEM program or subroutine.

In order to be able to modify these parameters via PLC, an OEM subroutine containing the relevant variables must be executed using the CNCEX command.

For the CNC to assume the new values, operate according to the indicators associated with the machine parameters.

Character	Type of update
//	It is necessary to press the keystroke sequence: [SHIFT] + [RESET] or turn the CNC off and back on.
/	[RESET] must be pressed.
	The rest of the parameters (those unmarked) will be updated automatically, only by changing them.

Read-only variables

MPGn

Returns the value assigned to general machine parameter (n).

```
CNCRD (MPG 8,R100,M102)
```
Loads register R100 with the value of general machine parameter INCHES (P8), If mm, R100 = 0; and if inch, R100 =1.

MP(X-C)n

Returns the value assigned to the machine parameter (n) of the indicated axis (X-C).

```
CNCRD (MPY 1,R100,M102)
```
Assigns the value of Y axis machine parameter DFORMAT (P1) to register R100.

MPSn

Returns the value assigned to the indicated machine parameter (n) of the main spindle.

MPLCn

Returns the value assigned to the indicated machine parameter (n) of the PLC.

11.4 Variables associated with work zones

The values of the limits are given in the units set by g.m.p. INCHES.

If, INCHES = 0, in ten-thousandths of a millimeter (±999999999).

If INCHES = 1, in hundred-thousandths of an inch (±393700787).

If rotary axis, in ten-thousandths of a degree (±999999999).

The status of the work zones are defined according to the following code:

0 = Disabled.

1 = Enabled as no-entry zone.

2 = Enabled as no-exit zone.

Read-and-write variables

FZONE

It returns the status of work zone 1.

FZLO(X-C)

Lower limit of zone 1 along the selected axis (X-C).

FZUP(X-C)

Upper limit of zone 1 along the selected axis (X-C).

The following example shows how it is possible to define as forbidden zone for the X axis the area between coordinates 0 and 100mm (1000000 tenths of microns).

```
<condition>        = MOV 0 R1            = CNCWR(R1, FZLOX, M1)
                   = MOV 1000000 R1      = CNCWR(R1, FZUPX, M1)
                   = MOV 1 R1            = CNCWR(R1, FZONE, M1)
```

SZONE

Status of work zone 2.

SZLO(X-C)

Lower limit of zone 2 along the selected axis (X-C).

SZUP(X-C)

Upper limit of zone 2 along the selected axis (X-C).

TZONE

Status of work zone 3.

TZLO(X-C)

Lower limit of zone 3 along the selected axis (X-C)

TZUP(X-C)

Upper limit of zone 3 along the selected axis (X-C).

FOZONE

Status of work zone 4.

FOZLO(X-C)

Lower limit of zone 4 along the selected axis (X-C).

FOZUP(X-C)

Upper limit of zone 4 along the selected axis (X-C).

FIZONE

Status of work zone 5.

FIZLO(X-C)

Lower limit of zone 5 along the selected axis (X-C).

FIZUP(X-C)

Upper limit of zone 5 along the selected axis (X-C).

11.5 Variables associated with feedrates

Read-only variables associated with the real (actual) feedrate

FREAL

It returns the CNC's real feedrate. It takes into account the feedrate override and the acc/dec of the machine.

In 0.0001 mm/min. or 0.00001 inch/min units.

On Laser cutting machines, it is recommended to use this variable to make the power of the Laser proportional to the actual feedrate at all times.

FREAL(X-C)

It returns the actual (real) CNC feedrate of the selected axis.

In 0.0001 mm/min. or 0.00001 inch/min units.

FTEO(X-C)

It returns the theoretical CNC feedrate of the selected axis.

In 0.0001 mm/min. or 0.00001 inch/min units.

Read-only variables associated with function G94

FEED

It returns the feedrate selected at the CNC by function G94. In mm/minute or inches/minute.

This feedrate may be indicated by program, by PLC or by DNC; the CNC selects one of them, the one indicated by DNC has the highest priority and the one indicated by program has the lowest priority.

DNCF

It returns the feedrate, in mm/minute or inches/minute selected by DNC. If it has a value of 0 it means that it is not selected.

PRGF

It returns the feedrate, in mm/minute or inches/minute selected by program. If it has a value of 0 it means that it is not selected.

Read-write variables associated with function G94

PLCF

It returns the feedrate, in mm/minute or inches/minute selected by PLC. If it has a value of 0 it means that it is not selected.

Read-only variables associated with function G95

FPREV

It returns the feedrate selected at the CNC by function G95. In mm/turn or inches/turn.

This feedrate may be indicated by program, by PLC or by DNC; the CNC selects one of them, the one indicated by DNC has the highest priority and the one indicated by program has the lowest priority.

DNCFPR

It returns the feedrate, in mm/turn or inches/turn selected by DNC. If it has a value of 0 it means that it is not selected.

PRGFPR

It returns the feedrate, in mm/turn or inches/turn selected by program. If it has a value of 0 it means that it is not selected.

Read-write variables associated with function G95

PLCFPR

It returns the feedrate, in mm/turn or inches/turn selected by PLC. If it has a value of 0 it means that it is not selected.

Read-only variables associated with function G32

PRGFIN

It returns the feedrate selected by program, in 1/min.

Likewise, the CNC variable FEED, associated with G94, indicates the resulting feedrate in mm/min or inches/min.

Read-only variables associated with the override

FRO

It returns the feedrate override (%) currently selected at the CNC. It is given in integer values between 0 and "MAXFOVR" (maximum 255).

This feedrate percentage may be indicated by program, by PLC, by DNC or by the front panel; the CNC selects one of them and the priority (from the highest to the lowest) is: by program, by DNC, by PLC and from the front panel switch.

DNCFRO

It returns the feedrate override % currently selected by the DNC. If it has a value of 0 it means that it is not selected.

CNCFRO

It returns the feedrate override % currently selected by the switch.

PRGFRO

This variable may be used to read or modify the feedrate override percentage currently selected by program. It is given in integer values between 0 and "MAXFOVR" (maximum 255). If it has a value of 0 it means that it is not selected.

Read-write variables associated with the override

PLCFRO

It returns the feedrate override % currently selected by the PLC. If it has a value of 0 it means that it is not selected.

PLCCFR

It returns the feedrate percentage currently selected by the PLC's execution channel. It is only set from the PLC, using an integer between 0 and 255.

11.6 Variables associated with coordinates

The values are given in the units set by g.m.p. INCHES.

If, INCHES = 0, in ten-thousandths of a millimeter (±999999999).

If INCHES = 1, in hundred-thousandths of an inch (±393700787).

If rotary axis, in ten-thousandths of a degree (±999999999).

Read-only variables

Accessing any of the variables POS(X-C), TPOS(X-C), APOS(X-C), ATPOS(X-C), DPOS(X-C) or FLWE(X-C) interrupts block preparation and the CNC waits for that command to be executed before resuming block preparation.

POS(X-C)

It returns the real tool base position value referred to machine reference zero (home).

On limit-less rotary axes, this variable takes into account the value of the active zero offset. The values of the variable are between the active zero offset and ±360º (ORG* ± 360º).

If ORG* = 20º it displays between 20º and 380º / displays between -340º and 20º.

If ORG* = -60º it displays between -60º and 300º / displays between -420 and -60º

At the lathe model CNC, the coordinates of each axis are given as follows:

- When read from the CNC, they are given in radius or diameter, depending on the active units system. Check the DIAM variable to know the active units system.
- When read from the PLC, they are always given in radius.

TPOS(X-C)

It returns the theoretical position value (real coordinate + following error) of the tool base referred to machine reference zero (home).

On limit-less rotary axes, this variable takes into account the value of the active zero offset. The values of the variable are between the active zero offset and ±360º (ORG* ± 360º).

If ORG* = 20º it displays between 20º and 380º / displays between -340º and 20º.

If ORG* = -60º it displays between -60º and 300º / displays between -420 and -60º

At the lathe model CNC, the coordinates of each axis are given as follows:

- When read from the CNC, they are given in radius or diameter, depending on the active units system. Check the DIAM variable to know the active units system.
- When read from the PLC, they are always given in radius.

APOS(X-C)

It returns the real tool base position value, referred to part zero, of the selected axis.

At the lathe model CNC, the coordinates of each axis are given as follows:

- When read from the CNC, they are given in radius or diameter, depending on the active units system. Check the DIAM variable to know the active units system.
- When read from the PLC, they are always given in radius.

ATPOS(X-C)

It returns the theoretical position value (real coordinate + following error) of the tool base referred to part zero.

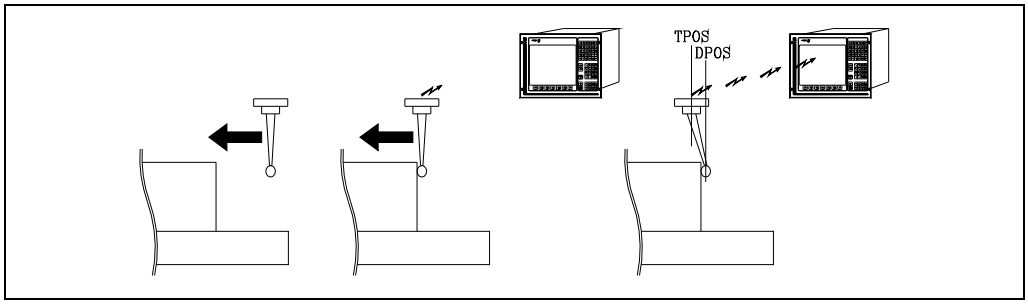
At the lathe model CNC, the coordinates of each axis are given as follows:

- When read from the CNC, they are given in radius or diameter, depending on the active units system. Check the DIAM variable to know the active units system.
- When read from the PLC, they are always given in radius.

DPOS(X-C)

The CNC updates this variable when probing, functions G75 and G76.

When the digital probe communicates with the CNC via infrared beams, there could be some delay (milliseconds) from the time the probe touches the part to the instant the CNC receives the probe signal.



Although the probe keeps moving until the CNC receives the probing signal, the CNC takes into account the value assigned to general machine parameter PRODEL and provides the following information in the variables TPOS(X-C) and DPOS(X-C).

TPOS(X-C)	Actual position of the probe when the CNC receives the probe signal.
DPOS(X-C)	Theoretical position of the probe when the probe touched the part.

FLWE(X-C)

It returns the following error of the selected axis.

DPLY(X-C)

It returns the position value (coordinate) shown on the screen for the selected axis.

Read-and-write variables

DIST(X-C)

These variables may be used to read or modify the distance traveled by the selected axis. This value is accumulative and is very useful when it is required to perform an operation which depends on the distance traveled by the axes, their lubrication for example.

Accessing any of the DIST(X-C) variables interrupts block preparation and the CNC waits for that command to be executed before resuming block preparation.

LIMPL(X-C) **LIMMI(X-C)**

With these variables, it is possible to set a second travel limit for each axis: LIMPL for the upper limit and LIMMI for the lower one.

The PLC activates and deactivates these second limits through general logic input ACTLIM2 (M5052).

The second travel limit will be taken into account if the first one has been set using axis machine parameters LIMIT+ (P5) and LIMIT- (P6).

11.7 Variables associated with electronic handwheels

Read-only variables

HANPF **HANPS** **HANPT** **HANPFO**

They return the pulses of the first (HANPF), second (HANPS), third (HANPT) or fourth (HANPFO) handwheel received since the CNC was turned on.

HANDSE

For handwheels with axis selector button, it indicates whether that button has been pressed or not. A value of ·0· means that it has not been pressed.

Read-and-write variables

HANFCT

It returns the multiplying factor set by PLC for each handwheel.

It must be used when using several electronic handwheels or when using a single handwheel but different multiplying factors (x1, x10, x100) are to be applied to each axis.

C			B			A			W			V			U			Z			Y			X			
c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	lsb

Once the switch has been turned to one of the handwheel positions, the CNC checks this variable and, depending on the values assigned to each axis bit (c, b, a) it applies the multiplying factor selected for each one of them.

c	b	a	
0	0	0	The value indicated at the front panel or keyboard switch.
0	0	1	x1 factor
0	1	0	x10 factor
1	0	0	x100 factor

If there are more than one bit set to "1" for an axis, the least significant bit will be considered. Thus:

c	b	a	
1	1	1	x1 factor
1	1	0	x10 factor

The screen always shows the value selected at the switch.

HBEVAR

It must be used when having a Fagor HBE handwheel.

It indicates whether the HBE handwheel is enabled or not, the axis to be jogged and the multiplying factor to be applied (x1, x10, x100).

				C			B			A			W			V			U			Z			Y			X			
*	^			c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	lsb

(*) Indicates whether the HBE handwheel pulses will be taken into account or not in jog mode.

0 = They are ignored.

1 = They are taken into account.

(^) When the machine has a general handwheel and individual handwheels (associated with an axis), it indicates which handwheel has priority when both are turned at the same time.

0 = The individual handwheel has priority. The relevant axis ignores the pulses from the general handwheel, the rest of the axes don't.

1 = The general handwheel has priority. It ignores the pulses from the individual handwheel.

(a, b, c) Indicate the axis to be moved and the selected multiplying factor.

c	b	a	
0	0	0	The value indicated at the front panel or keyboard switch.
0	0	1	x1 factor
0	1	0	x10 factor
1	0	0	x100 factor

If several axes are selected, the following order of priority is applied: X, Y, Z, U, V, W, A, B, C.

If there are more than one bit set to "1" for an axis, the least significant bit will be considered. Thus:

c	b	a	
1	1	1	x1 factor
1	1	0	x10 factor

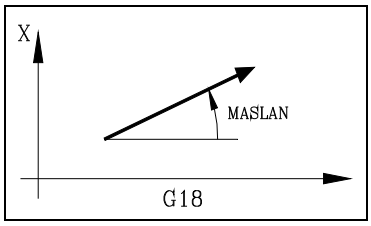
The HBE handwheel has priority. That is, regardless of the mode selected at the CNC switch (continuous or incremental JOG, handwheel), HBEVAR is set to other than "0", the CNC goes into handwheel mode.

It shows the selected axis in reverse video and the multiplying factor selected by the PLC. When the HBEVAR variable is set to "0", it shows the mode selected by the switch again.

See *"5.12 Fagor handwheels: HBA, HBE and LGB"* on page 242.

MASLAN

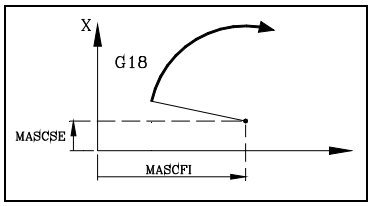
It must be used when the path-handwheel or the path-jog is selected.



Indicates the angle of the linear path.

MASCFI **MASCSE**

They must be used when the path-handwheel or the path-jog is selected.



On circular paths (arcs), they indicate the center coordinates.

11.8 Variables associated with feedback

ASIN(X-C)

"A" signal of the CNC's sinusoidal feedback for the X-C axis.

BSIN(X-C)

"B" signal of the CNC's sinusoidal feedback for the X-C axis.

ASINS

"A" signal of the CNC's sinusoidal feedback for the spindle.

BSINS

"B" signal of the CNC's sinusoidal feedback for the spindle.

11.9 Variables associated with the main spindle

Variables associated with the real speed

SREAL

It returns the actual (real) turning speed of the main spindle. Its value is given in 0.0001 rpm units.

FTEOS

It returns the theoretical turning speed of the main spindle.

Variables associated with spindle speed

PLCS is a read-write variable; the rest are read-only.

SPEED

It returns the turning speed of the main spindle currently selected at the CNC. Its value is given in 0.0001 rpm units.

This turning speed may be indicated by program, by PLC or by DNC; the CNC selects one of them, the one indicated by DNC has the highest priority and the one indicated by program has the lowest priority.

DNCS

It returns the spindle speed limit selected via DNC. If it has a value of 0 it means that it is not selected.

PLCS

It returns the spindle speed limit selected via PLC. If it has a value of 0 it means that it is not selected.

PRGS

It returns the spindle speed limit selected by programa. If it has a value of 0 it means that it is not selected.

Variables associated with constant cutting speed (lathe model)

PLCCSS is a read-write variable, the rest are read-only.

CSS

It returns the constant surface speed selected at the CNC.

This constant surface speed may be indicated by program, by PLC or by DNC; the CNC selects one of them, the one indicated by DNC has the highest priority and the one indicated by program has the lowest priority.

The values are given in the units set by g.m.p. INCHES.

If INCHES = 0, in m/min (±999999999).

If INCHES = 1, in ft/min (±393700787).

DNCCSS

It returns the constant surface speed selected via DNC. Its value is given in m/min or ft/min and it is 0 it means that it is not currently selected.

PLCCSS

It returns the constant surface speed selected by PLC. Its value is given in m/min or ft/min.

PRGCSS

It returns the constant surface speed selected by program. Its value is given in m/min or ft/min.

Variables associated with the spindle override

The variable PLCSSO is a read-write variable, the rest are read-only.

SSO

It returns the turning speed override (%) of the main spindle currently selected at the CNC. It is given in integer values between 0 and "MAXFOVR" (maximum 255).

This turning speed percentage of the main spindle may be indicated by program, by PLC, by DNC or by the front panel; the CNC selects one of them and the priority (from the highest to the lowest) is: by program, by DNC, by PLC and from the front panel.

PRGSSO

This variable may be used to read or modify the speed override percentage of the main spindle currently selected by program. It is given in integer values between 0 and "MAXFOVR" (maximum 255). If it has a value of 0 it means that it is not selected.

DNCSSO

It returns the turning speed override % of the main spindle currently selected via DNC. If it has a value of 0 it means that it is not selected.

PLCSSO

It returns the turning speed override % of the main spindle currently selected by PLC. If it has a value of 0 it means that it is not selected.

CNCSSO

It returns the turning speed override % of the main spindle currently selected from the front panel.

Speed limit related variables

PLCSL and MDISL are read-write variables, the rest are read-only.

SLIMIT

It returns the value set in rpm at the CNC for the turning speed limit of the main spindle.

This limit may be indicated by program, by PLC or by DNC; the CNC selects one of them, the one indicated by DNC has the highest priority and the one indicated by program has the lowest priority.

DNCSL

It returns the speed limit of the main spindle in rpm currently selected via DNC. If it has a value of 0 it means that it is not selected.

PLCSL

It returns the speed limit of the main spindle in rpm currently selected by PLC. If it has a value of 0 it means that it is not selected.

PRGSL

It returns the speed limit of the main spindle in rpm currently selected by program.

MDISL

Maximum machining spindle speed. This variable is also updated (refreshed) when programming function G92 via MDI.

Position related variables

POSS

It returns the real position of the main spindle. Its value is given in 0.0001 degree units within ±999999999º.

RPOSS

It returns the real position of the main spindle in 360º module. Its value is given in 0.0001 degree units within 0 and 360º.

TPOSS

It returns the theoretical position of the main spindle (real position + lag). Its value is given in 0.0001 degree units within ±999999999º.

RTPOSS

It returns the theoretical position of the main spindle (real position + lag) in 360º module. Its value is given in 0.0001 degree units within 0 and 360º.

PRGSP

Position programmed in M19 via program for the main spindle. This variable may be read from the CNC, from the PLC and from DNC.

Read-only variables associated with the following error (axis lag)

FLWES

Following error (lag) of the main spindle. Its value is given in 0.0001 degree units within ±999999999º.

11.10 Variables associated with local and global parameters

The CNC has two types of general purpose variables. These two variables may be used to read and write the following arithmetic parameters:

- Local parameters (P0-P25).
- Global parameters (P100-P299).
- User parameters (P1000 - P1255).
- OEM parameters (P2000-P2255).

It is possible to assign local parameters to more than one subroutine. Up to 6 nesting levels of the local parameters are possible within the 15 nesting levels for the subroutines. Therefore, each time a local parameter must be referred to, it is necessary to indicate its current nesting level.

Local and global parameters may be assigned a value within +2147483647.

Reading these parameters using functions GUP and LUP gives an integer number ignoring its decimals. Likewise, if the parameter value is greater than ±2147483647, the obtained value will be the maximum allowed, i.e. either 2147483647 or -2147483647.

Read-and-write variables

GUP n

It may be used to read or modify the indicated (n) global parameter (P100-P299), user parameter (P1000-P1255) (n) or OEM parameter (P2000-P2255) (n).

```
CNCRD (GUP 155, R100, M102)
```
Loads register R100 with the value of global parameter P155.

```
CNCWR (R101, GUP 155, M102)
```
It assigns the value of global parameter P155 to register R100.

LUP a b

It permits reading or modifying the indicated local parameter (P0-P25) (b) of the indicated nesting level (a).

```
CNCRD (LUP 3 15, R100, M102)
```
It assigns the value of local parameter P15 of nesting level 3 to register R100.

```
CNCWR (R101, GUP 2 15, M102)
```
It assigns the value of local parameter P15 of nesting level 2 to register R101.

11.11 Operating-mode related variables

Read-only variables related to the standard mode

OPMODE

It returns the code corresponding to the selected operating mode.

0 = Main menu.

10 = Automatic execution.
11 = Single block execution.
12 = MDI in EXECUTION.
13 = Tool inspection.
14 = Repositioning.
15 = Block search executing G.
16 = Block search executing G, M, S, T.

20 = Theoretical path simulation.
21 = G function simulation.
22 = G, M, S and T function simulation.
23 = Simulation with movement in the main plane.
24 = Simulation with rapid movement.
25 = Rapid simulation with S=0.

30 = Normal editing.
31 = User editing.
32 = TEACH-IN editing.
33 = Interactive editor.

40 = Movement in continuous JOG.
41 = Movement in incremental JOG.
42 = Movement with electronic handwheel.
43 = HOME search in JOG.
44 = Position preset in JOG.
45 = Tool calibration.
46 = MDI in JOG.
47 = User JOG operation.

50 = Zero offset table.
51 = Tool offset table.
52 = Tool table.
53 = Tool magazine table.
54 = Global parameter table.
55 = Local parameter table.
56 = User parameter table.
57 = OEM parameter table.

60 = Utilities.

70 = DNC status.

71 = CNC status.

80 = PLC file editing.
81 = PLC program compilation.
82 = PLC monitoring.
83 = Active PLC messages.
84 = Active PLC pages.
85 = Save PLC program.
86 = Restore PLC program.
87 = PLC usage maps.
88 = PLC statistics.

90 = Customizing.

100 = General machine parameter table.
101 = Axis machine parameter tables.
102 = Spindle machine parameter table.
103 = Serial line related machine parameter tables.
104 = PLC machine parameter table.
105 = M function table.
106 = Leadscrew error compensation tables and cross compensation tables.

110 = Diagnosis: configuration.
111 = Diagnosis: hardware test.
112 = Diagnosis: RAM memory test.
113 = Diagnosis: Flash memory test.
114 = User diagnosis.

11.12 Other variables

Read-only variables

NBTOOL

Indicates the tool number being managed. This variable can only be used within the tool change subroutine.

Example: There is a manual tool changer. Tool T1 is currently selected and the operator requests tool T5.

The subroutine associated with the tools may contain the following instructions:

```
(P103 = NBTOOL)
(MSG "SELECT T?P103 AND PRESS CYCLE START")
```

Instruction (P103 = NBTOOL) assigns the number of the tool currently being managed to parameter P103. Therefore, P103=5.

The message displayed by the CNC will be ""SELECT T5 AND PRESS CYCLE START".

Note: The NBTOOL variable is refreshed in all simulations including those where T functions are not executed. In other words, it may not correspond with the active tool (TOOL).

PRGN

Returns the program number being executed. Should none be selected, a value of -1 is returned.

BLKN

It returns the label number of the last executed block.

GGSA

It returns the status of functions G00 through G24. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G24	G23	G22	G21	G20	...	G04	G03	G02	G01	G00

```
CNCRD (GGSA, R110, M10)
```
Loads register R110 with the status of functions G00 through G24.

GGSB

It returns the status of functions G25 through G49. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G49	G48	G47	G46	G45	...	G29	G28	G27	G26	G25

GGSC

It returns the status of functions G50 through G74. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G74	G73	G72	G71	G70	...	G54	G53	G52	G51	G50

GGSD

It returns the status of functions G75 through G99. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G99	G98	G97	G96	G95	...	G79	G78	G77	G76	G75

GGSE

It returns the status of functions G100 through G124. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G124	G123	G122	G121	G120	...	G104	G103	G102	G101	G100

GGSF

It returns the status of functions G125 through G149. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G149	G148	G147	G146	G145	...	G129	G128	G127	G126	G125

GGSG

It returns the status of functions G150 through G174. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G174	G173	G172	G171	G170	...	G154	G153	G152	G151	G150

GGSH

It returns the status of functions G175 through G199. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G199	G198	G197	G196	G195	...	G179	G178	G177	G176	G175

GGSI

It returns the status of functions G200 through G224. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G224	G223	G222	G221	G220	...	G204	G203	G202	G201	G200

GGSJ

It returns the status of functions G225 through G249. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G249	G248	G247	G246	G245	...	G229	G228	G227	G226	G225

GGSK

It returns the status of functions G250 through G274. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G274	G273	G272	G271	G270	...	G254	G253	G252	G251	G250

GGSL

It returns the status of functions G275 through G299. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G299	G298	G297	G296	G295	...	G279	G278	G277	G276	G275

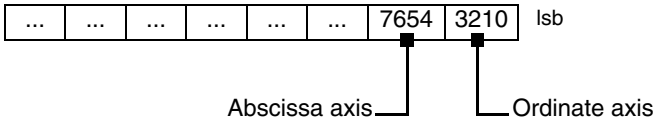
GGSM

It returns the status of functions G300 through G320. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G320	G319	G318	G317	G316	...	G304	G303	G302	G301	G300

PLANE

Returns data on the abscissa axis (bits 4 to 7) and the ordinate axis (bits 0 to 3) of the active plane in 32 bits and in binary.



The axes are coded in 4 bits and indicate the axis number according to the programming order.

Example: If the CNC controls the X, Y and Z axes and the ZX plane (G18) is selected.

(CNCRD PLANE, R100, M33) assigns the hexadecimal value $31 to register R100.

0000	0000	0000	0000	0000	0000	0011	0001	LSB

Abscissa axis = 3 (0011) => Z axis

Ordinate axis = 1 (0001) => X axis

LONGAX

This variable can only be used at the mill model. It returns the number according to the programming order corresponding to the longitudinal axis. This will be the one selected with the G15 function and by default the axis perpendicular to the active plane, if this is XY, ZX or YZ.

Example:

If the CNC controls the X, Y and Z axes and the Z axis is selected.

(CNCRD LONGAX, R22, M34) assigns the value of 3 to register R22.

MIRROR

Returns in the least significant bits of the 32-bit group, the status of the mirror image of each axis, 1 in the case of being active and 0 if not.

Bit 8	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0	LSB
						Axis 3	Axis 2	Axis 1	

The axis name corresponds to the number according to the programming order for them.

Example: If the CNC controls the X, Y and Z axes, axis1=X, axis2 = Y, axis 3=Z.

SCALE

It returns the general scaling factor being applied. It will be multiplied by 10000.

SCALE(X-C)

Returns the specific scaling factor of the indicated axis (X-C). It will be multiplied by 10000.

ORGROT

This variable can only be used at the mill model. It returns the rotation angle of the coordinate system currently selected with G73. Its value in 0.0001 degree units.

PRBST

Returns probe status.

0 = the probe is not touching the part.

1 = the probe is touching the part.

CLOCK

Returns the time in seconds indicated by the system clock. Possible values 0..4294967295.

TIME

Returns the time in hours-minutes-seconds format.

(CNCRD TIME, R100, M102) ; assigns the time to register R100. For example, if the time is 18h 22m 34s, R100 will show 182234.

DATE

Returns the date in year-month-day format.

(CNCRD DATE, R101, M102) ; assigns the date to register R101. For example: if the date is April 25th 1992, R101 = 920425.

CYTIME

It returns the amount of time (in hundredths of a second) elapsed executing the part. It ignores the time the execution has been interrupted. Possible values 0..4294967295.

The CNC will consider the execution of the program finished after executing its last block or after executing a block containing an M02 or M30 miscellaneous function.

FIRST

Indicates whether it is the first time that a program has been run or not. It returns a value of 1 if it is the first time and 0 if not.

A first-time execution is considered as being one which is done:

- After turning on the CNC.
- After pressing [SHIFT]+[RESET].
- Every time a new program is selected.

ANAIn

It returns the status in tenths of a volt (±5 V range) of the indicated analog input (n) and it is possible to select one of the 8 analog inputs (1 through 8).

CNCERR

Returns the Error code active at the CNC. If none, it returns "0".

DNCERR

Returns the Error code generated via DNC. If none, it returns "0".

DNCSTA

DNC transmission status, even when not having this option. There is on bit that will be set to ·1· when a transmission is in progress.

TIMEG

It shows the timing status of the timer programmed with G4 K in the CNC channel. This variable, returns the time remaining to end the timing block in hundredths of a second.

RIP

Linear theoretical feedrate resulting from the next loop (in mm/min).

The calculation of the resulting feedrate ignores the rotary axes, slave axes (gantry, coupled and synchronized) as well as DRO axes.

FBDIF(X-C)

Variable that can be read via CNC, PLC and DNC and permits monitoring the position difference between the first and second feedback at the oscilloscope with Fagor digital servo system.

If the difference between the two feedback readings exceeds the value set in a.m.p. FBACKDIF (P100), the CNC will show the relevant error message.

This monitoring depends on the value of a.m.p. FBACKAL (P11).



Monitoring of the difference between the first and second feedbacks is not possible on CAN axes.

DISBLO

Variable that indicates the value of thetotal distance programmed in blocks with look-ahead. This variable is read-only from the CNC and PLC and DNC and may be viewed at the oscilloscope.

Read-and-write variables

TIMER

This variable allows reading or modifying the time, in seconds, indicated by the clock enabled by the PLC. Possible values 0..4294967295.

The CNC will set this value to 0 when changing the software version or when a checksum error occurs.

PARTC

The CNC has a part counter whose count increases, in all modes except simulation, every time M30 or M02 is executed and this variable allows its value to be read or modified. This value will be between 0 and 4294967295

The CNC will set this value to 0 when changing the software version or when a checksum error occurs.

KEY

It allows reading the last accepted keystroke or simulating the CNC keyboard assigning the desired key code to it.

```
CNCRD (KEY, R110, M10)
    Loads register R110 with the value of the last key accepted.
```

To simulate the CNC keyboard from the PLC, follow these steps:

```
R111=1   R110=0
```

```
CNCWR (R111, KEYSCR, M10)
```

Indicates to the CNC that only keystrokes coming from the PLC must be processed (CNC keyboard inhibited).

```
CNCWR (R101, KEY, M10)
```

It indicates to the CNC that a key has been pressed whose code is indicated in register R101.

```
CNCWR (R110, KEYSCR, M10)
```

Process only keystrokes coming from the CNC.

KEYSRC

This variable allows reading or modifying the source of keystrokes, possible values being:

0 = Keyboard.

1 = PLC.

2 = DNC.

The CNC only allows modification of this variable if it is set to "0" or "1".

Once the keystroke simulation is finished, it is advisable to re-enable the CNC keyboard in order to be able to access the various operating modes of the CNC.

The CNC will assign a value of 0 to this variable on power-up and after pressing [SHIFT]+[RESET].

ANAOn

This variable allows the required analog output (n) to be read or modified. The value assigned will be expressed in 0.0001 volt units and within ±10 V.

The analog outputs which are free among the eight (1 through 8) available at the CNC may be modified, the corresponding error being displayed if an attempt is made to write in one which is occupied.

SELPRO

When having two probe inputs, it allows selecting the active input.

On power-up, it assumes the value of ·1· thus selecting the first probe input. To select the second probe input, set it to a value of ·2·.

Accessing this variable from the CNC interrupts block preparation.

DIAM

In the lathe model, it changes the programming mode for X axis coordinates between radius and diameter. When changing the value of this variable, the CNC assumes the new way to program the following blocks.

When the variable is set to ·1·, the programmed coordinates are assumed in diameter; when is set to ·0·, the programmed coordinates are assumed in radius.

This variable affects the display of the real value of the X axis in the coordinate system of the part and the reading of variables PPOSX, TPOSX and POSX.

On power-up, after executing an M02 or M30 and after an emergency or a reset, the variable is initialized according to the value of the DFORMAT parameter of the X axis. If this parameter has a value equal to or greater than 4, the variable takes a value of 1; otherwise, it takes the value of ·0·.

PRBMOD

It indicates whether a probing error is to issued or not in the following cases, even if general machine parameter PROBERR (P119) =YES.

- When a G75 probing move finishes before the probe has touched part.
- When a G76 probing move finishes but the probe is still touching the part.

The PRBMOD variable takes the following values.

Value	Meaning
0	An error message is issued.
1	No error message is issued.

Default value 0.

The PRBMOD variable can be read and written from the CNC and the PLC an read from the DNC.

RETREJ

It indicates that the retraction in drilling, or the mill type threading or lathe type threading cycle has finished.

This variable is set to ·1· at the end of the retraction and is set to ·0· when pressing [START].

In lathe, it indicates that the CNC has carried out a withdrawal from the thread. This variable takes the value of ·1· when the withdrawal distances are reached and stays at ·1· until pressing [START] or executing an M30 or a RESET. After executing one of these functions, it will take the value of ·0·.

RIGIER

It indicates the offset in mm/inches between the projection of the following error of the spindle onto the longitudinal axis and the following error of the longitudinal axis. This variable may be displayed on the oscilloscope and on the screen for following error.

The screen for following error only displays the offset value during rigid tapping while tapping is in progress. Once the tapping is completed, the data will disappear.

To make the taping smoother and easier on the tool, the value of the "RIGIER" variable must be as close to zero as possible. That will require retouching the following errors of the longitudinal axis and of the spindle. Since adjusting the spindle in closed loop is usually harder than adjusting an axis, we recommend to first adjust the spindle as best as possible and then the following error of the longitudinal axis so the displayed value of the "RIGIER" variable is as small as possible.

11.

ACCESS TO INTERNAL CNC VARIABLES

Other variables

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

AXES CONTROLLED FROM THE PLC

12

The PLC offers actions CNCEX and CNCEX1 to send commands to the CNC.

CNCEX	sends commands to the CNC so it executes movements on one or several axes.
CNCEX1	sends commands to the CNC so it executes any kind of block.

The CNCEX action is executed through the execution channel of the PLC.

The CNCEX1 action is executed via main channel of the CNC and as long as the JOG keyboard is enabled. Its execution can be interrupted by pressing [CYCLE STOP] or even canceled by pressing [RESET].

If a CNCEX1 action is received when the JOG keyboard is disabled, the CNC ignores this command.

The programming format for these actions is:

CNCEX	(ASCII block, Mark)
CNCEX1	(ASCII block, Mark)

By means of these actions, the PLC sends to the CNC the command indicated in the "ASCII Block" to be executed.

If the "ASCII Block" has been accepted by the CNC, the PLC will set the indicated mark to "0" or to "1" if otherwise. The CNC only indicates that the "ASCII Block" has been accepted. It is up to the operator to verify whether the command has actually been executed by the CNC or not.

```
CNCEX (G1 U125 V300 F500, M200)
```
Sends to the CNC the command "G1 U125 V300 F500" so it executes a linear interpolation of the U and V axes at a feedrate of F500 being the end point: U125 V300.

```
CNCEX1 (T5, M200)
```
Selects the tool T5 in the tool changer.

Example of how to use action CNCEX1 when using a tool changer controlled by the PLC.

1. The T executed last at the CNC is T1. Therefore, it is the active T.
2. A new tool is selected, for example T5.

   If carried out by means of action CNCEX1, the change is made by the CNC and it assumes T5 as the new active tool.

   If not carried out by means of action CNCEX1, the change is made by the PLC and T1 remains as the active tool.
3. Then, an operation programmed with T1 is carried out.

   If the change was made with action CNCEX1, the CNC detects the tool change (from T5 to T1) and carries out the change.

   If the change was not made with action CNCEX1, the CNC does not detect the tool change (T1), it does not make the change and carries out the operation with the selected tool T5 with the problems this may cause.

12.1 PLC execution channel

The CNC offers a parallel execution channel to execute commands received from the PLC. This channel will have its own history and it permits the execution of blocks programmed from the PLC regardless of the operating mode being selected at the CNC at the time.

When the CNC receives a command from the PLC and it is executing another command received earlier, it will store the new one in an internal buffer. This new command will be executed after finishing the one being executed.

The internal buffer can store up to 3 commands received from the PLC besides the one currently in execution.

12.1.1 Considerations

Screen editor

The a.m.p. "AXISTYPE" of each axis of the machine must be set properly indicating whether that axis is controlled by the CNC or from the PLC.

The axes of the PLC channel can only be governed from the PLC.

They may be edited and part-programs may be generated with axes of the PLC channel. This permits generating part-programs or subroutines associated with the PLC channel.

It issues an error message when trying to execute, from the CNC channel, a program block that includes a PLC axis.

When all the axes of the machine are set to be governed from the CNC, with the CNCEX action only blocks programmed in high level language may be executed through the PLC execution channel.

Axis control

To govern axes managed by PLC, use the following marks associated with “Feed-hold” and “Transfer Inhibit”:

/FEEDHOP	(M5004)	Similar to the /FEEDHOL signal
FHOUTP	(M5504)	Similar to the /FHOUT signal
/XFERINP	(M5005)	Similar to the /XFERINH signal

Auxiliary M functions

To control the M functions managed by the PLC, the following marks and registers are generated:

MBCDP1 through MBCDP7 (R565 through R571)

similar to signals MBCD1 through MBCD7.

AUXENDP (M5006)

Similar to the AUXEND signal.

MSTROBEP (M5505)

Similar to the MSTROBE signal.

Data transfer

If when executing at the PLC the action "CNCEX (ASCII Block, Mark)", the CNC detects that the contents of the ASCII block being received is erroneous, it will set the indicated Mark to "1". The PLC program will keep executing while it is up to the programmer to check whether the function was executed correctly or not.

The CNC considers the contents of the ASCII block incorrect in the following instances:

- When the syntax is incorrect.
- When programming a not-permitted preparatory function (G code).
- When programming an auxiliary function M, S, T or tool offset D.
- When programming a high level language block.
- When the axis to be moved cannot be controlled from the PLC.
- When the internal buffer for PLC command storage is full.

Errors during execution

When the CNC detects an execution error in one of the two execution channels (for example, travel limit overrun), it will show the corresponding error code.

If it must also stop the movement of the axes and the spindle rotation, the CNC will stop the movement of all the axes regardless of whether they are controlled from the CNC or the PLC.

Also, if the detected error stops the program execution, the CNC will stop the execution of both channels and each one of them will act as follows:

CNC channel

Once the cause of the error has been removed, select again the execution or simulation mode and continue with the program execution.

PLC channel

The PLC program does not stop and continues running.

The commands sent by means of action "CNCEX" will not be executed until removing the cause of the error.

Once the cause of the error removed, the CNC will execute all the new commands sent by the PLC.

To know from the PLC program whether any CNC error is active, this information can be requested by accessing the internal CNC variable "CNCERR". This variable indicates the error number being active at the CNC and if none is active, it returns a 0 value.

12.1.2 Blocks which can be executed from the PLC

It is possible to execute blocks that contain G codes, axis position values, feedrates, M functions and high level language programming.

Auxiliary functions S, T and D cannot be programmed.

The ASCII block to be sent to the CNC by means of the action CNCEX to be executed in the PLC execution channel must be written in the CNC's own programming format.

Preparatory functions

The preparatory functions which can be used in the PLC execution channel are the following:

G00	Rapid travers
G01	Linear interpolation
G02	Clockwise circular (helical) interpolation
G03	Counterclockwise circular (helical) interpolation
G04	Interrupt block preparation of the PLC channel.
G04 K	Dwell
G05	Round corner
G06	Circle center in absolute coordinates
G07	Square corner
G09	Arc defined by three points
G16	Main plane selection by two addresses and longitudinal axis
G32	Feedrate "F" as an inverted function of time.
G50	Controlled corner rounding
G52	Movement until making contact
G53	Programming with respect to machine zero
G70	Programming in inches
G71	Metric programming
G74	Home search
G75	Probing move until touching
G76	Probing move while touching
G90	Absolute programming:
G91	Incremental programming
G92	Coordinate preset
G93	Polar origin preset
G94	Feedrate in millimeters (inches) per minute
G95	Feedrate in millimeters (inches) per revolution.

All these functions must be programmed as described in the programming manual.

Move the axes

Only those axes set by means of a.m.p. AXISTYPE (P0) for each axis as to be controlled by the PLC can be mentioned.

The position values of these axes, which can be either linear or rotary, can be programmed in either Cartesian or polar coordinates.

These coordinates can also be defined via parametric programming using any global arithmetic parameters (P100 thru P299)

When using parametric programming, it is recommended to previously assign a value to the corresponding global parameter by means of the instruction: CNCWR.

```
... = MOV 150 R1
```
Assigns the value of 150 to register R1.

```
... = CNCWR (R1, GUP200, M100)
```
Assigns the value of R1 to parameter P200, (P200=150).

```
... = CNCEX (G90 G1 U P200, M100)
```
Requests the CNC to execute the command: G90 G1 U150. The U axis will go to position 150.

To govern axes managed by PLC, use the following marks associated with “Feed-hold” and “Transfer Inhibit”:

/FEEDHOP	(M5004)	Similar to the /FEEDHOL signal
FHOUTP	(M5504)	Similar to the /FHOUT signal
/XFERINP	(M5005)	Similar to the /XFERINH signal

Feedrate of the axes

The programming format for the axis feedrate (F5.5) depends on the function (G94 or G95) and on the work units selected for this execution channel.

- If G94, in mm/min. or inches/min.
- If G95, in mm/rev or inches/rev.

It must be borne in mind that this feedrate depends on the actual spindle rpm which is in the main execution channel.

If the moving axis is rotary, the CNC interprets that the programmed feedrate is in degrees/minute.

Modify the feedrate (override)

The PLCCFR variable sets, from the PLC, the % of feedrate selected by the execution channel of the PLC.

G.m.p. MAXFOVR (P18) limits the value of the percentage applied to both execution channels (main and PLC).

The OVRCAN (M5020) mark sets the feedrate override of the main channel to 100%. It does not affect the feedrate override of the PLC channel

Same as with the main channel, the following movements have a special treatment:

- When searching home, the value of PLCCFR is ignored.
- In G0, it considers the value of g.m.p. "RAPIDOVR (P17)"

  If "P17=NO" always 100%, except if PLCCFR=0. In that case, the movement stops.

  If "P17=YES" considers PLCCFR, but it limits its value to 100%.
- In G1, G2, G3 it is always applied except when operating at maximum feedrate (F0).
- In that case, it is limited to 100%. In G75, G76, it is only applied when g.m.p. FOVRG75 (P126) = YES.

Blocks programmed in high-level language

The high-level instructions that can be used in the PLC execution channel are:

(IF condition <action1> ELSE <action2>)

(CALL (expression))

```
CNCEX ((CALL 100), M1000)
```
Sends the (CALL 100) command to the CNC so it executes (calls) subroutine 100

```
CNCEX ((P100=P100+2), M1000)
```
Sends the (P100=P100+2) command to the CNC to increment the value of parameter P100 in 2 units.

Programming high-level blocks has the following restrictions:

- The programmed blocks can only work with global parameters.
- Up to 5 nesting levels of standard subroutines are allowed (neither parametric nor global).

Example in mm:

Move the W axis to the coordinate indicated by register R101.

When the PLC works with integers (32 bits), the value of register R2 is given in tenths of microns (0.0001 mm).

```
CNCWR (R101, GUP 155, M101)
```
Assigns the value indicated in R101 to global parameter P155.

```
CNCEX ((P155=P155/10000), M101)
```
Converts the value of P155 into mm.

```
CNCEX (G1 WP155 F2000, M101)
```
Movement of the W axis.

Interrupt block preparation

Same as in the CNC channel, blocks are also prepared in advance in the PLC channel.

```
CNCEX (G1 W100, M101)
```
Movement of the W axis.

```
CNCEX (IF P100=0 <action1>)
```
P100 is analyzed during block preparation.

The value of P100 may be different before, during and after the movement of the W axis. If it is to be analyzed after moving the axis, function G4 must be programmed.

```
CNCEX (G1 W100, M101)
```
Movement of the W axis.

```
CNCEX (G4, M102)
```
Interrupts block preparation.

```
CNCEX (IF P100=0 <action1>)
```
P100 is analyzed after moving the axis.

Likewise, every time a PLC resource is accessed (I, O, M, R), block preparation is interrupted.

```
CNCEX (G1 W100, M101)
```
Movement of the W axis.

```
CNCEX (IF PLCI8=1 <action2>)
```
I8 is checked after moving the axis.

Auxiliary M functions

The M functions programmed in the PLC channel may be defined in the M function table.

In the PLC channel, the following functions cannot be programmed: M0, M1, M2, M3, M4, M5, M6, M19, M30, M41, M42, M43 and M44.

The following marks and registers are generated for managing the M functions, :

```
MBCDP1 through MBCDP7 (R565 through R571)
```
similar to signals MBCD1 through MBCD7.

```
AUXENDP (M5006)
```
Similar to the AUXEND signal.

```
MSTROBEP (M5505)
```
Similar to the MSTROBE signal.

12.1.3 Control of the PLC program from the CNC

The section of the PLC program regarding the "axes controlled from the PLC" can be controlled from the CNC itself.

To do this, the inputs, outputs, marks, registers, timers or counters of the PLC itself are used.

The CNC has the following PLC related variables to read or change the status of the selected resource.

PLCI	To read or modify up to 32 PLC inputs.
PLCO	To read or modify up to 32 PLC outputs.
PLCM	To read or modify up to 32 PLC marks (internal relays).
PLCR	To read or modify the status of a register.
PLCT	To read or modify the count of a timer.
PLCC	To read or modify the count of a counter.

With these variables, the desired values will be assigned, in the part-program of the CNC, to the PLC resources used in the communication. The setting of these values will be carried out whenever an axis or axes are to be controlled from the PLC.

In turn, the PLC program must check the status of such resources and when detecting that one of them is activated, it must execute the corresponding section of the PLC program.

It is also possible to transfer data from the CNC to the PLC via global and local arithmetic parameters. The PLC has the following variables related to those CNC parameters:

GUP	To read or modify a global parameter of the CNC.
LUP	To read or modify a local parameter of the CNC.

Example:

The "U" axis is controlled by the PLC and we want to command it from any part-program of the CNC in such way that we could select the type of move (G00 or G01), the positioning coordinate and the feedrate for that move.

In order to command it from any part-program, it is convenient to have in a subroutine the section of the CNC program allowing the data transfer with the PLC.

This example uses subroutine SUB1 and, for data exchange, it uses global CNC parameters.

P100	Type of move. If P100 = 0, then G00; If P100 = 1, then G01.
P101	"U" axis positioning coordinate.
P102	Feedrate. It only makes sense when moving in G01.

To indicate to the PLC that it must execute this move, it activates the following PLC resource:

M1000	Command to begin movement.

Any part-program of the CNC may contain a block of the type:

(PCALL 1, G1, U100, F1000)

This block calls subroutine SUB1 and it transfers the local parameters G, U and F with the following information:

G Type of move.

U "U" axis positioning coordinate.

F Feedrate for the movement.

Subroutine SUB1 can be programmed as follows:

```
( SUB 1)
(P100 = G, P101 = U, P102 = F)
    Data transfer to global parameters.
( PLCM1000 = PLCM1000 OR 1 )
    Execution command for the PLC.
(RET)
```

The PLC program, in turn, will have to contain the following instructions:

```
M1000 = CNCEX (G90 GP100 UP101 FP102, M111)
;When mark  M1000 is active, it sends the indicated block to the CNC.
NOT M111 = RES M1000
If the CNC accepts this block, it resets mark M1000.
```

12.2 Action CNCEX1

The CNCEX1 action is executed via main channel of the CNC and as long as the JOG keyboard is enabled. Its execution can be interrupted by pressing [CYCLE STOP] or even canceled by pressing [RESET].

If a CNCEX1 action is received when the JOG keyboard is disabled, the CNC ignores this command.

The block to be executed must be written in the programming format of the CNC itself.

Any type of block can be sent which is edited in ISO or high level language. It admits preparatory functions, auxiliary functions, calls to subroutines, etc.

12.

AXES CONTROLLED FROM THE PLC

Action CNCEX1

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

PLC PROGRAMMING EXAMPLE 13

It is a three-axes machine (X, Y, Z) having a spindle with two speed ranges.

The PLC, besides controlling the 3 axes and the spindle, is in charge of lubricating the axes as well as turning the coolant on and off.

CNC configuration

The PLC has 512 inputs and 512 outputs. Some of them, depending on the CNC configuration, communicate with external devices.



Input I1 is the emergency input of the CNC and must be supplied with 24V. Regardless of how it is treated by the PLC program, this signal is processed directly by the CNC at all times.

Output O1 is normally at 24V, high logic level, and it is set low, 0V, whenever an ALARM or an ERROR occurs at the PLC output O1.

13.1 Definition of symbols (mnemonics)

It is a possible to associate a symbol (name) to any PLC resource. It may have up to 8 characters so long as the name does not coincide with any of the reserved instructions. It may not contain the following characters: blank-space " ", equal sign "=", parenthesis "(" or ")", comma "," or semi-colon ";".

These symbols or names must always be defined at the beginning of the program. Duplicate symbols cannot be defined; but several symbols may be assigned to the same resource.

For better clarification, the symbols used in this program are grouped by subjects.

Used in: Basic and necessary programming.

DEF	I-EMERG	I1	External emergency input.
DEF	I-CONDI	I70	Conditional mode. The CNC interrupts part-program execution when executing auxiliary function M01
DEF	SERVO-OK	I71	The servo drives are O.K.
DEF	O-EMERG	O1	Emergency output. It must be normally high.

Used in: Treatment of the axis travel limit switches.

DEF	I-LIMTX1	I72	X axis positive overtravel limit switch
DEF	I-LIMTX2	I73	X axis negative overtravel limit switch
DEF	I-LIMTY1	I74	Y axis positive overtravel limit switch
DEF	I-LIMTY2	I75	Y axis negative overtravel limit switch
DEF	I-LIMTZ1	I76	Z axis positive overtravel limit switch
DEF	I-LIMTZ2	I77	Z axis negative overtravel limit switch

Used in: Treatment of the machine reference (home) switches.

DEF	I-REF0X	I78	X axis home switch
DEF	I-REF0Y	I79	Y axis home switch
DEF	I-REF0Z	I80	Z axis home switch

Used in: Treatment of M, S, T functions.

DEF	M-03	M1003	Auxiliary mark. Indicates that M03 must be executed
DEF	M-04	M1004	Auxiliary mark. Indicates that M04 must be executed
DEF	M-08	M1008	Auxiliary mark. Indicates that M08 must be executed
DEF	M -41	M1041	Auxiliary mark. Indicates that M41 must be executed
DEF	M -42	M1042	Auxiliary mark. Indicates that M42 must be executed

Used in: Machine way lubrication.

DEF	I-LUBING	I81	Operator request to lubricate the ways of the machine
DEF	O-LUBING	O2	Ways lubrication output

Used in: Coolant treatment.

DEF	I-COOLMA	I82	The operator control the coolant. Manual mode.
DEF	I-COOLAU	I83	The CNC controls the coolant. Automatic mode.
DEF	O-COOL	O3	Coolant output

Used in: Spindle turning control.

DEF	O-S-ENAB	O4	Spindle enable output

Used in: Treatment of the spindle gear change.

DEF	O-GEAR1	O5	Move gears to select range 1 (gear 1)
DEF	O-GEAR2	O6	Move gears to select range 2 (gear 1)
DEF	I-GEAR1	I84	Indicates that Gear 1 is selected
DEF	I-GEAR2	I85	Indicates that Gear 2 is selected

Used in: Keyboard simulation.

DEF	I-SIMULA	I86	The operator requests the simulation of program P12
DEF	SENDKEY	M1100	Indicates that the code of a key is to be sent out to the CNC
DEF	KEYCODE	R55	Indicates the code of the key to be simulated
DEF	LASTKEY	R56	Indicates which is the last key accepted by the CNC
DEF	SENTOK	M1101	Indicates that the key code has been sent correctly
DEF	KEYBOARD	R57	Used to indicate to the CNC the source of the keys
DEF	CNCKEY	0	Used to indicate that the keys come from the CNC keyboard
DEF	PLCKEY	1	Used to indicate that the keys come from the PLC

DEF	MAINMENU	$FFF4	Code of the "MAIN MENU" key
DEF	SIMULATE	$FC01	Code of the "SIMULATE" key (F2)
DEF	KEY1	$31	Code of the "1" key
DEF	KEY2	$32	Code of the "2" key
DEF	ENTER	$0D	Code of the "ENTER" key
DEF	THEOPATH	$FC00	Code of the "THEORETICAL PATH" key (F1)
DEF	START	$FFF1	Code of the "START" key

13.2 First cycle module.

CY1

( ) = ERA O1 512 = ERA C1 256 = ERA T1 256 = ERA R1 256 = ERA M1 2000

( ) = ERA M4000 4127 = ERA M4500 4563 = ERA M4700 4955

Initializes all PLC resources to low logic level "0".

( ) = TG1 2 120000

Initializes the timer which controls the lubrication of the machine ways on power-up. This operation will be performed for 2 minutes.

( ) = TG2 4 3600000

Initializes the timer which controls the amount of time the axes are moving before they are lubricated. This lubrication lasts 5 minutes and it takes place after the axes have been moving for 1 hour.

END

13.3 Main module.

PRG

REA

---- Basic and necessary programming ----

( ) = /STOP

Permission to execute the part-program

() = /FEEDHOL

Permission to move the axes

( ) = /XFERINH

Permission to execute the next block

I-EMERG AND (rest of conditions) = /EMERGEN

If the external emergency input is activated or any other emergency occurs, the general logic input /EMERGEN of the CNC. When there is no emergency, this signal must remain high.

/ALARM AND CNCREADY = O-EMERG

The emergency output, O1, of the PLC (O-EMERG) must be normally high

If an alarm or emergency is detected at the CNC (/ALARM) or a problem was detected when powering the CNC up (CNCREADY), the emergency output O-EMERG must be brought low.

I-CONDI = M01STOP

When the operator selects the conditional mode (I-CONDI), the CNC general logic input M01STOP must be activated. It interrupts the program when executing M01.

START AND (rest of conditions) = CYSTART

When the cycle START key is pressed, the CNC activates the general logic output START.

The PLC must check that the rest of the conditions (hydraulic, safety devices, etc.) are met before setting the general input CYSTART high in order to start executing the program

SERVO-OK AND NOT LOPEN = SERVO1ON = SERVO2ON = SERVO3ON

If the servo drives are OK and the CNC does not detect any errors in the positioning loop of the axes (LOPEN), the positioning loop must be closed on all axes. Axis logic inputs of the CNC: SERVO1ON, SERVO2ON, SERVO3ON.

----- Treatment of the axis overtravel limit switches -----

I-LIMTX1 = LIMIT+1

I-LIMTX2 = LIMIT-1

I-LIMTY1 = LIMIT+2

I-LIMTY2 = LIMIT-2

I-LIMTZ1 = LIMIT+3

I-LIMTZ2 = LIMIT-3

----- Treatment of the machine reference (home) switches -----

I-REF0X = DECEL1

I-REF0Y = DECEL2

I-REF0Z = DECEL3

----- Message treatment -----

The PLC allows displaying the corresponding PLC message at the CNC screen by activating marks MSG1 through MSG255. This text must be previously edited at the PLC message table.

The following example shows how to generate a message to remind the operator to home the axes after powering the machine up.

(MANUAL OR MDI OR AUTOMAT) AND NOT (REFPOIN1 AND REFPOIN2 AND REFPOIN3) = MSG5

The message (MSG5) appears in the JOG, MDI or Automatic modes and only when the axes of the machine have not been referenced (homed). The CNC logic outputs "REFPOIN" indicate that the axes have been homed. ----- Error treatment -----

----- Error message treatment -----

The PLC permits displaying the corresponding error message on the CNC screen by activating marks ERR1 through ERR128 as well as interrupting the CNC program execution stopping the axes and the spindle. The activation of any of these marks does not activate the external CNC Emergency output.

Because the PLC program is not interrupted by these marks, it is advised to make it possible to change their status via accessible external inputs; otherwise, the CNC will keep receiving the same error at every PLC scan (cycle) thus preventing access to any PLC mode.

The text associated to the error message must be previously edited at the PLC error table.

The next example shows how to generate the X axis overtravel limit overrun error when one of the overtravel limit switches is pressed.

NOT I-LIMTX1 OR NOT I-LIMTX2 = ERR10

----- Treatment of M, S, T functions -----

The CNC activates the general logic output MSTROBE to "tell" the PLC to execute the M functions indicated at the variables MBCD1 through MBCD7.

It also activates: the SSTROBE output when the S function indicated at variable SBCD must be executed, the TSTROBE output when the T function indicated at variable TBCD must be executed and the T2STROBE output when the T function indicated at variable T2BCD must be executed.

Whenever the CNC activates one of these signals, it is convenient to deactivate the general CNC input AUXEND in order to interrupt the execution of the CNC. When the PLC concludes the processing of the required function, this AUXEND signal must be activated back so that the CNC resumes the execution of the interrupted program.

This example deactivates the AUXEND signal for 100 milliseconds using the timer T1.

MSTROBE OR SSTROBE OR TSTROBE OR T2STROBE = TG1 1 100

The activation of the STROBE signals activates timer T1 in the mono-stable mode for 100 milliseconds.

Whenever timer T1 is active, the PLC must set the AUXEND signal low as described in: "Treatment of the general CNC input AUXEND".

When the CNC activates the MSTROBE signal, the contents of variables MBCD1 through MBCD7 must be analyzed in order to know which auxiliary functions are to be executed. All MBCD variables may be analyzed at the same time by using "MBCD*".

This example SETs the auxiliary marks so they can be analyzed later. Once analyzed, they must be RESet so that the PLC does not analyze them again on the next cycle (scan).

DFU MSTROBE AND CPS MBCD* EQ $0 = RES M-08

DFU MSTROBE AND CPS MBCD* EQ $2 = RES M-08

Functions M00 and M02 cancel the coolant (M08).

DFU MSTROBE AND CPS MBCD* EQ $3 = SET M-03 = RES M-04

DFU MSTROBE AND CPS MBCD* EQ $4 = SET M-04 = RES M-03

DFU MSTROBE AND CPS MBCD* EQ $5 = RES M-03 = RES M-04

Functions M03 and M04 are incompatible with each other and M05 cancels both.

DFU MSTROBE AND CPS MBCD* EQ $8 = SET M-08

DFU MSTROBE AND CPS MBCD* EQ $9 = RES M-08

DFU MSTROBE AND CPS MBCD* EQ $30 = RES M-08

Functions M09 and M30 cancel the coolant (M08)

DFU MSTROBE AND CPS MBCD* EQ $41 = SET M-41 = RES M-42

DFU MSTROBE AND CPS MBCD* EQ $42 = SET M-42 = RES M-41

Functions M41 and M42 are incompatible with each other.

----- Spindle turning control -----

The spindle enable output O-S-ENAB will be activated when selecting function M03 or M04.

M-03 OR M-04 = O-S-ENAB

----- Treatment of spindle gear change -----

The spindle in this example has two gears (high and low). To perform a gear change, proceed as follows:

- Deactivate the general CNC input AUXEND.
- Remove the control of the spindle back to the CNC Controlled by PLC.
- Output an oscillating velocity command to change gears.
- Move the gears.
- Verify that the gear change has been completed.
- Remove the oscillating velocity command.
- Return the control of the spindle back to the CNC.
- Activate the general CNC input AUXEND.

Deactivate the general CNC input AUXEND

> While changing gears (ranges), general CNC input AUXEND should be canceled in order to interrupt the execution of the CNC. "Treatment of the general CNC input AUXEND".

Remove the control of the spindle back to the CNC Controlled by PLC.

Output an oscillating velocity command to change gears.

DFU M-41 OR DFU M-42

When a range (gear) change is requested...

= MOV 2000 SANALOG

... A 0.610V analog command for the spindle is prepared and...

= SET PLCCNTL

... the PLC grabs the control of the spindle loop.

PLCCNTL AND M2011

While the PLC has the spindle control...

= SPDLEREV

... the spindle turning direction is changed every 400 milliseconds.

Move the gears.

> The corresponding gear output (O-GEAR) is kept active until the range selection is completed (I-GEAR).

M-41 AND NOT I-GEAR1 = O-GEAR1

M-42 AND NOT I-GEAR2 = O-GEAR2

Verify that the gear change has been completed.

Remove the oscillating velocity command.

Return the control of the spindle back to the CNC.

(M-41 AND I-GEAR1) OR (M-42 AND I-GEAR2)

Once the gear change has concluded, the following must be done:

= RES M-41 = RES M-42

... remove the request for a gear change (M-41, M-42), ...

= MOV 0 SANALOG

... remove the spindle velocity command,...

= RES PLCCNTL

... Return the control of the spindle to the CNC.

I-GEAR1 = GEAR1

I-GAMA2 = GEAR2

The corresponding CNC logic input (GEAR1, GEAR2) must be activated to confirm the gear change.

----- *Lubrication of the machine ways* -----

In this example, the machine axes are lubricated in the following instances:

- On machine power-up. For 2 minutes.
- When requesting a manual lubrication. For 5 minutes.
- After the axes have been moving for 1 hour. For 5 minutes.
- After an axis has travelled a specific distance since last lubricated. For 4 minutes.

Lubrication on machine power-up.

This operation will be performed for 2 minutes.

Whenever the machine is powered up, the PLC program starts running. Therefore, the first cycle module CY1 must activate timer T2 in the mono-stable mode for 2 minutes (120000 milliseconds).

( ) = TG1 2 120000

Manual lubrication.

This operation will last 5 minutes and it will be performed at operator's request.

DFU I-LUBING = TG1 3 300000

Whenever the operator requests the lubricating (lubing) operation, T3 must be activated in the mono-stable mode for 5 minutes (300000 milliseconds).

Lubrication every hour of axis motion.

This operation takes place when the axes of the machine have been moving for an accumulated time period of 1 hour. They will be lubricated for 5 minutes.

Timer T4 is used to keep track of the axis accumulated moving time and T5 to time the 5 minute lubrication period.

The first cycle module CY1 must activate timer T4 in the delayed activation mode with a time constant of 1 hour (3600 000 milliseconds).

( ) = TG2 4 3600000

ENABLE1 OR ENABLE2 OR ENABLE3 = TEN 4

T4 only times when any of the axis is moving.

T4 = TG1 5 300000

After having timed 1 hour, T5 must be activated in the mono-stable mode for 5 minutes. (300000 milliseconds)

T5 = TRS 4 = TG2 4 3600000

Resets the axis-motion timer T4 to zero.

Lubrication when an axis has traveled a specific distance since the last time it was lubricated

PLC machine parameters USER12 (P14), "USER13 (P15) and USER14 (P16) are used to indicate the distance each axis must travel before it gets lubricated.

( ) = CNCRD(MPLC12,R31,M302) = CNCRD(MPLC13,R32,M302) = CNCRD(MPLC14,R33,M302)

Assigns to registers R31, R32 and R33 the values of PLC machine parameters USER12 (P14), "USER13 (P15) and USER14 (P16)

( ) = CNCRD(DISTX,R41,M302) = CNCRD(DISTY,R42,M302) = CNCRD(DISTZ,R43,M302)

Assigns to registers R41, R42 and R43 the distance each axis has travelled.

CPS R41 GT R31 OR CPS R42 GT R32 OR CPS R43 GT R33

If the distance traveled by any axis exceeds the one set by machine parameter,.....

= TG1 6 240000

... .timer T6 must be activated in the mono-stable mode for 4 minutes (240000 milliseconds) and .....

= MOV 0 R39

= CNCWR(R39,DISTX,M302) = CNCWR(R39,DISTY,M302) = CNCWR(R39,DISTZ,M302)

... reset to "0" the count of the distance traveled by each axis.

Activate the lubricating (lubing) operation.

T2 OR T3 OR T5 OR T6 = O-LUBING

If any of these conditions is met, the lubing output will be activated.

DFD O-LUBING = TRS2 = TRS3 = TRS4 = TRS5 = TRS6

Once the lubricating operation has concluded, All timers must be reset to "0".

---- Coolant treatment ----

The CNC executes function M08 to turn the coolant on and function M09 to turn it off.

Also, in this case, the operator has a switch to select whether the coolant is activated manually by the operator or automatically by the CNC.

I-REFMAN The operator control the coolant. Manual mode.

I-REFAUT The operator control the coolant. Automatic mode.

O-REFRIG Coolant on/off output.

I-COOLMA OR (I-COOLAU AND M-08) = O-COOL

Coolant ON.

RESETOUT = NOT O-COOL = RES M-08

The coolant will be turned off when the CNC is reset to initial conditions (RESETOUT) or when executing functions M00, M02, M09 and M30.

This instruction does not contemplate functions M00, M02, M09 and M30 since the treatment of M, S, T functions turns mark M-08 off when activating any of them.

----- Treatment of the general CNC input AUXEND -----

It is advisable to have one single instruction to control each one of the logic CNC inputs, thus preventing undesired functioning.

When having several instructions which can activate or deactivate an input, the PLC will always assign the result of analyzing the last one of those instructions.

This example shows how to group in a single instruction all the conditions that activate or deactivate one logic CNC input.

NOT T1 AND NOT M-41 AND NOT M-42 = AUXEND

Input AUXEND will remain low while:

- The "Treatment of the MSTROBE, TSTROBE, STROBE signals" is in progress (timer T1 active)
- A spindle gear change is being performed (M-41, M-42)

----- Keyboard simulation -----

With this example it is possible to simulate the theoretical path of part-program P12 whenever the operator requests it.

To do this, follow these steps:

- Indicate to the CNC that from now on the keys will come from the PLC.
- Simulate all the necessary steps sending the code of each one of the keys.
- Indicate to the CNC that from now on the keys will be coming from the CNC keyboard, not from the PLC.

In order to make sending the keys easier, a subroutine is used which utilizes the following parameters:

ENVIATEC (Send Key) Calling parameter that must be activated whenever a key is to be sent.

CODTECLA (Code of the key) Calling parameter that must contain the code corresponding to the key being simulated.

ENVIOK (Sent OK) Outgoing parameter indicating that the key code has been sent successfully.

DFU I-SIMULA = SET M120 = ERA M121 126

Whenever the operator requests the simulation (I-SIMULA), marks M120 through M126 must be activated....

= MOV PLCKEY KEYBOARD = CNCWR (KEYBOARD, KEYSRC, M100)

.. indicate to the CNC that, from now on, the keys will be coming from the PLC (PLCKEY)

= MOV MAINMENU KEYCODE = SET SENDKEY

... and send the code for the "MAIN MENU" key.

M120 AND SENTOK = RES M120 = RES SENTOK = SET M121

If the previous key was sent out successfully (SENTOK), flags M120 and SENTOK will be turned off, the flag for the next stage (M121) is activated ....

= MOV SIMULATE KEYCODE = SET SENDKEY

... and the code for the SIMULATE key (F2) is sent out.

M121 AND SENTOK = RES M121 = RES SENTOK = SET M122

If the previous key was sent out successfully (SENTOK), flags M121 and SENTOK will be turned off, the flag for the next stage (M122) is activated ....

= MOV KEY1 KEYCODE = SET SENDKEY

... ...and the code for the "1" key is sent out.

M122 AND SENTOK = RES M122 = RES SENTOK = SET M123

If the previous key was sent out successfully (SENTOK), flags M122 and SENTOK will be turned off, the flag for the next stage (M123) is activated ....

= MOV KEY2 KEYCODE = SET SENDKEY

... ...and the code for the "2" key is sent out.

M123 AND SENTOK = RES M123 = RES SENTOK = SET M124

If the previous key was sent out successfully (SENTOK), flags M123 and SENTOK will be turned off, the flag for the next stage (M124) is activated ....

= MOV ENTER KEYCODE = SET SENDKEY

... ...and the code for the "ENTER" key is sent out.

M124 AND SENTOK = RES M124 = RES SENTOK = SET M125

If the previous key was sent out successfully (SENTOK), flags M124 and SENTOK will be turned off, the flag for the next stage (M125) is activated ....

= MOV THEOPATH KEYCODE = SET SENDKEY

... and the code for the "THEORETICAL PATH" (F1) is sent out.

M125 AND SENTOK = RES M125 = RES SENTOK = SET M126

If the previous key was sent out successfully (SENTOK), flags M125 and SENTOK will be turned off, the flag for the next stage (M126) is activated ....

= MOV START KEYCODE = SET SENDKEY

... and the code for the START key is sent out.

M126 AND SENTOK = RES M126 = RES SENTOK

If the last key was sent out successfully (SENTOK), flags M126 and SENTOK will be turned off....

= MOV CNCKEY KEYBOARD = CNCWR (KEYBOARD, KEYSRC, M100)

.. and the CNC is "told" that from now on the keys will be coming from CNC keyboard (CNCKEY), not from the PLC.

--- Subroutine used to send a key ---

SENDKEY =SET M100 =SET M101 =SET M102 =RES SENDKEY

To send a key (SENDKEY), set to "1" internal marks M100 through M102 and reset the SENDKEY flag to "0".

M100 = CNCWR (KEYCODE, KEY, M100)

Sends to the CNC the code of the key to be simulated (KEYCODE). If this command is not executed correctly (M100=1), the PLC will try again on the next cycle scan.

M101 AND NOT M100 = CNCRD (KEY, LASTKEY, M101)

If the previous command was executed correctly, (M100=0), it reads the last key accepted by the CNC (LASTKEY).

M102 AND NOT M101 AND CPS LASTKEY EQ KEYCODE

If the previous command was executed correctly (M101=0) and the CNC accepted the key sent to it (LASTKEY = KEYCODE), ...

= RES M102 = SET SENTOK

... the flag is turned off (M102=0) and the key is considered to be sent out successfully (SENTOK=1)...

= NOT M101

... But if the CNC did not accept the key sent to it, it waits until it does (M101=1).

End of subroutine.

END

End of the program.

13.

PLC PROGRAMMING EXAMPLE

Main module.

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

APPENDIX

A. CNC technical characteristics 425
B. Probe connection 429
C. Summary of internal CNC variables 431
D. Summary of PLC commands 437
E. Summary of PLC inputs and outputs 441
F. 2-digit BCD code output conversion table 447
G. Key codes 449
H. Logic outputs of key status 451
I. Key inhibiting codes 453
J. Machine parameter setting chart 455
K. M functions setting chart 461
L. Leadscrew error compensation table 463
M. Cross compensation table 465
N. Maintenance 467

FAGOR

CNC 8035

(SOFT M: V15.3x)
(SOFT T: V16.3x)

CNC TECHNICAL CHARACTERISTICS

The machine manufacturer must comply with the EN 60204-1 (IEC-204-1) standard in terms of protection against electrical shock due to faulty contacts with external power supply.

This unit MUST NOT be opened by unauthorized personnel.

To avoid overheating the internal circuitry, do not block the ventilation grooves and install a ventilation system that removes the hot air from the enclosure.

General characteristics

3 feedback inputs for the axes.
3 analog outputs to control the axes (±10 V).
1 feedback input for the spindle encoder.
1 analog output to control the spindle (±10 V).
2 feedback inputs for the electronic handwheels.
2 inputs for digital probe (TTL or 24 V DC).
Digital CAN servo.

0.0001mm or 0.00001 inch resolution.
Multiplying factor up to x 25 with sinewave input.
Feedrate from 0.0001 to 99999.9999 mm/min (0.00001 - 3937 inches/min).
Maximum travel: ±99999.9999 mm (±3937 inches).

1 RS232C communication line.
40 optocoupled digital inputs
24 optocoupled digital outputs

32-bit processor
Math coprocessor
Graphics coprocessor.
256 Kb CNC program memory.
Block processing time of 6.5 ms.
Configurable sample time: 4, 5 or 6 ms

Approximate weight 7.5 Kg.
Maximum consumption of 48 W in normal operation.

Color monitor

Technology: Color TFT LCD.
Diagonal display area dimension: 10,4”.
Resolution: VGA 3 x 640 x 480 pixels.
Number of colors: 262144 Colors (6 bit for each subpixel RGB).
Backlit with 2 cold-cathode fluorescent lamps.

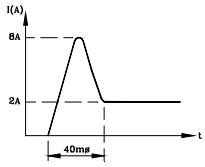
Due to the current state of the COLOR TFT LCD technology, all manufacturers accept the fact the LCD screens have a certain number of defective pixels.

Monochrome monitor

Technology: LCD STN.
Diagonal display area dimension: 7,5".
Resolution: 640 x 480 pixels.
8 gray ranges.
Backlit with 1 cold-cathode fluorescent lamp.

Due to the current state of the LCD technology, all manufacturers accept the fact the LCD screens have a certain number of defective pixels.

Power supply



Nominal voltage: 20 V minimum and 30 V maximum.
Ripple: 4 V.
Nominal current: 2 A.
Current peak on power-up: 8 A.
The figure shows the shape of the supply current on power-up

PLC

Memory: 100 kbytes.
Programming in mnemonics.
1 millisecond time unit.
512 inputs.
512 outputs.
2047 user marks.
256 32-bit registers.
256 32-bit counters.
256 32-bit timers.

5V probe input.

Typical value: 0.25 mA. ≅ Vin = 5V.
High threshold (logic level 1) VIH: A partir de +2,4 Vcc.
Low threshold (logic level 0) VIL: Por debajo de +0,9 Vcc
Maximum nominal voltage Vimax = +15 Vcc.

24V probe input.

Typical value: 0.30 mA. ≅ Vin = 24V.
High threshold (logic level 1) VIH: A partir de +12,5 Vcc.
Low threshold (logic level 0) VIL: Por debajo de +4 Vcc
Maximum nominal voltage Vimax = +35 Vcc.

Digital inputs

Nominal voltage + 24 Vdc.
Maximum nominal voltage + 30 Vdc.
Minimum nominal voltage + 18 Vdc.
High threshold (logic level 1) VIH: A partir de +18 Vcc.
Low threshold (logic level 0) VIL: Under +5 Vdc or not connected.
Typical consumption of each input 5 mA.
Maximum consumption of each input 7 mA.
Protection by means of galvanic isolation by optocouplers.
Protection against reverse connection up to -30 Vdc.

Digital outputs

Nominal supply voltage + 24 Vdc.
Maximum nominal voltage + 30 Vdc.
Minimum nominal voltage + 18 Vdc.
Output voltage Vout = Supply voltage (Vdc) -3 V
Maximum output current 100 mA
Protection by means of galvanic isolation by optocouplers.
Shortcircuit protection. Place external recovery diodes.

Analog outputs for axes and spindle

Command voltage within ±10 V, 16-bit solution
Minimum impedance of the connected connector 10 KΩ.
Shielded cable should be used.

Ambient conditions

Relative humidity: 30-95 % without condensation.
Operating temperature: between +5 ºC (41 ºF) and +40 ºC (104 ºF) with an average under +35 ºC (95 ºF).
Storage temperature: between -25 ºC (-13 ºF) and +70 ºC (158 ºF).
Maximum work altitude: Meets the “IEC 1131-2” standard.

Packaging

Meets the “EN 60068-2-32” standard

Vibration

When running 10-50 Hz amplitude 0.2 mm (1g).
While being shipped 10-50 Hz amplitude 1 mm (5g).
Free fall of packaged unit under Fagor ruling 1m.

Electromagnetic compatibility and safety

Refer to the section on safety conditions in the introduction of this manual.

Degree of protection

Central Unit : IP54 for the front panel and IP2X for the rear panel.
Accessible parts inside: IP1X.
Operator panel: IP54

Battery

3.5 V lithium battery
Estimated life: 3 years
As from error indication (low battery) the information contained in the memory will be kept for 10 days maximum, with the CNC off. It must be replaced.



Neither attempt to recharge the battery nor expose it to temperatures over 100 ºC (212 ºF).

Do not short-circuit the terminals for risk of explosion or combustion.

A.

APPENDIX

CNC technical characteristics

FAGOR

CNC 8035

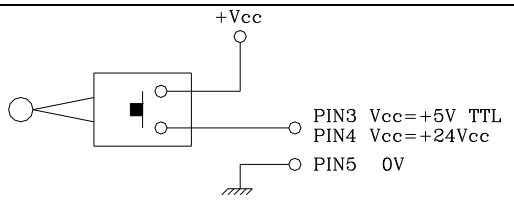
(SOFT M: V15.3X)
(SOFT T: V16.3X)

PROBE CONNECTION

The CNC has two probe inputs (of 5Vdc and 24Vdc) at connector X3.

Depending on the type of connection applied the g.m.p. “PRBPULSE” (P39) must be set, indicating whether it operates with the leading edge or trailing edge of the signal which the probe provides.

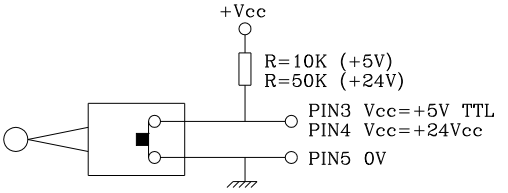
Probe with “normally open contact” output,



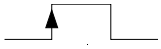
Active high (with an up flank).



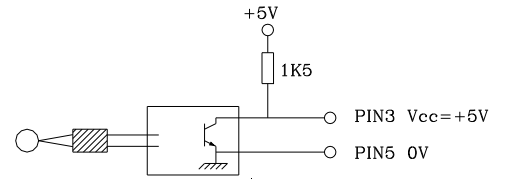
Probe with “normally closed contact” output.



Active high (with an up flank).



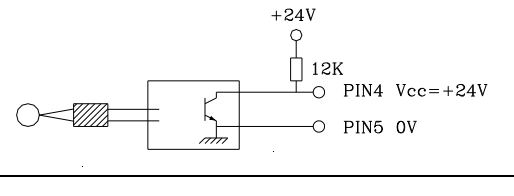
Interface with output in open collector Connection to +5 V.



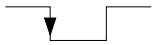
Active low (with a down flank).



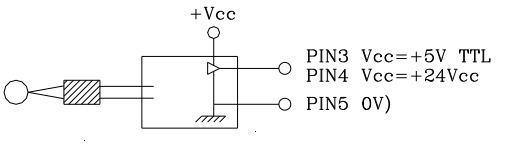
Interface with output in open collector Connection to +24 V.



Active low (with a down flank).



Interface with output in PUSH-PULL.



The active flank depends on the interface

B.

APPENDIX

Probe connection

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

SUMMARY OF INTERNAL CNC VARIABLES.

- The R symbol indicates that the variable can be read.
- The W symbol indicates that the variable can be modified.

Variables associated with tools.

Variable	CNC	PLC	DNC	(section 11.1)
TOOL	R	R	R	Number of the active tool.
TOD	R	R	R	Number of active tool offset.
NXTOOL	R	R	R	Number of the next requested tool waiting for M06.
NXTOD	R	R	R	Number of the next tool's offset.
TMZPn	R	R	-	(n) tool's position in the tool magazine.
TLFDn	R/W	R/W	-	(n) tool's offset number.
TLFFn	R/W	R/W	-	(n) tool's family code.
TLFNn	R/W	R/W	-	Nominal life assigned to tool (n).
TLFRn	R/W	R/W	-	Real life value of tool (n).
TMZTn	R/W	R/W	-	Contents of tool magazine position (n).
HTOR	R/W	R	R	Tool radius being used by the CNC to do the calculations.

Tool related variables (specific of the mill model).

TORn	R/W	R/W	-	Tool radius value of offset (n).
TOLn	R/W	R/W	-	Tool length value of offset (n).
TOIn	R/W	R/W	-	Tool radius wear of offset (n).
TOKn	R/W	R/W	-	Tool length wear of offset (n).

Tool related variables (specific of the lathe model).

TOXn	R/W	R/W	-	Tool length offset (n) along X axis.
TOZn	R/W	R/W	-	Tool length offset (n) along Z axis.
TOFn	R/W	R/W	-	Location code of offset (n).
TORn	R/W	R/W	-	Tool radius value of offset (n).
TOIn	R/W	R/W	-	Tool length wear of offset (n) along X axis.
TOKn	R/W	R/W	-	Tool length wear of offset (n) along Z axis.
NOSEAn:	R/W	R/W	-	Cutter angle of indicated tool.
NOSEWn	R/W	R/W	-	Width of indicated tool.
CUTAn	R/W	R/W	-	Cutting angle of indicated tool.

Variables associated with zero offsets.

Variable	CNC	PLC	DNC	(section 11.2)
ORG(X-C)	R	R	-	Active zero offset on the selected axis. The value of the additive offset indicated by the PLC is not included.
PORGF	R	-	R	Abscissa coordinate value of polar origin.
PORGS	R	-	R	Ordinate coordinate value of polar origin.
ORG(X-C)n	R/W	R/W	R	Zero offset (n) value of the selected axis.
PLCOF(X-C)	R/W	R/W	R	Value of the additive zero offset activated via PLC.
ADIOF(X-C)	R	R	R	Value for the selected axis of the zero offset with additive handwheel.

Variables associated with machine parameters.

Variable	CNC	PLC	DNC	(section 11.3)
MPGn	R	R	-	Value assigned to general machine parameter (n).
MP(X-C)n	R	R	-	Value assigned to (X-C) axis machine parameter (n).
MPSn	R	R	-	Value assigned to machine parameter (n) of the main spindle.
MPLCn	R	R	-	Value assigned to machine parameter (n) of the PLC.

Variables associated with work zones.

Variable	CNC	PLC	DNC	(section 11.4)
FZONE	R	R/W	R	Status of work zone 1.
FZLO(X-C)	R	R/W	R	Work zone 1. Lower limit along the selected axis (X/C).
FZUP(X-C)	R	R/W	R	Work zone 1. Upper limit along the selected axis (X-C).
SZONE	R	R/W	R	Status of work zone 2.
SZLO(X-C)	R	R/W	R	Work zone 2. Lower limit along the selected axis (X/C).
SZUP(X-C)	R	R/W	R	Work zone 2. Upper limit along the selected axis (X-C).
TZONE	R	R/W	R	Status of work zone 3.
TZLO(X-C)	R	R/W	R	Work zone 3. Lower limit along the selected axis (X/C).
TZUP(X-C)	R	R/W	R	Work zone 3. Upper limit along the selected axis (X-C).
FOZONE	R	R/W	R	Status of work zone 4.
FOZLO(X-C)	R	R/W	R	Work zone 4. Lower limit along the selected axis (X/C).
FOZUP(X-C)	R	R/W	R	Work zone 4. Upper limit along the selected axis (X-C).
FIZONE	R	R/W	R	Status of work zone 5.
FIZLO(X-C)	R	R/W	R	Work zone 5. Lower limit along the selected axis (X/C).
FIZUP(X-C)	R	R/W	R	Work zone 5. Upper limit along the selected axis (X-C).

Feedrate related variables.

Variable	CNC	PLC	DNC	(section 11.5)
FREAL	R	R	R	Real feedrate of the CNC in mm/min or inch/min.
FREAL(X-C)	R	R	R	Actual (real) CNC feedrate of the selected axis.
FTEO/X-C)	R	R	R	Theoretical CNC feedrate of the selected axis.

Variables associated with function G94.

FEED	R	R	R	Active feedrate at the CNC in mm/min or inch/min.
DNCF	R	R	R/W	Feedrate selected via DNC.
PLCF	R	R/W	R	Feedrate selected via PLC.
PRGF	R	R	R	Feedrate selected by program.

Variables associated with function G95.

FPREV	R	R	R	Active feedrate at CNC, in m/rev or inch/rev.
DNCFPR	R	R	R/W	Feedrate selected via DNC.
PLCFPR	R	R/W	R	Feedrate selected via PLC.
PRGFPR	R	R	R	Feedrate selected by program.

Variables associated with function G32.

PRGFIN	R	R	R	Feedrate selected by program, in 1/min.

Variables associated with feedrate override (%)

FRO	R	R	R	Feedrate Override (%) active at the CNC.
PRGFRO	R/W	R	R	Override (%) selected by program.
DNCFRO	R	R	R/W	Override (%) selected via DNC.
PLCFRO	R	R/W	R	Override (%) selected via PLC.
CNCFRO	R	R	R	Override (%) selected from the front panel knob.
PLCCFR	R	R/W	R	Override (%) of the PLC execution channel.

Coordinate related variables.

Variable	CNC	PLC	DNC	(section 11.6)
PPOS(X-C)	R	-	-	Programmed theoretical position value (coordinate).
POS(X-C)	R	R	R	Machine coordinates. Real coordinates of the tool base.
TPOS(X-C)	R	R	R	Machine coordinates. Theoretical coordinates of the tool base.
APOS(X-C)	R	R	R	Part coordinates. Real coordinates of the tool base.
ATPOS(X-C)	R	R	R	Part coordinates. Theoretical coordinates of the tool base.
DPOS(X-C)	R	R	R	Theoretical position of the probe when the probe touched the part.
FLWE(X-C)	R	R	R	Following error of the indicated axis.
DIST(X-C)	R/W	R/W	R	Distance traveled by the indicated axis.
LIMPL(X-C)	R/W	R/W	R	Second upper travel limit.
LIMMI(X-C)	R/W	R/W	R	Second lower travel limit.
DPLY(X-C)	R	R	R	Coordinate of the selected axis displayed on the screen.
GPOS(X-C)n p	R	-	-	Coordinate of the selected axis, programmed in the (n) block of the program (p).

Variables associated with electronic handwheels.

Variable	CNC	PLC	DNC	(section 11.7)
HANPF	R	R	-	Pulses received from 1st handwheel since the CNC was turned on.
HANPS	R	R	-	Pulses received from 2nd handwheel since the CNC was turned on.
HANPT	R	R	-	Pulses received from 3rd handwheel since the CNC was turned on.
HANPFO	R	R	-	Pulses received from 4th handwheel since the CNC was turned on.
HANDSE	R	R		For handwheels with a selector button, it indicates whether that button has been pressed or not.
HANFCT	R	R/W	R	Multiplying factor different for each handwheel (when having several).
HBEVAR	R	R/W	R	HBE handwheel. Reading enabled, axis being jogged and multiplying factor (x1, x10, x100).
MASLAN	R/W	R/W	R/W	Linear path angle for "Path handwheel" or "Path Jog" mode.
MASCFI	R/W	R/W	R/W	Arc center coordinates for "Path handwheel mode" or "Path jog".
MASCSE	R/W	R/W	R/W	Arc center coordinates for "Path handwheel mode" or "Path jog".

Feedback related variables.

Variable	CNC	PLC	DNC	(section 11.8)
ASIN(X-C)	R	R	R	A signal of the CNC's sinusoidal feedback for the selected axis.
BSIN(X-C)	R	R	R	B signal of the CNC's sinusoidal feedback for the selected axis.
ASINS	R	R	R	"A" signal of the CNC's sinusoidal feedback for the spindle.
BSINS	R	R	R	"B" signal of the CNC's sinusoidal feedback for the spindle.

Variables associated with the main spindle.

Variable	CNC	PLC	DNC	(section 11.9)
SREAL	R	R	R	Real spindle speed
FTEOS	R	R	R	Theoretical spindle speed.

Variables associated with spindle speed.

SPEED	R	R	R	Active spindle speed at the CNC.
DNCS	R	R	R/W	Spindle speed selected via DNC.
PLCS	R	R/W	R	Spindle speed selected via PLC.
PRGS	R	R	R	Spindle speed selected by program.

Variables associated with constant cutting speed (lathe model).

CSS	R	R	R	Constant surface speed active at the CNC.
DNCCSS	R	R	R/W	Constant surface speed selected via DNC.
PLCCSS	R	R/W	R	Constant surface speed selected via PLC.
PRGCSS	R	R	R	Constant surface speed selected by program.

Variables associated with the spindle override.

SSO	R	R	R	Spindle Speed Override (%) active at the CNC.
PRGSSO	R/W	R	R	Override (%) selected by program.
DNCSSO	R	R	R/W	Override (%) selected via DNC.
PLCSSO	R	R/W	R	Override (%) selected via PLC.
CNCSSO	R	R	R	Spindle Speed Override (%) selected from front panel.

Speed limit related variables.

SLIMIT	R	R	R	Spindle speed limit active at the CNC.
DNCSL	R	R	R/W	Spindle speed limit selected via DNC.
PLCSL	R	R/W	R	Spindle speed limit selected via PLC.
PRGSL	R	R	R	Spindle speed limit selected by program.
MDISL	R	R/W	R	Maximum machining spindle speed.

Position related variables.

POSS	R	R	R	Real Spindle position. Reading from the PLC in ten-thousandths of a degree (within ±999999999) and from the CNC in degrees (within ±99999.9999).
RPOSS	R	R	R	Real Spindle position. Reading from the PLC in ten-thousandths of a degree (between 0 and 3600000) and from the CNC in degrees (between 0 and 360).
TPOSS	R	R	R	Theoretical spindle position. Reading from the PLC in ten-thousandths of a degree (within ±999999999) and from the CNC in degrees (within ±99999.9999).
RTPOSS	R	R	R	Theoretical spindle position. Reading from the PLC in ten-thousandths of a degree (between 0 and 3600000) and from the CNC in degrees (between 0 and 360).
PRGSP	R	R	R	Position programmed in M19 via program for the main spindle.

Variables related to the following error.

FLWES	R	R	R	Spindle following error.

PLC related variables.

Variable	CNC	PLC	DNC	
PLCMSG	R	-	R	Number of the active PLC message with the highest priority.
PLCIn	R/W	-	-	32 PLC inputs starting from (n).
PLCOn	R/W	-	-	32 PLC outputs starting from (n).
PLCMn	R/W	-	-	32 PLC marks starting from (n).
PLCRn	R/W	-	-	(n) Register.
PLCTn	R/W	-	-	Indicated (n) Timer's count.
PLCCn	R/W	-	-	Indicated (n) Counter's count.
PLCMMn	R/W	-	-	Modifies the (n) mark of the PLC.

Variables associated with local and global parameters.

Variable	CNC	PLC	DNC	(*section 11.10*)
GUP n	-	R/W	-	Global parameter (P100-P299) (n), user parameter (P1000-P1255) (n), OEM parameter (P2000-P2255) (n).
LUP (a,b)	-	R/W	-	Indicated local (P0-P25) parameter (b) of the nesting level (a).
CALLP	R	-	-	Indicates which local parameters have been defined by means of a PCALL or MCALL instruction (calling a subroutine).

Operating-mode related variables.

Variable	CNC	PLC	DNC	(*section 11.11*)
OPMODE	R	R	R	Operating mode.

Other variables.

Variable	CNC	PLC	DNC	(*section 11.12*)
NBTOOL	R	-	R	Number of the tool being managed..
PRGN	R	R	R	Number of the program in execution.
BLKN	R	R	R	Label number of the last executed block.
GSn	R	-	-	Status of the indicated G function (n).
GGSA	-	R	R	Status of functions G00 thru G24.
GGSB	-	R	R	Status of functions G25 thru G49.
GGSC	-	R	R	Status of functions G50 thru G74.
GGSD	-	R	R	Status of functions G75 thru G99.
GGSE	-	R	R	Status of functions G100 thru G124.
GGSF	-	R	R	Status of functions G125 thru G149.
GGSG	-	R	R	Status of functions G150 thru G174.
GGSH	-	R	R	Status of functions G175 thru G199.
GGSI	-	R	R	Status of functions G200 thru G224.
GGSJ	-	R	R	Status of functions G225 thru G249.
GGSK	-	R	R	Status of functions G250 thru G274.
GGSL	-	R	R	Status of functions G275 thru G299.
GGSM	-	R	R	Status of functions G300 thru G320.
MSn	R	-	-	Status of the indicated M function (n)
GMS	-	-	R	Status of M functions: M (0..6, 8, 9, 19, 30, 41..44).
PLANE	R	R	R	Abscissa and ordinate axes of the active plane.

Variable	CNC	PLC	DNC	(section 11.12)
LONGAX	R	R	R	Axis affected by the tool length compensation (G15).
MIRROR	R	R	R	Active mirror images.
SCALE	R	R	R	General scaling factor applied. Reading from the PLC in ten-thousandths.
SCALE(X-C)	R	R	R	Scaling Factor applied only to the indicated axis. Reading from the PLC in ten-thousandths.
ORGROT	R	R	R	Rotation angle (G73) of the coordinate system.
ROTPF	R	-	-	Abscissa of rotation center.
ROTPS	R	-	-	Ordinate of rotation center.
PRBST	R	R	R	Returns probe status.
CLOCK	R	R	R	System clock in seconds.
TIME	R	R	R/W	Time in Hours, minutes and seconds.
DATE	R	R	R/W	Date in Year-Month-Day format
TIMER	R/W	R/W	R/W	Clock activated by PLC, in seconds.
CYTIME	R	R	R	Time to execute a part in hundredths of a second.
PARTC	R/W	R/W	R/W	Parts counter of the CNC.
FIRST	R	R	R	First time a program is executed.
KEY	R/W	R/W	R/W	keystroke code.
KEYSRC	R/W	R/W	R/W	Source of the keys.
ANAIn	R	R	R	Voltage (in volts) of the indicated analog input (n).
ANAOn	R/W	R/W	R/W	Voltage (in volts) to apply to the indicated output (n).
CNCERR	-	R	R	Active CNC error number.
PLCERR	-	-	R	Active PLC error number.
DNCERR	-	R	-	Number of the error generated during DNC communications.
DNCSTA	-	R	-	DNC transmission status.
TIMEG	R	R	R	Remaining time to finish the dwell block (in hundredths of a second).
SELPRO	R/W	R/W	R	When having two probe inputs, it selects the active input.
DIAM	R/W	R/W	R	It changes the programming mode for X axis coordinates between radius and diameter.
PRBMOD	R/W	R/W	R	Indicates whether a probing error must be displayed or not.
RETREJ	R/W	R/W	R	It indicates that the retraction in drilling, or the mill type threading or lathe type threading has finished.
RIP	R	R	R	Linear theoretical feedrate resulting from the next loop (in mm/min).
RIGIER	R	R	R	Offset between the projection of the following error of the spindle onto the longitudinal axis and the following error of the longitudinal axis.

The "KEY" variable can be "written" at the CNC only via the user channel.

The "NBTOOL" variable can only be used within the tool change subroutine.

C.

APPENDIX

Summary of internal CNC variables.

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

SUMMARY OF PLC COMMANDS

PLC Resources.

Inputs:	I 1/512
Outputs:	O 1/512
User marks:	M 1/2000
Arithmetic flag marks:	M 2003
Clock marks:	M 2009/2024
Fixed status marks:	M 2046/2047
Marks associated with messages:	M 4000/4127
Marks associated with errors:	M 4500/4563
Screen marks:	M 4700/4955
CNC communication marks:	M 5000/5957
Timers:	T 1/256
Counters:	C 1/256
User registers	R 1/499
Registers for communication with the CNC	R 500/559

The value stored in each register will be considered by the PLC as a signed integer which could be referred to in the following formats:

Decimal	Integer within ±2147483647.
Hexadecimal	Number preceded by the $ sign and between 0 and FFFFFFFF.
Binary	Number preceded by the letter B and made up of up to 32 bits (1 or 0).

Directing instructions.

PRG	Main module.
CY1	First cycle module.
PE t	Periodic module. It will be executed every t time (in milliseconds).
END	End of module.
L 1/256	Label.
DEF	Symbol definition.
REA	The consultations will use real values.
IMA	The consultations will use image values.
IRD	Updates the "I" resources with the values of the physical inputs.
MRD	Updates resources M5000/5957 and R500/559 with the values of the logic CNC outputs.
OWR	Updates the physical outputs with the real values of the "O" resources.
MWR	Updates the logic CNC inputs (internal variables) with the values of resources M5000/5957 and R500/599
TRACE	Captures data for the Logic Analyzer while executing the PLC cycle.

Simple consulting Instructions.

I	1/512	Inputs
O	1/512	Outputs
M	1/5957	Marks
T	1/256	Timers
C	1/256	Counters
B	0/31 R 1/499	Register bit

Flank detection consulting Instructions.

DFU (Up flank detection) DFD (Down flank detection)	I 1/512 O 1/512 M 1/5957

Comparison consulting Instructions.

CPS	T 1/256 C 1/256 R 1/559 #	GT GE EQ NE LE LT	T 1/256 C 1/256 R 1/559 #

Operators.

NOT	Inverts the result of the consulting instruction it precedes.
AND	Performs the logic function “AND” between consulting instructions.
OR	Performs the logic function “OR” between consulting instructions.
XOR	Performs the logic function “EXCLUSIVE OR” between consulting instructions.

Assignment Binary Action Instructions.

= I	1/512	Inputs.
= O	1/512	Outputs.
= M	1/5957	Marks.
= TEN	1/256	Timer enable.
= TRS	1/256	Timer reset.
= TGn	1/256 n/R	Timer trigger input.
= CUP	1/256	Counter count up.
= CDW	1/256	Counter count down.
= CEN	1/256	Counter enable.
= CPR	1/256 n/R	Counter preset.
= B	0/31 R 1/499	Register Bits.

Conditioned binary actions instructions.

= SET If the logic expression is “1”, this action assigns a “1” to the resource.

= RES If the logic expression is “1”, this action assigns a “0” to the resource.

= CPL If the logic expression is “1”, this action complements the logic state of the resource.

= SET = RES = CPL	I 1/512 O 1/512 M 1/5957 B 0/31 R 1/559

Sequence breaking action instructions.

= JMP L 1/256	Unconditional jump.
= RET	Return or End of a subroutine.
= CAL L 1/256	Call to a subroutine.

Arithmetic action instructions.

= MOV Transfers the logic states of the indicated source to the indicated destination.

= NGU Complements all register bits.

= NGS Changes the sign of the Register contents.

= ADS Adds the contents of a two registers or a number and a register content.

= SBS Subtracts between the contents of two registers or between a number and a register content.

= MLS Multiplies the contents of two registers or a number and a register content.

= DVS Divides the contents of two registers or a number and a register content.

= MDS Module between registers contents or between a number and a register content.

	Origin	Destination	Source code	Destination code	Number of bits to transmit
MOV	I 1/512 O 1/512 M 1/5957 T 1/256 C 1/256 R 1/559 #	I 1/512 O 1/512 M 1/5957 R 1/559	0 (Bin) 1 (BCD)	0 (Bin) 1 (BCD)	32 28 24 20 16 12 8 4

ADS SBS MLS DVS MDS	R1/559 #	R1/559 #	R1/559

Logic action instructions.

= AND Logic AND operation between register contents or between a number and a register content.

= OR Logic OR operation between register contents or between a number and a register content.

= XOR Logic XOR operation between register contents or between a number and a register content.

= RR 1/2 Clockwise register rotation.

= RR 1/2 Counterclockwise register rotation.

AND OR XOR	R1/559 #	R1/559 #	R1/559

	Origin	Number of repetitions	Destination
RR1 RR2 RL1 RL2	R1/559	R1/559 0/31	R1/559

Specific action instructions.

= ERA Group erase

= CNCRD CNCRD (Variable, R1/559, M1/5957)
Reading of internal CNC variables.

= CNCWR CNCWR (R1/559, Variable, M1/5957)
Writing of internal CNC variables.

= PAR PAR R1/559 M1/5957
Parity of a register

ERA	I	1/512	1/512
	O	1/512	1/512
	M	1/5957	1/5957
	T	1/256	1/256
	C	1/256	1/256
	R	1/559	1/559

SUMMARY OF PLC INPUTS AND OUTPUTS

GENERAL LOGIC INPUTS.

/EMERGEN	M5000	Stops the axes and the spindle. Displays the error.
/STOP	M5001	Stops execution of the part program, maintaining spindle rotation.
/FEEDHOL	M5002	Stops axis feed momentarily, maintaining spindle rotation.
/XFERINH	M5003	Prevents the next block from being executed, but finishes the one being executed.
CYSTART	M5007	Starts program execution..
SBLOCK	M5008	The CNC changes to single block execution mode.
MANRAPID	M5009	Selects rapid travel for all the movements that are executed in JOG mode.
OVRCAN	M5010	Selects feedrate override at 100%.
LATCHM	M5011	The axes will move from the moment the corresponding JOG key is pressed until the STOP key is pressed.
ACTGAIN2	M5013	Indicates that the CNC assumes the 2nd set of gains.
RESETIN	M5015	Initial machining conditions selected by machine parameter.
AUXEND	M5016	Indicates that the execution of the M, S and T functions has completed.
TIMERON	M5017	Enables the timer:
TREJECT	M5018	Rejection of tool in use.
PANELOFF	M5019	Deactivation of keyboard.
PLCABORT	M5022	Possibility to abort the PLC channel
PLCREADY	M5023	PLC without errors.
INT1	M5024	Executes the interruption subroutine indicated in g.m.p. P35
INT2	M5025	Executes the interruption subroutine indicated in g.m.p. P36
INT3	M5026	Executes the interruption subroutine indicated in g.m.p. P37
INT4	M5027	Executes the interruption subroutine indicated in g.m.p. P38
BLKSKIP1	M5028	The “/ and /1” block skip condition is met.
BLKSKIP2	M5029	The “/2” block skip condition is met.
BLKSKIP3	M5030	The “/3” block skip condition is met.
M01STOP	M5031	Stops execution of the part program when the auxiliary M01 function is executed.
RETRACE	M5051	It activates the Retrace function.
ACTLM2	M5052	Activates the second travel limits.
HNLINARC	M5053	Type of path with "Path Handwheel" or "Path jog".
MASTRHND	M5054	It activates the "Path Handwheel" or "Path jog" mode.
EXRAPID	M5057	Selects rapid travel for all the movements that are executed in execution mode.
FLIMITAC	M5058	Limit the feedrate of each axis to the value set in its machine parameter FLIMIT (P75).
SLIMITAC	M5059	Limit the spindle speed to the value set in its machine parameter SLIMIT (P66).
BLOABOR	M5060	Finish the movement in progress and start executing the next block.
ACTGAINT	M5063	Indicates that the CNC assumes the 3rd set of gains.
SKIPCYCL	M5064	Go on to the next hole after a retraction in a drilling or a mill type threading.
RETRACYC	M5065	It indicates that the Z axis has been stopped before starting the retraction.

AXIS LOGIC INPUTS.

	Axis 1	Axis 2	Axis 3	
LIMIT+	M5100	M5150	M5200	Travel limit overrun. Stops the axes and the spindle. Displays the error.
LIMIT-*	M5101	M5151	M5201	Travel limit overrun. Stops the axes and the spindle. Displays the error.
DECEL*	M5102	M5152	M5202	Home switch pressed.
INHIBIT*	M5103	M5153	M5203	Inhibits axis movement.
MIRROR*	M5104	M5154	M5204	Applies mirror image.
SWITCH*	M510	M5155	M5205	Swaps commands (axes with 1 drive)
DRO*	M5106	M5156	M5206	DRO axis. (DRO*=1 and SERVOON*=0).
SERVO*ON	M5107	M5157	M5207	Servo signal. (=1) closes the position loop
AXIS+*	M5108	M5158	M5208	Moves the axis in JOG mode. Similar to JOG keys.
AXIS-*	M5109	M5159	M5209	Moves the axis in JOG mode. Similar to JOG keys.
SPENA*	M5110	M5160	M5210	With CAN. Speed enable signal of the drive.
DRENA*	M5111	M5161	M5211	With CAN. Drive enable signal of the drive.
ELIMINA*	M5113	M5163	M5213	It does not display the axis and cancels the feedback alarms.
SMOTOF*	M5114	M5164	M5214	Cancels the SMOTIME filter, a.m.p. SMOTIME (P58).
LIM*OFF	M5115	M5165	M5215	It ignores the software limits.
MANINT*	M5116	M5166	M5216	Activate the additive handwheel in each axis.

SPINDLE LOGIC INPUTS.

Main		
LIMIT+S	M5450	Travel limit overrun. Stops the axes and the spindle. Displays the error.
LIMIT -S	M5451	Travel limit overrun. Stops the axes and the spindle. Displays the error.
DECELS	M5452	Home switch pressed.
SPDLEINH	M5453	Outputs a zero command for the spindle.
SPDLEREV	M5454	Reverses the spindle turning direction.
SMOTOFS	M5455	Cancels the SMOTIME filter, s.m.p. SMOTIME (P46).
SERVOSON	M5457	Servo signal. (=1) to move the spindle in closed loop (M19).
GEAR1	M5458	Spindle gear 1 selected.
GEAR2	M5459	Spindle gear 2 selected.
GEAR3	M5460	Spindle gear 3 selected.
GEAR4	M5461	Spindle gear 4 selected.
SPENAS	M5462	With CAN. Speed enable signal of the drive.
DRENAS	M5463	With CAN. Drive enable signal of the drive.
PLCFM19	M5464	Rapid synchronization feedrate, in M19.
M19FEED	R505	Rapid synchronization feedrate, in M19.
PLCCNTL	M5465	Spindle controlled directly by the PLC.
SANALOG	R504	Spindle velocity command. Only for spindle controlled by PLC.
ELIMIS	M5456	The CNC does not display the spindle although it keeps controlling it.

KEY INHIBITING LOGIC INPUTS.

KEYDIS1	R500	Inhibit the operation of the panel keys.
KEYDIS2	R501	
KEYDIS3	R502	
KEYDIS4	R503	

LOGIC INPUTS OF THE PLC CHANNEL

/FEEDHOP	M5004	Stops the PLC axes momentarily, maintaining spindle rotation.
/XFERINP	M5005	Prevents the next block from being executed in the PLC channel, but finishes the one being executed.
AUXENDP	M5006	Indicates that the execution of the M, S and T functions has completed.
BLOABORP	M5061	Possibility to abort the PLC channel

GENERAL LOGIC OUTPUTS.

CNCREADY	M5500	CNC without errors.
START	M5501	The CYCLE START key of the front panel has been pressed.
FHOUT	M5502	Indicates that program execution has been interrupted.
RESETOUT	M5503	Indicates that the CNC is set to initial conditions.
LOPEN	M5506	Indicates that the positioning loop for the axes is open.
/ALARM	M5507	An alarm or emergency condition was detected.
JOG	M5508	The manual operation (JOG) mode has been selected.
AUTOMAT	M5509	The automatic operation mode has been selected.
MDI	M5510	The MDI mode has been selected.
SBOUT	M5511	The single block execution mode has been selected.
INCYCLE	M5515	The part program is being executed.
RAPID	M5516	A rapid traverse is being executed (G00).
TAPPING	M5517	A tapping cycle is being executed (G84).
THREAD	M5518	A threading block is being executed (G33).
PROBE	M5519	A probing movement is being executed (G75/G76).
ZERO	M5520	A machine reference search is being executed (G74).
RIGID	M5521	A rigid tapping block in execution. Milling model.
RETRAEND	M5522	Retrace function. All possible blocks have been retraced.
CSS	M5523	The G96 function is selected.
SELECT0	M5524	Position selected at the front panel switch.
SELECT1	M5525	Position selected at the front panel switch.
SELECT2	M5526	Position selected at the front panel switch.
SELECT3	M5527	Position selected at the front panel switch.
SELECT4	M5528	Position selected at the front panel switch.
SELECT5	M5529	Position selected at the front panel switch.
SELECT6	M5530	Position selected at the front panel switch.
SELECT7	M5531	Position selected at the front panel switch.
SELECTOR	R564	Position selected at the front panel switch.

MSTROBE	M5532	Indicates that the auxiliary M functions which are indicated in registers R550 to R556 must be executed.
SSTROBE	M5533	Indicates that the auxiliary S function of register R557 must be executed.
TSTROBE	M5534	Indicates that the auxiliary T function of register R558 must be executed.
T2STROBE	M5535	Indicates that the auxiliary T function of register R559 must be executed.
ADVINPOS	M5537	For punch presses, it indicates that punching may begin.
INTEREND	M5538	It indicates that the interpolation is finished.
INPOS	M5539	The axes are in position.
DM00	M5547	Program interrupted after executing the auxiliary function M00.
DM01	M5546	Program interrupted after executing the auxiliary function M01.
DM02	M5545	The execution of the program has stopped after executing the auxiliary M02 function.
DM03	M5544	The spindle is turning clockwise (M03).
DM04	M5543	The spindle is turning counterclockwise (M04).
DM05	M5542	The spindle is stopped (M05).
DM06	M5541	The auxiliary M06 function has been executed.
DM08	M5540	The coolant output has been activated (M08).
DM09	M5555	The coolant output has been canceled (M09).
DM19	M5554	A block with spindle stop has been executed (M19).
DM30	M5553	The program concluded after executing the auxiliary M30 function.
DM41	M5552	First spindle speed gear (range) selected (M41).
DM42	M5551	Second spindle speed gear (range) selected (M42).
DM43	M5550	Third spindle speed gear (range) selected (M43).
DM44	M5549	Fourth spindle speed gear (range) selected (M44).
RETRACT	M5567	It indicates that the drilling stop or the mill type threading stop or the retraction in a lathe type threading has finished.

AXIS LOGIC OUTPUTS.

	Axis 1	Axis 2	Axis 3	
ENABLE*	M5600	M5650	M5700	Enables axis movement.
DIR*	M5601	M5651	M5701	Indicate axis moving direction.
REFPOIN*	M5602	M5652	M5702	Home search done.
DRSTAF*	M5603	M5653	M5703	With CAN. They indicate servo drive status.
DRSTAS*	M5604	M5654	M5704	With CAN. They indicate servo drive status.
ANT*	M5606	M5656	M5706	If distance < MINMOVE (P54), ANT*=1
INPOS*	M5607	M5657	M5707	Axis in position.

SPINDLE LOGIC OUTPUTS.

Main		
ENABLES	M5950	Enables spindle movement.
DIRS	M5951	Spindle turning direction
REFPOINS	M5952	The spindle has been already referenced (homed).
DRSTAFS	M5953	With CAN. They indicate servo drive status.
DRSTASS	M5954	With CAN. They indicate servo drive status.
REVOK	M5956	Spindle rpm correspond to programmed speed.
INPOSS	M5957	Spindle in position.

AUXILIARY M, S, T FUNCTION TRANSFER.

MBCD1	R550	Auxiliary M function to be executed in the main channel.
MBCD2	R551	Auxiliary M function to be executed in the main channel.
MBCD3	R552	Auxiliary M function to be executed in the main channel.
MBCD4	R553	Auxiliary M function to be executed in the main channel.
MBCD5	R554	Auxiliary M function to be executed in the main channel.
MBCD6	R555	Auxiliary M function to be executed in the main channel.
MBCD7	R556	Auxiliary M function to be executed in the main channel.
MBCDP1	R565	Auxiliary M function to be executed in the PLC channel.
MBCDP2	R566	Auxiliary M function to be executed in the PLC channel.
MBCDP3	R567	Auxiliary M function to be executed in the PLC channel.
MBCDP4	R568	Auxiliary M function to be executed in the PLC channel.
MBCDP5	R569	Auxiliary M function to be executed in the PLC channel.
MBCDP6	R570	Auxiliary M function to be executed in the PLC channel.
MBCDP7	R571	Auxiliary M function to be executed in the PLC channel.
SBCD	R557	Spindle speed in BCD (2 or 8 digits).
TBCD	R558	Indicates the magazine position of the tool to be placed in the spindle.
T2BCD	R559	Magazine position (pocket) for the tool.

LOGIC OUTPUTS OF KEY STATUS.

KEYBD1	R560	Indicate whether a key of the operator panel is pressed.
KEYBD2	R561	
KEYBD3	R562	
KEYBD4	R563	

E.

APPENDIX

Summary of PLC inputs and outputs

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

2-DIGIT BCD CODE OUTPUT CONVERSION TABLE

Programmed S	S BCD
0	00
1	20
2	26
3	29
4	32
5	34
6	35
7	36
8	38
9	39
10-11	40
12	41
13	42
14-15	43
16-17	44
18-19	45
20-22	46
23-24	47
25-27	48
28-31	49
32-35	50
36-39	51
40-44	52
45-49	53

Programmed S	S BCD
50-55	54
56-62	55
63-70	56
71-79	57
80-89	58
90-99	59
100-111	60
112-124	61
125-139	62
140-159	63
160-179	64
180-199	65
200-223	66
224-249	67
250-279	68
280-314	69
315-354	70
355-399	71
400-449	72
450-499	73
500-559	74
560-629	75
630-709	76
710-799	77

Programmed S	S BCD
800-899	78
900-999	79
1000-1119	80
1120-1249	81
1250-1399	82
1400-1599	83
1600-1799	84
1800-1999	85
2000-2239	86
2240-2499	87
2500-2799	88
2800-3149	89
3150-3549	90
3550-3999	91
4000-4499	92
4500-4999	93
5000-5599	94
5600-6299	95
6300-7099	96
7100-7999	97
8000-8999	98
9000-9999	99

F.

APPENDIX

2-digit BCD code output conversion table

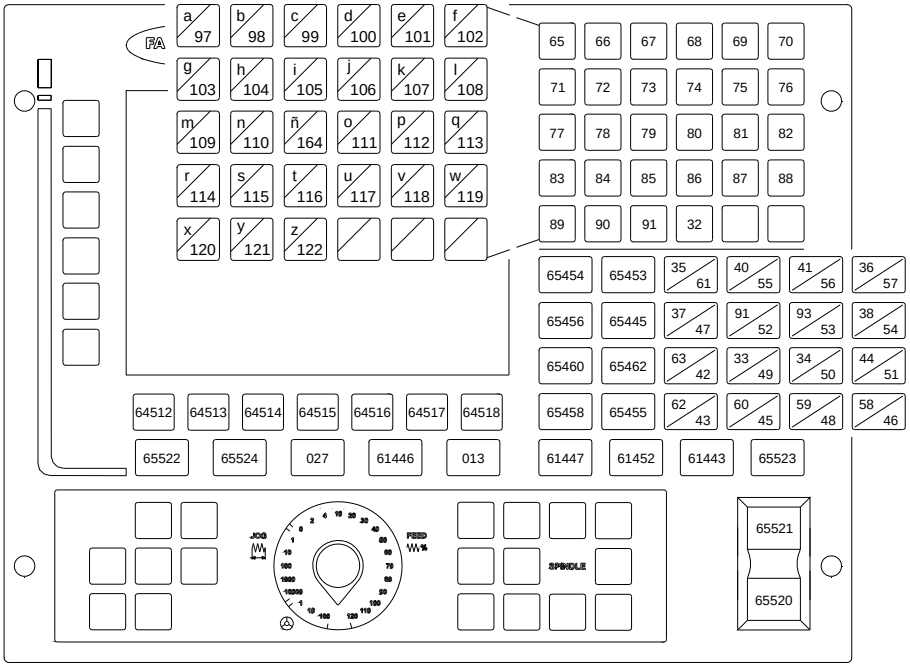
FAGOR

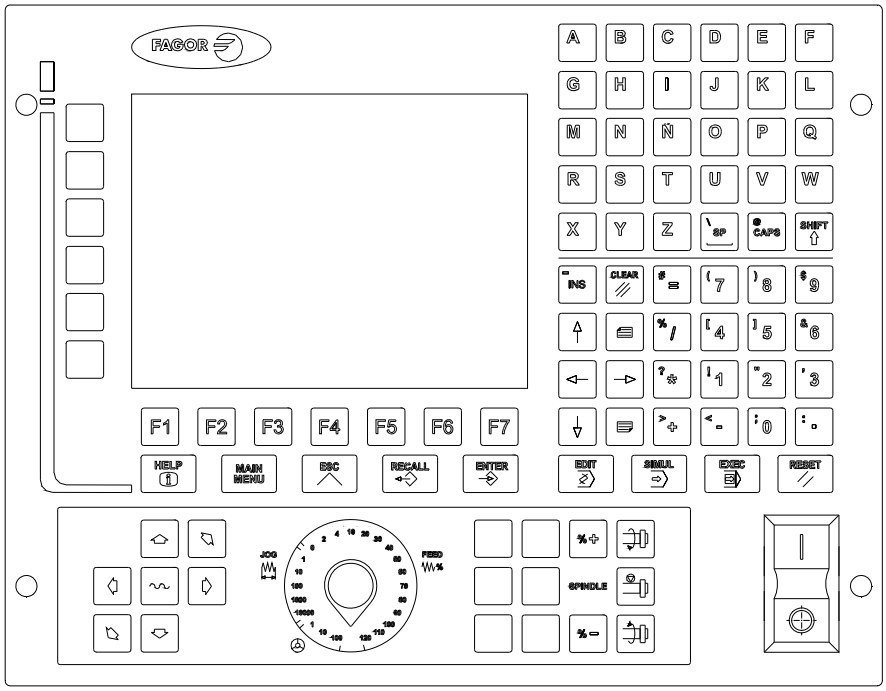
CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

KEY CODES

Alphanumeric operator panel (M-T models)





FAGOR

CNC 8035

(SOFT M: V15.3x)
(SOFT T: V16.3x)

G.

APPENDIX
Key codes

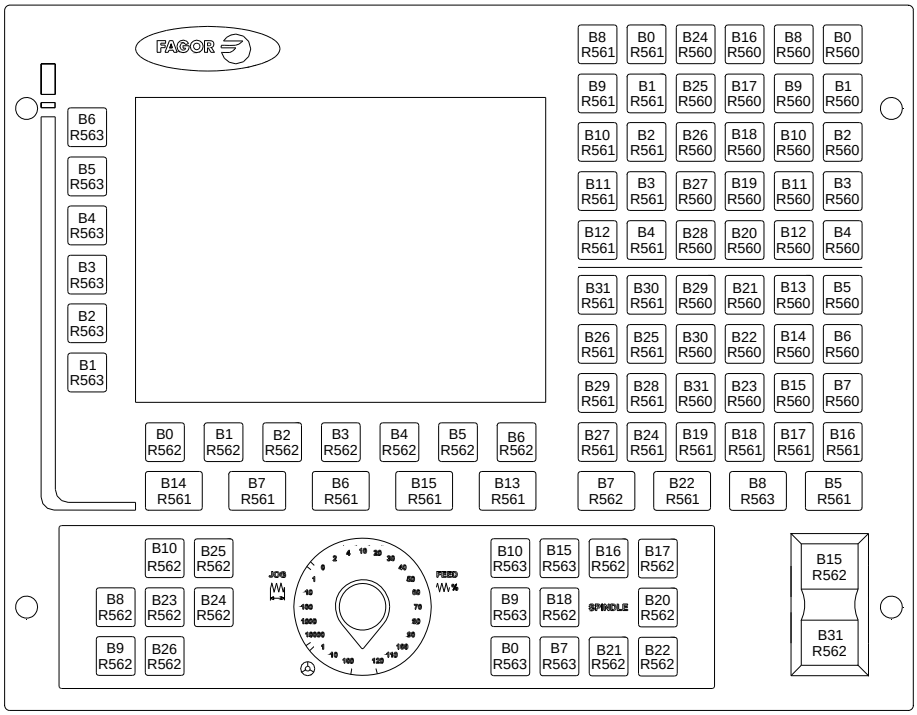
FAGOR

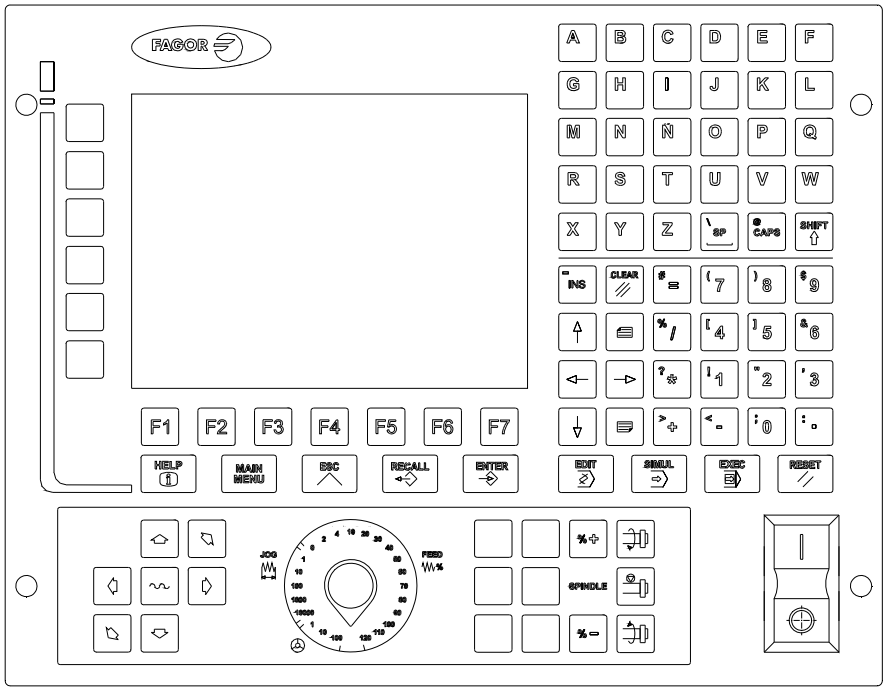
CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

LOGIC OUTPUTS OF KEY STATUS

Alphanumeric operator panel (M-T models)





FAGOR

CNC 8035

(SOFT M: V15.3x)
(SOFT T: V16.3x)

H.

APPENDIX

Logic outputs of key status

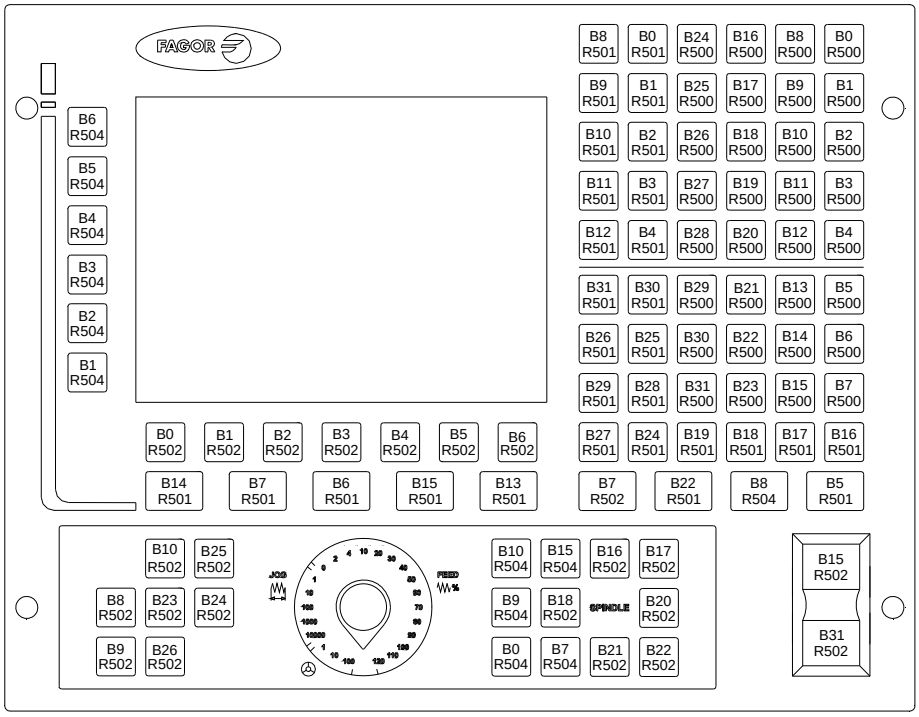
FAGOR

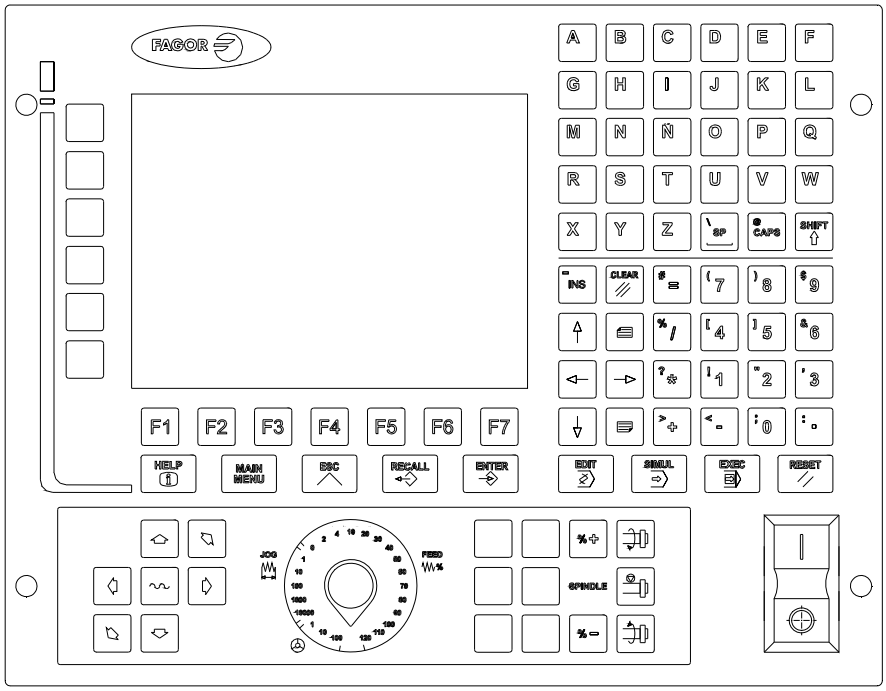
CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

KEY INHIBITING CODES

Alphanumeric operator panel (M-T models)





FAGOR

CNC 8035

(SOFT M: V15.3x)
(SOFT T: V16.3x)

I.

APPENDIX
Key inhibiting codes

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

MACHINE PARAMETER SETTING CHART

General machine parameters

P0		P50		P100		P150	
P1		P51		P101		P151	
P2		P52		P102		P152	
P3		P53		P103		P153	
P4		P54		P104		P154	
P5		P55		P105		P155	
P6		P56		P106		P156	
P7		P57		P107		P157	
P8		P58		P108		P158	
P9		P59		P109		P159	
P10		P60		P110		P160	
P11		P61		P111		P161	
P12		P62		P112		P162	
P13		P63		P113		P163	
P14		P64		P114		P164	
P15		P65		P115		P165	
P16		P66		P116		P166	
P17		P67		P117		P167	
P18		P68		P118		P168	
P19		P69		P119		P169	
P20		P70		P120		P170	
P21		P71		P121		P171	
P22		P72		P122		P172	
P23		P73		P123		P173	
P24		P74		P124		P174	
P25		P75		P125		P175	
P26		P76		P126		P176	
P27		P77		P127		P177	
P28		P78		P128		P178	
P29		P79		P129		P179	
P30		P80		P130		P180	
P31		P81		P131		P181	
P32		P82		P132		P182	
P33		P83		P133		P183	
P34		P84		P134		P184	
P35		P85		P135		P185	
P36		P86		P136		P186	
P37		P87		P137		P187	
P38		P88		P138		P188	
P39		P89		P139		P189	
P40		P90		P140		P190	
P41		P91		P141		P191	
P42		P92		P142		P192	
P43		P93		P143		P193	
P44		P94		P144		P194	
P45		P95		P145		P195	
P46		P96		P146		P196	
P47		P97		P147		P197	
P48		P98		P148		P198	
P49		P99		P149		P199	

_____ axis machine parameters

P0		P50		P100		P150	
P1		P51		P101		P151	
P2		P52		P102		P152	
P3		P53		P103		P153	
P4		P54		P104		P154	
P5		P55		P105		P155	
P6		P56		P106		P156	
P7		P57		P107		P157	
P8		P58		P108		P158	
P9		P59		P109		P159	
P10		P60		P110		P160	
P11		P61		P111		P161	
P12		P62		P112		P162	
P13		P63		P113		P163	
P14		P64		P114		P164	
P15		P65		P115		P165	
P16		P66		P116		P166	
P17		P67		P117		P167	
P18		P68		P118		P168	
P19		P69		P119		P169	
P20		P70		P120		P170	
P21		P71		P121		P171	
P22		P72		P122		P172	
P23		P73		P123		P173	
P24		P74		P124		P174	
P25		P75		P125		P175	
P26		P76		P126		P176	
P27		P77		P127		P177	
P28		P78		P128		P178	
P29		P79		P129		P179	
P30		P80		P130		P180	
P31		P81		P131		P181	
P32		P82		P132		P182	
P33		P83		P133		P183	
P34		P84		P134		P184	
P35		P85		P135		P185	
P36		P86		P136		P186	
P37		P87		P137		P187	
P38		P88		P138		P188	
P39		P89		P139		P189	
P40		P90		P140		P190	
P41		P91		P141		P191	
P42		P92		P142		P192	
P43		P93		P143		P193	
P44		P94		P144		P194	
P45		P95		P145		P195	
P46		P96		P146		P196	
P47		P97		P147		P197	
P48		P98		P148		P198	
P49		P99		P149		P199	

_____ axis machine parameters

P0		P50		P100		P150	
P1		P51		P101		P151	
P2		P52		P102		P152	
P3		P53		P103		P153	
P4		P54		P104		P154	
P5		P55		P105		P155	
P6		P56		P106		P156	
P7		P57		P107		P157	
P8		P58		P108		P158	
P9		P59		P109		P159	
P10		P60		P110		P160	
P11		P61		P111		P161	
P12		P62		P112		P162	
P13		P63		P113		P163	
P14		P64		P114		P164	
P15		P65		P115		P165	
P16		P66		P116		P166	
P17		P67		P117		P167	
P18		P68		P118		P168	
P19		P69		P119		P169	
P20		P70		P120		P170	
P21		P71		P121		P171	
P22		P72		P122		P172	
P23		P73		P123		P173	
P24		P74		P124		P174	
P25		P75		P125		P175	
P26		P76		P126		P176	
P27		P77		P127		P177	
P28		P78		P128		P178	
P29		P79		P129		P179	
P30		P80		P130		P180	
P31		P81		P131		P181	
P32		P82		P132		P182	
P33		P83		P133		P183	
P34		P84		P134		P184	
P35		P85		P135		P185	
P36		P86		P136		P186	
P37		P87		P137		P187	
P38		P88		P138		P188	
P39		P89		P139		P189	
P40		P90		P140		P190	
P41		P91		P141		P191	
P42		P92		P142		P192	
P43		P93		P143		P193	
P44		P94		P144		P194	
P45		P95		P145		P195	
P46		P96		P146		P196	
P47		P97		P147		P197	
P48		P98		P148		P198	
P49		P99		P149		P199	

_____ axis machine parameters

P0		P50		P100		P150	
P1		P51		P101		P151	
P2		P52		P102		P152	
P3		P53		P103		P153	
P4		P54		P104		P154	
P5		P55		P105		P155	
P6		P56		P106		P156	
P7		P57		P107		P157	
P8		P58		P108		P158	
P9		P59		P109		P159	
P10		P60		P110		P160	
P11		P61		P111		P161	
P12		P62		P112		P162	
P13		P63		P113		P163	
P14		P64		P114		P164	
P15		P65		P115		P165	
P16		P66		P116		P166	
P17		P67		P117		P167	
P18		P68		P118		P168	
P19		P69		P119		P169	
P20		P70		P120		P170	
P21		P71		P121		P171	
P22		P72		P122		P172	
P23		P73		P123		P173	
P24		P74		P124		P174	
P25		P75		P125		P175	
P26		P76		P126		P176	
P27		P77		P127		P177	
P28		P78		P128		P178	
P29		P79		P129		P179	
P30		P80		P130		P180	
P31		P81		P131		P181	
P32		P82		P132		P182	
P33		P83		P133		P183	
P34		P84		P134		P184	
P35		P85		P135		P185	
P36		P86		P136		P186	
P37		P87		P137		P187	
P38		P88		P138		P188	
P39		P89		P139		P189	
P40		P90		P140		P190	
P41		P91		P141		P191	
P42		P92		P142		P192	
P43		P93		P143		P193	
P44		P94		P144		P194	
P45		P95		P145		P195	
P46		P96		P146		P196	
P47		P97		P147		P197	
P48		P98		P148		P198	
P49		P99		P149		P199	

Machine parameters for the main spindle

P0		P50		P100		P150	
P1		P51		P101		P151	
P2		P52		P102		P152	
P3		P53		P103		P153	
P4		P54		P104		P154	
P5		P55		P105		P155	
P6		P56		P106		P156	
P7		P57		P107		P157	
P8		P58		P108		P158	
P9		P59		P109		P159	
P10		P60		P110		P160	
P11		P61		P111		P161	
P12		P62		P112		P162	
P13		P63		P113		P163	
P14		P64		P114		P164	
P15		P65		P115		P165	
P16		P66		P116		P166	
P17		P67		P117		P167	
P18		P68		P118		P168	
P19		P69		P119		P169	
P20		P70		P120		P170	
P21		P71		P121		P171	
P22		P72		P122		P172	
P23		P73		P123		P173	
P24		P74		P124		P174	
P25		P75		P125		P175	
P26		P76		P126		P176	
P27		P77		P127		P177	
P28		P78		P128		P178	
P29		P79		P129		P179	
P30		P80		P130		P180	
P31		P81		P131		P181	
P32		P82		P132		P182	
P33		P83		P133		P183	
P34		P84		P134		P184	
P35		P85		P135		P185	
P36		P86		P136		P186	
P37		P87		P137		P187	
P38		P88		P138		P188	
P39		P89		P139		P189	
P40		P90		P140		P190	
P41		P91		P141		P191	
P42		P92		P142		P192	
P43		P93		P143		P193	
P44		P94		P144		P194	
P45		P95		P145		P195	
P46		P96		P146		P196	
P47		P97		P147		P197	
P48		P98		P148		P198	
P49		P99		P149		P199	

Machine parameters for serial line 1

P0 P1		P8 P9		P16 P17		P24 P25	
P2 P3		P10 P11		P18 P19		P26 P27	
P4 P5		P12 P13		P20 P21		P28 P29	
P6 P7		P14 P15		P22 P23		P30 P31	

PLC machine parameters

P0 P1		P22 P23		P44 P45		P66 P67	
P2 P3		P24 P25		P46 P47		P68 P69	
P4 P5		P26 P27		P48 P49		P70 P71	
P6 P7		P28 P29		P50 P51		P72 P73	
P8 P9		P30 P31		P52 P53		P74 P75	
P10 P11		P32 P33		P54 P55		P76 P77	
P12 P13		P34 P35		P56 P57		P78 P79	
P14 P15		P36 P37		P58 P59		P80 P81	
P16 P17		P38 P39		P60 P61		P82 P83	
P18 P19		P40 P41		P62 P63		P84 P85	
P20 P21		P42 P43		P64 P65		P86 P87	

M FUNCTIONS SETTING CHART

M function	Associated subroutine	Setting bits 7	6	5	4	3	2	1	0
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								

M function	Associated subroutine	Setting bits 7	6	5	4	3	2	1	0
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								

K.

APPENDIX

M functions setting chart

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

LEADSCREW ERROR COMPENSATION TABLE

Axis _____			
Point	Position	Error	Error (-)
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E

Axis _____			
Point	Position	Error	Error (-)
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E

Axis _____			
Point	Position	Error	Error (-)
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E
P P		E E	E E

L.

APPENDIX

Leadscrew error compensation table

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

CROSS COMPENSATION TABLE

Moving axis _____		MOVAXIS (P32)
Point	Position	Error
P P		E E
P P		E E
P P		E E
P P		E E
P P		E E
P P		E E
P P		E E
P P		E E
P P		E E

Axis to be compensated _____		COMPAXIS (P33)
Point	Position	Error
P P		E E
P P		E E
P P		E E
P P		E E
P P		E E
P P		E E
P P		E E
P P		E E
P P		E E

M.

APPENDIX

Cross compensation table

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)

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 CNC and, consequently, possible malfunctions.

On the other hand, accumulated dirt can sometimes act as an electrical conductor and shortcircuit 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 75º alcohol.

Do not use highly compressed air to clean the unit because it could generate electrostatic discharges.

The plastics used on the front panel are resistant to :

- Grease and mineral oils.
- Bases and bleach.
- Dissolved detergents.
- Alcohol.



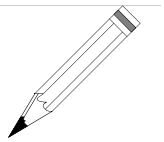
Fagor Automation shall not be held responsible for any material or physical damage derived from the violation of these basic safety requirements.

To check the fuses, first unplug the unit from mains If the CNC does not turn on when flipping the power switch, check that the fuses are the right ones and they are in good condition.

Avoid solvents. The action of solvents such as chlorine hydrocarbons, benzole, esters and ether may damage the plastics used to make the front panel of the unit.

Do not get into the inside of the unit. Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not handle the connectors with the unit connected to main AC power. Before handling these connectors (I/O, feedback, etc.), make sure that the unit is not connected to main AC power.



N.

APPENDIX

FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)



N.

FAGOR

CNC 8035

(SOFT M: V15.3x)
(SOFT T: V16.3x)

N.



FAGOR

CNC 8035

(SOFT M: V15.3X)
(SOFT T: V16.3X)