CNC 8055

Installation manual

Ref.1501
Soft: V01.6x

FAGOR

FAGOR AUTOMATION

DUAL-USE PRODUCTS

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

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

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

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

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

The content of this manual and its validity for the product described here has been verified. Even so, involuntary errors are possible, hence no absolute match is guaranteed. However, the contents of this document are regularly checked and updated implementing the necessary corrections in a later edition. We appreciate your suggestions for improvement.

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

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 13
Safety conditions 15
Warranty terms 19
Material returning terms 21
Additional remarks 23
Fagor documentation 25

CHAPTER 1 8055 CNC CONFIGURATION

1.1 CNC structure 27
1.2 Central unit 29
1.2.1 –CPU– Module 33
1.2.2 –Vpp Axes– module 43
1.2.3 –Vpp SB Axes– module 50
1.2.4 –I/O– module (inputs - outputs) 57
1.3 Monitors 62
1.3.1 11" LCD Monitor 63
1.3.2 11" LCD monitor + M, T, MC or TC keyboard 65
1.4 Operator panel 67
1.4.1 MC, TC, MCO/TCO and alphanumeric operator panels 68
1.4.2 Alphanumeric keyboard (optional) 69

CHAPTER 2 CNC 8055I CONFIGURATION

2.1 CNC structure 73
2.1.1 Connectors 77
2.2 Operator panel 102
2.2.1 Alphanumeric keyboard (optional) 104

CHAPTER 3 HEAT DISSIPATION

3.1 Heat dissipation by natural convection 108
3.2 Heat dissipation by forced convection with inside fan 109
3.3 Heat dissipation by air flow to the outside using a fan 110

CHAPTER 4 REMOTE MODULES (BUS CAN WITH CANOPEN PROTOCOL)

4.1 Installation of the modules 113
4.2 Power supply 114
4.3 Digital inputs and digital outputs (single module) 120
4.4 Digital inputs and digital outputs (double module) 122
4.5 Electrical characteristics of the inputs and outputs 124
4.6 Numbering of the digital inputs and outputs 126

CHAPTER 5 MACHINE AND POWER CONNECTION

5.1 Digital inputs and outputs 131
5.2 Analog inputs and outputs 132
5.3 Setup 133
5.4 Connection of the emergency input and output 137

CHAPTER 6 MACHINE PARAMETERS

6.1 Parameter matching between the CNC and the Sercos drive 145
6.2 Parameters that may be modified from the oscilloscope, OEM program or OEM subroutine 146
6.3 General machine parameters 148
6.4 Axis parameters 206
6.5 Spindle parameters 239
6.5.1 Machine parameters for the main and 2nd spindles 240
6.5.2 Machine parameters for auxiliary spindle 259
6.6 Drive parameters 261
6.6.1 Friction compensation 264

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6X

6.7 Serial line parameters 265
6.8 Ethernet parameters 268
6.9 PLC Parameters 272
6.10 Tables 280
6.10.1 Miscellaneous (M) function table 280
6.10.2 Leadscrew error compensation table 282
6.10.3 Cross compensation parameter table 284

CHAPTER 7 CONCEPTS

7.1 Axes and coordinate systems 285
7.1.1 Rotary axes 288
7.1.2 Gantry axes 290
7.1.3 Slaved axes and synchronized axes 291
7.1.4 Incline axis 292
7.1.5 Tandem axes 294
7.2 Jog 300
7.2.1 Relationship between the axes and the JOG keys 300
7.2.2 Path-jog mode 301
7.3 Movement with an electronic handwheel 303
7.3.1 Standard handwheel 304
7.3.2 Path handwheel 305
7.3.3 Feed handwheel mode 306
7.3.4 "Additive handwheel" mode 307
7.4 feedback system 309
7.4.1 Counting speed limitation 310
7.4.2 Resolution 311
7.5 Axis adjustment 315
7.5.1 Drive adjustment 316
7.5.2 Gain setting 317
7.5.3 Proportional gain setting 318
7.5.4 Feed-forward gain setting 319
7.5.5 Derivative (AC-forward) gain setting 320
7.5.6 Leadscrew backlash compensation. 321
7.5.7 Leadscrew error compensation 322
7.5.8 Circle geometry test 324
7.6 Reference systems 326
7.6.1 Home search 327
7.6.2 Setting on systems without distance-coded feedback 331
7.6.3 Setting on systems with distance-coded feedback 333
7.6.4 Axis travel limits (software limits) 334
7.7 Unidirectional approach 335
7.8 Auxiliary M, S, T function transfer 336
7.8.1 Transferring M, S, T using the AUXEND signal 338
7.8.2 Transferring the auxiliary (miscellaneous) M functions without the AUXEND signal . 339
7.9 Main and second spindle 340
7.9.1 Spindle types 342
7.9.2 Spindle speed (S) control 343
7.9.3 Spindle gear change 345
7.9.4 Spindle in closed loop 347
7.10 Auxiliary spindle controlled by PLC 353
7.11 Treatment of emergency signals 354
7.12 Digital servo (CAN or Sercos) 357
7.12.1 Communications channel 358
7.12.2 Drive's absolute feedback 361
7.13 Axes (2) controlled by a single drive 362
7.13.1 "C" axis and spindle with a single feedback 365
7.14 Additive coupling between axes 368
7.15 Fagor handwheels: HBA, HBE and LGB 370
7.16 Machine safety related functions 374
7.16.1 Maximum machining spindle speed 374
7.16.2 Cycle start disabled when hardware errors occur. 375
7.17 Configuring a CNC as two and a half axes 376
7.17.1 Machine parameter setting 377
7.17.2 PLC program 378
7.18 Tool magazine 380
7.18.1 Tool change via PLC 380
7.18.2 Tool magazine management 381
7.19 Gear ratio management on axes and spindle 382
7.19.1 Axis example: Encoder at the motor 383
7.19.2 Axis example: external feedback device without a gear box 384
7.19.3 Axis example: external feedback device with gear box 388
7.19.4 Spindle example: Encoder at the motor 391
7.19.5 Spindle example: external encoder without gear box 393
7.19.6 Spindle example: external encoder with gear box 396

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6X

7.20 Feedback combination for SERCOS axes with external feedback to the CNC 399
7.21 Open communication 400
7.21.1 Reception setting 402
7.21.2 Incompatibility with WINDNC via RS232 402
7.21.3 Trace of characters sent and received 403
7.21.4 Examples of open communication 404
7.22 Auto-adjustment of axis machine parameter DERGAIN 408

CHAPTER 8 INTRODUCTION TO THE PLC

8.1 PLC resources 412
8.2 PLC program execution 413
8.3 Loop time 416
8.4 Modular structure of the program 417
8.4.1 First cycle module (CY1) 417
8.4.2 Main module (PRG) 417
8.4.3 Periodic execution module (PE t) 418
8.4.4 Priority of execution of the PLC modules 419

CHAPTER 9 PLC RESOURCES

9.1 Inputs 421
9.2 Outputs 422
9.3 Marks 423
9.4 Registers 425
9.5 Timers 426
9.5.1 Monostable mode. TG1 input 429
9.5.2 Delayed activation mode. TG2 input 431
9.5.3 Delayed deactivation mode. TG3 input 433
9.5.4 Signal limiting mode. TG4 Input 435
9.6 Counters 437
9.6.1 Operating mode of a counter 440

CHAPTER 10 PLC PROGRAMMING

10.1 Module structure 442
10.2 Directing instructions 443
10.3 Consulting instructions 446
10.4 Operators and symbols 448
10.5 Action instruction 449
10.5.1 Binary assignment instructions 450
10.5.2 Conditional binary action instructions 451
10.5.3 Sequence breaking action instructions 452
10.5.4 Arithmetic action instructions 453
10.5.5 Logic action instructions 455
10.5.6 Specific action instructions 457

CHAPTER 11 CNC-PLC COMMUNICATION

11.1 Auxiliary M, S, T functions 460
11.2 Auxiliary M, S, T function transfer 463
11.2.1 Transferring M, S, T using the AUXEND signal 464
11.2.2 Transferring the auxiliary (miscellaneous) M functions without the AUXEND signal 465
11.3 Displaying messages, errors and screens 466
11.4 Access to the PLC from the CNC 468
11.5 Access to the PLC from a PC, via DNC 469

CHAPTER 12 LOGIC CNC INPUTS AND OUTPUTS

12.1 General logic inputs 472
12.2 Axis logic inputs. 481
12.3 Spindle logic inputs. 486
12.4 Logic inputs of the auxiliary spindle 491
12.5 Key inhibiting logic inputs. 492
12.6 Logic inputs of the PLC channel 493
12.7 General logic outputs 495
12.8 Logic outputs of the axes 503
12.9 Spindle logic outputs. 505
12.10 Logic outputs of the auxiliary spindle 507
12.11 Logic outputs of key status 508

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6X

CHAPTER 13 ACCESS TO INTERNAL CNC VARIABLES

13.1 Variables associated with tools. 511
13.2 Variables associated with zero offsets. 515
13.3 Variables associated with function G49 516
13.4 Variables associated with machine parameters 518
13.5 Variables associated with work zones 519
13.6 Variables associated with feedrates 521
13.7 Variables associated with coordinates 524
13.8 Variables associated with electronic handwheels 526
13.9 Variables associated with feedback 528
13.10 Variables associated with the main spindle 529
13.11 Variables associated with the second spindle 532
13.12 Variables associated with the live tool 535
13.13 Variables associated with local and global parameters 536
13.14 Sercos variables 537
13.15 Software & hardware configuration variables 538
13.16 Variables associated with telediagnosis 541
13.17 Operating-mode related variables 543
13.18 Other variables 547

CHAPTER 14 AXES CONTROLLED FROM THE PLC

14.1 PLC execution channel 558
14.1.1 Considerations 559
14.1.2 Blocks which can be executed from the PLC 561
14.1.3 Control of the PLC program from the CNC 565
14.2 Action CNCEX1 567
14.3 Synchronize a PLC axis with a CNC axis 568

CHAPTER 15 CUSTOMIZABLE SCREENS

15.1 Configuration file 570
15.2 Configuration language 572
15.3 Key words 573
15.4 Arithmetic operations 577
15.5 Conditional instructions. 579
15.6 Example of a configuration file 580
15.7 Error log file (P999500) 582

CHAPTER 16 CONFIGURABLE WORK MODE

16.1 Axis control 585
16.2 Tool control 586
16.3 Spindle control 587
16.4 MDI 588
16.5 Screens, subroutines and cycles 589
16.6 Associated keys 590
16.7 OEM text in several languages. 592
16.8 Associated programs 595
16.9 Associated subroutines 596
16.10 Configuration file 597
16.11 Error log file (P999500) 601
16.12 Cycle data entry 602
16.13 Example. Consult inputs and outputs 604
16.14 Example. Machining canned cycle 605

CHAPTER 17 PLC PROGRAMMING EXAMPLE

17.1 Definition of symbols (mnemonics) 608
17.2 First cycle module. 610
17.3 Main module. 611

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

APPENDIX

A	Technical characteristics of the 8055iCNC	621
B	Central unit of the 8055 CNC	625
C	11" LCD Monitor	629
D	Probe connection at the 8055i	631
E	Probe connection at the 8055 CNC	633
F	Summary of internal CNC variables	635
G	Summary of PLC commands	643
H	Summary of PLC inputs and outputs	647
I	2-digit BCD code output conversion table	653
J	Key code	655
K	Logic outputs of key status	665
L	Key inhibiting codes	675
M	Machine parameter setting chart	685
N	M functions setting chart	697
O	Leadscrew error compensation table	699
P	Cross compensation table	701
Q	Maintenance	703

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6X

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6X

ABOUT THE PRODUCT

BASIC CHARACTERISTICS OF THE DIFFERENT MODELS

	8055i FL EN	8055 FL 8055i FL	8055 Power 8055i Power
Built-in	8055i FL EN	8055i FL	8055i Power
Enclosure	-----	8055 FL	8055 Power
USB	Standard	Standard	Standard
Block processing time	1 ms	3.5 ms	1 ms
RAM memory	1Mb	1Mb	1 Mb
Software for 7 axes	-----	-----	Option
TCP transformation	-----	-----	Option
C axis (Lathe)	-----	-----	Option
Y axis (Lathe)	-----	-----	Option
Look-ahead	100 blocks	100 blocks	200 blocks
Flash Memory 512Mb / 2Gb	512Mb	Option	Option

HARDWARE OPTIONS OF THE 8055I CNC

	Analog	Digital	Engraving
Ethernet	Option	Option	Option
RS-232 serial line	Standard	Standard	Standard
16 digital inputs and 8 outputs (I1 to I16 and O1 to O8)	Standard	Standard	Standard
Another 40 digital inputs and 24 outputs (I65 to I104 and O33 to O56)	Option	Option	Option
Probe inputs	Standard	Standard	Standard
Spindle (feedback input and analog output)	Standard	Standard	Standard
Electronic handwheels	Standard	Standard	Standard
4 axes (feedback and velocity command)	Option	Option	- - -
Remote CAN modules, for digital I/O expansion (RIO).	Option	Option	- - -
Sercos servo drive system for Fagor servo drive connection.	- - -	Option	- - -
CAN servo drive system for Fagor servo drive connection.	- - -	Option	- - -

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

SOFTWARE OPTIONS OF THE 8055 AND 8055I CNCS.

	Model							
	GP	**M**	**MC**	**MCO**	**EN**	**T**	**TC**	**TCO**
Number of axes with standard software	4	4	4	4	3	2	2	2
Number of axes with optional software	7	7	7	7	-----	4 or 7	4 or 7	4 or 7
Electronic threading	-----	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.
Tool magazine management:	-----	Stand.	Stand.	Stand.	-----	Stand.	Stand.	Stand.
Machining canned cycles	-----	Stand.	Stand.	-----	Stand.	Stand.	Stand.	-----
Multiple machining	-----	Stand.	Stand.	-----	Stand.	-----	-----	-----
Solid graphics	-----	Stand.	Stand.	Stand.	-----	Stand.	Stand.	Stand.
Rigid tapping	-----	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.
Tool live monitoring	-----	Opt.	Opt.	Opt.	Stand.	Opt.	Opt.	Opt.
Probing canned cycles	-----	Opt.	Opt.	Opt.	Stand.	Opt.	Opt.	Opt.
DNC	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.
COCOM version	Opt.	Opt.	Opt.	Opt.	-----	Opt.	Opt.	Opt.
Profile editor	Stand.	Stand.	Stand.	Stand.	-----	Stand.	Stand.	Stand.
Tool radius compensation	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.
Tangential control	Opt.	Opt.	Opt.	Opt.	-----	Opt.	Opt.	Opt.
Retracing	-----	Opt.	Opt.	Opt.	Stand.	Opt.	Opt.	Opt.
Setup assistance	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.	Stand.
Irregular pockets with islands	-----	Stand.	Stand.	Stand.	-----	-----	-----	-----
TCP transformation	-----	Opt.	Opt.	Opt.	-----	-----	-----	-----
C axis (on Lathe)	-----	-----	-----	-----	-----	Opt.	Opt.	Opt.
Y axis (on Lathe)	-----	-----	-----	-----	-----	Opt.	Opt.	Opt.
Telediagnosis	Opt.	Opt.	Opt.	Opt.	Stand.	Opt.	Opt.	Opt.

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:

8055 / 8055i CNC

Consisting of the following modules and accessories:

MONITOR-8055, MONITOR-55-11-USB
OP-8055
KS 50/55, KB-40/55-ALFA, DVD AMPLI 8055
PSB-8055
CPU-KEY CF 8055 FL LARGE, CPU-KEY CF 8055 Power LARGE
AXES 8055 VPP
I/O 8055, COVER 8055, SERCOS 8055
Remote modules RIO
CNC 8055i FL, CNC 8055i Power
ANALOG 8055i-B, 40I/24O-8055i-B, ANALOG+40I/24O-B, COVER ANA+I/O-8055i-B
ETHERNET-CAN-SERCOS, ETHERNET-CAN-CAN AXES, ETHERNET-CAN AXES

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. Requirements and equipment 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

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
OPT-MC	Operating manual for the MC option.
OPT-TC	Operating manual for the TC option.
OPT-CO	Manual of the CO manual

Software V01.00 **October 2010**

First version.

Software V01.20 **April 2011**

List of features	Manual
Open communication.	INST
Improvements to Look Ahead machining.	INST
Blocks with helical interpolation in G51.	PRG
G84. Tapping with relief.	PRG

Software V01.08 **August 2011**

List of features	Manual
S.m.p. OPLDECTI (P86).	INST

Software V01.30 **September 2011**

List of features	Manual
Gear ratio management on Sercos spindles	INST
Improved feedrate limit management (FLIMIT).	INST
New type of penetration in lathe type threading cycles.	PRG
Improved lathe type thread repair. Partial repair.	PRG
MC option: Rigid tapping with relief.	OPT-MC
TC option: New type of penetration in threading cycles.	OPT-TC
TC option: Improved thread repair. Partial and multi-entry (start) thread repair.	OPT-TC
TC option: Zig-zag entry to the groove at the starting point of the groove.	OPT-TC

Software V01.31 **October 2011**

List of features	Manual
CNC 8055 FL Engraving model	INST / OPT/ PRG

Software V01.40 **January 2012**

List of features	Manual
Execution of M3, M4 and M5 using PLC marks	INST / PRG
Values 12 and 43 of variable OPMODE in conversational work mode.	INST / PRG

Software V01.60 **December 2013**

List of features	Manual
Auto-adjustment of axis machine parameter DERGAIN	INST
New value for axis machine parameter ACFGAIN (P46)	INST
Value 120 of the OPMODE variable.	INST / PRG

Software V01.65 **January 2015**

List of features	Manual
Block processing time of 1 ms on the "CNC 8055i FL Engraving" model	INST / OPT/ PRG

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.

- 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 y C=0,2 µF / 600 V.
  - AC motors. RC connected between phases, with values of R=300 Ω / 6 W y 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)

- Monitor enclosure (CNC 8055) or central unit (CNC 8055i)

  Guarantee the required gaps between the monitor or 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 (8055)

- "Axes" and "Inputs-Outputs" modules.

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

  They are protected by an external fast fuse (F) of 3.15 A 250V against overvoltage of the external power supply (over 33 Vdc) and against reverse connection of the power supply.

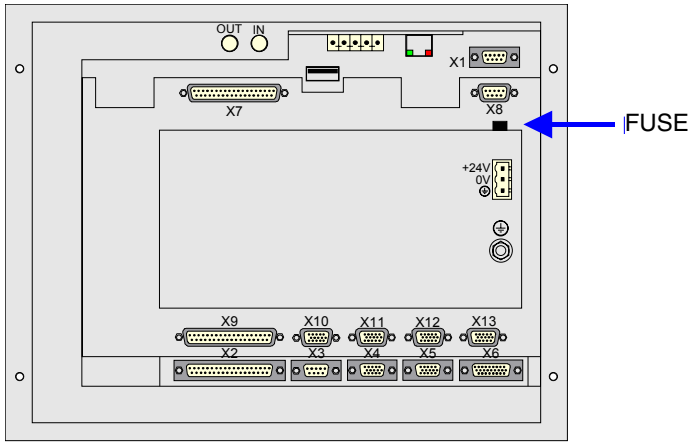
- Monitor.

  The type of protection fuse depends on the type of monitor. See identification label of the unit itself.

PROTECTIONS OF THE UNIT ITSELF (8055I)

- 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 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 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.



Symbol for danger or prohibition.
It indicates actions or operations that may cause damage to people or to units.



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 catalog.

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 authorized 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

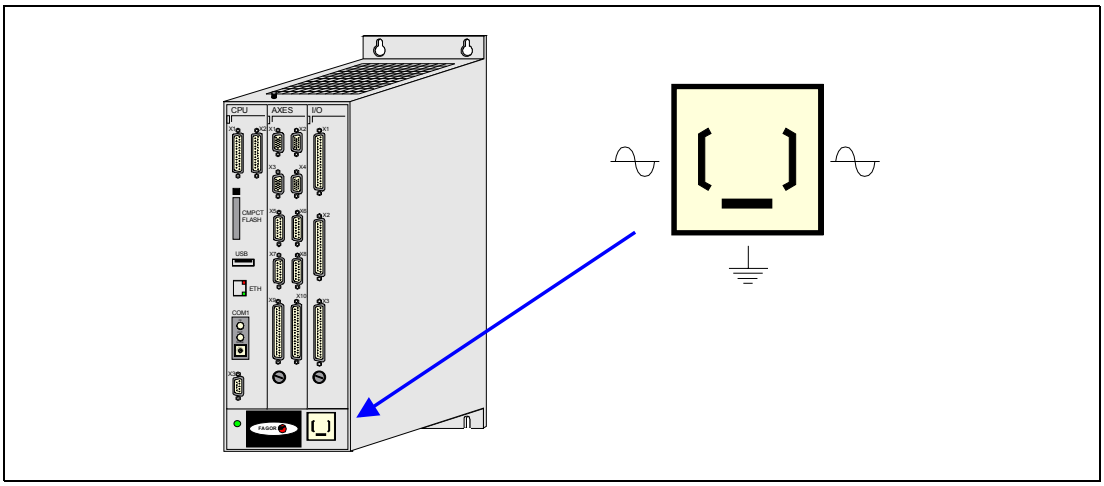
When sending the central nit or the remote modules, pack them in its original package and packaging material. If the original packaging material is not available, pack it as follows:

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.

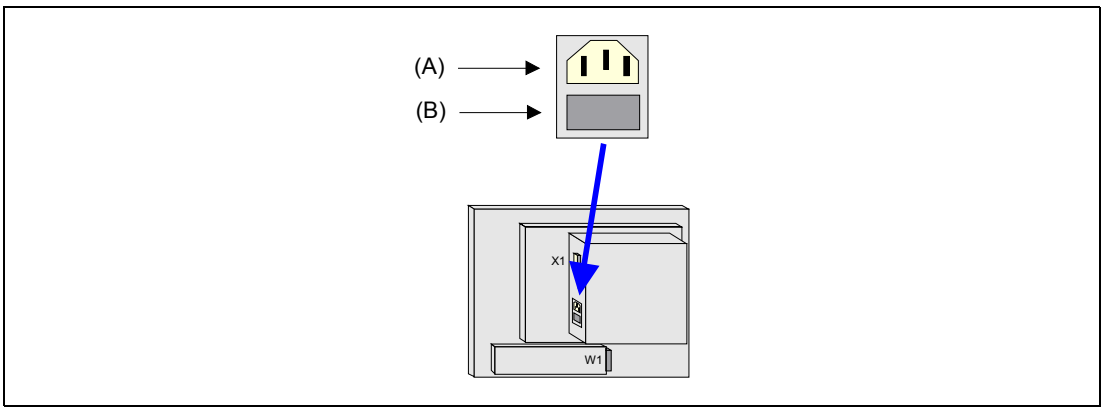
ADDITIONAL REMARKS

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.

To prevent electrical shock at the central unit of the 8055 CNC, use the proper mains AC connector at the power supply module. Use 3-wire power cables (one for ground connection).



To prevent electrical shock at the monitor of the 8055 CNC, use the proper mains AC connector (A) with 3-wire power cables (one of them for ground connection).



Before turning on the monitor of the 8055 CNC and verifying that the external AC line (B) fuse of each unit is the right one. See identification label of the unit itself.

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.

MC Manual

Directed to the end user.

It describes how to operate and program in MC mode.

It contains a self-teaching manual.

TC Manual

Directed to the end user.

It describes how to operate and program in TC mode.

It contains a self-teaching manual.

MCO/TCO model

Directed to the end user.

It describes how to operate and program in MCO and TCO mode.

Examples-M manual

Directed to the end user.

It contains programming examples for the M mode.

Examples-T manual

Directed to the end user.

It contains programming examples for the T mode.

WINDNC Manual

It is directed to people using the optional DNC communications software.

It is supplied in a floppy disk with the application.

WINDRAW55 Manual

Directed to people who use the WINDRAW55 to create screens.

It is supplied in a floppy disk with the application.

FAGOR

CNC 8055
CNC 8055i

8055 CNC CONFIGURATION

1

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

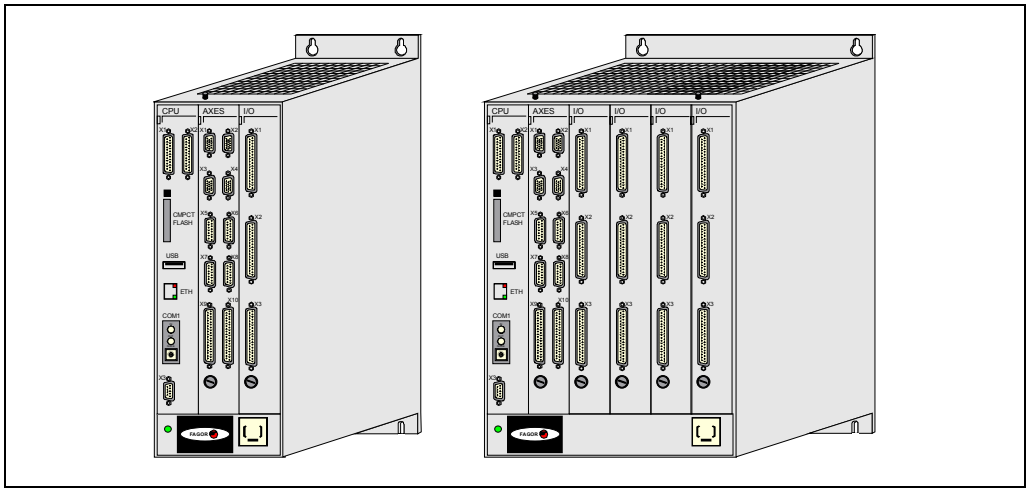
The CNC can control machine movements and devices.

1.1 CNC structure

The CNC consists of the following elements:

- Central Unit.
- Monitor.
- Keyboard.

The central unit (CPU) has a modular structure. There are 2 models: for 3 and 6 modules.

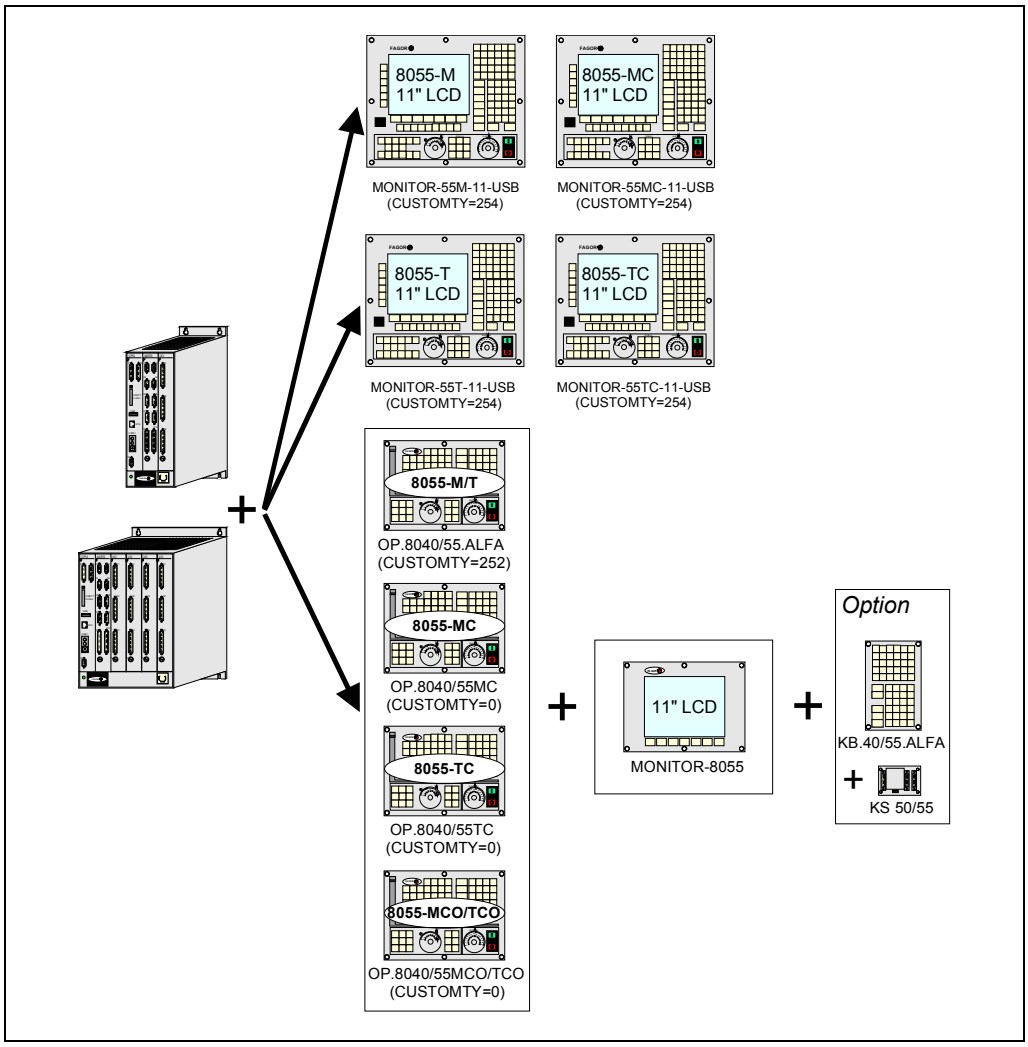


Either separate keyboard and monitor or keyboards with an incorporated monitor are available.

The monitors are 11" LCD.

The keyboards are specific for each model and work mode.

The following illustration shows the possible combinations. Each configuration shows the value of machine parameter CUSTOMTY (P92).



Use the video signal cable (up to 40 m) to connect the monitor with the central unit and the keyboard signal cable (up to 25 m) to connect the keyboard with the central unit.

Keyboard auto-identification

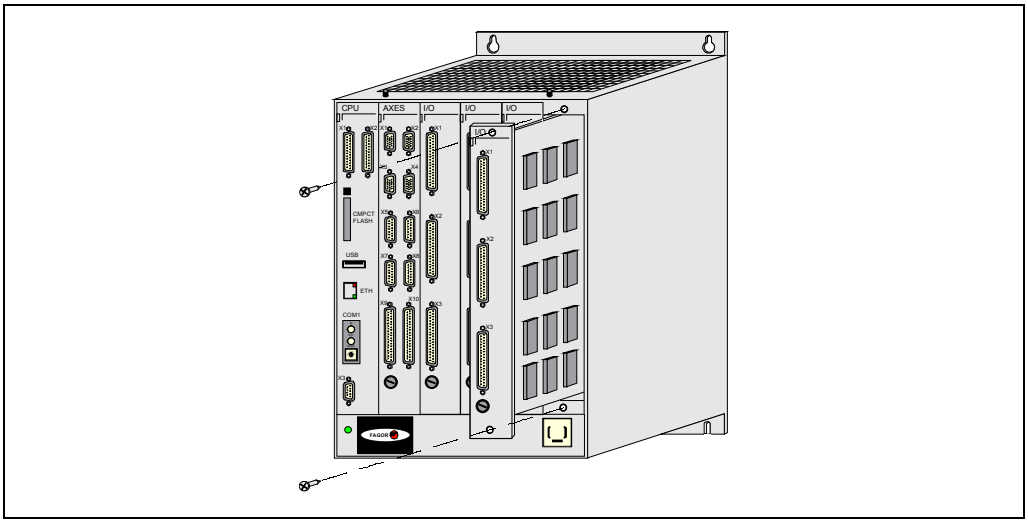
Certain keyboard models have a auto-identification system. With this type of keyboards, parameter CUSTOMTY is updated automatically; for the rest of the keyboards, this parameter must be set manually.

If the keyboard does not match the CNC model, it shows the relevant error message and it loads the key codes that correspond to the CNC model. For example, if a mill model keyboard is connected to a lathe CNC, the keyboard is set for a lathe and the CNC shows the error message.

1.2 Central unit

The central unit is usually located in the electrical cabinet, is modular and it comes in two models: There are 2 models: for 3 and 6 modules.

The modules are mounted using the screws located at their top and bottom.



Available modules

CPU

It contains the system software and carries out the CNC functions (editing, execution, simulation, display, etc.), process the information of the rest of the modules and generate the video signals for the monitor.

Optionally, it communicates with the drives via Sercos interface.

It must be part of all the configurations and mounted as the first module from the left.

Axes:

Besides controlling the spindle and the axes of the machine, it governs the first 40 digital PLC inputs and 24 digital PLC outputs.

It must be present in all configurations. Together with the –CPU– module makes up the basic system configuration.

I/Os

It is optional. It offers another 64 digital PLC inputs and 32 digital PLC outputs.

Central Unit (CPU) configuration

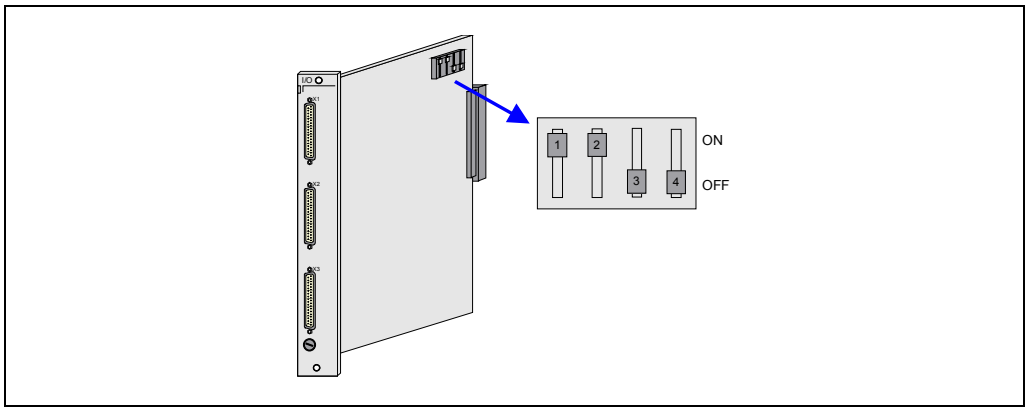
The configuration of the central unit depends on each application. The –CPU– and –AXES– modules must be part of all configurations.

The –CPU– module must be the first one from the left. The rest of the modules do not have to follow a particular order and may be interchanged according to personal preferences and connection possibilities of the machine.

The CNC has a PLUG&PLAY system that recognizes the configuration of the central unit. To do that, regardless of their physical location, each module has a logic address or device select code which identifies it within the internal configuration of the CNC. The logic address (device select code) for each module is factory set as follows:

–Axes– module	Logic address 2.
–I/Os– (1) module	Logic address 3.
–I/Os– (2) module	Logic address 4.
–I/Os– (3) module	Logic address 5.

Nevertheless, except in the axes module, these logic addresses may be modified at will. To do that, use the microswitches located in one of the corners of the printed circuit board.



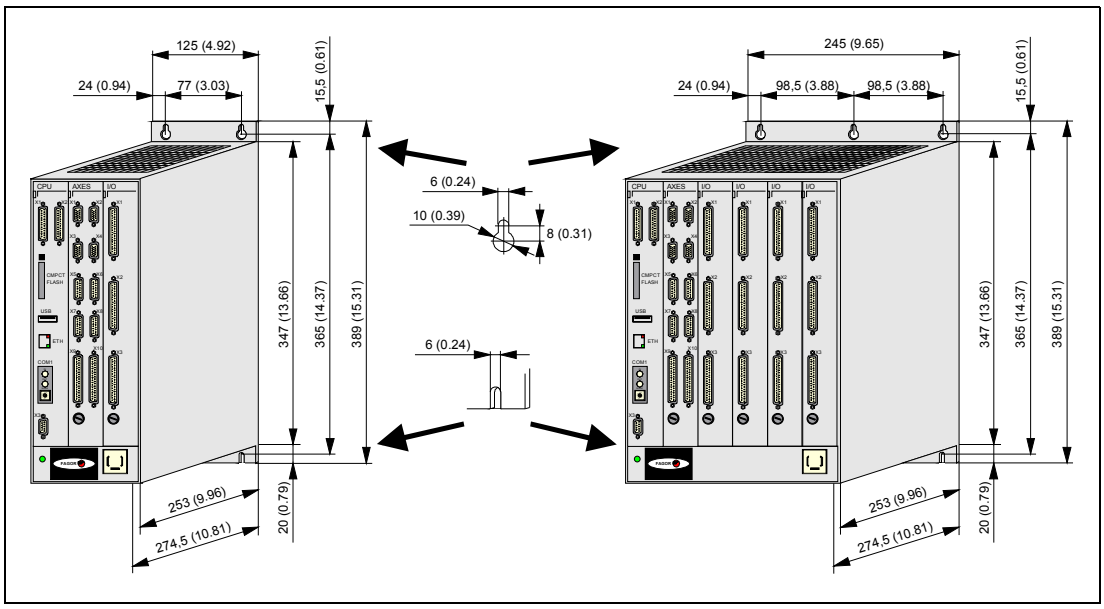
The logic address is set in binary code between 1 and 14. Logic address "0" and "15" are reserved. Logic addresses 0 and 15 are reserved.

Logic address	Micro switch position				Logic address	Micro switch position			
	1	2	3	4		1	2	3	4
0	off	off	off	off	8	on	off	off	off
1	off	off	off	on	9	on	off	off	on
2	off	off	on	off	10	on	off	on	off
3	off	off	on	on	11	on	off	on	on
4	off	on	off	off	12	on	on	off	off
5	off	on	off	on	13	on	on	off	on
6	off	on	on	off	14	on	on	on	off
7	off	on	on	on	15	on	on	on	on

When using several –I/O– modules, the CNC assumes the one with the lowest address as the first expansion module, as –I/O– (2) module the next address and as –I/O– (3) the one with the highest address number.

Dimensions and installation

The central unit is supplied with the requested configuration and its mounted on to the electrical cabinet by means of the holes located on its back for that purpose. Care must be taken to position the power supply at the bottom.



Central Unit (CPU) voltage supply

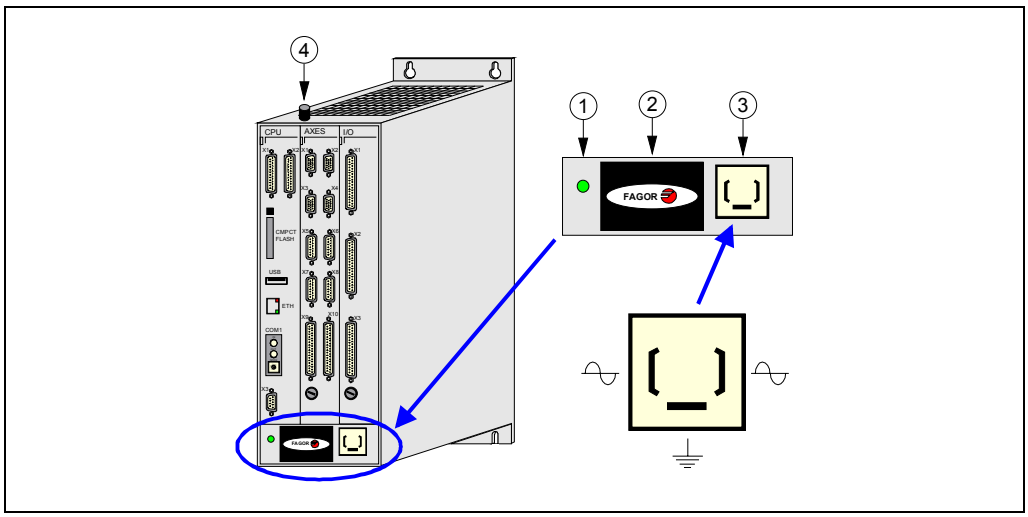
Do not get into the inside of the unit.

- *Only authorized personnel from Fagor Automation may do it.*

Do not handle the connectors with the unit connected to main AC power.

- *Before manipulating these connectors, make sure that the unit is not connected to main AC power.*

Power the central unit through a separate 110VA transformer with an output voltage between 84Vac and 264Vac and 50-60 Hz.



1. Indicator led. When ON, it indicates that the central unit is under power.
2. Lithium battery. Maintains the RAM memory data when the system is powered off.
3. Mains plug. Is used to power the central unit by connecting it to the transformer and to ground.
4. Ground terminal. All ground terminals of the machine must be connected to this terminal. Metric 6mm.

When detecting a voltage peak, wait for 3 minutes before turning it on again.

For further technical information, refer to the appendix at the end of this manual. See *"Central unit of the 8055 CNC"* on page 625.

1.2.1 –CPU– Module

Besides containing the system software, this module performs all the functions of the CNC (edit, execute, display, etc.) as well as processing the information from the rest of the modules and generating video signals for the monitor.

The connectors that permit connecting the central unit with the monitor and the keyboard are located in this module.

When replacing the–CPU– module, the contents of the internal RAM memory are kept for about 24 hours as long as it has been previously on for more than 1 minute; but the date and the time will be lost and will have to be set again.

Do not get into the inside of the unit.

- *Only personnel authorized by Fagor Automation may manipulate the inside of this unit.*

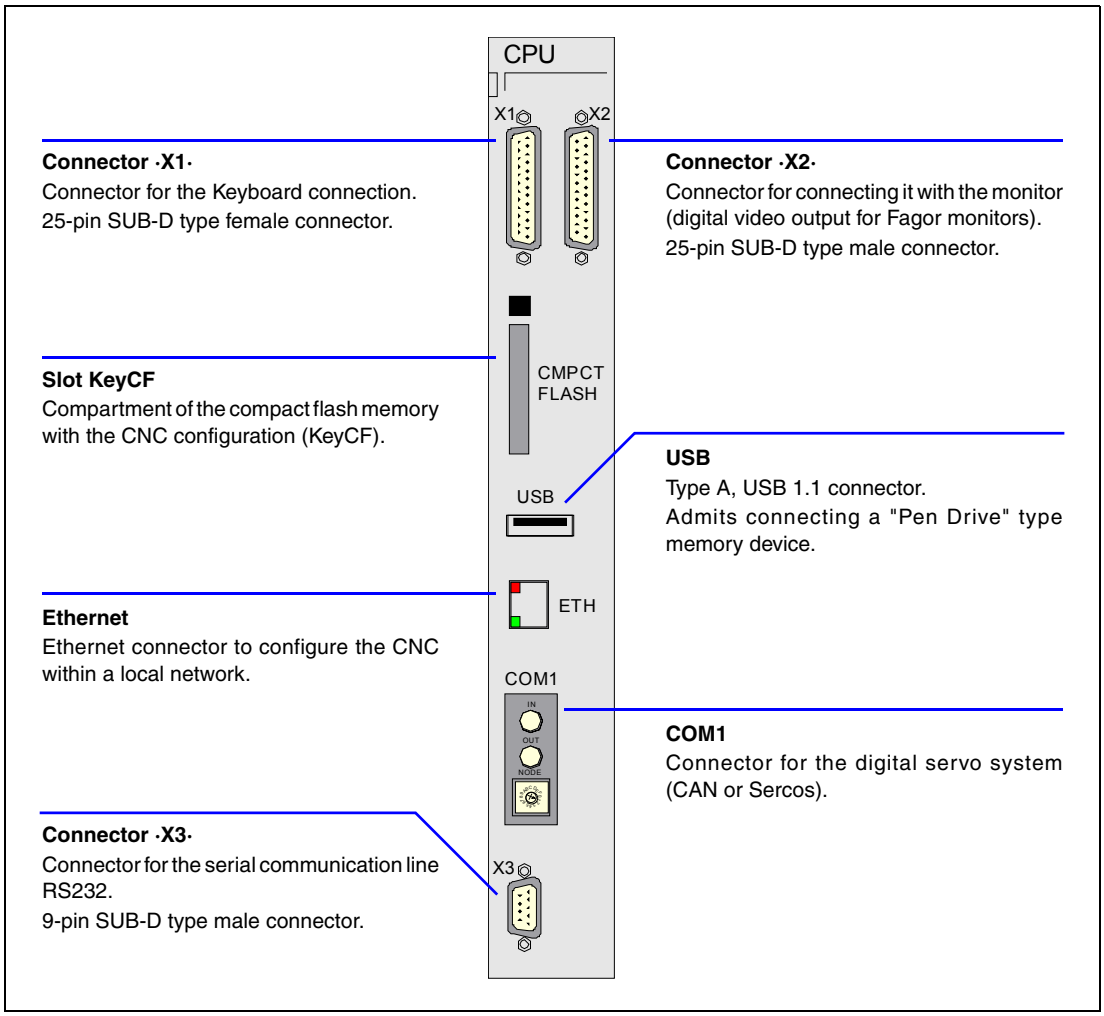
Do not handle the connectors with the unit connected to main AC power.

- *Before manipulating these connectors, make sure that the unit is not connected to main AC power.*

Peripheral connection and disconnection.

- *The CNC must be powered off when connecting or disconnecting any peripheral through connector X3 (RS232C).*
- *When the mains connection of the PC or peripheral device is not referenced to ground, it is recommended to connect the cable shield to the connector hood only at the CNC end.*

Elements



Connectors and connection

Connector X1 - for the Keyboard connection

SUB-D type 25-pin female connector to connect the central unit with the keyboard.

FAGOR AUTOMATION provides the cable necessary for this connection. This cable has two 25-pin male connectors of the 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 X2 - for Fagor monitors

25-pin male connector of the SUB-D type to connect the central unit with the monitor.

Fagor Automation provides the cable necessary for this connection. This cable has two 25-pin female connectors of the SUB-D type.

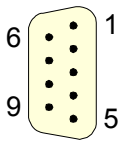
Both connectors have a latching system by means of two screws UNC4.40.

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 X3 - 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	DSR
	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.

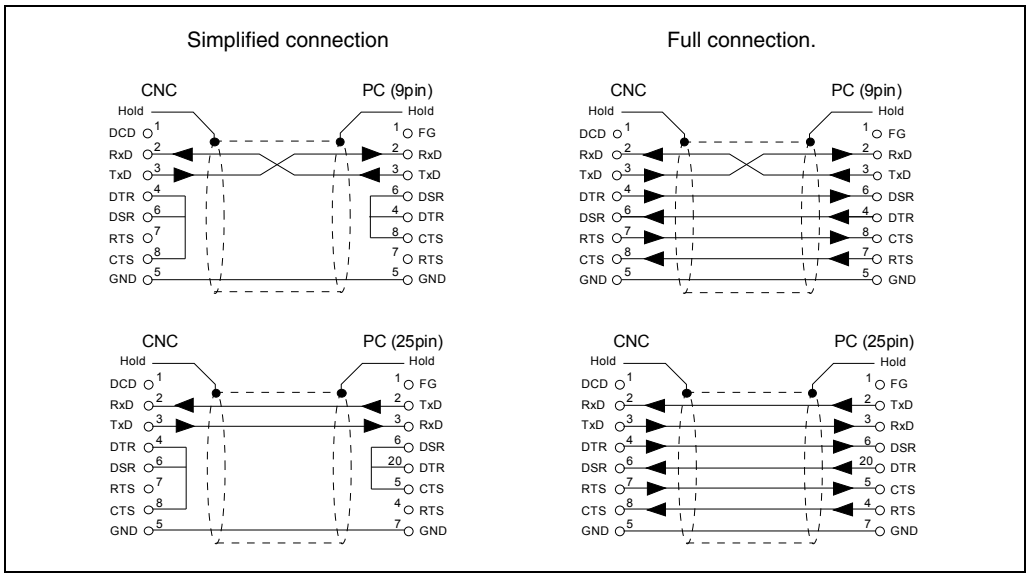
The CNC can operate at up to 115,200 Baud.

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 (-GND- pin) thus, avoiding reference points at different voltages especially in long cables.

Recommended RS232C interface connection.



Slot "KEYCF" - Compartment of the KeyCF (CNC configuration card)

It has a compact flash type hard disk to store user programs and for updating software versions. The hard disk may be accessed from the outside.

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 CNC will recognize the KeyCF as <Hard Disk>.

When having the Ethernet option, it is possible to use a PC directory as a remote hard disk.

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

The USB 1.1 port with type A connector, it admits connecting a "Pendrive" 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.

When using a USB extension cable, it must be a type A - type B cable and no longer than 3 m. The cable should have double shield.

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.

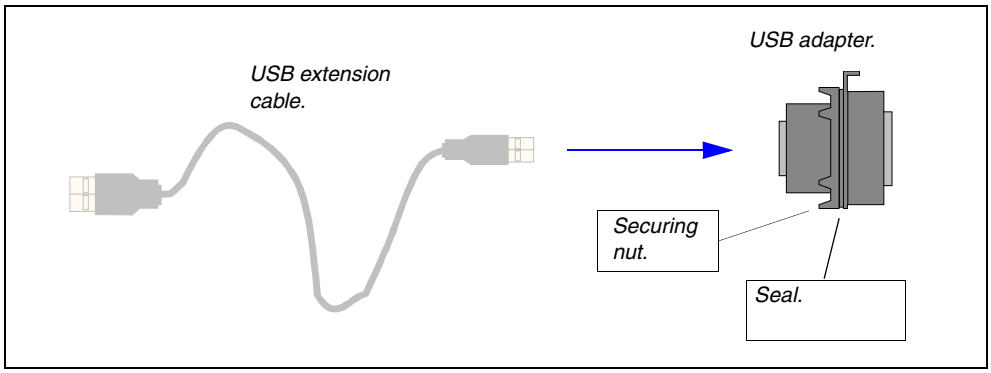
The CNC recognizes the connected device as USB Hard Disk. When it is connected, it will be shown as <USB hard disk> on the left panel of the <explorer>. To see its contents, press the <update> (refresh) softkey.

Within the USB device, the CNC will only recognize files with extensions *.f55 (software version), *fhw (firmware update files), part-programs, parameters, tables, pages (screens) and symbols. The CNC will not recognize any other type of file. Part-programs cannot be edited or executed from the USB hard disk.

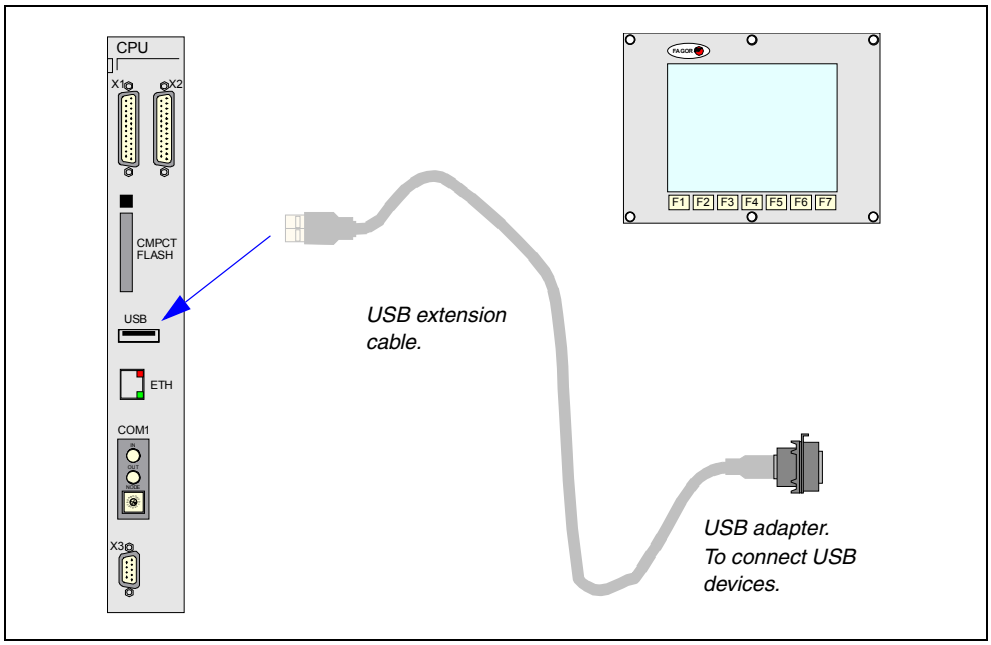
Separate monitors (without keyboard)

Connecting the USB extension set supplied by Fagor:

1. Connect the cable and the USB adapter. Check that the seal and the nut of the USB adaptor are secured as shown in the figure.



2. While the CNC is off, connect the extension cable to the USB connector of the CNC.

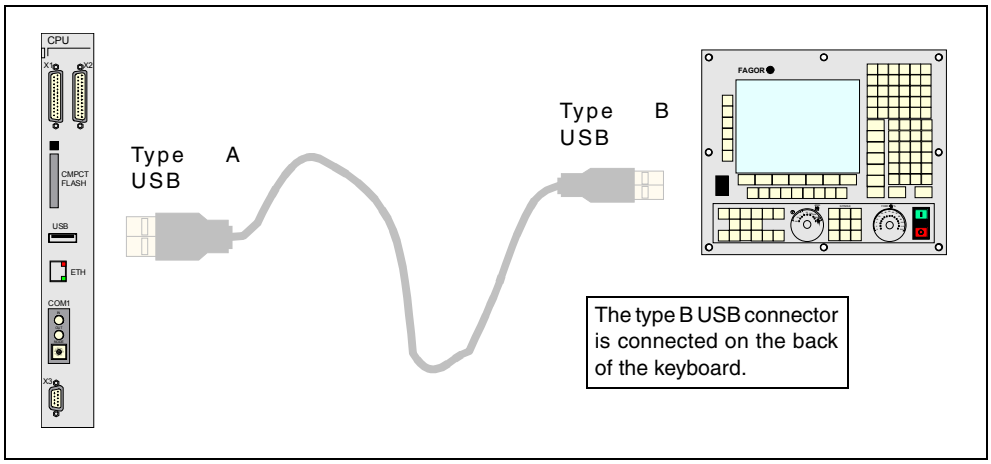


3. Once properly connected to the extension set, the USB devices may be connected through the adapter.

Monitors with keyboard

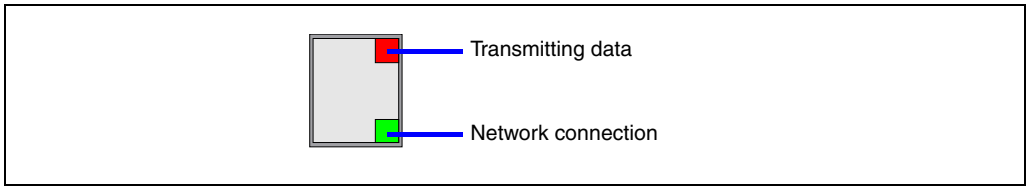
Connecting the USB extension cable supplied by Fagor

1. While the CNC is off, connect the extension cable to the CPU and to the keyboard.



2. Once the extension cable has been properly connected, the USB connector of the keyboard may be used.

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 or carrying out telediagnostic tasks.

The Ethernet option does not require having the DNC option available.

The Ethernet card has an RJ-45 connector and two LED's that inform on the status of the connection.

Red LED	Blinks while transmitting data.
Green LED	On while connected to the network.

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.

COM1 - Digital servo (CAN or Sercos)

The digital servo system (CAN or Sercos) is connected to COM1. This servo system is enabled through its corresponding software option.

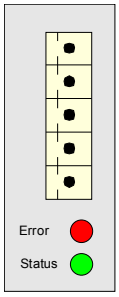
Two types of digital servo system may be used to communicate with Fagor servo drives:

- Sercos interface IEC1491.
- CAN field bus and standard CanOpen communication protocol. In general, their characteristics are similar to those of the Sercos ring with lower transmission speed.

The same system may have digital axes (CAN or Sercos) and analog axes. On the other hand, it is not possible to have digital axes with Sercos interface and CAN interface at the same time.

Digital CAN servo

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.

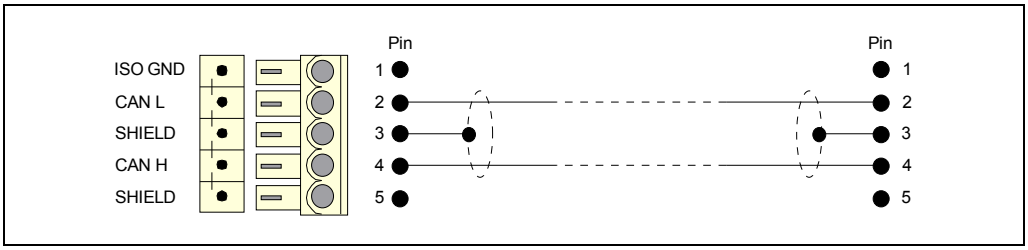
Characteristics of the CAN cable

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).

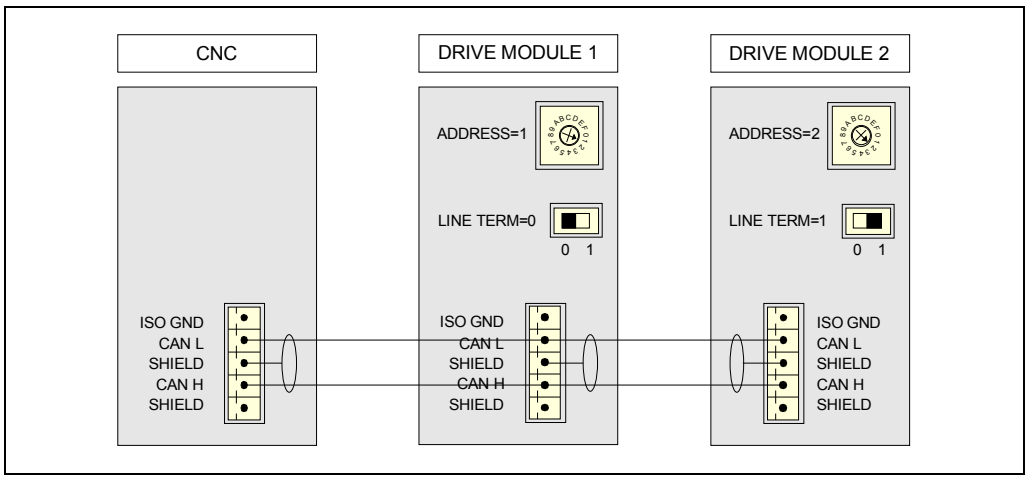


Signal	Description
ISO GND	Ground / 0 V.
CAN L	Bus signal (LOW).
SHIELD	CAN shield.
CAN H	Bus signal (HIGH).
SHIELD	CAN shield.

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

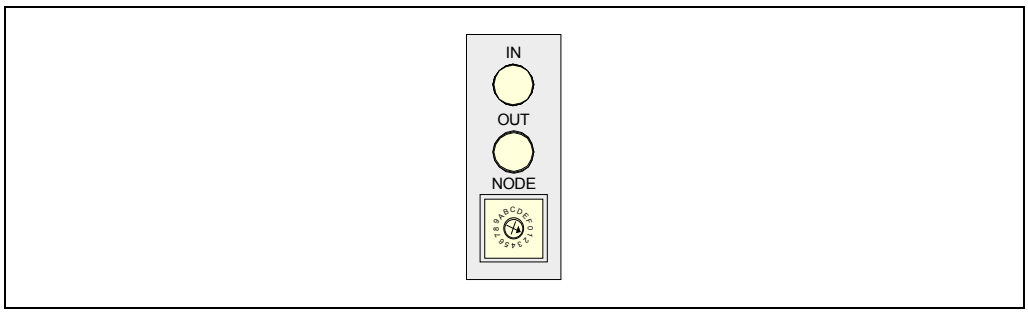
Interconnection of modules.

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



Sercos digital servo system

Module identification at the bus.

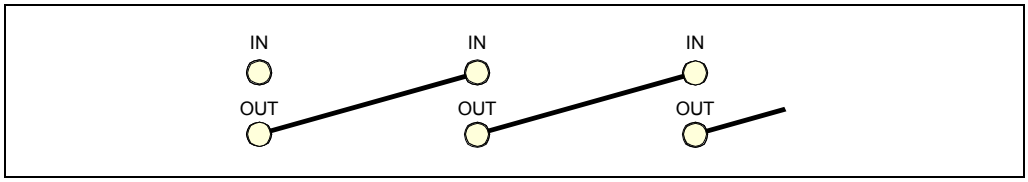


Each element integrated into the Sercos bus is identified with the 16-position rotary switch (0-15) "Address" (also known as Node_Select). This rotary switch selects the address (node) occupied by each element integrated in the bus.

The CNC must always occupy the "0" position and the rest will occupy the consecutive positions 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 fact that the drive identified with number 1 (for example) corresponds to the X axis, to Y or to another one is irrelevant. However, it is a good idea, for the sake of clarity, that the machine axes X, Y, Z, U, V, W, A, B and C are numbered sequentially in that order.

Sercos connector pinout.



Signal	Description
IN	Sercos connection input.
OUT	Sercos connection output.

Sercos cable characteristics.

Fagor Automation provides the fiber optic cables needed for Sercos communication. There are different types of cables depending on length and dynamic and static installation characteristics.

- Fiber optic cable with polymer core (SFO, SFO-FLEX) for up to 40 meters If the cable will be subject to dynamic conditions (it will be moving), the SFO-FLEX cable must always be used. If the cable will be subject to static conditions (it will not be moving), the SFO cable will be enough.
- Fiber optic cable with glass core (SFO-V-FLEX) for more than 40 meters.

Mechanical characteristics of the cables.

SFO cable	
Flexibility.	Normal. Minimum bending radius: 30 mm. Only to be used when the cable will not be moving (static conditions).
Covering.	PUR. Polyurethane resistant to chemical used on machines.
Temperature.	Work: -20 ºC / 80 ºC (-4 ºF / 176 ºF). Storage: -35 ºC / 85 ºC (-31 ºF / 158 ºF).

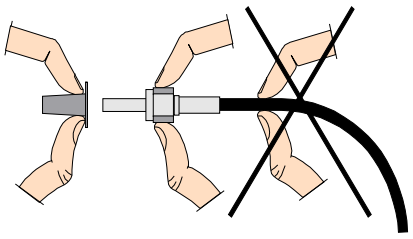
SFO-V-FLEX cable	
Flexibility.	Minimum bending radius: 16 mm. Special cables to be used in cable carrying chains.
Temperature.	Work: -65 ºC / 125 ºC (-85 ºF / 257 ºF).

SFO-FLEX cable	
Flexibility.	High. Minimum static bending radius of 50 mm and a dynamic radius of 70 mm. Special cables to be used in cable carrying chains.
Covering.	PUR. Polyurethane resistant to chemical used on machines.
Temperature.	Work: -20 ºC / 70 ºC (-4 ºF / 158 ºF). Storage: -40 ºC / 80 ºC (-40 ºF / 176 ºF).

Cable handling.

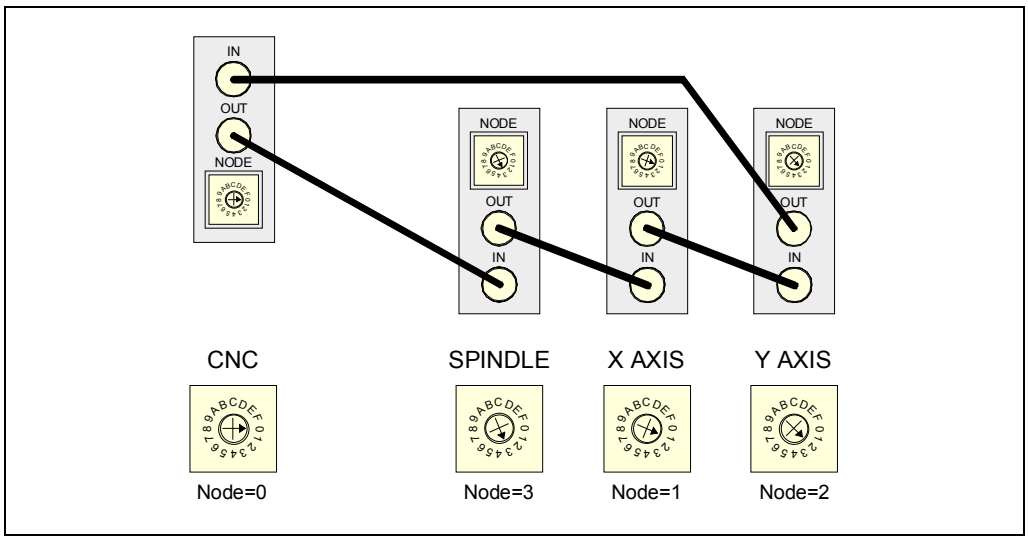
Fagor supplies the cable with the terminals protected by a cap. Remove the protection cap before connecting the cable.

To remove the terminal protecting cap or to connect or disconnect the cable, hold the cable by its terminal; never pull the cable by holding its plastic part because it could render it useless.



Interconnection of modules.

It is connected in a ring through optic fiber, by joining an OUT terminal with an IN terminal. The figure shows the Sercos connection of the CNC with Fagor spindle drives and axes X and Y.



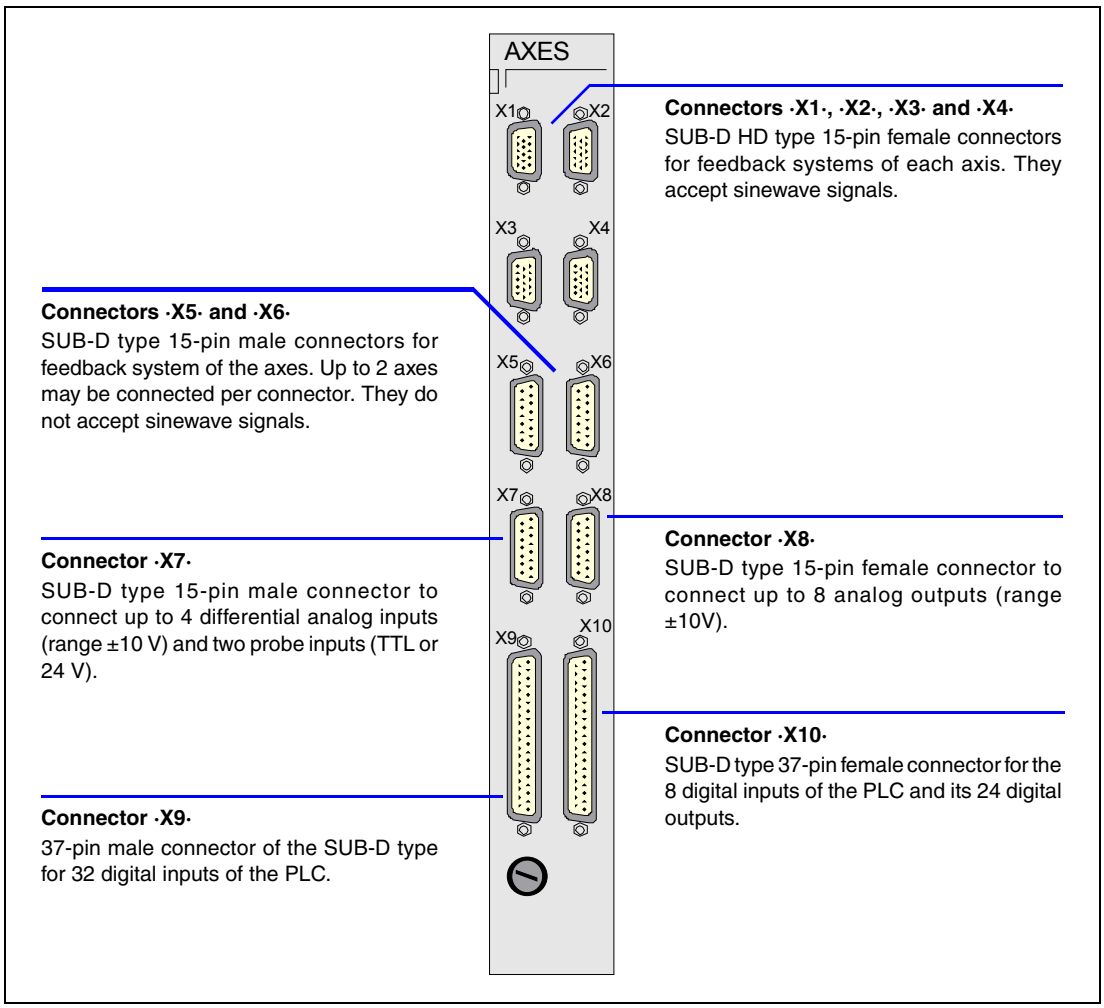
1.2.2 –Vpp Axes– module

This module will appear in <DIAGNOSIS> / <CONFIGURATION> / <HARDWARE> with the name of "Axis module 2".

Besides controlling the spindle and the axes of the machine, it governs the first 40 digital PLC inputs and 24 digital PLC outputs. This module offers the following to communicate with the outside world:

4	Feedback inputs admitting single-ended and double-ended (differential) TTL signals as well as 1Vpp sinusoidal signals.
4	Feedback inputs admitting single and double-ended (differential) TTL signals.
8	Analog command outputs for the servo drives.
4	Free differential analog inputs for controlling, monitoring or supervising systems.
2	Digital probe inputs.
24	Digital outputs ,optocoupled, commanded by the PLC.
40	Digital inputs ,optocoupled, read by the PLC.

Elements



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 manipulating these connectors, make sure that the unit is not connected to main AC power.

The machine manufacturer must comply with the "EN 60204-1 (IEC-204-1)", standard regarding protection against electrical shock due to I/O contact failures with external power supply when not hooking up this connector before turning the power supply on.

Signal adapters

The following signal adapters are available to be used with the –Vpp axes– module.

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

Technical characteristics of the feedback inputs

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

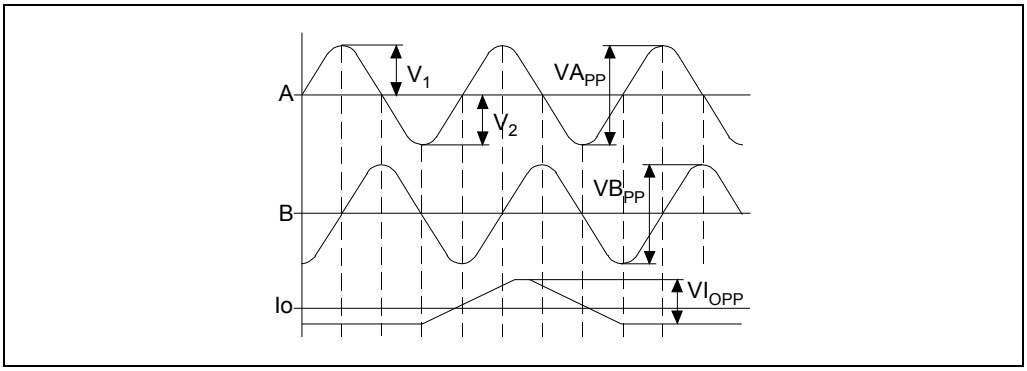
Work levels for differential square signal.

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.

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.



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

Connectors and connection

Connectors X1, X2, X3, X4 - Differential TTL and 1Vpp sinusoidal feedback signals

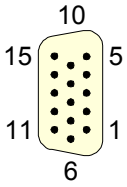
They are 15-pin female connectors of the SUB-D HD type and they are used for the feedback system connections of the axes. They admit differential TTL and 1Vpp sinusoidal feedback signals.

Each connector may be connected to one axis. g.m.p. AXIS1 (P0), AXIS2 (P1), AXIS3 (P2) and AXIS4 (P3) must be set to let the CNC know which axis has been connected to each one of them.

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. The wires of the shielded cables cannot be unshielded for more than 75mm (about 3 inches).

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

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

Handwheel connection

When connecting handwheels, they must be differential as well as the axis selector signal. The axis selector signal must be connected to pins 5 and 6.

Non-differential handwheels (for example Fagor 100P) may be connected using either the Fagor signal adapter "SA-TTL-TTLD" (from "non-differential TTL to differential TTL) or connectors X5 and X6.

Protection at the connectors

It detects overcurrents or short-circuits at the feedback devices issuing the relevant error message.

"Supply voltage error on axis feedback *".

Connectors X5, X6 - TTL and differential TTL feedback signals

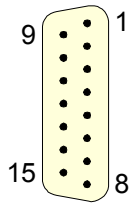
They are 15-pin male connectors of the SUB-D type used for feedback system connections. They admit differential (double-ended) and non-differential (single-ended) TTL feedback.

Each connector may be connected to up to 2 axes. g.m.p. AXIS5 (P4), AXIS6 (P5), AXIS7 (P6) and AXIS8 (P7) must be set to let the CNC know which axis has been connected to each one of them.

The cables 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. The wires of the shielded cables cannot be unshielded for more than 75mm (about 3 inches).

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



Pin	Signal and function	
1	A	Differential squarewave feedback signal.
2	/A	
3	B	
4	/B	
5	I0	Home signals.
6	/I0	
7	+5 V	Voltage supply for feedback systems.
8	0 V	
9	A	Differential squarewave feedback signal.
10	/A	
11	B	
12	/B	
13	I0	Home signals.
14	/I0	
15	- - -	

Handwheel connection

When using a FAGOR 100P model handwheel, the axis selecting signal must be connected to the reference mark (I0) pin of the corresponding axis 5 or 13 of this connector.

Protection at the connectors

It detects the error in axis pairs issuing the relevant error message.

"Supply voltage error on axis feedback *".

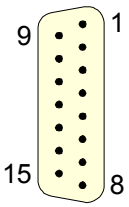
Connector X7 - Differential analog inputs and touch probes

It is a 15-pin male connector of the SUB-D type used to connect the two touch probes and the analog inputs.

Up to 4 analog inputs may be connected for supervision, monitoring, etc. The signals may be within a range of ±5 V or ±10V; the range is selected by PLC parameter "IANA5V (P130)".

There are 4 probe inputs (two for 5V and two for 24V) and pin ·7· (0V probe input) must be connected to the 0V of the external power supply.

The cable shield must be connected to the metallic hood at each end. The wires of the shielded cables cannot be unshielded for more than 75mm (about 3 inches).

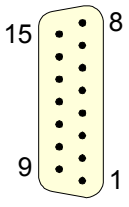
	Pin	Signal and function	
	1 2 3 4	+I1 +I2 +I3 +I4	Positive differential analog inputs.
	5 6	PRB1_5 PRB2_5	5 V input of probe 1. 5 V input of probe 2.
	7	GND	Probe's 0 V input.
	8	- - -	
	9 10 11 12	-I1 -I2 -I3 -I4	Negative differential analog inputs.
	13 14	PRB1_24 PRB2_24	24 V input of probe 1. 24 V input of probe 2.
	15	+5V.	+5V supply for the probes.

Connector X8 - Analog outputs.

It is a 15-pin female connector of the SUB-D type used for the analog command outputs

Each one of the outputs (O1 thru O8) correspond to the feedback inputs X1 thru X6. The name of the axis connected to each one of them is determined by setting g.m.p. AXIS1 (P0) thru AXIS8 (P7).

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

	Pin	Signal and function	
	1 2 3 4 5 6 7 8	O1 O2 O3 O4 O5 O6 O7 O8	Analog command outputs with a range of ±10V.
	9 10 11 12 13 14	GND GND GND GND GND GND	Velocity command reference signals.
	15		Chassis - Shield.

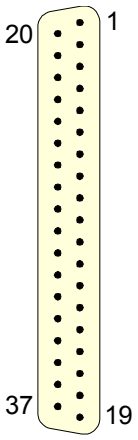
Connector X9 - PLC inputs

It is a 37-pin male connector of the SUB-D type used for the PLC inputs.

Since the response time of the EMERGENCY signal must be very short, the CNC has assigned input I1 (pin 2) for this purpose. Thus, the CNC will treat this input immediately regardless of how the PLC program uses it.

The 0V of the power supply used for these inputs must be connected to pins 18 and 19 of the connector.

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



Pin	Signal and function	
1	- - -	
2	I1	/Emergency stop.
3	I3	
4	I5	
5	I7	
6	I9	
7	I11	
8	I13	
9	I15	
10	I17	
11	I19	
12	I21	
13	I23	
14	I25	
15	I27	
16	I29	
17	I31	
18	0 V	External power supply.
19	0 V	External power supply.
20	- - -	
21	I2	
22	I4	
23	I6	
24	I8	
25	I10	
26	I12	
27	I14	
28	I16	
29	I18	
30	I20	
31	I22	
32	I24	
33	I26	
34	I28	
35	I30	
36	I32	
37		Chassis - Shield.

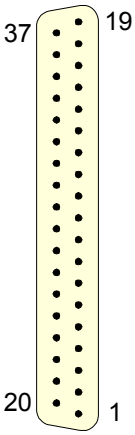
Connector X10 - PLC inputs and outputs.

It is a 37-pin female connector of the SUB-D type used for the inputs and outputs of the PLC.

When certain errors are issued, the CNC, besides indicating it to the PLC (/ALARM mark), activates output O1 (pin 2) of this connector. This way, regardless of how this signal is treated by the PLC program, the electrical cabinet can process this signal immediately.

Both 24V and 0V of the power supply used to power these I/Os must be connected to pins 18 and 19 (for 0V) and pins 1 and 20 (for the 24V).

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



Pin	Signal and function	
1	24 V	External power supply.
2	O1	/Emergency output.
3	O3	
4	O5	
5	O7	
6	O9	
7	O11	
8	O13	
9	O15	
10	O17	
11	O19	
12	O21	
13	O23	
14	I33	
15	I35	
16	I37	
17	I39	
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	O10	
26	O12	
27	O14	
28	O16	
29	O18	
30	O20	
31	O22	
32	O24	
33	I34	
34	I36	
35	I38	
36	I40	
37		Chassis - Shield.

The Emergency output, which coincides with O1 of the PLC, will be activated (change from logic level 1 to 0) when certain errors come up at the CNC or when the PLC output O1 is set to 0 (logic level 0).

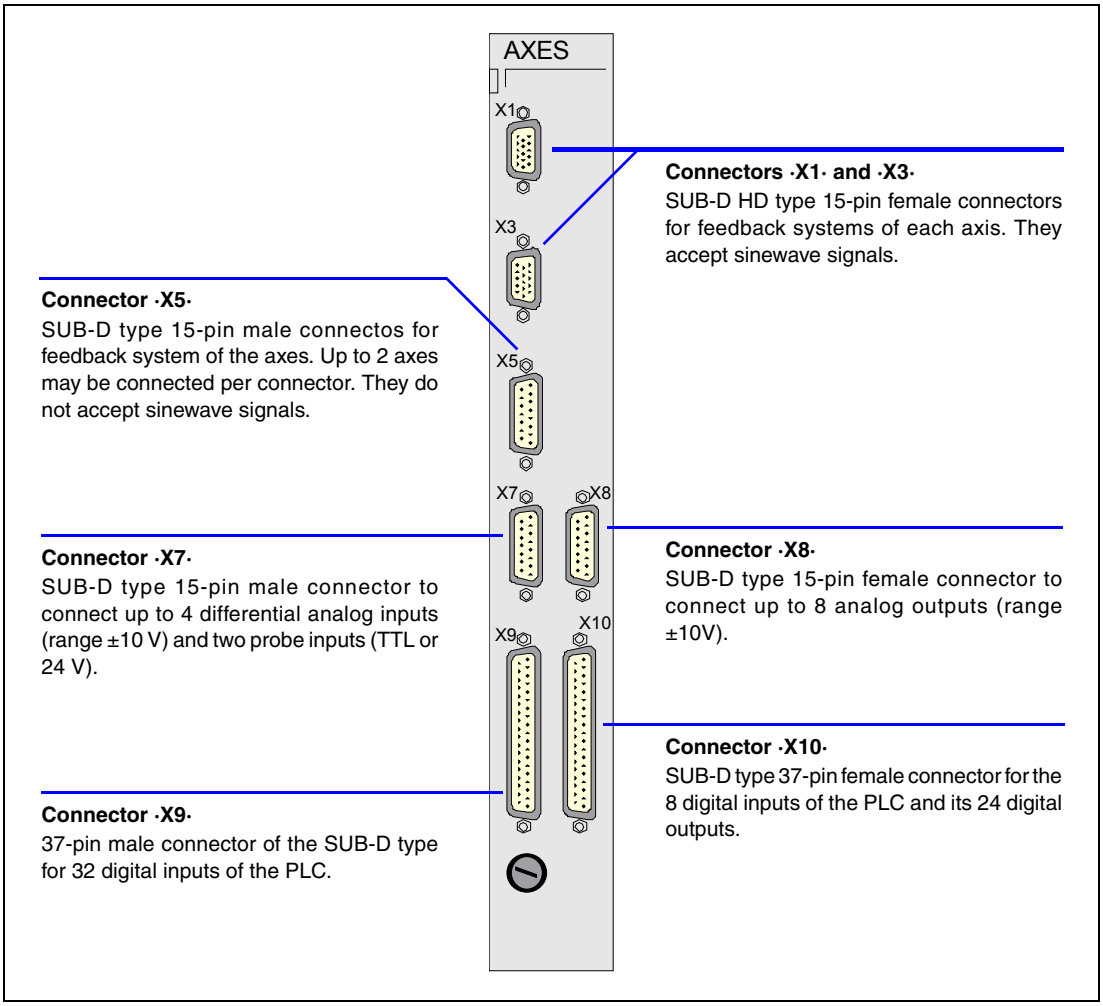
1.2.3 –Vpp SB Axes– module

The axes module is used in Sercos configurations and will appear in <DIAGNOSIS> / <CONFIGURATION> / <HARDWARE> under the name of "SB axes module".

Besides controlling the spindle and the axes of the machine, it governs the 40 digital PLC inputs and 24 digital PLC outputs. This module offers the following to communicate with the outside world:

2	Feedback inputs admitting single-ended and double-ended (differential) TTL signals as well as 1Vpp sinusoidal signals.
2	Feedback inputs admitting single and double-ended (differential) TTL signals.
8	Analog command outputs for the servo drives.
4	Free differential analog inputs for controlling, monitoring or supervising systems.
2	Digital probe inputs.
24	Digital outputs ,optocoupled, commanded by the PLC.
40	Digital inputs ,optocoupled, read by the PLC.

Elements



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 manipulating these connectors, make sure that the unit is not connected to main AC power.

The machine manufacturer must comply with the "EN 60204-1 (IEC-204-1)", standard regarding protection against electrical shock due to I/O contact failures with external power supply when not hooking up this connector before turning the power supply on.

Signal adapters

The following signal adapters are available to be used with the –Vpp axes– module.

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

Technical characteristics of the feedback inputs

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

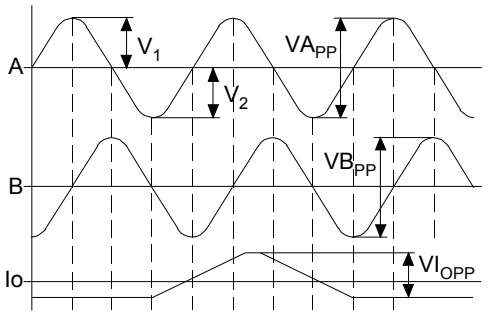
Work levels for differential square signal.

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.

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.



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

Connectors and connection

Connectors X1, X3 - Differential TTL and 1Vpp sinusoidal feedback signals

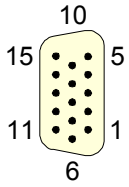
They are 15-pin female connectors of the SUB-D HD type and they are used for the feedback system connections of the axes. They admit differential TTL and 1Vpp sinusoidal feedback signals.

Each connector may be connected to one axis. g.m.p. AXIS1 (P0) and AXIS3 (P2) must be set to let the CNC know which axis has been connected to each one of them.

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. The wires of the shielded cables cannot be unshielded for more than 75mm (about 3 inches).

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

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

Handwheel connection

When connecting handwheels, they must be differential as well as the axis selector signal. The axis selector signal must be connected to pins 5 and 6.

Non-differential handwheels (for example Fagor 100P) may be connected using either the Fagor signal adapter "SATTLTTLD" (from "non-differential TTL to differential TTL) or connector X5.

Protection at the connectors

It detects overcurrents or short-circuits at the feedback devices issuing the relevant error message.

"Supply voltage error on axis feedback *".

Connector X5 - TTL and differential TTL feedback signals

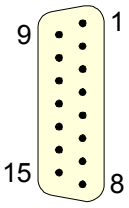
It is a 15-pin male connectors of the SUB-D type used for feedback system connection. It admits differential (double-ended) and non-differential (single-ended) TTL feedback.

This connector may be connected to up to 2 axes. g.m.p. AXIS5 (P4) and AXIS6 (P5) must be set to let the CNC know which axis has been connected to each one of them.

The cables 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. The wires of the shielded cables cannot be unshielded for more than 75mm (about 3 inches).

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

	Pin	Signal and function	
	1 2 3 4	A /A B /B	Differential squarewave feedback signal.
	5 6	I0 /I0	Home signals.
	7 8	+5 V 0 V	Voltage supply for feedback systems.
	9 10 11 12	A /A B /B	Differential squarewave feedback signal.
	13 14	I0 /I0	Home signals.
	15	- - -	

Handwheel connection

When using a FAGOR 100P model handwheel, the axis selecting signal must be connected to the reference mark (I0) pin of the corresponding axis 5 or 13 of this connector.

Protection at the connectors

It detects the error in axis pairs issuing the relevant error message.

"Supply voltage error on axis feedback *".

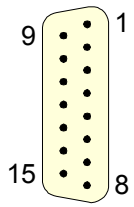
Connector X7 - Differential analog inputs and touch probes

It is a 15-pin male connector of the SUB-D type used to connect the two touch probes and the analog inputs.

Up to 4 analog inputs may be connected for supervision, monitoring, etc. The signals may be within a range of ±5 V or ±10V; the range is selected by PLC parameter "IANA5V (P130)".

There are 4 probe inputs (two for 5V and two for 24V) and pin ·7· (0V probe input) must be connected to the 0V of the external power supply.

The cable shield must be connected to the metallic hood at each end. The wires of the shielded cables cannot be unshielded for more than 75mm (about 3 inches).

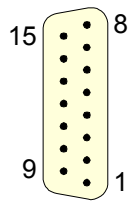
	Pin	Signal and function	
	1 2 3 4	+I1 +I2 +I3 +I4	Positive differential analog inputs.
	5 6	PRB1_5 PRB2_5	5 V input of probe 1. 5 V input of probe 2.
	7	GND	Probe's 0 V input.
	8	- - -	
	9 10 11 12	-I1 -I2 -I3 -I4	Negative differential analog inputs.
	13 14	PRB1_24 PRB2_24	24 V input of probe 1. 24 V input of probe 2.
	15	+5V.	+5V supply for the probes.

Connector X8 - Analog outputs.

It is a 15-pin female connector of the SUB-D type used for the analog command outputs

Each one of the outputs (O1 thru O8) correspond to the feedback inputs X1 thru X6. The name of the axis connected to each one of them is determined by setting g.m.p. AXIS1 (P0) thru AXIS8 (P7).

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

	Pin	Signal and function	
	1 2 3 4 5 6 7 8	O1 O2 O3 O4 O5 O6 O7 O8	Analog command outputs with a range of ±10V.
	9 10 11 12 13 14	GND GND GND GND GND GND	Velocity command reference signals.
	15		Chassis - Shield.

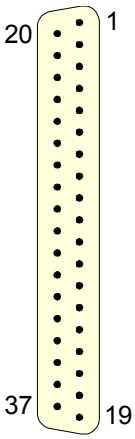
Connector X9 - PLC inputs

It is a 37-pin male connector of the SUB-D type used for the PLC inputs.

Since the response time of the EMERGENCY signal must be very short, the CNC has assigned input I1 (pin 2) for this purpose. Thus, the CNC will treat this input immediately regardless of how the PLC program uses it.

The 0V of the power supply used for these inputs must be connected to pins 18 and 19 of the connector.

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



Pin	Signal and function	
1	- - -	
2	I1	/Emergency stop.
3	I3	
4	I5	
5	I7	
6	I9	
7	I11	
8	I13	
9	I15	
10	I17	
11	I19	
12	I21	
13	I23	
14	I25	
15	I27	
16	I29	
17	I31	
18	0 V	External power supply.
19	0 V	External power supply.
20	- - -	
21	I2	
22	I4	
23	I6	
24	I8	
25	I10	
26	I12	
27	I14	
28	I16	
29	I18	
30	I20	
31	I22	
32	I24	
33	I26	
34	I28	
35	I30	
36	I32	
37		Chassis - Shield.

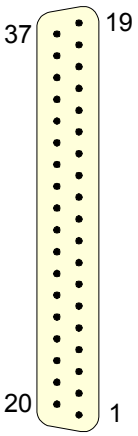
Connector X10 - PLC inputs and outputs.

It is a 37-pin female connector of the SUB-D type used for the inputs and outputs of the PLC.

When certain errors are issued, the CNC, besides indicating it to the PLC (/ALARM mark), activates output O1 (pin 2) of this connector. This way, regardless of how this signal is treated by the PLC program, the electrical cabinet can process this signal immediately.

Both 24V and 0V of the power supply used to power these I/Os must be connected to pins 18 and 19 (for 0V) and pins 1 and 20 (for the 24V).

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

	Pin	Signal and function	
	1	24 V	External power supply.
	2	O1	/Emergency output.
	3	O3	
	4	O5	
	5	O7	
	6	O9	
	7	O11	
	8	O13	
	9	O15	
	10	O17	
	11	O19	
	12	O21	
	13	O23	
	14	I33	
	15	I35	
	16	I37	
	17	I39	
	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	O10	
	26	O12	
	27	O14	
	28	O16	
	29	O18	
	30	O20	
	31	O22	
	32	O24	
	33	I34	
	34	I36	
	35	I38	
	36	I40	
	37		Chassis - Shield.

The Emergency output, which coincides with O1 of the PLC, will be activated (change from logic level 1 to 0) when certain errors come up at the CNC or when the PLC output O1 is set to 0 (logic level 0).

1.2.4 –I/O– module (inputs - outputs)

This module is used to expand the basic configuration and may be used to expand the number of digital inputs and outputs of the PLC.

Each module provides:

64 Optocoupled digital inputs.

32 Optocoupled digital outputs.

The numbering of the various inputs and outputs of each module depends on the logic address assigned to the module and it is as follows:

Module	Inputs	Outputs
–Vpp Axes– module –Vpp SB Axes– module	I1-I40	O1-O24
Input - output module I/O (1)	I65-I128	O33-O64
Input - output module I/O (2)	I129-I192	O65-O96
Input - output module I/O (3)	I193-I256	O97-O128

The PLC can control up to 512 inputs and 512 outputs although it can only communicate to the outside world through the ones indicated above.

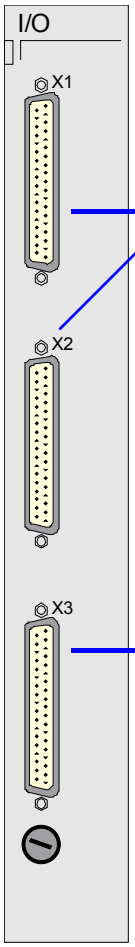


Do not get into the inside of the unit. Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not handle the connectors with the unit connected to main AC power. Before manipulating these connectors, make sure that the unit is not connected to main AC power.

The machine manufacturer must comply with the "EN 60204-1 (IEC-204-1)", standard regarding protection against electrical shock due to I/O contact failures with external power supply when not hooking up this connector before turning the power supply on.

Elements



Connectors ·X1· & ·X2·
37-pin male connector of the SUB-D type for 64 digital inputs of the PLC.

Connector ·X3·
37-pin female connector of the SUB-D type for 32 digital outputs of the PLC.

Connectors and connection

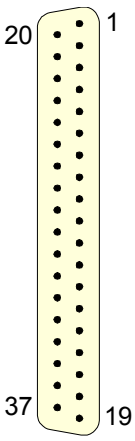
Connectors X1, X2

37-pin male connectors of the SUB-D type used for the PLC inputs.

The 0V of the power supply used for these inputs must be connected to pins 18 and 19 of each connector.

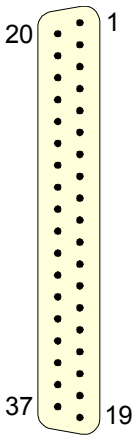
All shields must only be connected to ground at the CNC end leaving the other end free. The wires of the shielded cables cannot be unshielded for more than 75mm (about 3 inches).

Connector X1.



Pin	Signal and function	
1	- - -	
2	I97	
3	I99	
4	I101	
5	I103	
6	I105	
7	I107	
8	I109	
9	I111	
10	I113	
11	I115	
12	I117	
13	I119	
14	I121	
15	I123	
16	I125	
17	I127	
18	0 V	External power supply.
19	0 V	External power supply.
20	- - -	
21	I98	
22	I100	
23	I102	
24	I104	
25	I106	
26	I108	
27	I110	
28	I112	
29	I114	
30	I116	
31	I118	
32	I120	
33	I122	
34	I124	
35	I126	
36	I128	
37		Chassis - Shield.

Connector X2.

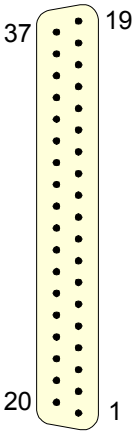
	Pin	Signal and function	
	1	- - -	
	2	I65	
	3	I67	
	4	I69	
	5	I71	
	6	I73	
	7	I75	
	8	I77	
	9	I79	
	10	I81	
	11	I83	
	12	I85	
	13	I87	
	14	I89	
	15	I91	
	16	I93	
	17	I95	
	18	0 V	External power supply.
	19	0 V	External power supply.
	20	- - -	
	21	I66	
	22	I68	
	23	I70	
	24	I72	
	25	I74	
	26	I76	
	27	I78	
	28	I80	
	29	I82	
	30	I84	
	31	I86	
	32	I88	
	33	I90	
	34	I92	
	35	I94	
	36	I96	
	37		Chassis - Shield.

Connector X3.

37-pin female connector of the SUB-D type used for the PLC outputs.

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

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



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	O49	
11	O51	
12	O53	
13	O55	
14	O57	
15	O59	
16	O61	
17	O63	
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	O50	
30	O52	
31	O54	
32	O56	
33	O58	
34	O60	
35	O62	
36	O64	
37		Chassis - Shield.

1.3 Monitors

There are independent monitors and those with incorporated keyboard. See *"1.1 CNC structure"* on page 27.

Independent monitors

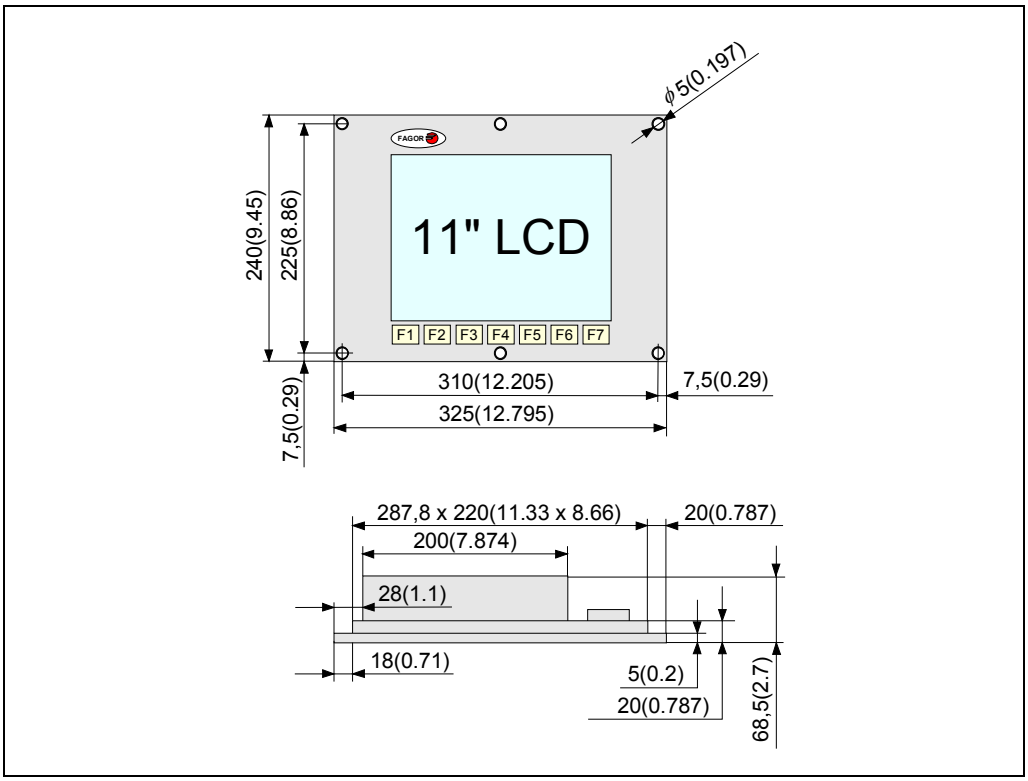
Monitor	Reference
11" LCD	MONITOR-8055

Monitors with incorporated keyboard

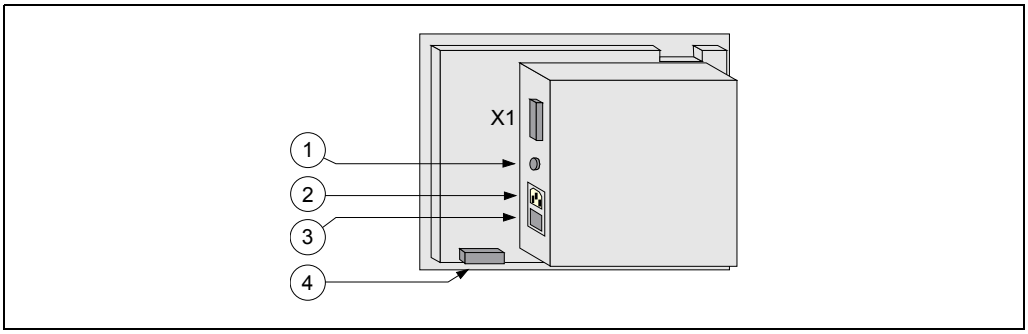
Monitor + Keyboard	Reference
11" LCD + M keyboard	MONITOR-55M-11-USB
11" LCD + T keyboard	MONITOR-55T-11-USB
11" LCD + MC keyboard	MONITOR-55MC-11-USB
11" LCD + TC keyboard	MONITOR-55TC-11-USB

1.3.1 11" LCD Monitor

Dimensions

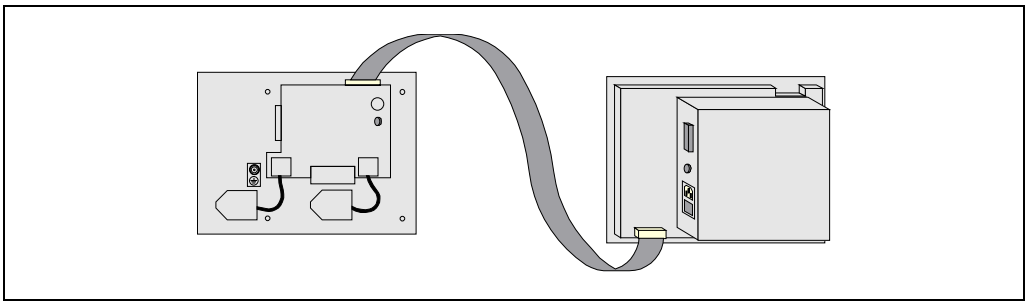


Elements



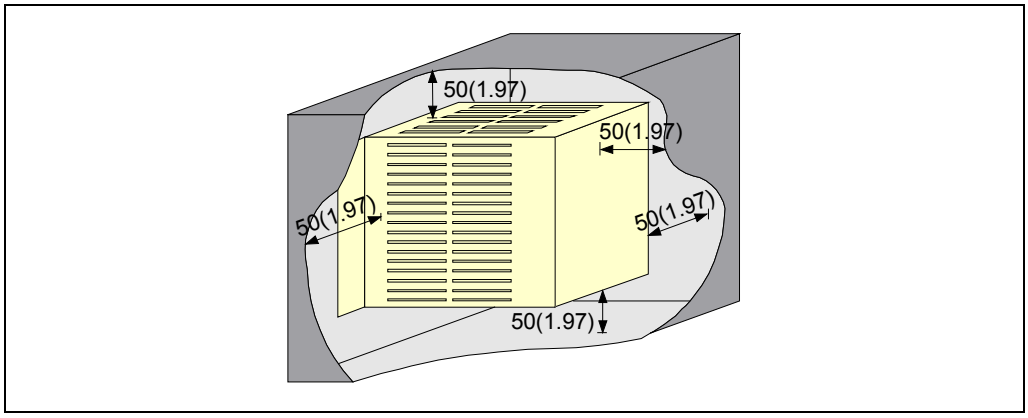
X1 SUB-D type 25-pin male connector to connect the video signals with the central unit.

1. Ground terminal. All ground terminals of the machine must be connected to this terminal. Metric 6mm.
2. Mains plug. Use the connector provided for 220Vac power and ground connection.
3. On/off power switch.
4. SUB-D type 25-pin female connector to connect the keyboard.



Enclosure

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



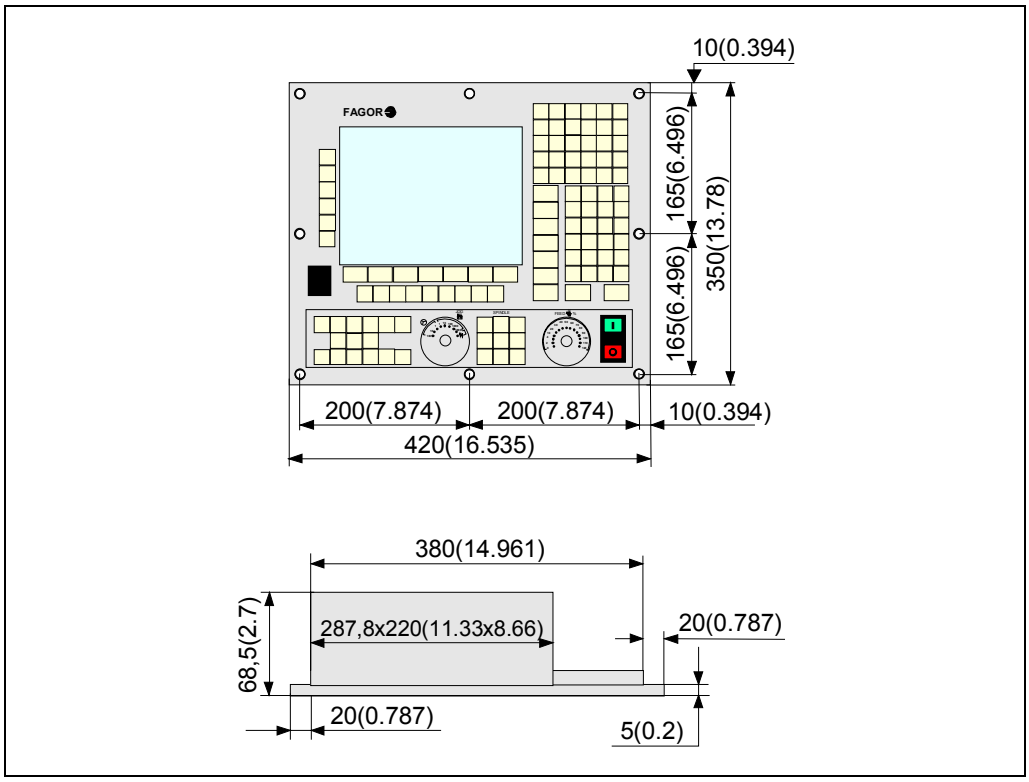
When using a fan to better ventilate the enclosure, a DC FAN must be used since an AC fan may generate electromagnetic interference resulting in distorted images being displayed by the CRT.

The temperature inside the enclosure must be between 0 and 50ºC (32ºF and 122ºF).

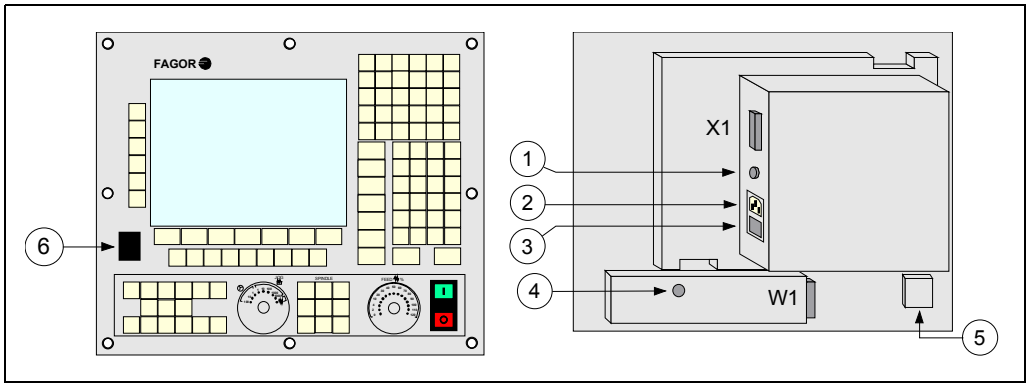
Do not get into the inside of the unit. Only personnel authorized by Fagor Automation may manipulate the inside of this unit.
Do not handle the connectors with the unit connected to main AC power. Before manipulating these connectors, make sure that the unit is not connected to main AC power.

1.3.2 11" LCD monitor + M, T, MC or TC keyboard

Dimensions



Elements

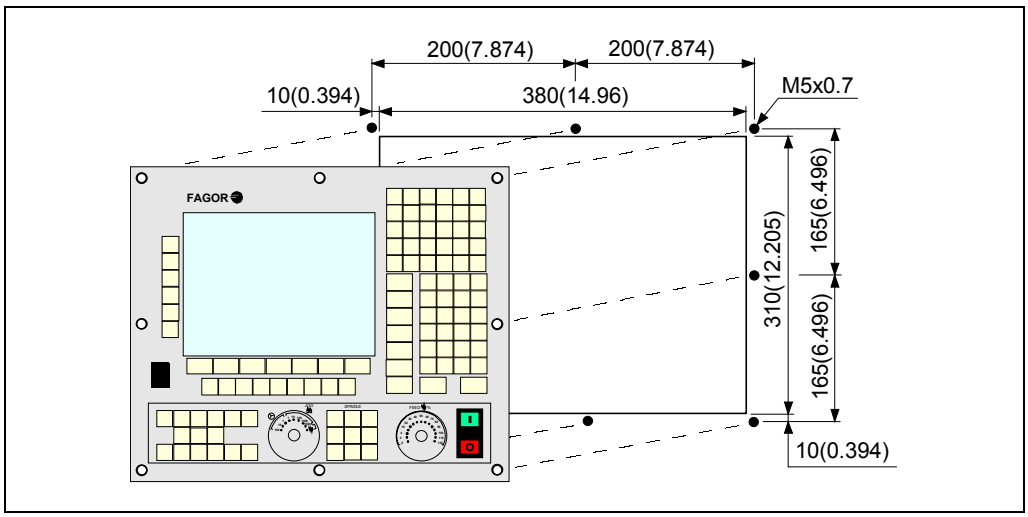


W1 SUB-D type 25-pin female connector to connect the keyboard signals.

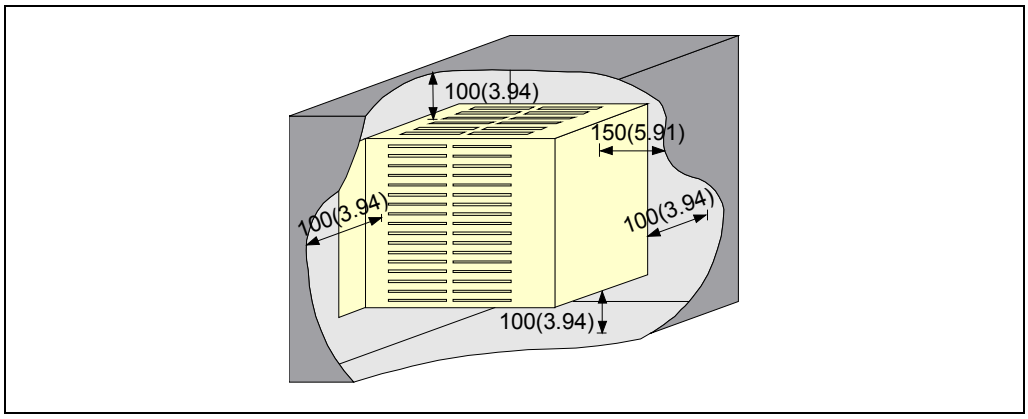
X1 SUB-D type 25-pin male connector to connect the video signals.

1. Ground terminal. All ground terminals of the machine must be connected to this terminal. Metric 6mm.
2. Mains plug. Use the connector supplied for A.C. power and ground connection.
3. On/off power switch.
4. Buzzer.
5. Type B USB connector to connect the extension cable between the CPU and the keyboard.
6. Type A USB connector to connect "pendrive" type memory devices. This connector is covered with a plug fitted into the connector, remove the plug to use the connector.

Enclosure



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



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.

The temperature inside the enclosure must be between 0 and 50ºC (32ºF and 122ºF).



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 manipulating these connectors, make sure that the unit is not connected to main AC power.

1.4 Operator panel

Independent keyboards are called operator panels. See *"1.1 CNC structure"* on page 27.

Operator panels available for the mill model

Monitor	Reference
Alphanumeric operator panel	Ref. OP.8040/55.ALFA
MC operator panel	Ref. OP.8040/55.MC
MCO operator panel	Ref. OP.8040/55.MCO/TCO

Operator panels available for the lathe model

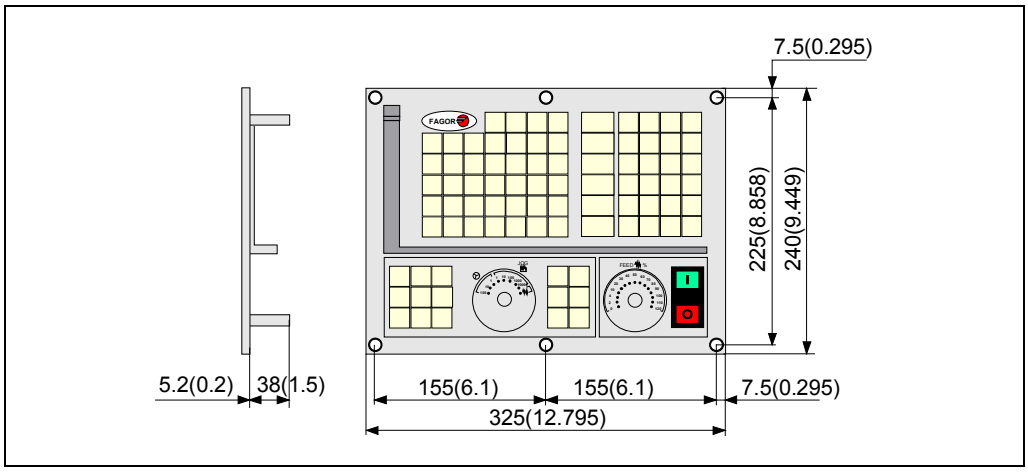
Monitor	Reference
Alphanumeric operator panel	Ref. OP.8040/55.ALFA
TC operator panel	Ref. OP.8040/55.TC
TCO operator panel	Ref. OP.8040/55.MCO/TCO

All operator panels differ only on the keyboard; they have the same size and are connected the same way with the central unit.

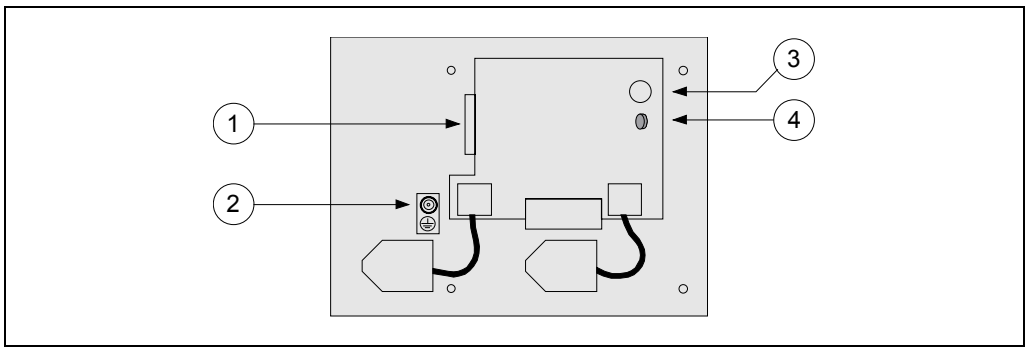
They all can have an optional alphanumeric keyboard where each key has a single letter or number assigned to it (KB.40/55.ALFA). This keyboard is connected to the operator panel through the KS50/55 adapter.

1.4.1 MC, TC, MCO/TCO and alphanumeric operator panels

Dimensions

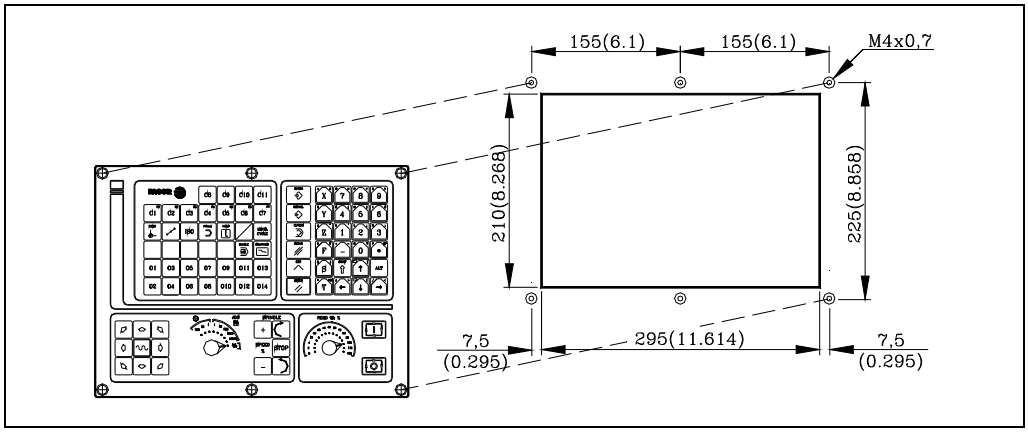


Elements



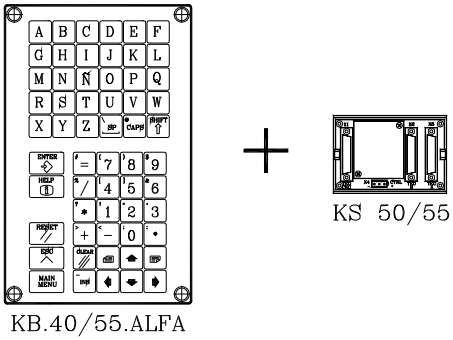
1. SUB-D type 25-pin female connector to connect the keyboard with the central unit or with a switching board.
2. Ground terminal.
3. Buzzer.
4. Potentiometer for adjusting the buzzer volume.

Enclosure

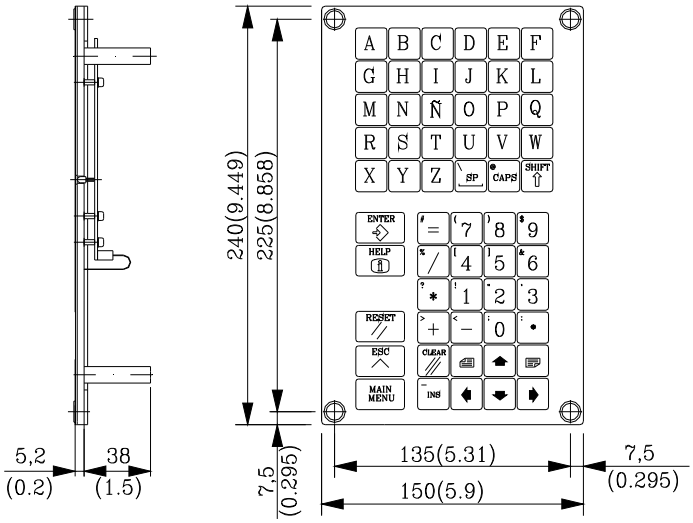


1.4.2 Alphanumeric keyboard (optional)

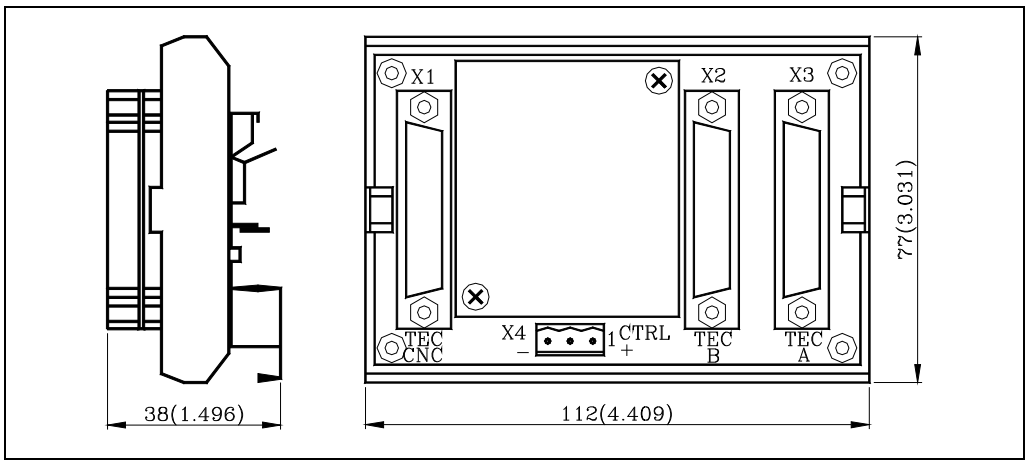
MC, TC and MCO/TCO can have an optional alphanumeric keyboard where each key has a single letter or number assigned to it (KB.40/55.ALFA). This keyboard is connected to the operator panel through the KS50/55 adapter.



Dimensions



Dimensions and elements of the KS50/55



X1 25-pin female SUB-D type (normal density) connector to connect the "central unit + monitor".

X2 25-pin female SUB-D type (normal density) connector to connect the "alphanumeric keyboard + monitor".

X3 SUB-D type 25-pin female connector to connect with the operator panel.

X4 3-pin male Phoenix connector, 7,62 mm pitch, to select the keyboard to be attended by the central unit.

If connector X4 is not supplied with voltage, the CNC attends to the operator panel.

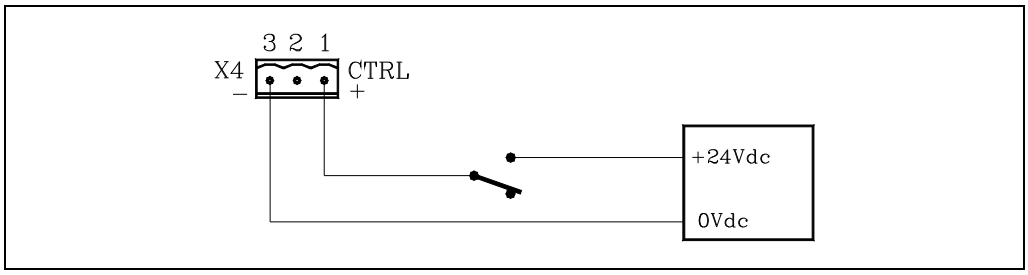
Pin	Value	Meaning
1	0 V 24 V	The CNC attends to the operator panel. The CNC attends to the alphanumeric keyboard.
2 3	- - - GND	Not being used. External power supply.

The maximum cable length allowed between the "central unit + monitor" and the "operator panel" or "alphanumeric keyboard" is 25 m (82 ft).

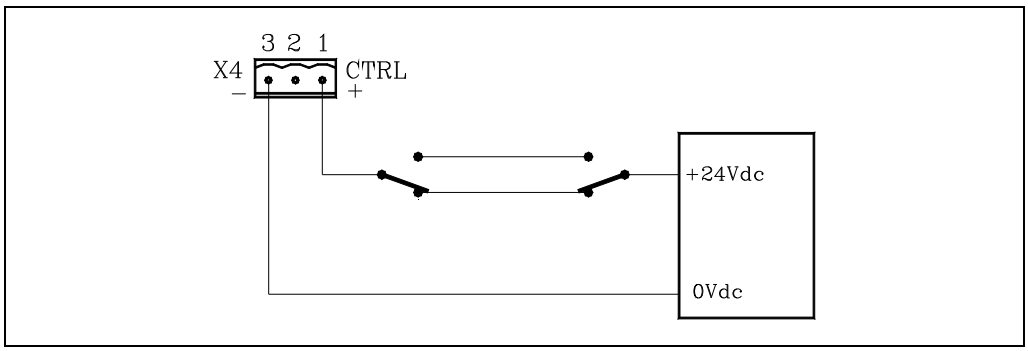
When using 2 keyboards, a keyboard switching board must be installed.

Here are some examples for keyboard selection.

With a switcher.



With two switchers.



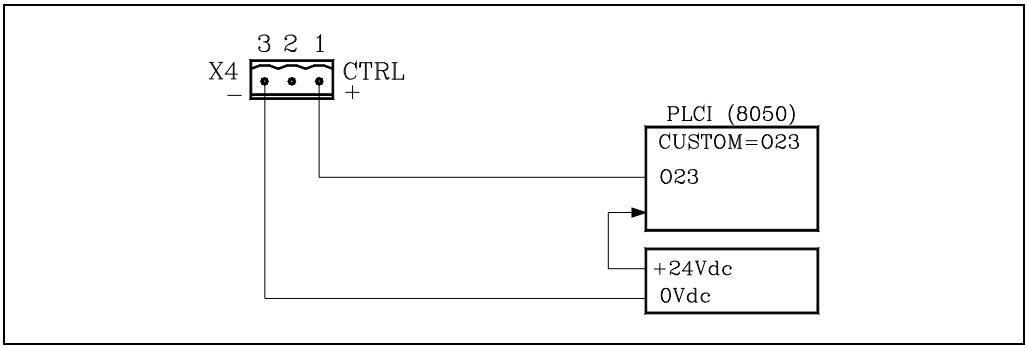
With a PLC.

CNC general logic output CUSTOM (M5512) indicates to the PLC the currently selected work mode.

CUSTOM (M5512) = 0	M or T work mode.
CUSTOM (M5512) = 1	MC, TC, MCO or TCO work mode.

If CUSTOM=O23 is programmed at the PLC, output O23 indicates the work mode selected at the CNC.

Therefore, when doing the connection of the figure, every time the work mode is changed, the corresponding keyboard will be selected.



1.

8055 CNC CONFIGURATION
Operator panel

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

CNC 8055I CONFIGURATION

2

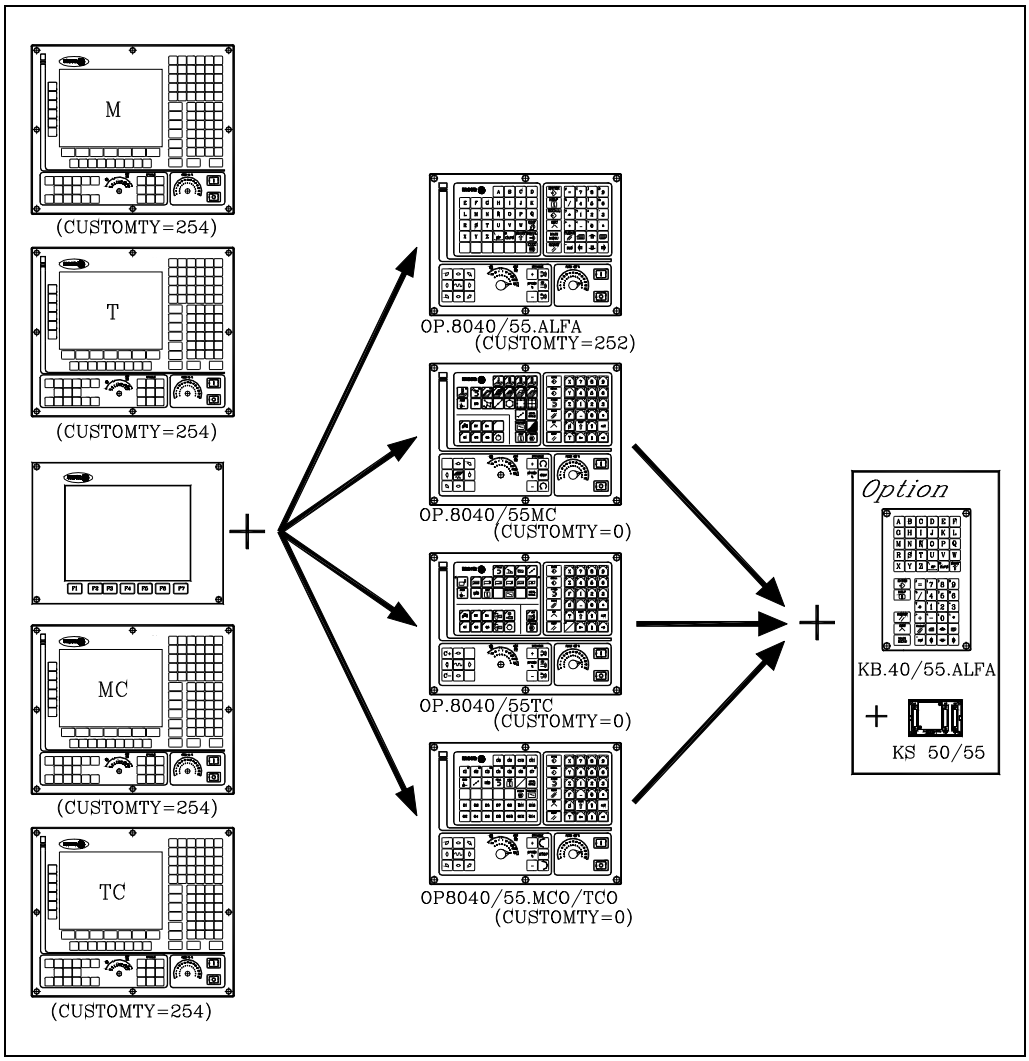
The CNC is prepared to be used in industrial environments, especially on milling machines, lathes, etc.

The CNC can control machine movements and devices.

2.1 CNC structure

The central unit is located on the rear of the monitor. The monitor may be either integrated into the operator panel or separate from it.

The following illustration shows the possible combinations. Each configuration shows the value of general machine parameter CUSTOMTY (P92).



Operator panels with integrated monitor

CNC 8055i FL-GP-K	CNC 8055i Power-GP-K
CNC 8055i FL-M-K	CNC 8055i Power-M-K
CNC 8055i FL-MC-K	CNC 8055i Power-MC-K
CNC 8055i FL-T-K	CNC 8055i Power-T-K
CNC 8055i FL-TC-K	CNC 8055i Power-TC-K

All these keyboards have an auto-identification system. See *"Keyboard auto-identification"* on page 74.

Independent monitors

CNC 8055i FL-GP	CNC 8055i Power-GP
CNC 8055i FL-M	CNC 8055i Power-M
CNC 8055i FL-T	CNC 8055i Power-T

Independent operator panels

M model	OP-8055-ALFA operator panel
T model	OP-8055-ALFA operator panel
MC model	OP-8055-MC operator panel
TC model	OP-8055-TC operator panel
MCO/TCO model	OP-8055-MCO/TCO operator panel

MC, TC and MCO/TCO operator panels can have an optional alphanumeric keyboard where each key has a single letter or number assigned to it (KB-40/55-ALFA). This keyboard is connected to the operator panel through the KS 50/55 adapter.

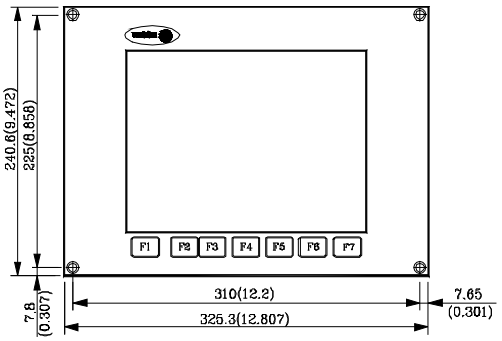
All these keyboards have an auto-identification system. See *"Keyboard auto-identification"* on page 74.

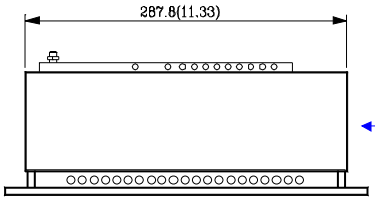
Keyboard auto-identification

Certain keyboard models have a auto-identification system. With this type of keyboards, parameter CUSTOMTY is updated automatically; for the rest of the keyboards, this parameter must be set manually.

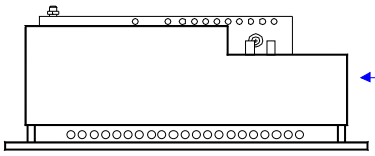
If the keyboard does not match the CNC model, it shows the relevant error message and it loads the key codes that correspond to the CNC model. For example, if a mill model keyboard is connected to a lathe CNC, the keyboard is set for a lathe and the CNC shows the error message.

Dimensions

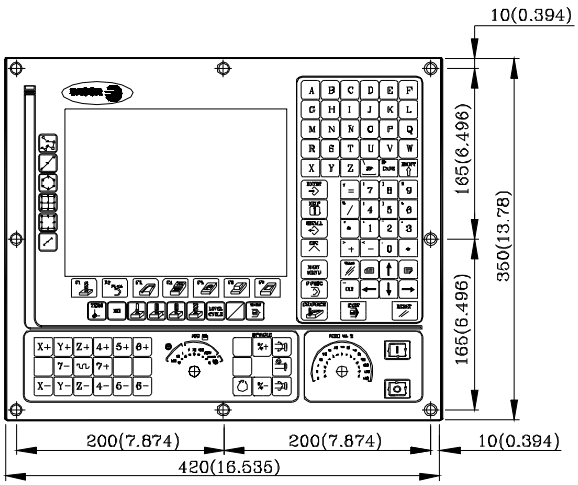


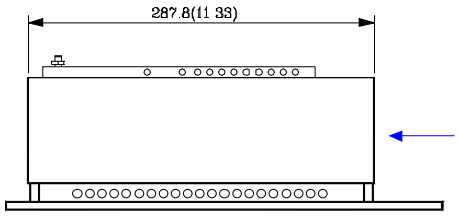


Top view of the CNC with the communications board "CAN - CAN - Ethernet".

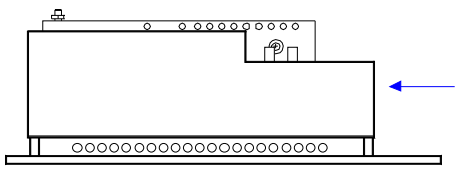


Top view of the CNC with the communications board "Sercos - CAN - Ethernet".



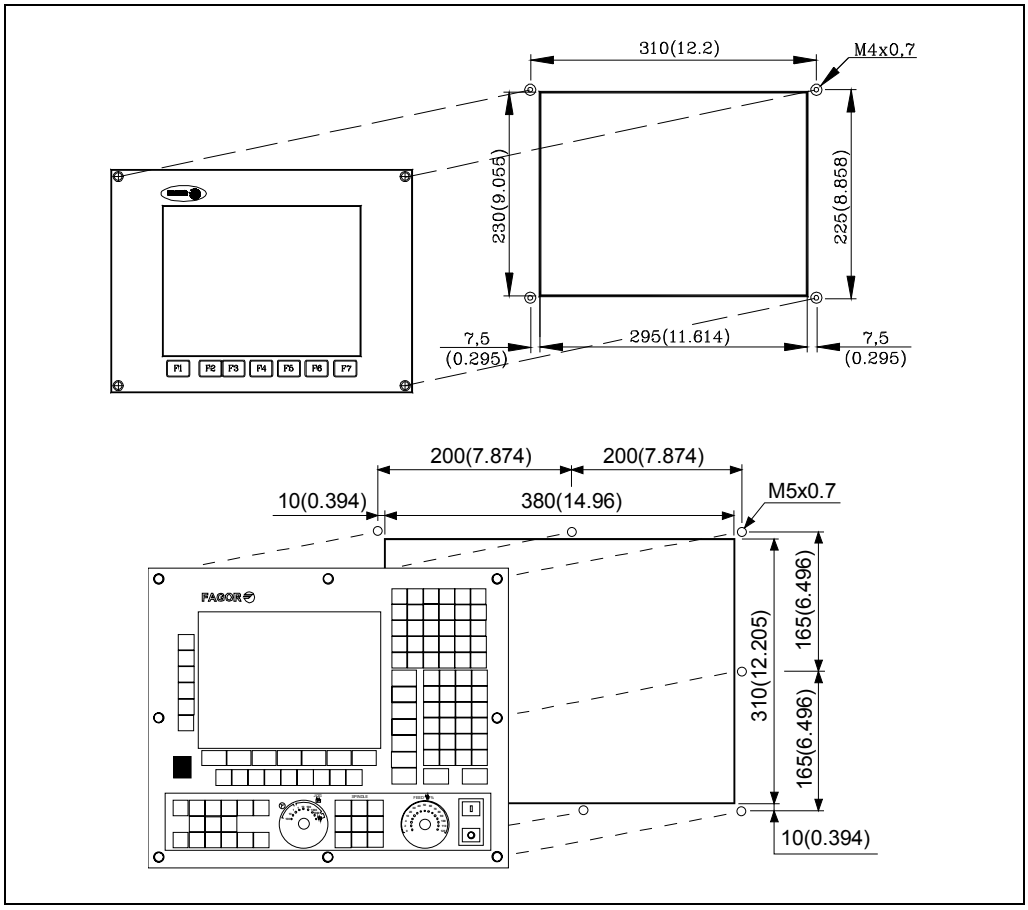


Top view of the CNC with the communications board "CAN - CAN - Ethernet".

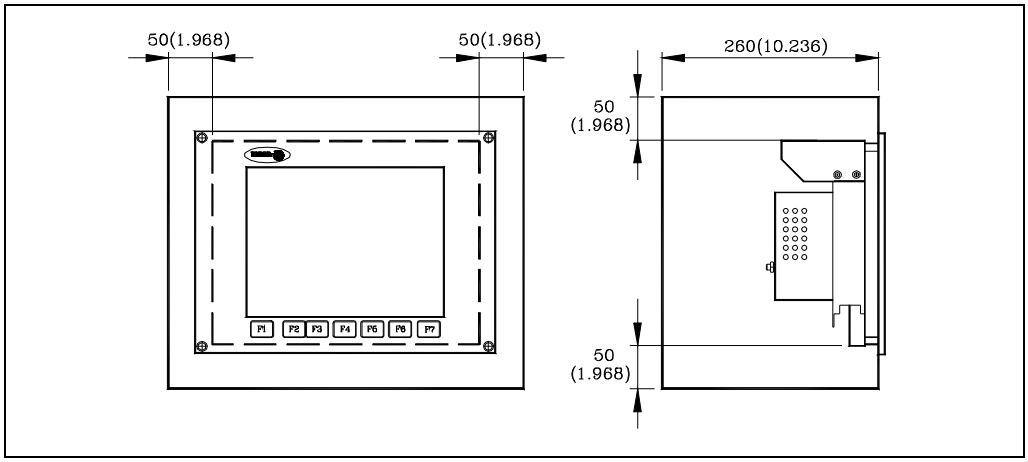


Top view of the CNC with the communications board "Sercos - CAN - Ethernet".

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)

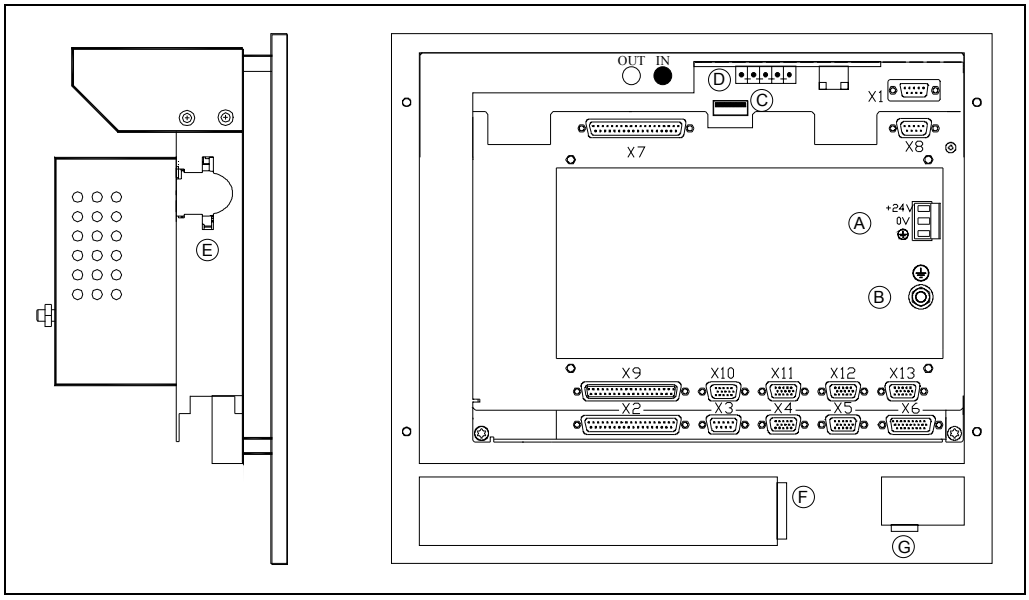
Relative humidity between 5% and 95% non condensing

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.

2.1.1 Connectors

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) or USB extension cable.
(D)	Communications board
(E)	Compact flash memory with the CNC configuration (KeyCF).
(F)	To connect the keyboard signal.
(G)	To connect the USB extension cable to the keyboard.
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 I/O connection (I97 to I104 and O33 to O56).
X8	For axis velocity command connection.
X9	For digital input connection (I65 to I96).
X10	For feedback connection of the first axis.
X11	For feedback connection of the second axis.
X12	For feedback connection of the third axis.
X13	For feedback connection of the fourth axis.

Do not get into the inside of the unit. Only authorized personnel from Fagor Automation may do it.
Do not handle the connectors with the unit connected to main AC power. Before manipulating these connectors, make sure that the unit is not connected to main AC power.
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.

Protection at the connectors

It detects over-currents or short-circuits at the +5 V of the feedback of the handwheels, spindle and probe and it issues the relevant error message.

Besides this, it also detects the 24V of the external supply at the digital outputs.

Hardware protections

The axis board installed in the CNC includes the recognition of the 24V at the inputs and outputs.

Signal adapters

The following signal adapters are available to be used with the feedback inputs.

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

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).

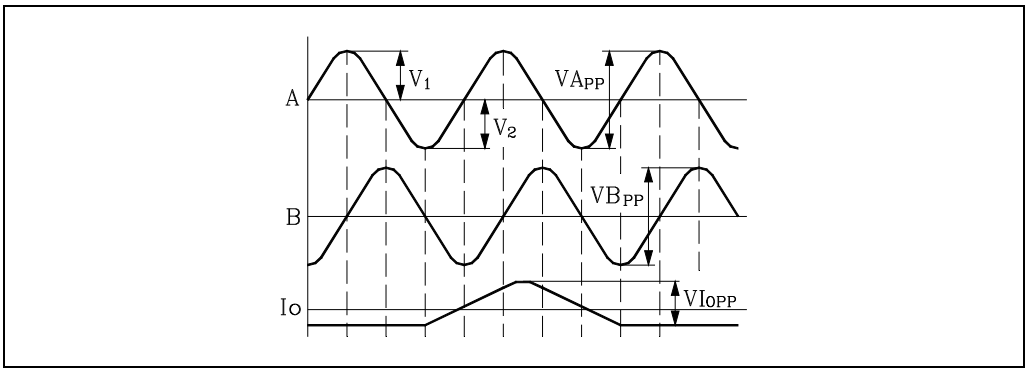
Work levels for differential square signal (axes and spindle).

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:	400 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.

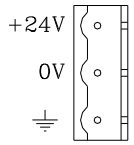
Work levels for non-differential square signal.

Maximum frequency:	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.

Connectors and connection

Power supply

3-prong male Phoenix connector, 7.65 mm pitch.

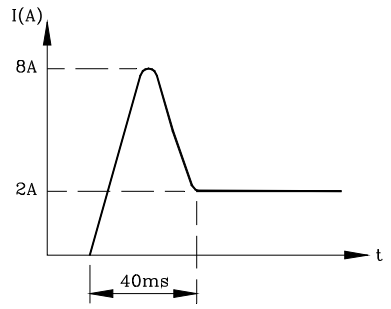
	Pin	Signal and function	
	1	+ 24 V	Power supply.
	2	0 V	Power supply.
	3	Chassis	Shield.

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.

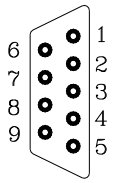
The supply current has the following shape on power-up:



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	DSR
	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.

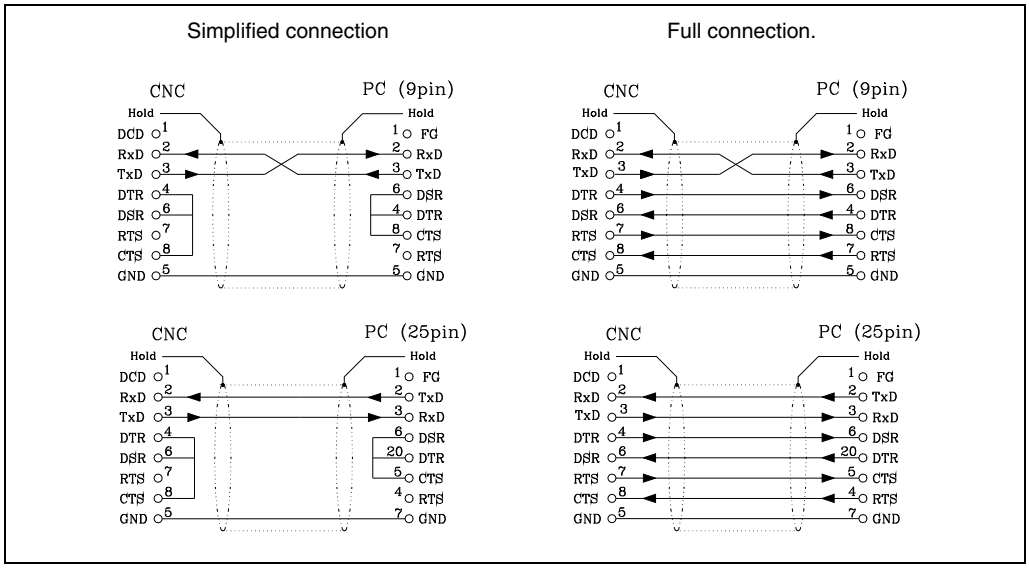
The CNC can operate at up to 115,200 Baud.

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 (-GND- pin) thus, avoiding reference points at different voltages especially in long cables.

Recommended RS232C interface connection



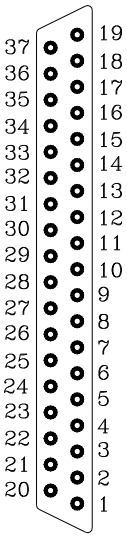
Connector X2 - Digital inputs (I1 to I16) and digital outputs (O1 to O8)

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.

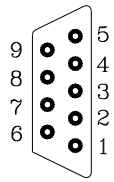
The emergency output, which coincides with O1 of the PLC, will be activated (change from logic level 1 to 0) when an ALARM or ERROR occurs at the CNC or when the PLC output O1 is set to 0 (logic level 0).



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	/ Emergency input.
	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			
	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). See *"Technical characteristics of the 8055iCNC"* on page 621.

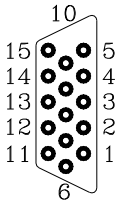
The appendix of this manual describes the recommended connection circuits. See *"Probe connection at the 8055i"* on page 631.

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).

The 5 V outputs are protected against short-circuit.

Connector X4 - For analog spindle connection

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

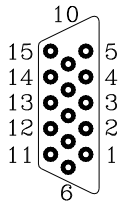
Pin			
	1	A	Feedback signals.
	2	/A	
	3	B	
	4	/B	
	5	I0	
	6	/I0	
	7	- - -	
	8	- - -	
	9	+5 V	+5 V output for feedback.
	10	ana_out	Velocity command output.
	11	GND	0 V output for feedback.
	12	GND	0 V output for velocity command.
	13	- - -	
	14	- - -	
	15	Chassis	Shield.

It admits differential TTL and 1Vpp sinusoidal feedback signals.

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 2 3 4	A1 /A1 B1 /B1	Feedback signals of first handwheel.
	5 6 7 8	A2 /A2 B2 /B2	Feedback signals of second handwheel.
	9 10 11 12 13 14 15	+5 V +5 V GND GND 100P - - - Chassis	Supply output. Supply output. Supply output. Supply output. Push button of Fagor 100P handwheel. 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 union cable required for this connection. It consists of a cable hose, one high density SUB-D type 26-pin male connector and a SUB-D type 25-pin male connector.

Both connectors have a latching system by means of two screws UNC4.40. The cable hose shield is soldered to the metal hoods covering both connectors.

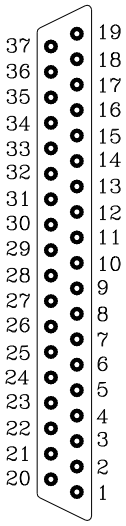
Cable connection.

Connector of the operator panel (25 pin)	CNC connector X6 (26 pin)
1	1
2	13
3	5
4	23
5	15
6	7
7	25
8	17
9	9
10	19
11	11
12	3
13	21
14	4
15	22
16	14
17	6
18	24
19	16
20	8
21	26
22	10
23	2
24	20
25	12

Connector X7 - Digital inputs (I97 to I104) and digital outputs (O33 to O56)

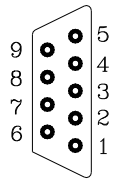
It is a 37-pin normal density SUB-D type female connector.

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 2 3 4 5	24 V O33 O35 O37 O39	External power supply.
	6 7 8 9 10	O41 O43 O45 O47 O49	
	11 12 13 14 15	O51 O53 O55 I97 I99	
	16 17 18 19	I101 I103 0 V 0 V	 External power supply. External power supply.
	20 21 22 23 24	24 V O34 O36 O38 O40	External power supply.
	25 26 27 28 29	O42 O44 O46 O48 O50	
	30 31 32 33 34	O52 O54 O56 I98 I100	
	35 36 37	I102 I104 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	Velocity command output for the fourth axis.
	6	GND	Velocity command reference signals.
	7	GND	
	8	GND	
	9	GND	

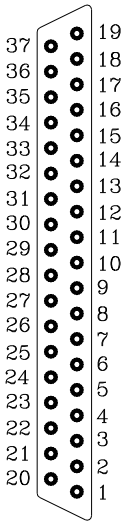
The cable shield must be connected to the metallic hood at each end.

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

Connector X9 - Digital inputs (I65 to I96)

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	I89	
	15	I91	
	16	I93	
	17	I95	
	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	I90	
	34	I92	
	35	I94	
	36	I96	
	37	Chassis	Shield.

Conectores X10, X11, X12, X13 - Feedback inputs for the axes

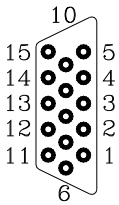
X10 For feedback connection of the first axis.

X11 For feedback connection of the second axis.

X12 For feedback connection of the third axis.

X13 For feedback connection of the fourth 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.

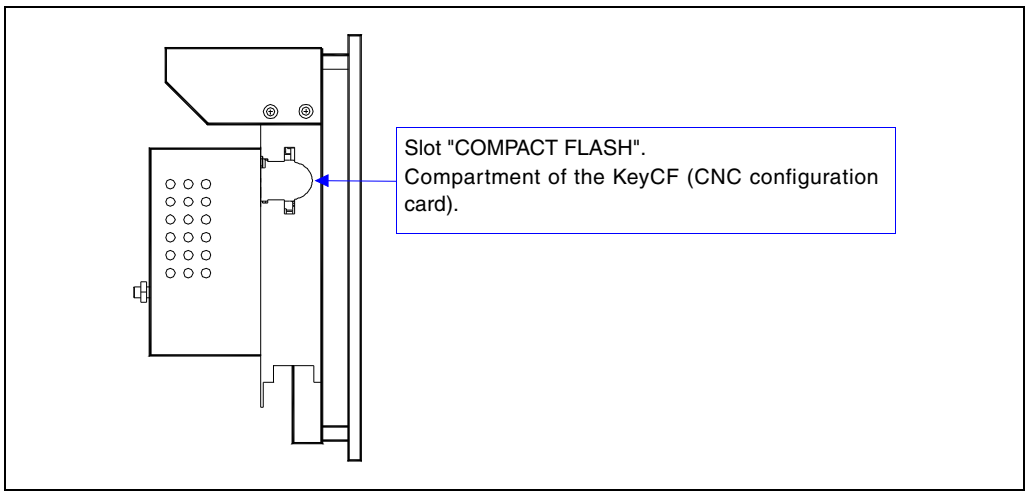
Protection at the connectors

It detects over-current or short-circuits at the feedback devices issuing the relevant error message.

"Supply voltage error on axis feedback *".

Slot "COMPACT FLASH" - Compartment of the KeyCF (CNC configuration card)

The COMPACT FLASH slot is located on the left side of the CNC. This slot is used for the KeyCF and for upgrading software versions.



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

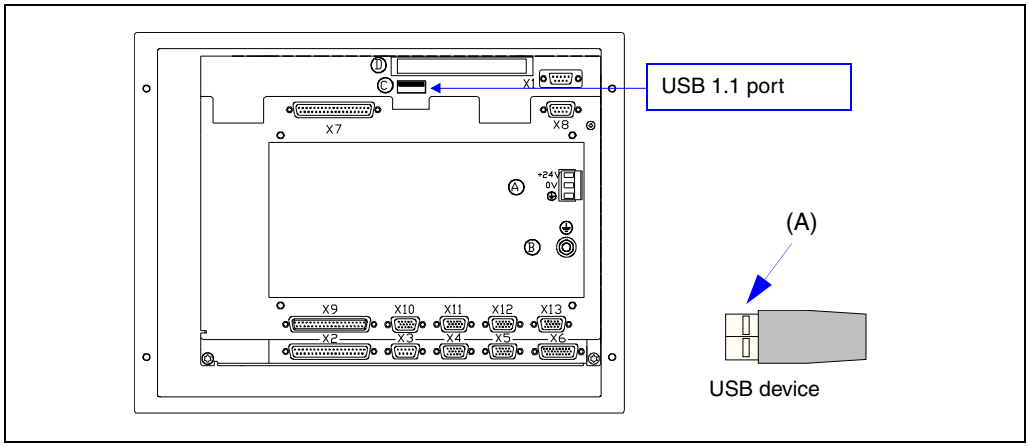
- The card id (all the cards are different).
- The software features that have been purchased for that unit

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 CNC will recognize the KeyCF as <Hard Disk>, it can also be accessed from the CNC.

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

The USB 1.1 port with type A connector, it admits connecting a "Pendrive" 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.



Use a USB extension cable to prevent undesired short-circuits with the metal housing (A) of the USB device at the CNC. The extension cable must not be longer than 3 m and it must be plugged in while the CNC is off.

We recommend to use the extension set supplied by Fagor.

Once this cable has been plugged in, USB devices may be connected or disconnected through it while the CNC is on.

The CNC recognizes the connected device as USB Hard Disk. 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.

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.

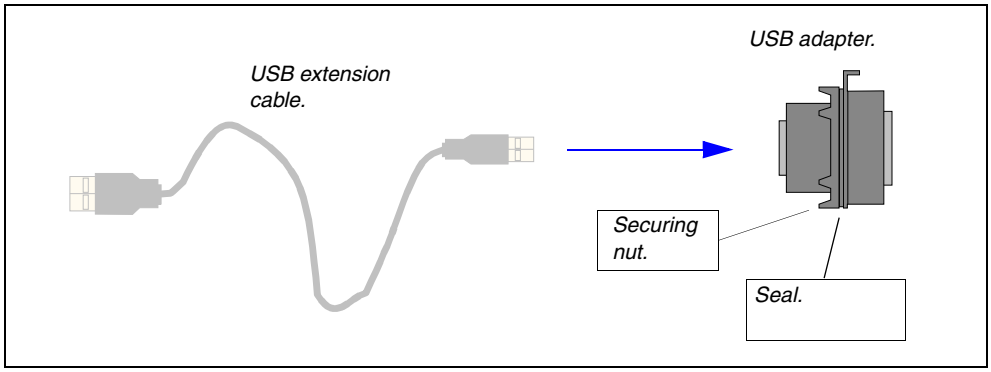
Within the USB device, the CNC will only recognize files with extensions *.f55 (software version), *fhw (firmware update files), part-programs, parameters, tables, pages (screens) and symbols. The CNC will not recognize any other type of file.

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

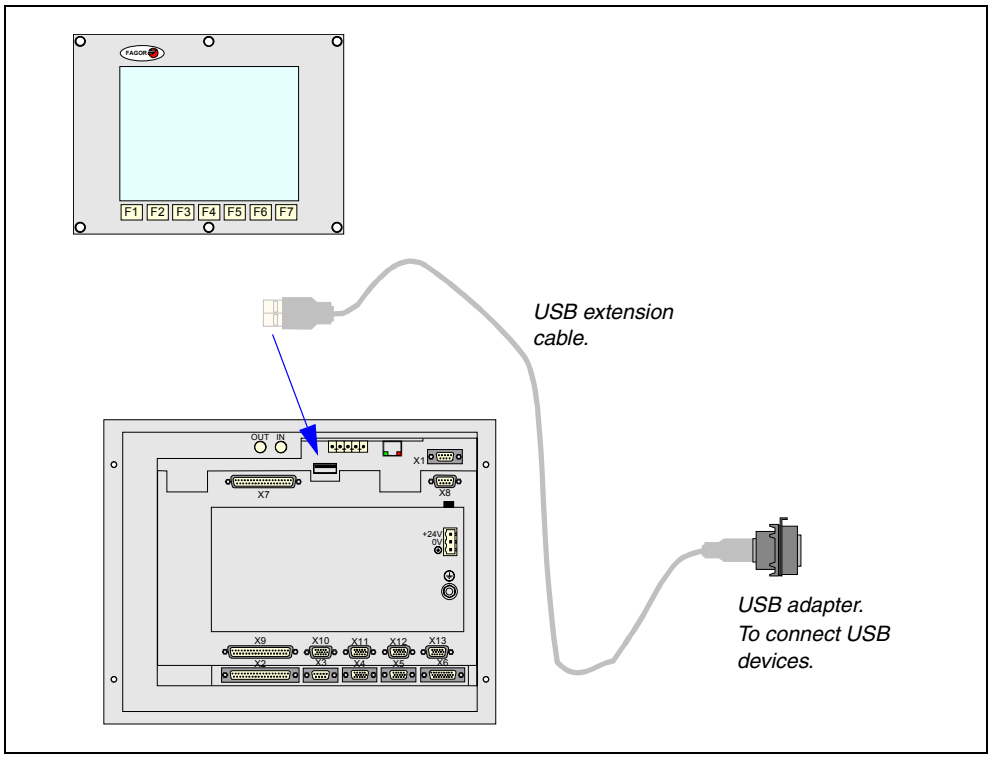
Separate monitors (without keyboard)

Connecting the USB extension set supplied by Fagor:

1. Connect the cable and the USB adapter. Check that the seal and the nut of the USB adaptor are secured as shown in the figure.



2. While the CNC is off, connect the extension cable to the USB connector of the CNC.



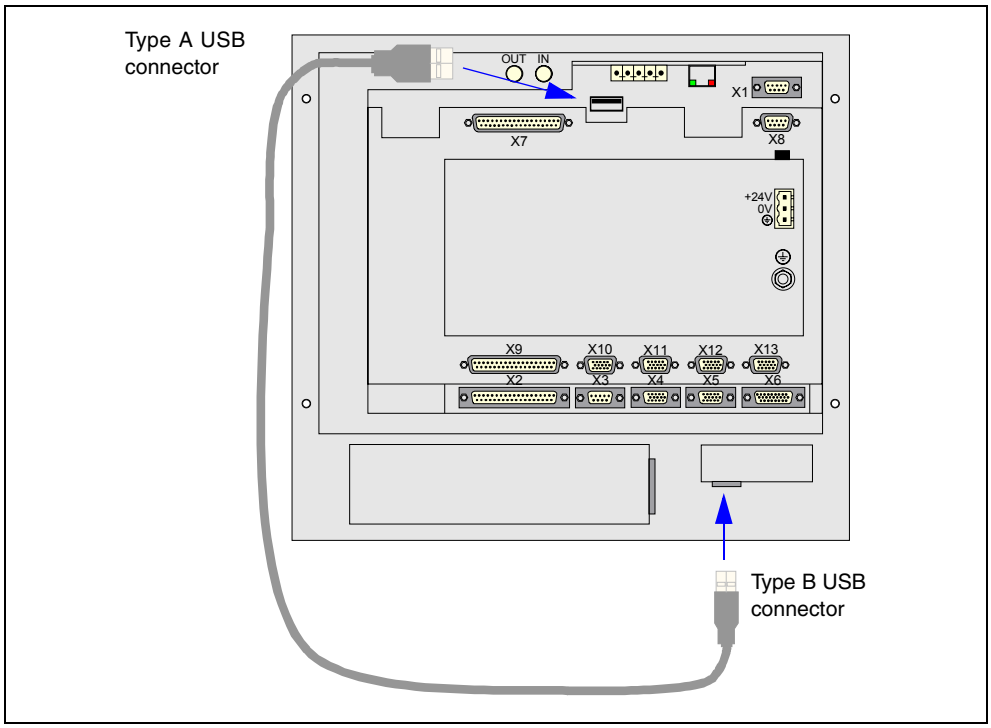
3. Once the extension set has been properly connected, it will be possible to connect and disconnect USB devices to the CNC while it is on.

Monitors with keyboard

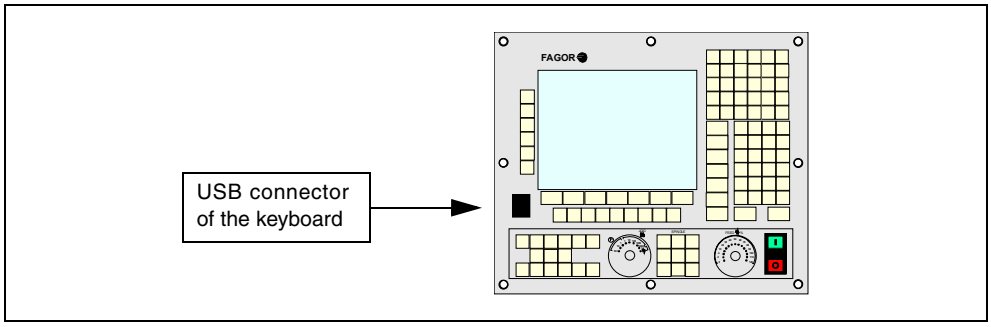
Connecting the USB extension cable supplied by Fagor:

The USB extension cable for monitors with an integrated keyboard is supplied with the CNC.

1. Connect the extension cable as shown in the image while the CNC is off.



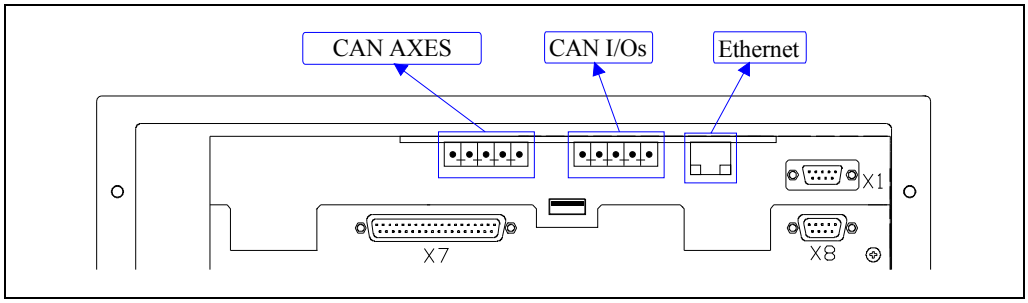
2. Once the extension cable is connected, it will be possible to USB connector of the front of the keyboard, while the CNC is on.



Sercos, CAN and Ethernet - communications board

There are 3 communications boards:

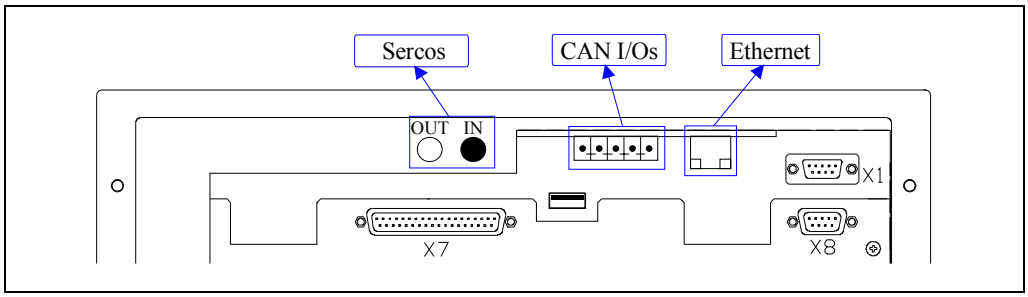
CAN - CAN - Ethernet.



This board has the following connections:

- CAN servo system bus.
- Bus CAN Open for remote digital I/O
- Ethernet bus

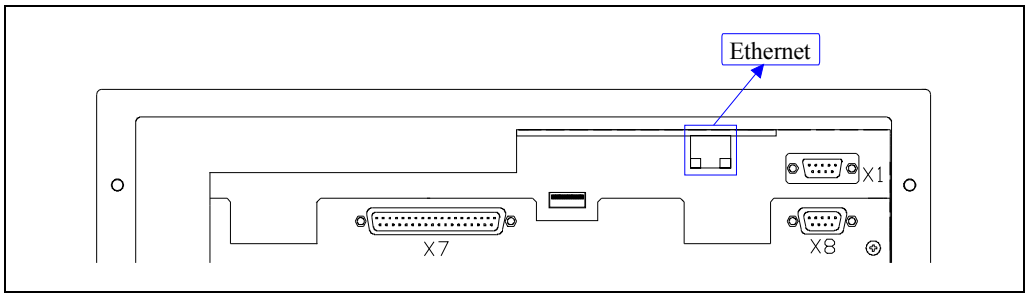
Sercos - CAN - Ethernet.



This board has the following connections:

- Sercos servo system bus.
- Bus CAN Open for remote digital I/O
- Ethernet bus

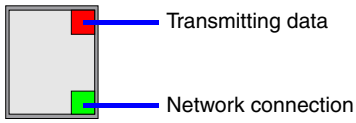
Ethernet.



This board has the following connection:

- 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 or carrying out telediagnostic tasks.

The Ethernet option does not require having the DNC option available.

The Ethernet card has an RJ-45 connector and two LED's that inform on the status of the connection.

Red LED	Blinks while transmitting data.
Green LED	On while connected to the network.

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.

CAN I/Os - Expansion of inputs and outputs (remote I/O)

The CAN bus connection allows connecting up to 4 remote modules to the central unit to expand the number of digital or analog inputs and outputs,

The transmission speed depends on the length of the cable or total CAN connection distance. Machine parameter IOCANSPE (P88) must be set.

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.

The CNC's communications board does not have a switch and always assumes position 0 inside the bus. The rest of the modules occupy consecutive positions starting from ·1·.

Although both CAN buses are independent, the CAN addresses of the drives cannot be the same as the CAN addresses of the I/O modules. If address ·1· is used at the drive's CAN bus, there cannot be any module with that address in the CAN bus of the I/O's.

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.

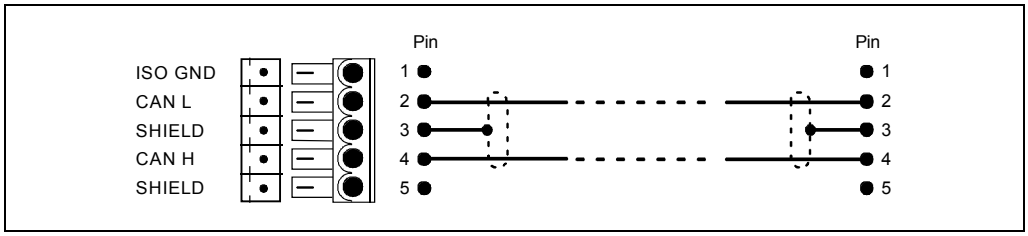
The switch position of the terminating elements must be "1" and that of the rest of the elements "0".

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 CNC does not have a "Line_Term" switch and always has the terminating resistor activated.

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.

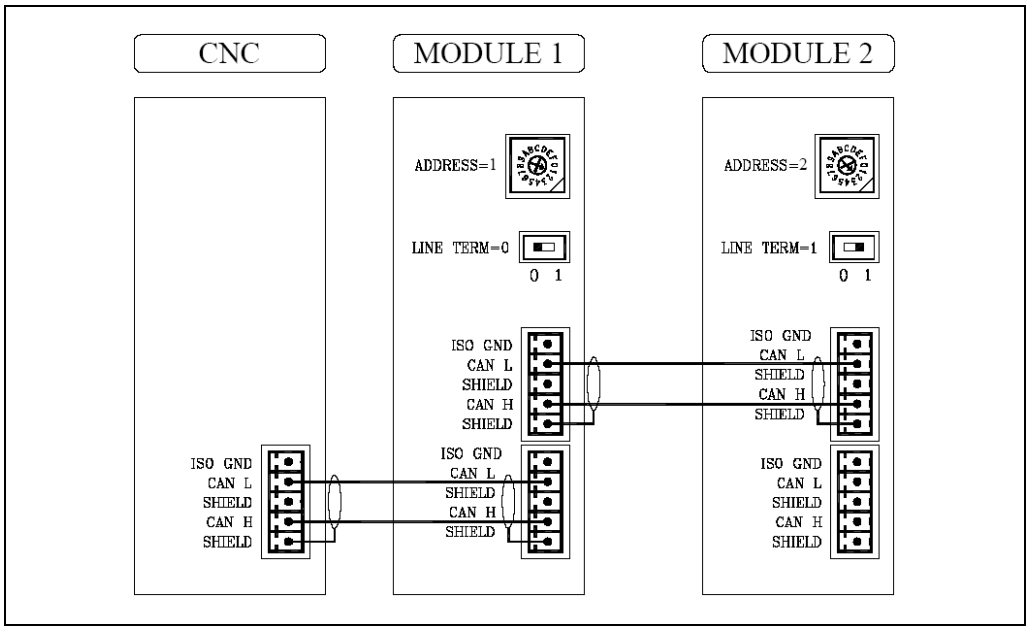
Characteristics of the CAN cable.

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).

Interconnection of modules.

It is a serial connection and any of the two connectors may be used. The figure shows the CAN connection between the central unit and 2 groups of remote modules.



DIGITAL DRIVES - Digital servo (CAN or Sercos)

Two types of digital servo system may be used to communicate with Fagor servo drives:

- Sercos interface IEC1491.
- CAN field bus and standard CanOpen communication protocol. In general, their characteristics are similar to those of the Sercos ring with lower transmission speed.

The same system may have digital axes (CAN or Sercos) and analog axes. On the other hand, it is not possible to have digital axes with Sercos interface and CAN interface at the same time.

Digital CAN servo

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.

Although both CAN buses are independent, the CAN addresses of the drives cannot be the same as the CAN addresses of the I/O modules. If address ·1· is used at the drive's CAN bus, there cannot be any module with that address in the CAN bus of the I/O's.

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.

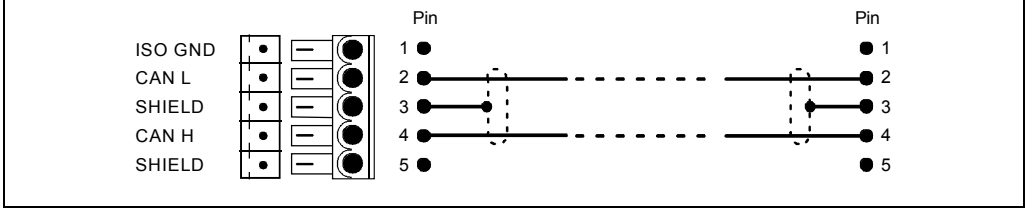
Characteristics of the CAN cable

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).

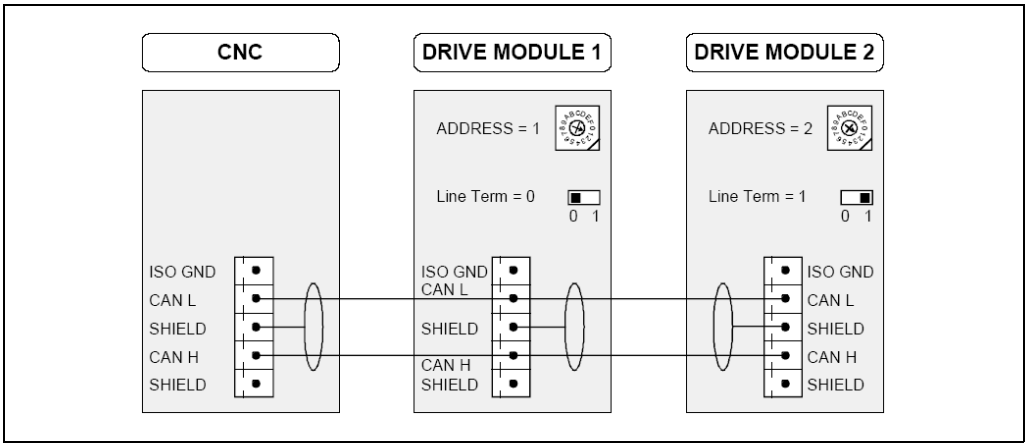


Signal	Description
ISO GND	Ground / 0 V.
CAN L	Bus signal (LOW).
SHIELD	CAN shield.
CAN H	Bus signal (HIGH).
SHIELD	CAN shield.

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.



Sercos digital servo system

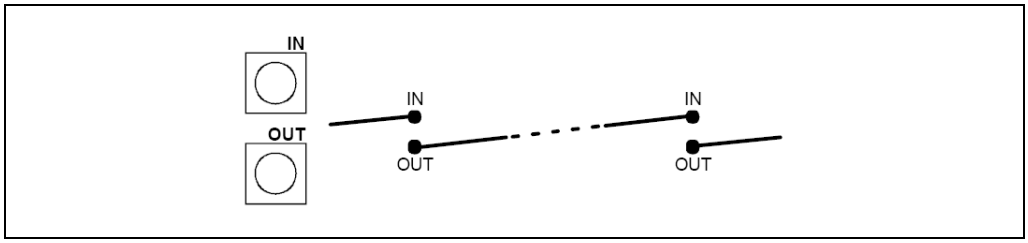
Module identification at the bus.

Each element integrated into the Sercos bus is identified with the 16-position rotary switch (0-15) "Address" (also known as Node_Select). This rotary switch selects the address (node) occupied by each element integrated in the bus.

The CNC's communications board does not have a switch and always assumes position 0 inside the bus. The rest of the modules occupy consecutive positions starting from ·1·.

The fact that the drive identified with number 1 (for example) corresponds to the X axis, to Y or to another one is irrelevant. However, it is a good idea, for the sake of clarity, that the machine axes X, Y, Z, U, V, W, A, B and C are numbered sequentially in that order.

Sercos connector pinout.



Signal	Description
IN	Sercos connection input.
OUT	Sercos connection output.

Sercos cable characteristics.

Fagor Automation provides the fiber optic cables needed for Sercos communication. There are different types of cables depending on length and dynamic and static installation characteristics.

- Fiber optic cable with polymer core (SFO, SFO-FLEX) for up to 40 meters If the cable will be subject to dynamic conditions (it will be moving), the SFO-FLEX cable must always be used. If the cable will be subject to static conditions (it will not be moving), the SFO cable will be enough.
- Fiber optic cable with glass core (SFO-V-FLEX) for more than 40 meters.

Mechanical characteristics of the cables.

SFO cable	
Flexibility.	Normal. Minimum bending radius: 30 mm. Only to be used when the cable will not be moving (static conditions).
Covering.	PUR. Polyurethane resistant to chemical used on machines.
Temperature.	Work: -20 ºC / 80 ºC (-4 ºF / 176 ºF). Storage: -35 ºC / 85 ºC (-31 ºF / 158 ºF).

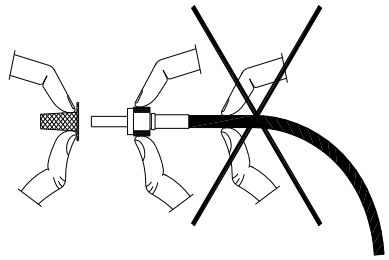
SFO-FLEX cable	
Flexibility.	High. Minimum static bending radius of 50 mm and a dynamic radius of 70 mm. Special cables to be used in cable carrying chains.
Covering.	PUR. Polyurethane resistant to chemical used on machines.
Temperature.	Work: -20 ºC / 70 ºC (-4 ºF / 158 ºF). Storage: -40 ºC / 80 ºC (-40 ºF / 176 ºF).

SFO-V-FLEX cable	
Flexibility.	Minimum bending radius: 16 mm. Special cables to be used in cable carrying chains.
Temperature.	Work: -65 ºC / 125 ºC (-85 ºF / 257 ºF).

Cable handling.

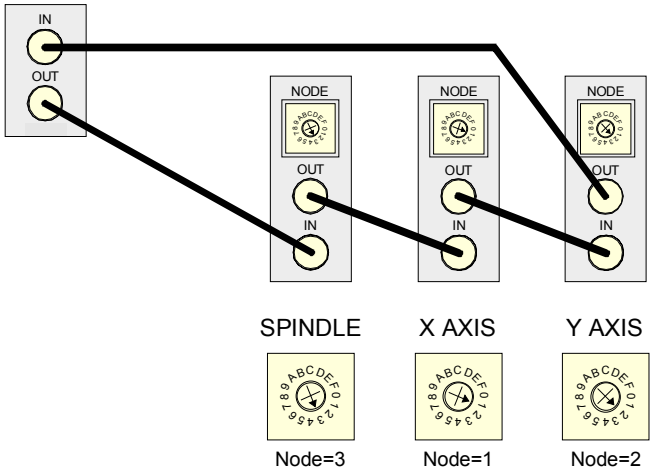
Fagor supplies the cable with the terminals protected by a cap. Remove the protection cap before connecting the cable.

To remove the terminal protecting cap or to connect or disconnect the cable, hold the cable by its terminal; never pull the cable by holding its plastic part because it could render it useless.



Interconnection of modules.

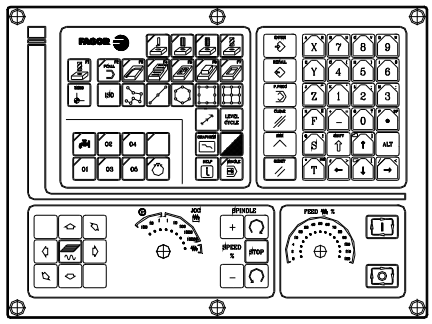
It is connected in a ring through optic fiber, by joining an OUT terminal with an IN terminal. The figure shows the Sercos connection of the CNC with Fagor spindle drives and axes X and Y.



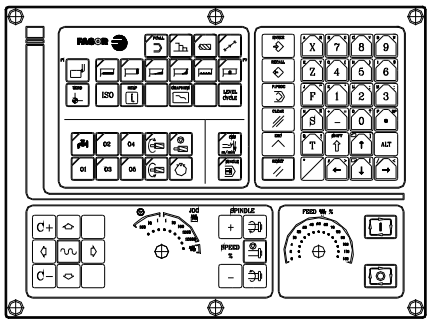
2.2 Operator panel

The operator panel depends on the CNC model.

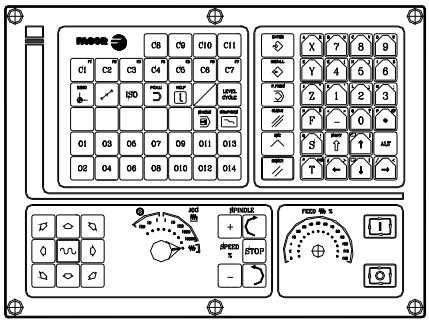
M model	OP-8055-ALFA operator panel
T model	OP-8055-ALFA operator panel
MC model	OP-8055-MC operator panel
TC model	OP-8055-TC operator panel
MCO/TCO model	OP-8055-MCO/TCO operator panel



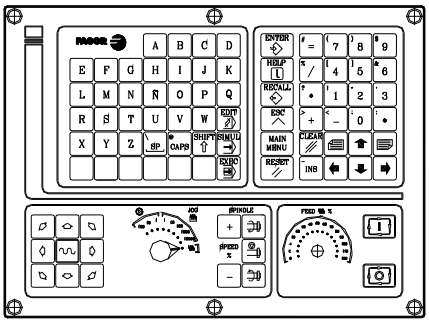
OP.8040/55.MC



OP.8040/55.TC

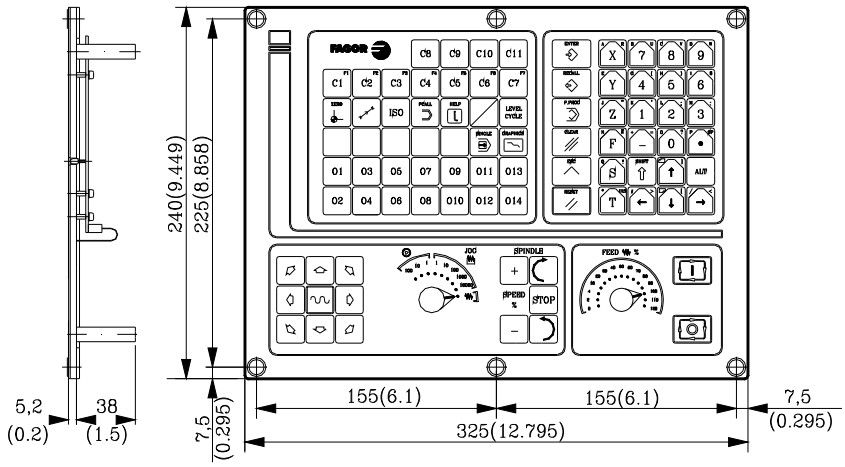


OP.8040/55.MCO/TCO

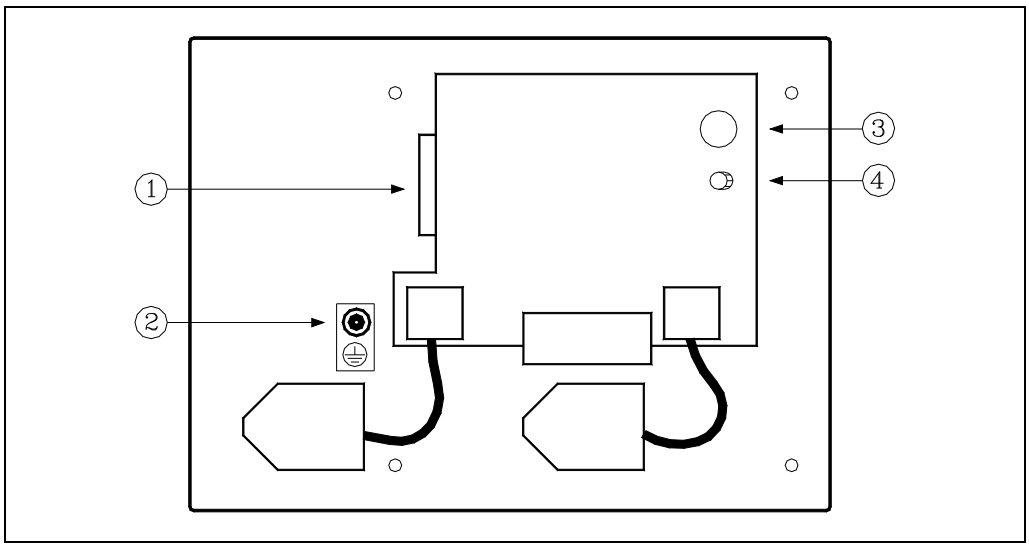


OP.8040/55.ALFA

Dimensions

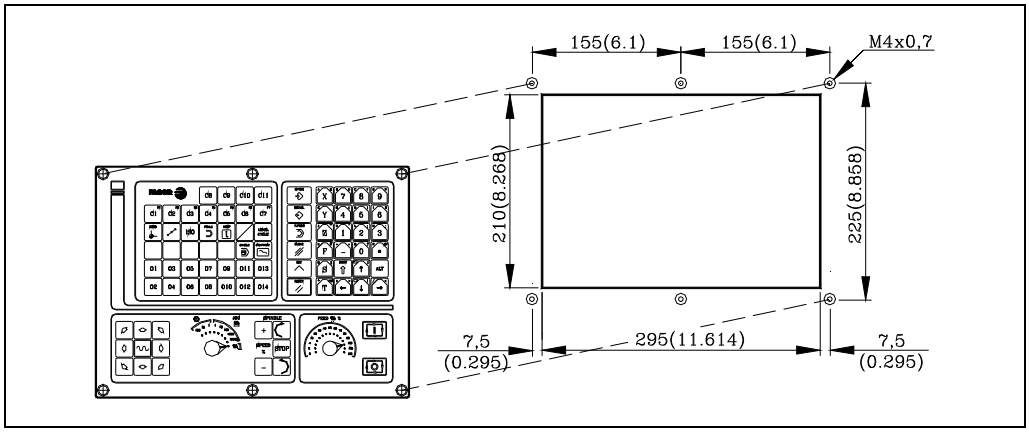


Elements



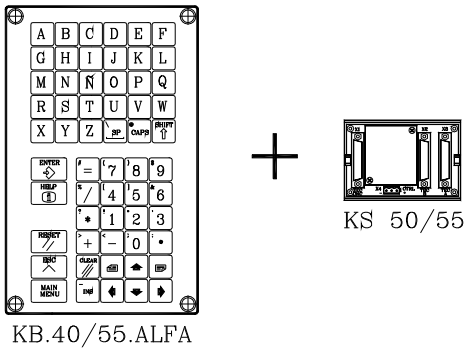
1. SUB-D type 25-pin female connector to connect the keyboard with the central unit or with a switching board.
2. Ground terminal.
3. Buzzer.
4. Potentiometer for adjusting the buzzer volume.

Enclosure

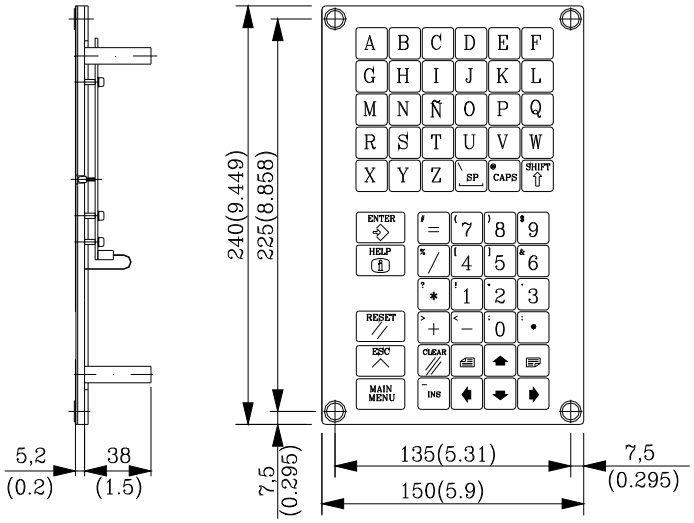


2.2.1 Alphanumeric keyboard (optional)

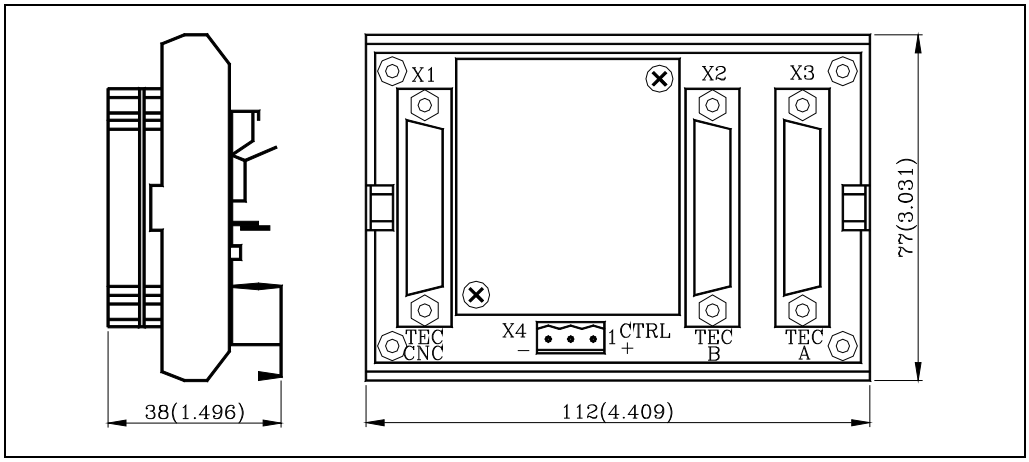
MC, TC and MCO/TCO can have an optional alphanumeric keyboard where each key has a single letter or number assigned to it (KB-55-ALFA). This keyboard is connected to the operator panel through the KS 50/55 adapter.



Dimensions



Dimensions and elements of the KS 50/55



X1 25-pin female SUB-D type (normal density) connector to connect the "central unit + monitor".

X2 25-pin female SUB-D type (normal density) connector to connect the "alphanumeric keyboard + monitor".

X3 SUB-D type 25-pin female connector to connect with the operator panel.

X4 3-pin male Phoenix connector, 7,62 mm pitch, to select the keyboard to be attended by the central unit.

If connector X4 is not supplied with voltage, the CNC attends to the operator panel.

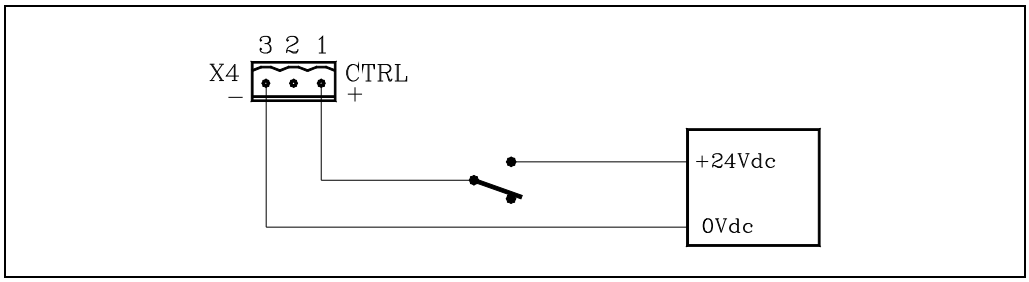
Pin	Value	Meaning
1	0 V 24 V	The CNC attends to the operator panel. The CNC attends to the alphanumeric keyboard.
2 3	- - - GND	Not being used. External power supply.

The maximum cable length allowed between the "central unit + monitor" and the "operator panel" or "alphanumeric keyboard" is 25 m (82 ft).

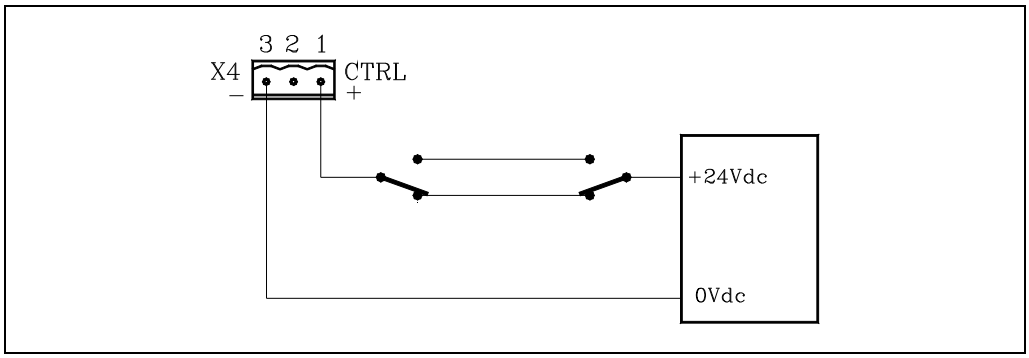
When using 2 keyboards, a keyboard switching board must be installed.

Here are some examples for keyboard selection.

With a switcher.



With two switchers.



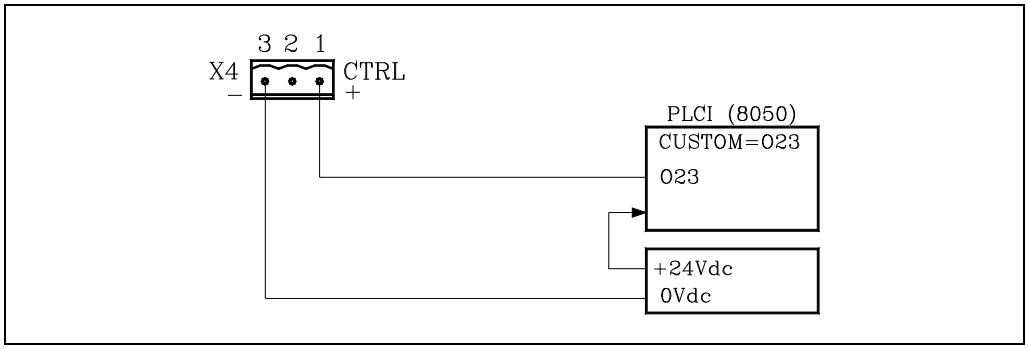
With a PLC.

CNC general logic output CUSTOM (M5512) indicates to the PLC the currently selected work mode.

CUSTOM (M5512) = 0	M or T work mode.
CUSTOM (M5512) = 1	MC, TC, MCO or TCO work mode.

If CUSTOM=O23 is programmed at the PLC, output O23 indicates the work mode selected at the CNC.

Therefore, when doing the connection of the figure, every time the work mode is changed, the corresponding keyboard will be selected.



HEAT DISSIPATION

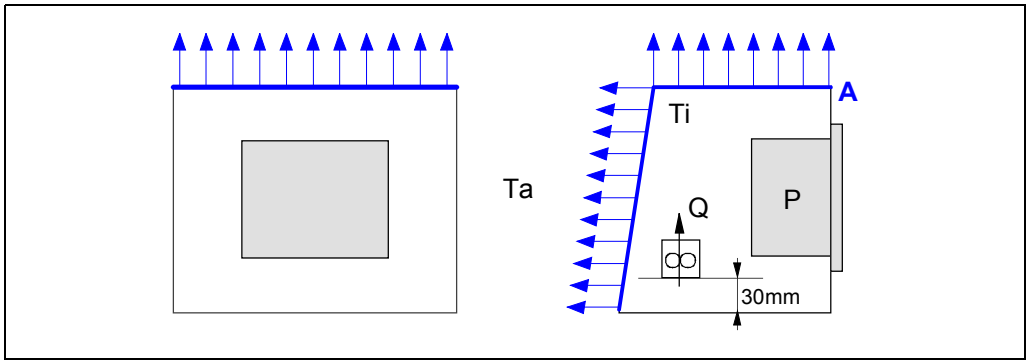
3

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.

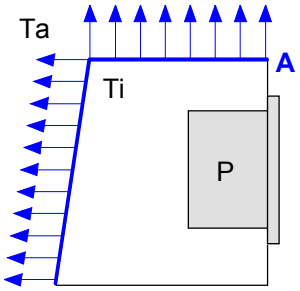


A	(m^2)	Total surface required.
P	(W)	Total power dissipated by all the elements that generate heat inside the enclosure, including the power supply and the fan if there is one.
Ta	(ºC)	Ambient temperature outside the enclosure.
Ti	(ºC)	Temperature inside the enclosure.
Δt	(ºC)	Temperature difference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

3.1 Heat dissipation by natural convection



Surface without paint.

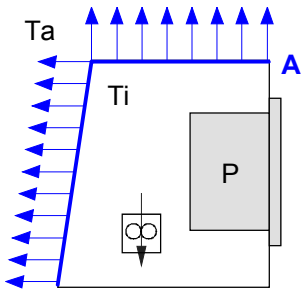
$$A = \frac{P}{5 \cdot \Delta T}$$

Surface with smooth metallic enamel.

$$A = \frac{P}{5.7 \cdot \Delta T}$$

3.2 Heat dissipation by forced convection with inside fan

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



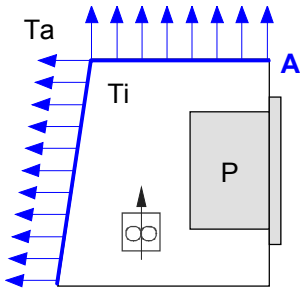
Surface without paint.

$$A = \frac{P}{5.6 \cdot \Delta T}$$

Surface with smooth metallic enamel.

$$A = \frac{P}{7.6 \cdot \Delta T}$$

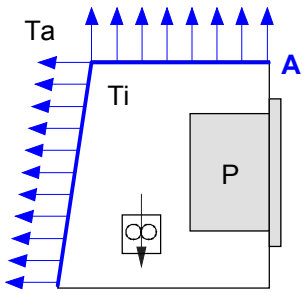
Fan whose air flow is Q = 13.6 m^3/h facing up.



Surface without paint.

$$A = \frac{P}{5.8 \cdot \Delta T}$$

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



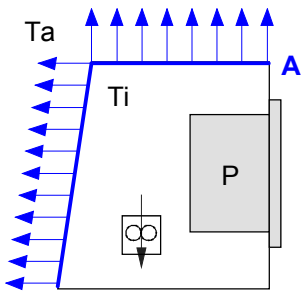
Surface without paint.

$$A = \frac{P}{6.75 \cdot \Delta T}$$

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.

$$A = \frac{P}{7.5 \cdot \Delta T}$$

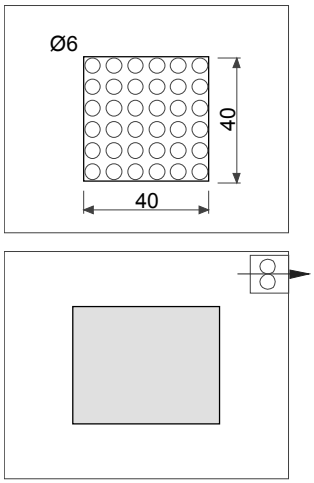
Surface with smooth metallic enamel.

$$A = \frac{P}{9.8 \cdot \Delta T}$$

3.3 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.

$$V = \frac{3.8 \cdot P}{\Delta T}$$

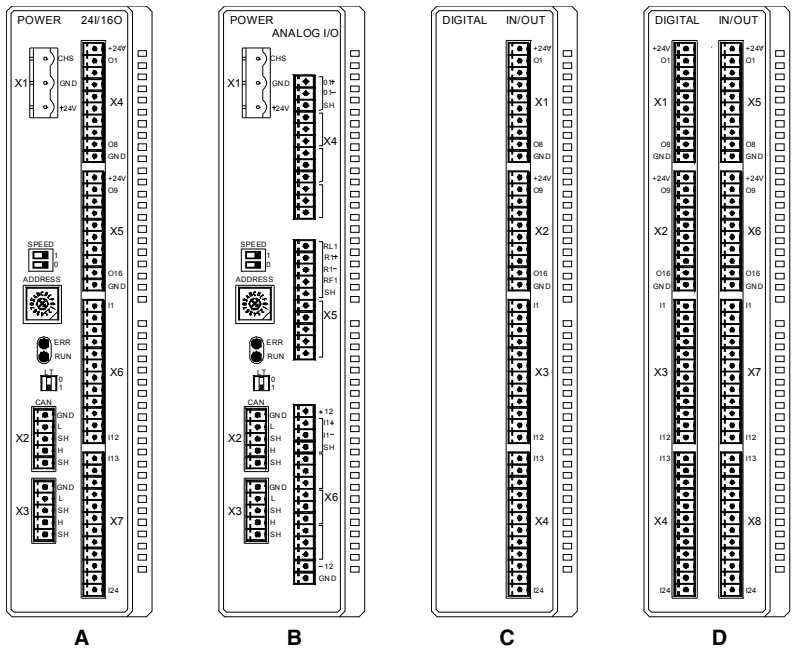
Bear in mind that this air flow through the unit extracts hot air to the outside, but it allows dirt into the enclosure. Thus, a filter should be installed to maintain the ambient conditions allowed.

REMOTE MODULES (BUS CAN WITH CANOPEN PROTOCOL)

4

Remote modules may be used to have an additional number of digital and analog inputs and outputs (remote I/O) as well as inputs for temperature sensors. The remote modules are distributed by groups and are connected to the central unit through the CAN bus.

Up to 4 groups may be connected to the CAN bus, where each group may consist of 1 or 2 of the following elements.



A. Power supply with 24 digital inputs and 16 digital outputs.

This module must be powered at 24 V DC and connected to the system CAN bus.

B. Power supply with 4 analog inputs, 4 analog outputs and 2 inputs for temperature sensors.

This module must be powered at 24 V DC and connected to the system CAN bus.

C. Digital Inputs / Outputs (single module).

Each module has 24 digital inputs and 16 digital outputs.

D. Digital Inputs / Outputs (double module).

Each module has 48 digital inputs and 32 digital outputs.

Consumption

The consumption of each group is 1.2 A not including the consumption of the outputs.

Some general points to consider

When mounting the groups, bear in mind the following considerations.

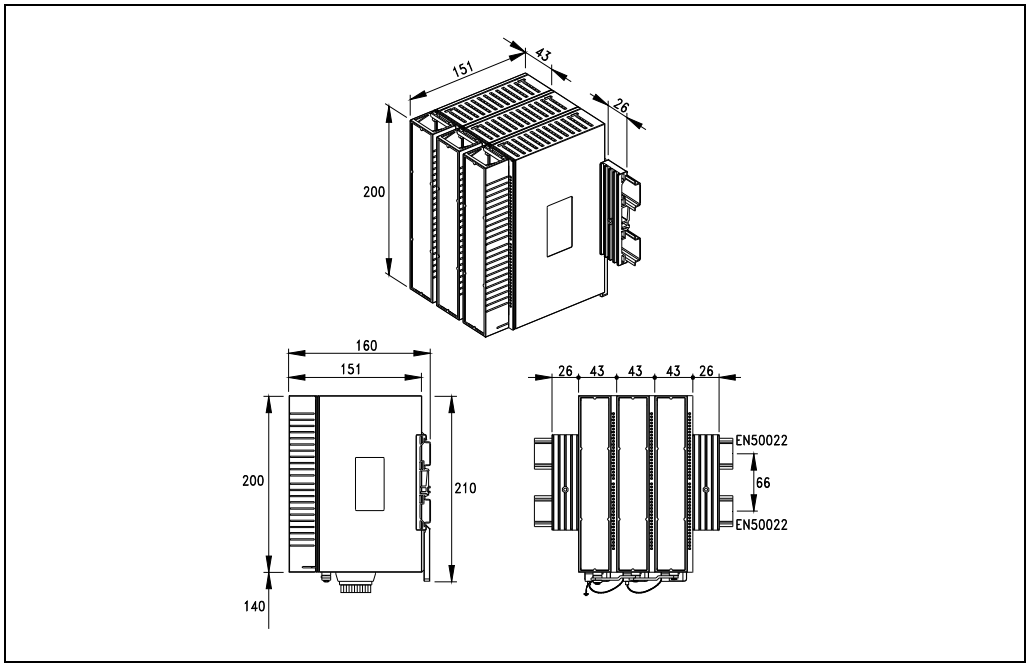
- One of the power supply modules must be present in each group.
- Two power supply modules cannot be in the same group.
- Power supplies of both models may be connected to the same CAN bus.
- The system will only support the treatment of two analog I/O boards.

4.1 Installation of the modules

Place the modules on 2 profiles, according to the UNE 50022 standard, with 2 securing ends, one at each end of the group; they help securing the modules besides maintaining the right gap between the profiles.

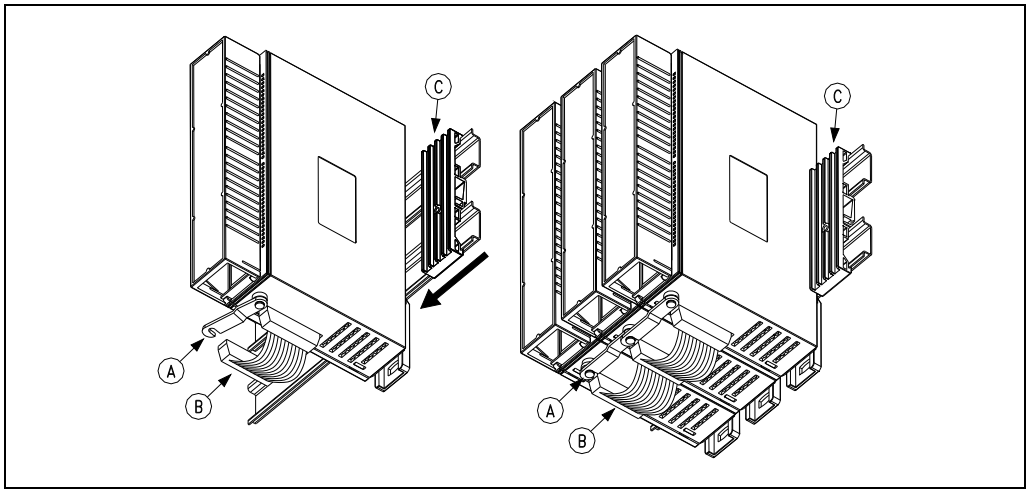
Dimensions of the modules

Always leave a 140 mm gap under the modules for ventilation and later handling.



Connection of the modules

The modules of the group are connected as follows:



A. For ground connection.

B. Ribbon cable to interconnect the modules.

C. Securing studs.

Each group is connected to the system (CPU, Keyboard, etc.) through the CAN bus as described later on.

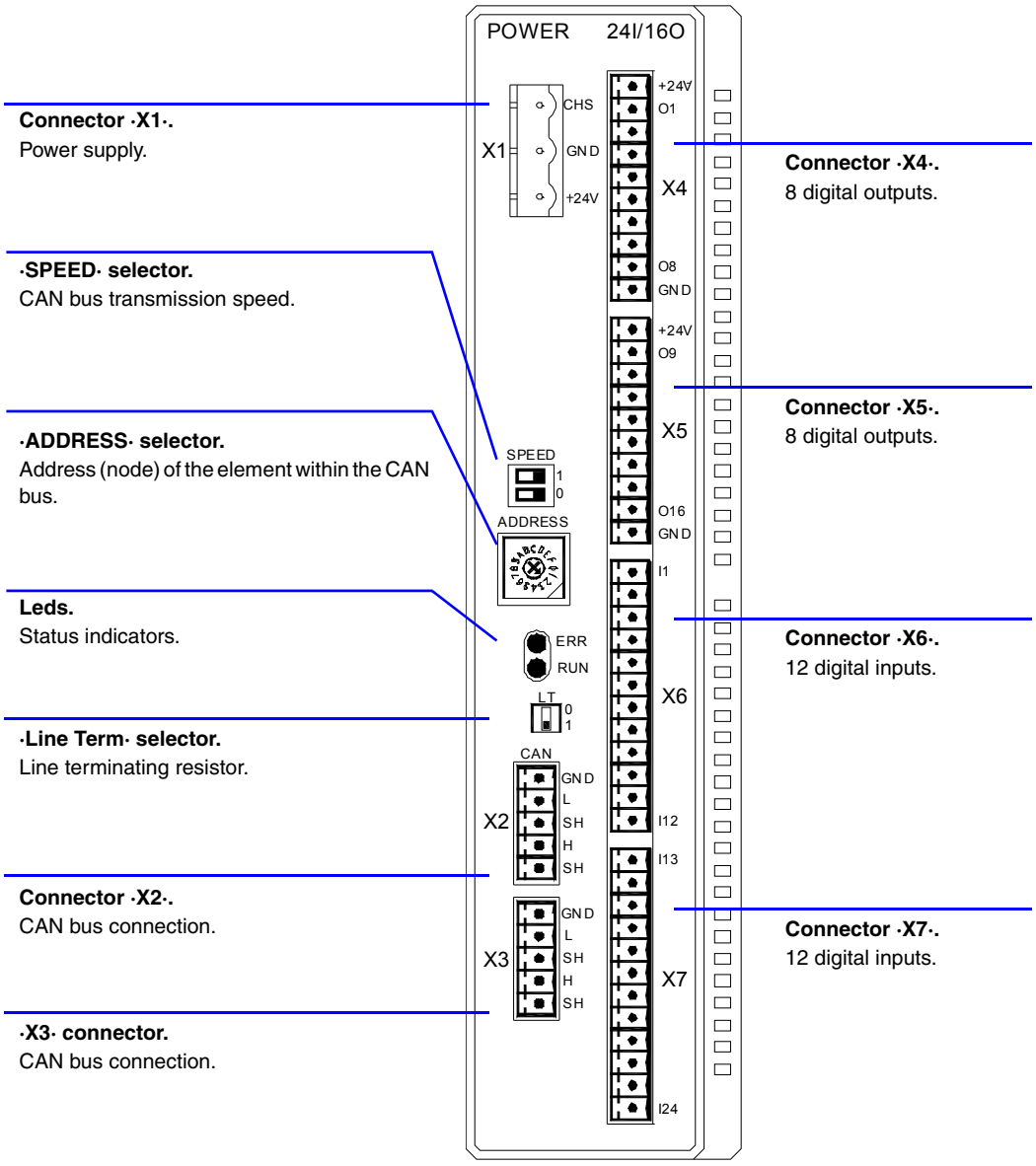
4.2 Power supply

The power supply must be powered at 24 V DC and connected to the CAN bus of the system.

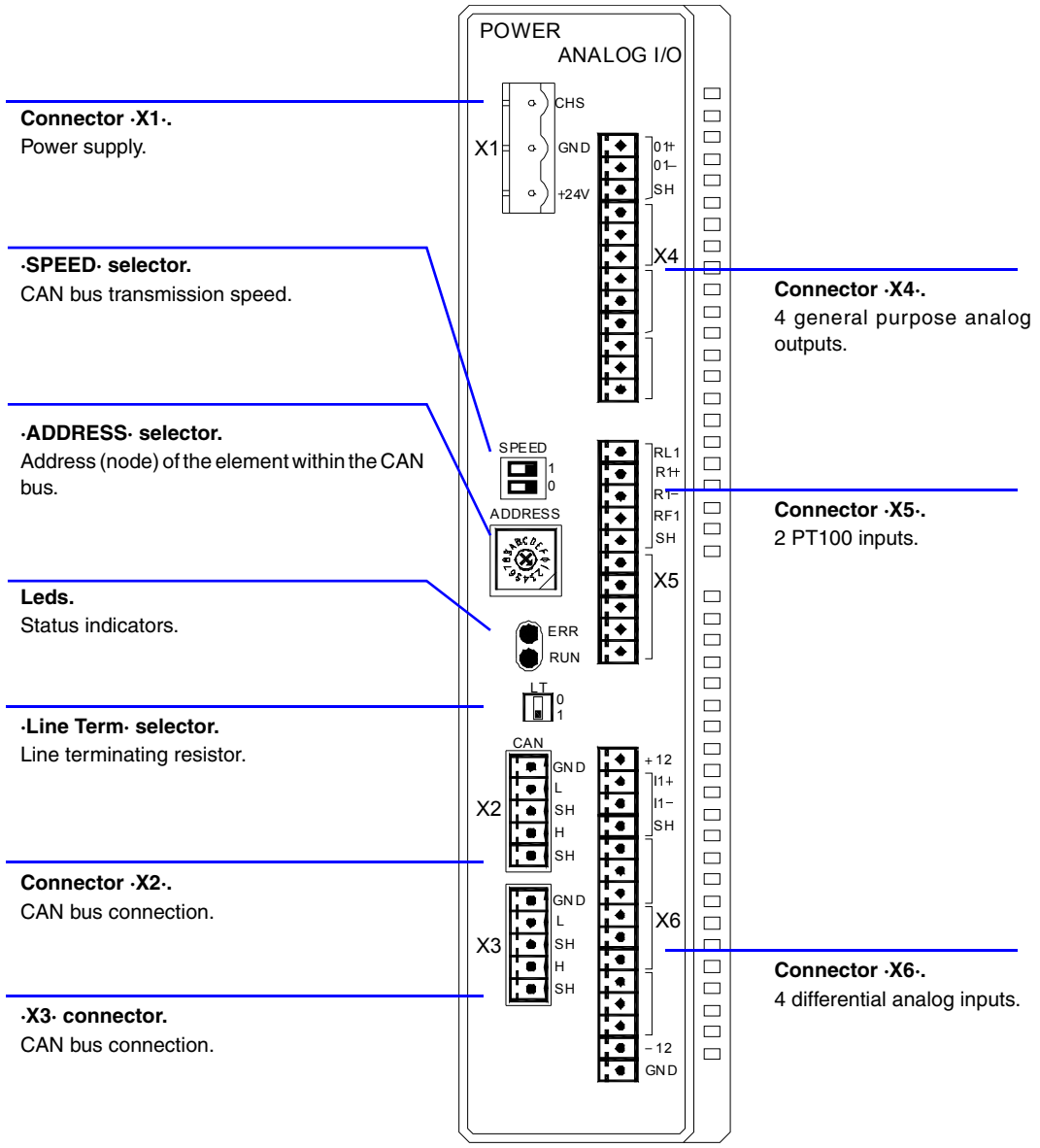
There are two power supply models.

- Power supply with 24 digital inputs and 16 digital outputs.
- Power supply with 4 analog inputs, 4 analog outputs and 2 inputs for temperature sensors.

Power supply with digital inputs and digital outputs.



Power supply with analog inputs and outputs.



Connector ·X1·.
Power supply.

·SPEED· selector.
CAN bus transmission speed.

·ADDRESS· selector.
Address (node) of the element within the CAN bus.

Leds.
Status indicators.

·Line Term· selector.
Line terminating resistor.

Connector ·X2·.
CAN bus connection.

·X3· connector.
CAN bus connection.

Connector ·X4·.
4 general purpose analog outputs.

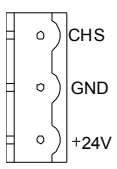
Connector ·X5·.
2 PT100 inputs.

Connector ·X6·.
4 differential analog inputs.

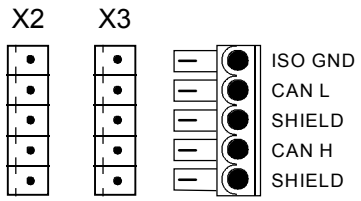
Description of the connectors (power supply)

Connector ·X1·. Power supply.

3-pin male Phoenix connector (7.62 mm pitch).

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

Connector ·X2· & ·X3·. CAN bus connection.

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

·SPEED· selector. CAN bus transmission speed.

When using the CANopen protocol, the transmission speed at the bus is defined in each node. All the nodes must work at the same speed.

The transmission speed depends on the total length of the cable. Use the following approximate values. Assigning other values may cause communication errors due to signal distortion.

Selector	Speed:	Length of the CAN bus.
SPEED 1 0	1000 kHz	Up to 20 meters.
SPEED 1 0	800 kHz	From 20 to 40 meters.
SPEED 1 0	500 kHz	From 40 to 100 meters.
SPEED 1 0	500 kHz	From 40 to 100 meters.

·ADDRESS· selector. Address (node) of the element within the CAN 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"). The CNC must always occupy position "0" and the rest of the elements of the bus will occupy consecutive positions starting with 1.

·ERR· LED Status indicator LED.

The LED is red. Its meaning depends on the blinking frequency.

Type of blinking.	Meaning
LED off.	The module works properly.
Blinking fast.	Module configuration stage.
Single blinking.	Warning. Poor transmission.
Double blinking.	There is no communication with the cpu.
LED on.	Error. Too many errors.

·RUN· LED. Status indicator LED.

The LED is green. Its meaning depends on the blinking frequency.

Type of blinking.	Meaning
LED on.	The module works properly.
Single blinking.	Module stopped.
Blinking fast.	Module configuration stage.
Continuous blinking.	Start-up stage or error.

·Line Term· selector.

Line terminating resistor. 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 switch of the elements at the ends must be in the ·1· position and the rest of the elements in the ·0· position.

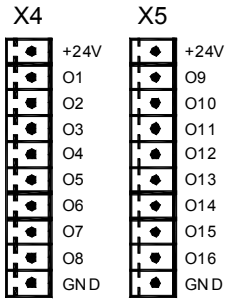
The central unit must always be at one end of the line. The other end will be the last one of the remote module groups.

Description of the connectors (digital inputs and outputs)

Connector ·X4· & ·X5·. Digital outputs (8 outputs in each connector).

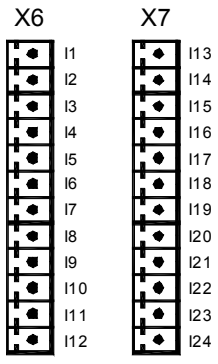
10-pin male Phoenix minicombicon connector (3.5 mm pitch).

Both connectors must be powered with 24 V DC and GND.

	Signal	Function
	+ 24 V	Power supply.
	O1 - O8	Digital outputs.
	O9 - O16	Digital outputs.
	GND	Power supply.

Connector ·X6· & ·X7·. Digital inputs (12 inputs in each connector).

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

	Signal	Function
	I1 - I12	Digital inputs.
	I13 - I24	Digital inputs.

Description of the connectors (analog inputs and outputs)

Connector ·X4·. General purpose analog outputs (4 outputs).

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

	Signal	Function
X4 O1+ O1- SH O2+ O2- SH O3+ O3- SH O4+ O4- SH	O1+ O1-	Analog output.
	O2+ O2-	Analog output.
	O3+ O3-	Analog output.
	O4+ O4-	Analog output.
	SH	Grid connection.

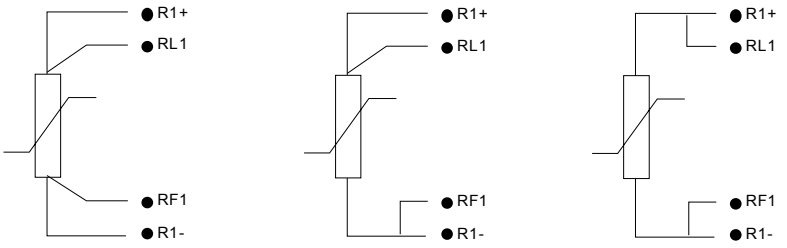
Each analog output has three pins (O+, O-, SH). Use shielded cables connecting their meshes to the corresponding shield pin.

Connector ·X5·. Inputs for the temperature sensor PT100 (2 inputs).

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

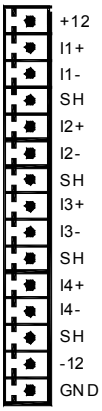
	Signal	Function
X5 RL1 R1+ R1- RF1 SH RL2 R2+ R2- RF2 SH	R1+ R1- RL1 RF1	Signals of the PT100 sensor.
	R2+ R2- RL2 RF2	Signals of the PT100 sensor.
	SH	Grid connection.

Each input has 5 pins (RL, R+, R-, RF1, SH). Use shielded cables connecting their meshes to the corresponding shield pin.



Connector ·X6·. Differential analog inputs (4 inputs).

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

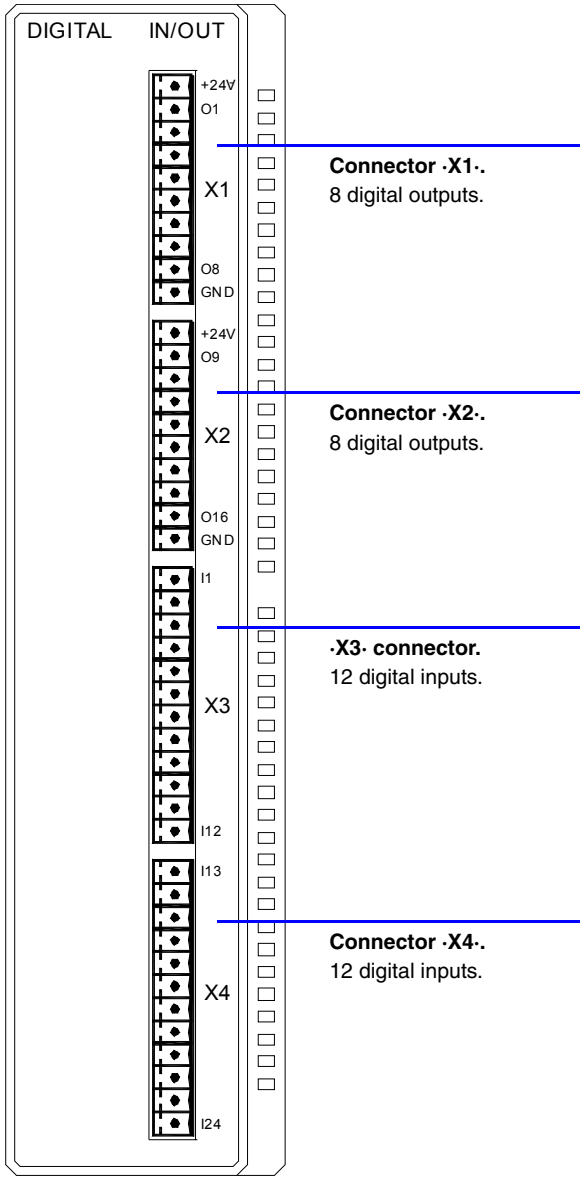
	Signal	Function
X6  	I1+ I1-	Analog input
	I2+ I2-	Analog input
	I3+ I3-	Analog input
	I4+ I4-	Analog input
	SH	Grid connection.
	+12 -12 GND	Reference outputs.

Each analog input has three pins (I+, I-, SH). Use shielded cables connecting their meshes to the corresponding shield pin.

4.3 Digital inputs and digital outputs (single module)

This module is used to expand the digital inputs and outputs (remote I/O). Each module has 24 digital inputs and 16 digital outputs.

Digital inputs and outputs (single module).

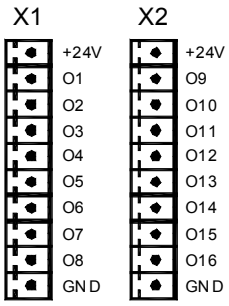


Description of the connectors (digital inputs and outputs)

Connector ·X1· & ·X2·. Digital outputs (8 outputs in each connector).

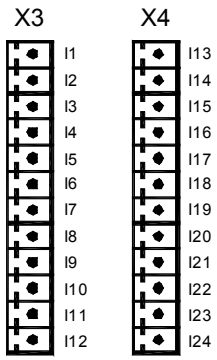
10-pin male Phoenix minicombicon connector (3.5 mm pitch).

Both connectors must be powered with 24 V DC and GND.

	Signal	Function
	+ 24 V	Power supply.
	O1 - O8	Digital outputs.
	O9 - O16	Digital outputs.
	GND	Power supply.

Connector ·X3· & ·X4·. Digital inputs (12 inputs in each connector).

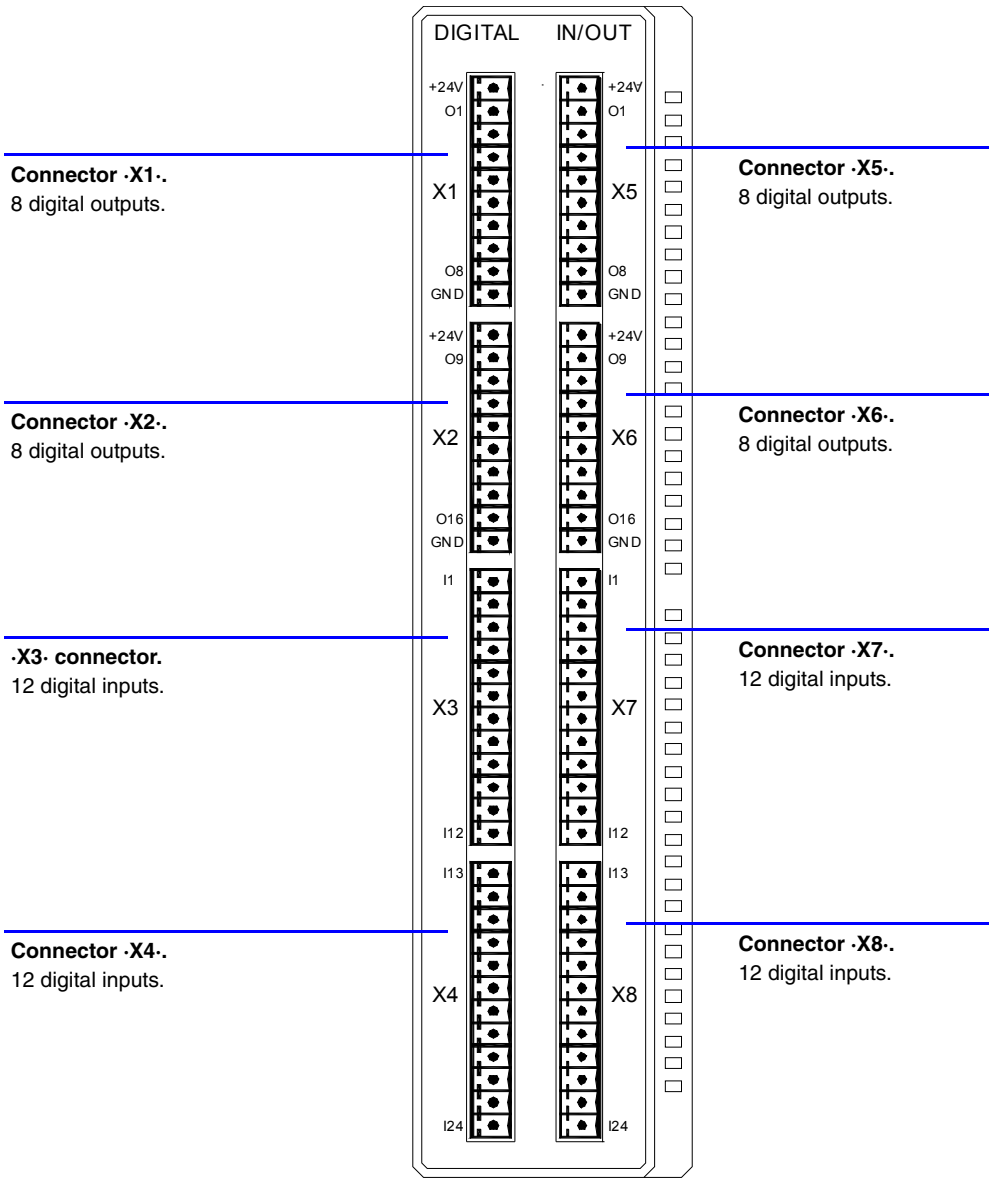
12-pin male Phoenix minicombicon connector (3.5 mm pitch).

	Signal	Function
	I1 - I12	Digital inputs.
	I13 - I24	Digital inputs.

4.4 Digital inputs and digital outputs (double module)

This module is used to expand the digital inputs and outputs (remote I/O). Each module has 48 digital inputs and 32 digital outputs.

Digital inputs and outputs (double module).

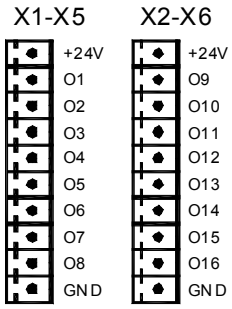


Description of the connectors (digital inputs and outputs)

Connector ·X1· & ·X2· & ·X5· & ·X6·. Digital outputs (8 outputs in each connector).

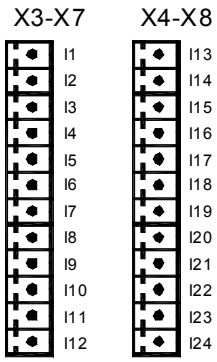
10-pin male Phoenix minicombicon connector (3.5 mm pitch).

Both connectors must be powered with 24 V DC and GND.

	Signal	Function
	+ 24 V	Power supply.
	O1 - O8	Digital outputs.
	O9 - O16	Digital outputs.
	GND	Power supply.

Connector ·X3· & ·X4· & ·X7· & ·X8·. Digital inputs (12 inputs in each connector).

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

	Signal	Function
	I1 - I12	Digital inputs.
	I13 - I24	Digital inputs.

4.5 Electrical characteristics of the inputs and outputs

Digital inputs

All digital inputs are galvanically isolated through opto-couplers. The electrical characteristics of the inputs are:

Nominal voltage	+24 V DC (between +18 V and +30 V DC).
High threshold "1"	From +18 V DC up.
Low threshold "0"	Under +9 V DC.
Typical consumption of each input	5 mA.
Maximum consumption of each input	7 mA.

Digital outputs

All digital outputs are galvanically isolated through opto-couplers. The electrical characteristics of the outputs are:

Nominal voltage	+24 V DC (between +18 V and +30 V DC).
Output voltage	2 V less than the supply voltage.
Maximum output current	500 mA per output.

The digital outputs have a fuse inside for protection against over-voltage (over 33 V DC) and against reverse connection of the power supply.

Analog inputs

Use shielded cables connecting their meshes to the corresponding shield pin. All the analog inputs have the following characteristics:

Voltage range	±10 V
Resolution	12 bits
Input impedance	20 kΩ
Maximum cable length (unshielded)	75 mm.

Analog outputs

Use shielded cables connecting their meshes to the corresponding shield pin. All the analog outputs have the following characteristics:

Command voltage within range	±10 V
Resolution	16 bits
Minimum impedance of the connected device	10 kΩ.
Maximum cable length (unshielded)	75 mm.

Inputs for the temperature sensor PT100

Use shielded cables connecting their meshes to the corresponding shield pin. The electrical characteristics of the inputs are:

Type of sensor.	PT100
Temperature range	Between -200 ºC (-328 ºF) and +850 ºC (1562 ºF).
Resolution	0.1 ºC
Typical consumption of each input	2 mA.
Maximum cable length (unshielded)	75 mm.

4.6 Numbering of the digital inputs and outputs

The following PLC machine parameters identify each of the 4 possible remote modules. The following group of parameters must be defined for each one of them.

IOCANID*	Node address.
ICAN*	Number of digital inputs of the group.
OCAN*	Number of digital outputs of the group.
NUICAN*	Number of the first digital input of the group.
NUOCAN*	Number of the first digital output of the group.
IANALOG*	Number of analog inputs of the group.
OANALOG*	Number of analog outputs of the group.
PT100_*	Number of physical connections for PT100 probes of the group.
NUIANA*	Number of the first analog input of the group.
NUOANA*	Number of the first analog output of the group.

Refer to the parameters and examples of how to set the remote modules. See *"6.9 PLC Parameters"* on page 272.

MACHINE AND POWER CONNECTION

5

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 8055 CNC's central unit

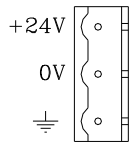
It has a three-prong plug for mains and ground connection.

Power the central unit through a separate 110VA transformer with an output voltage between 84 V AC and 264 V AC and 50-60 Hz.

The monitor & keyboard unit must be powered with 220Vac.

Mains connection of the 8055i CNC's central unit

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

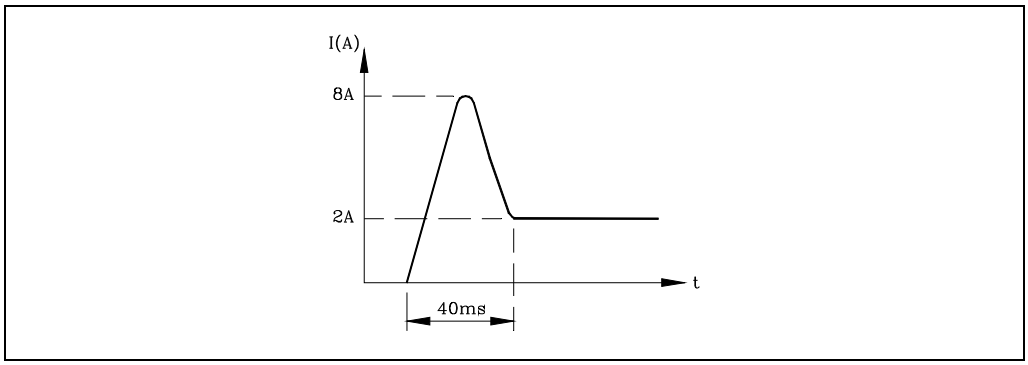
	Pin	Signal and function	
	1	+ 24 V	Power supply.
	2	0 V	Power supply.
	3	Chassis	Shield.

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.

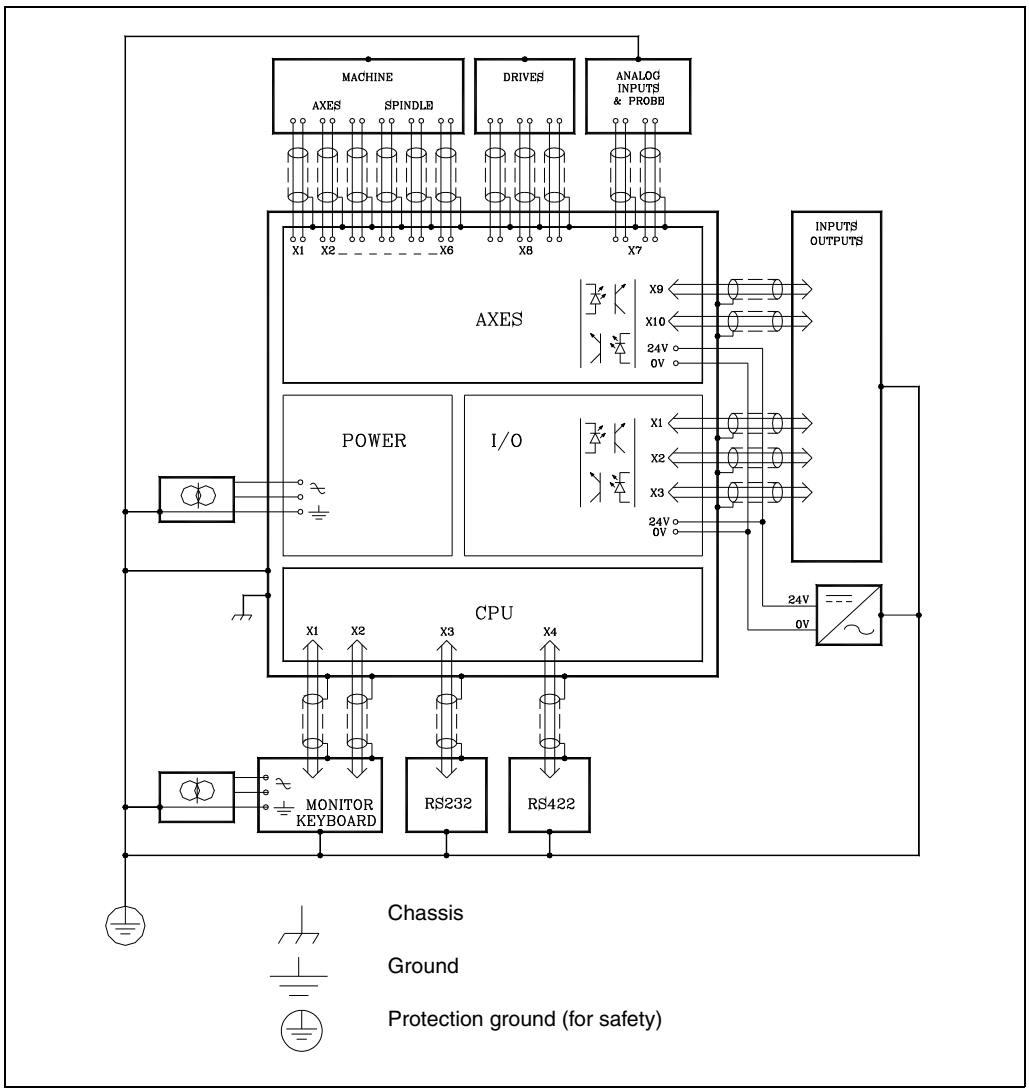
Proper ground installation reduces the effects of electrical interference. But, signal cables also require additional protections. This is generally achieved by using twisted-pair cables that are also covered with antistatic shielding mesh-wire. This shield must be connected to a specific point avoiding 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).

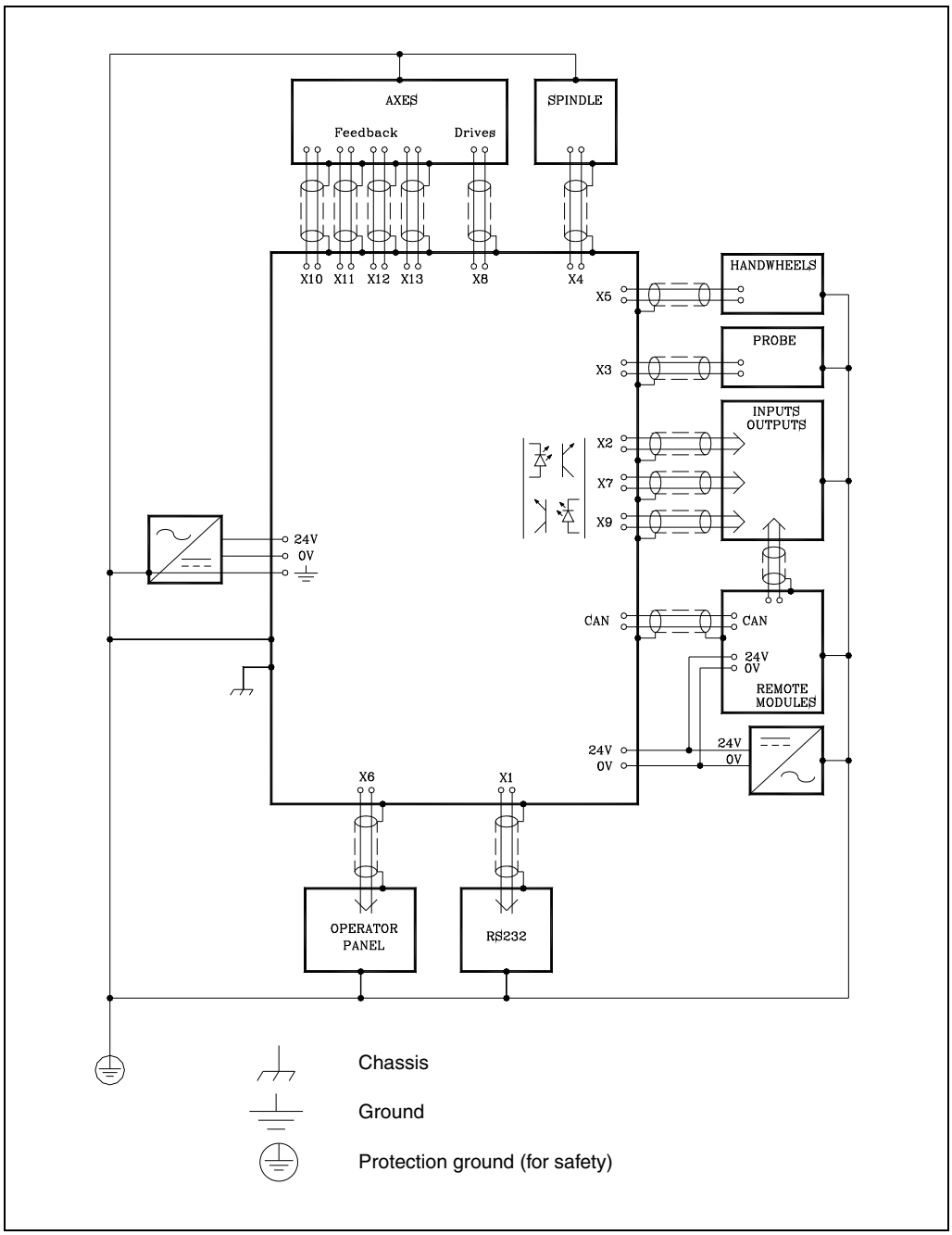
When a second point is necessary, it is recommended to join both points with a cable whose section is no smaller than 8 mm^2.

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

CNC 8055. Ground connection diagram



8055i CNC. Ground connection diagram



5.1 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 8055i 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.

5.2 Analog inputs and outputs

Analog outputs

They may be used for axis, spindle drives and as a servo for other devices.

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 8055 CNC configuration"*. See chapter *"2 CNC 8055i configuration"*.



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

Analog inputs

They may be used for monitoring, supervising, controlling, etc. external devices.

The electrical characteristics of these inputs are:

Voltage range	±5 V.
Input impedance	20 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 8055 CNC configuration"*. See chapter *"2 CNC 8055i configuration"*.

5.3 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 *"6 Machine parameters"*.

These values, shown in the parameter tables, may be modified manually from the CNC's keyboard or from a peripheral (pendrive, computer, etc.) via the two serial communication ports RS 232C and USB.

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 labeled 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 labeled 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.

5.4 Connection of the emergency input and output

The emergency input of the CNC corresponds with the I01 input of the PLC and must be supplied with 24V.

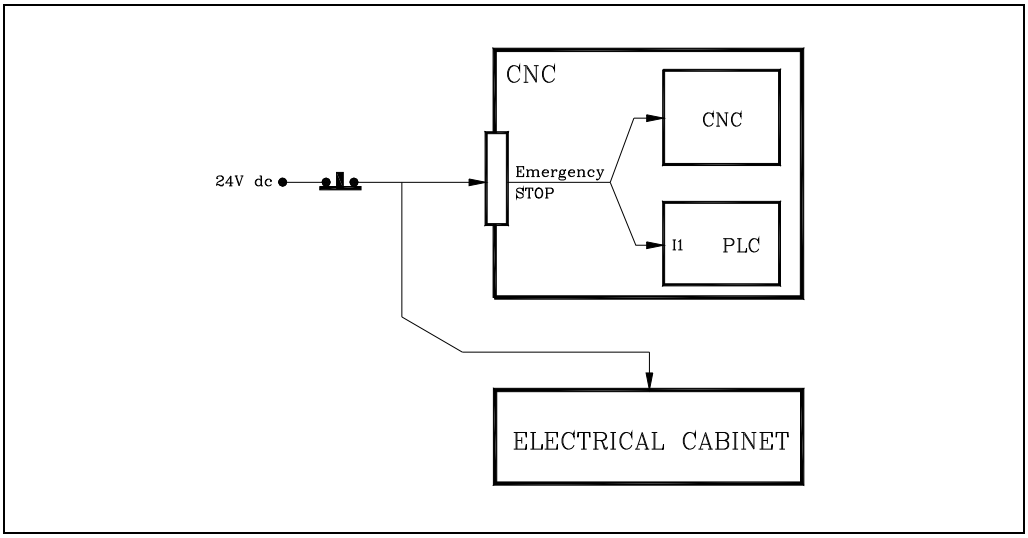
8055 CNC

pin 2 of connector X9 of the –Axes– module.

8055i CNC.

pin 10 of connector X2

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.

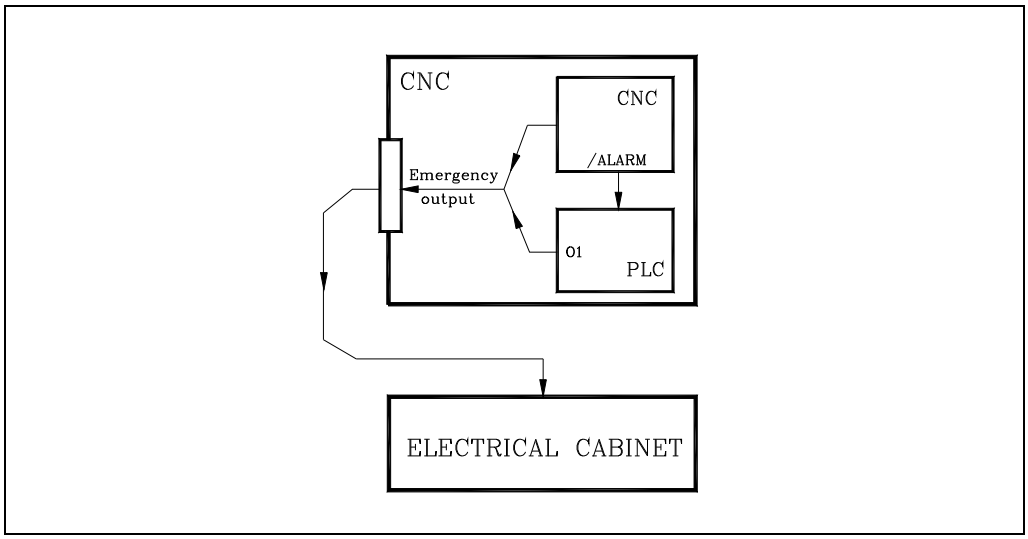
The interface of the electrical cabinet will take into account all the elements that could cause this type of error. Among such elements are:

- 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.

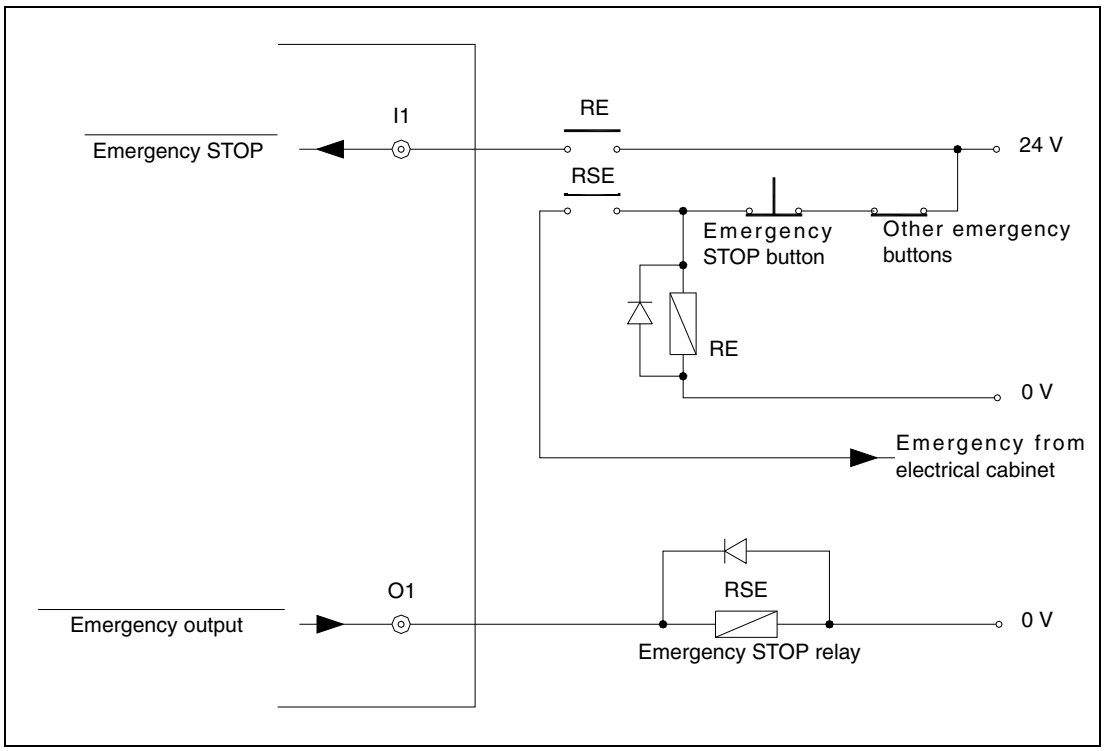
When the CNC detects an error, it will let the PLC know through the general logic output "/ALARM" and it will activate the emergency output (logic level 0).

Pin 2 of connector X10 of the –Axes– module.

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:

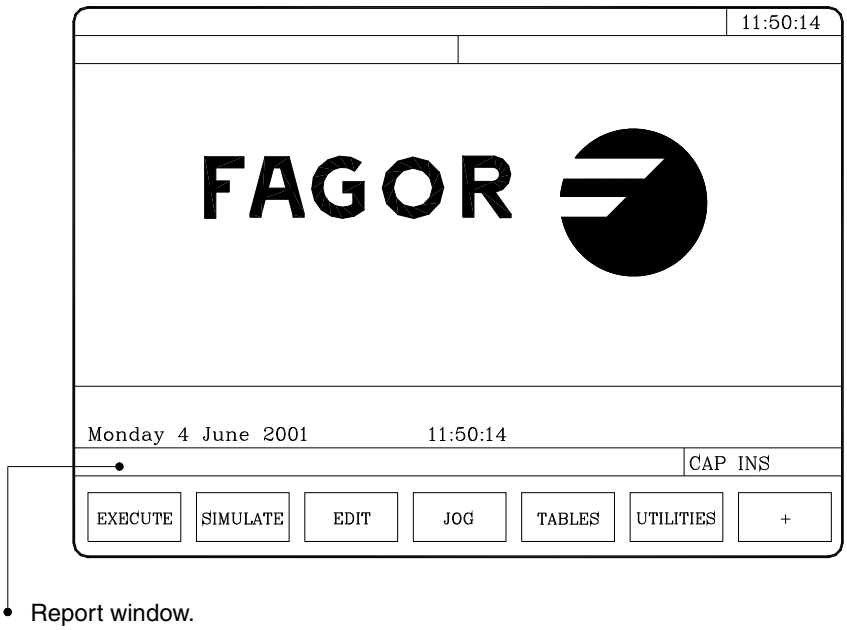


MACHINE PARAMETERS

6

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)
- Machine parameters for the spindles (main, second and auxiliary).
- Drive parameters (access to the parameters of each drive).
- Parameters for the serial line and Ethernet.
- 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	To 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. These compensation tables 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).

VERIFICATIONS BEFORE STARTUP

In order for the CNC to perform properly, after a setup, check that the following machine parameters have been set with proper values. This is very important for safety reasons, because if those values are not the right ones, the machine's performance could be deficient.

General machine parameters

PRODEL (P106)

This parameter the time delay between the instant probing takes place and the CNC is aware of it. G.m.p. PRODEL (P106) must be set to 0.

STOPTAP (P116)

This parameter indicates whether general inputs /STOP (M5001), /FEEDHOL (M5002) and /XFERINH (M5003) are enabled or not while executing function G84. G.m.p. STOPTAP (P116) must be set to NO.

INSFEED (P117)

This parameter sets the tool inspection feedrate. G.m.p. INSFEED (P117) must be set with a proper value (other than 0).

RAPIDEN (P130)

This parameter indicates whether the "rapid" key affects the execution and the simulation or not. G.m.p. RAPIDEN (P130) must be set to 0.

MAXOFFI (P165)

This parameter indicates the maximum amount of wear that may be entered for "I" when modifying the wear offsets from tool inspection mode. G.m.p. MAXOFFI (P165) must be set with a proper value (other than 0).

MAXOFFK (P166)

This parameter indicates the maximum amount of wear that may be entered for "K" when modifying the wear offsets from tool inspection mode. G.m.p. MAXOFFK (P166) must be set with a proper value (other than 0).

FEEDTYPE (P170)

This parameter indicates the behavior of the feedrate when programming F0. G.m.p. FEEDTYPE (P170) must be set to 0.

MAXOFFJ (P182)

This parameter indicates the maximum incremental value allowed for Y axis wear compensation. G.m.p. MAXOFFJ (P182) must be set with a proper value (other than 0).

Axis parameters

LIMIT+ (P5) and LIMIT- (P6)

These parameters set the travel limits for the axis (positive and negative). A.m.p. LIMIT+ (P5) and LIMIT- (P6) must have the right values

FBACKAL (P11)

This parameter indicates whether the feedback alarm for this axis will be ON or OFF. A.m.p. FBACKAL (P11) must be set to ON.

FBALTIME (P12)

This parameter indicates the maximum time that the axis may stay without properly responding to the CNC's command. A.m.p. FBALTIME (P12) must be set with a proper value (other than 0).

MAXFLWE1 (P21)

This parameter indicates the maximum following error allowed when this axis moves. The value of a.m.p. MAXFLWE1 (P21) must be slightly higher than the amount of following error (lag) in G0.

MAXFLWE2 (P22)

This parameter indicates the maximum following error allowed when this axis is stopped. The value of MAXFLWE2 (P22) must be snug, i.e. the value must not be too large.

MAXCOUPE (P45)

This parameter indicates the maximum difference allowed between the following errors of the Gantry axes that electronically coupled. A.m.p. MAXCOUPE (P45) must be set with a proper value (other than 0). Only for Gantry or slaved axes.

FLIMIT (P75)

This parameter indicates the maximum safety limit the axis feedrate. A.m.p. FLIMIT (P75) must be set with a proper value (other than 0).

ADIFEED (P84)

This parameter sets indicates the maximum feedrate allowed due to the additive handwheel. A.m.p. ADIFEED (P84) must be set with a proper value (other than 0).

FRAPIDEN (P85)

This parameter indicates the maximum axis feedrate when activating the EXRAPID mark and when pressing the rapid key in execution or in simulation with motion. A.m.p. FRAPIDEN (P85) must be set with a proper value (other than 0).

Spindle parameters

FBACKAL (P15)

This parameter indicates whether the feedback alarm for the spindle will be ON or OFF. S.m.p. FBACKAL (P15) must be set to ON.

MAXFLWE1 (P21)

This parameter indicates the maximum following error allowed for the spindle when moving in closed loop (M19). The value of s.m.p. MAXFLWE1 (P21) must be slightly higher than the amount of following error (lag) in G0.

MAXFLWE2 (P22)

This parameter indicates the maximum following error allowed for the spindle when stopped in closed loop (M19). The value of MAXFLWE2 (P22) must be snug, i.e. the value must not be too large.

SYNPOSOF (P53)

This parameter indicates the maximum shift (offset) between the main spindle and the second one when they are synchronized in position. S.m.p. SYNPOSOF (P53) must be set with a proper value (other than 0).

SYNSPEOF (P54)

This parameter indicates the maximum speed error (offset) between the main spindle and the second one when they are synchronized in speed. S.m.p. SYNSPEOF (P54) must be set with a proper value (other than 0).

SYNMAXSP (P63)

This parameter indicates the maximum turning speed when the spindle are synchronized. S.m.p. SYNMAXSP (P63) must be set with a proper value (other than 0).

SLIMIT (P66)

This parameter sets the maximum safety limit for the spindle speed. S.m.p. SLIMIT (P66) must be set with a proper value (other than 0).

PLC Parameters.

WDGPRG (P0)

This parameter sets the Watchdog time-out period for the main PLC program. PLC.m.p. WDGPRG (P0) must be set with a proper value.

WDGPER (P1)

This parameter sets the Watch-Dog time-out period for the periodic module of the PLC. Only if a periodic subroutine has been defined. PLC.m.p. WDGPER (P1) must be set with a proper value.

6.1 Parameter matching between the CNC and the Sercos drive

Some CNC and drive parameters must be set accordingly (matching each other) to ensure proper system performance. While initializing the Sercos ring, on CNC power-up and when validating the machine parameters of the axes and spindles, the CNC updates at the drives the parameters to ensure that they match.

The CNC matches the parameters when the drive is an ACSD.

Understanding the parameter matching table.

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

CNC — List of CNC machine parameters.

DRIVE — List of drive parameters that are equivalent to each CNC parameter.

Feedback — It indicates whether or not writing the parameter at the drive depends on the type of axis feedback, internal or external.

CNC	DRIVE	Feedback	Remarks
FBACKDIF (P100)	PP5	External	If 2nd drive feedback
INPREV (P87)	NP121		If INPREV, OUTPREV and PITCHB are other than 0. Affects all the sets.
OUTPREV (P88)	NP122		If INPREV, OUTPREV and PITCHB are other than 0. Affects all the sets.
PITCHB (P86)	NP123		If INPREV, OUTPREV and PITCHB are other than 0. Affects all the sets.
I0TYPE (P52)	PP115	External	It writes bits 0, 1, 3.
	PV200		If 2nd drive feedback and I0TYPE =1 or 2. It writes 0.
I0CODI2 (P69)	NP165		If 2nd drive feedback and I0TYPE =1 or 2.
I0CODI1 (P68)	NP166		If 2nd drive feedback and I0TYPE =1 or 2.
ABSOFF (P53)	PP177 PP178	Internal External	If 1st drive feedback. If 2nd drive feedback.

If a.m.p. DRISET(P91) is other than 0, drive parameters NP121, NP122 and NP123 are updated with the set indicated in that axis machine parameter.

6.2 Parameters that may be modified from the oscilloscope, 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 *"13.4 Variables associated with machine parameters"* on page 518.

General machine parameters:

Parameter	Number	Update
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.

6.3 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	Auxiliary spindle. Live tool.
2	Y axis.	14	Second main spindle
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.

For the 8055 CNC model.
The connectors refer to the axis board.

Parameter	Feedback (connector)	Command (connector X8)	Default value	
			- M -	- T -
AXIS1 (P0)	X1	O1 - Pin 1	1 (X axis)	1 (X axis)
AXIS2 (P1)	X2	O2 - Pin 2	2 (Y axis)	3 (Z axis)
AXIS3 (P2)	X3	O3 - Pin 3	3 (Z axis)	10 (spindle)
AXIS4 (P3)	X4	O4 - Pin 4	4 (U axis)	11 (handwheel)
AXIS5 (P4)	X5 (1-6)	O5 - Pin 5	5 (V axis)	0 (free)
AXIS6 (P5)	X5 (9-14)	O6 - Pin 6	10 (spindle)	0 (free)
AXIS7 (P6)	X6 (1-6)	O7 - Pin 7	11 (handwheel)	0 (free)
AXIS8 (P7)	X6 (9-14)	O8 - Pin 8	0 (free)	0 (free)

For the 8055i CNC model.

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)	4th axis	X13	X8 - Pin 5	4 (U axis)	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 3 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 *"7.3 Movement with an electronic handwheel"* on page 303.

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 OFF).
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

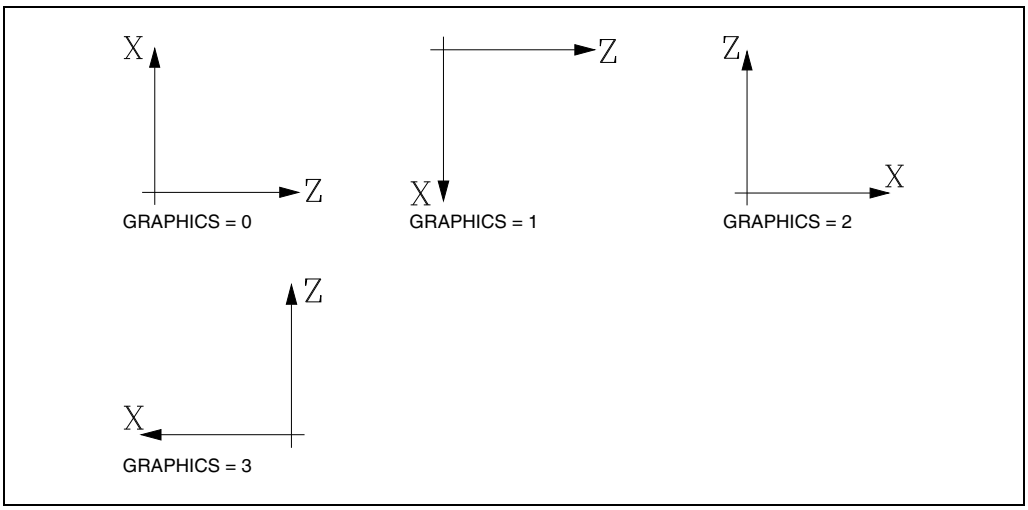
GRAPHICS (P16)

On T, TC and TCO models, this parameter indicates the axes coordinates system to be used for the graphic representation. On these models, 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.

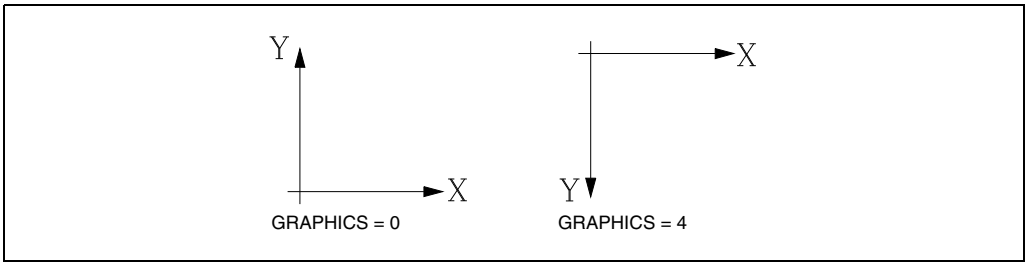
Default value: 0



On M, MC, and MCO models, this parameter 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



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 *"7.8 Auxiliary M, S, T function transfer"* on page 336.

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)

It indicates whether the probe functions of the CNC act on an up flank (positive pulse) or with the down flank (negative pulse) of the signal provided by the probe.

At an 8055 CNC, the probe is connected through connector X7 of the AXES module. At an 8055i CNC, it is possible to have up to two probes connected through connector X3.

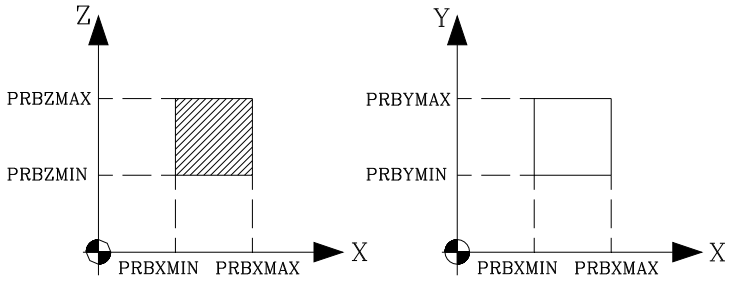
Value	Meaning
+ sign	Positive pulse (24 V or 5 V).
- sign	Negative pulse (0 V).

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)

Not being used.

NPCROSS2 (P54)

Indicates the number of points available in the second 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 (none)

MOVAXIS2 (P55)

Used in the second 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)

COMAXIS2 (P56)

Used in the second 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 NPCROSS2=15, MOVAXIS2=2 and COMAXIS2=8, the CNC will allow access to the second cross compensation table.
Each one of these 15 points (NPCROSS2) of this table will indicate the X position value and the error suffered by the B axis when the Y axis is positioned at this point.
This way, the CNC will apply the compensation of the Y axis table on to the B axis.

NPCROSS3 (P57)

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

This compensation is used when the movement of one axis causes a position change on another axis. The CNC offers a third 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 (none)

MOVAXIS3 (P58)

Used in the third 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)

COMAXIS3 (P59)

Used in the third 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 NPCROSS3=25, MOVAXIS3=3 and COMAXIS=4, the CNC will allow access to the third cross compensation table.

Each one of these 25 points (NPCROSS3) of this table will indicate the X position value and the error suffered by the U axis when the Z axis is positioned at this point.

This way, the CNC will apply this compensation onto the U axis.

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)

It is used on the lathe model CNC.

It indicates whether or not the "C" axis is only deactivated by the typical spindle related "M" functions (M03, M04, M05, etc.).

Value	Meaning
YES	Only the typical functions cancel it.
NO	It is canceled in all the cases.

Default value: NO

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).
1··6	Period in milliseconds.

Default value: 2 (Power model)
Default value: 0 (FL model)

At the FL model, if general machine parameter LOOPTIME (P72) is set with a value lower than 4, the period will be 4 ms.

IPOTIME (P73)

Not being used

COMPTYPE (P74)

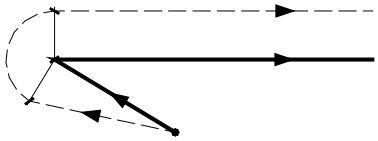
It determines how tool radius compensation is applied. This parameter has three digits.

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

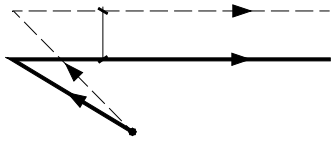
The units set the type of beginning/end of tool radius compensation applied by the CNC.

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



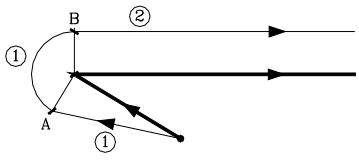
COMPTYPE = x1

(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.

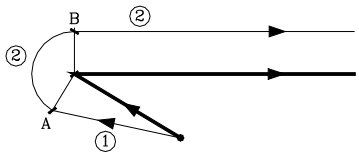
Value	Meaning
x00	It is executed at the end of the current block.
x10	It is executed at the beginning of the next block with compensation.

Default value: 00



COMPTYPE = 00

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

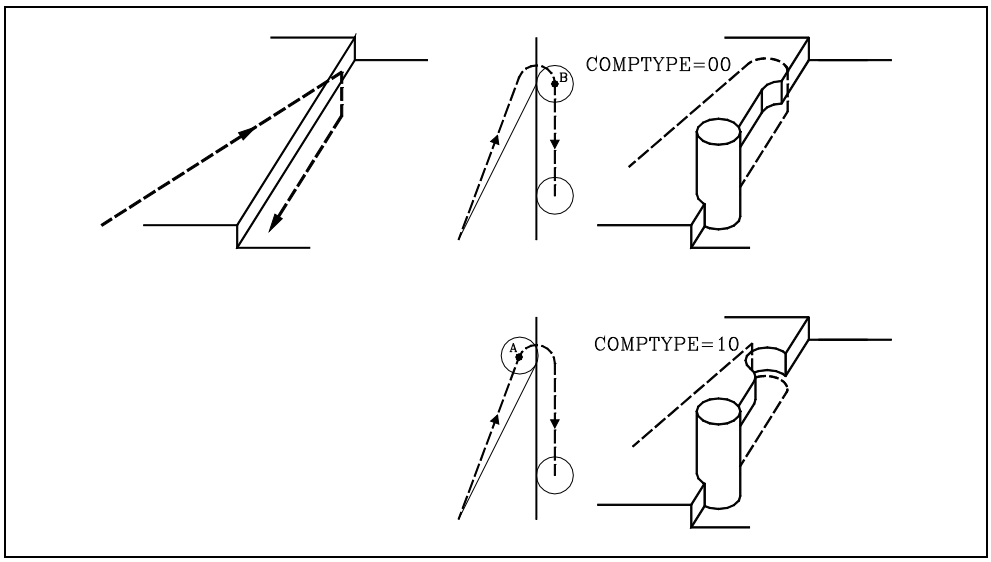


COMPTYPE = 10

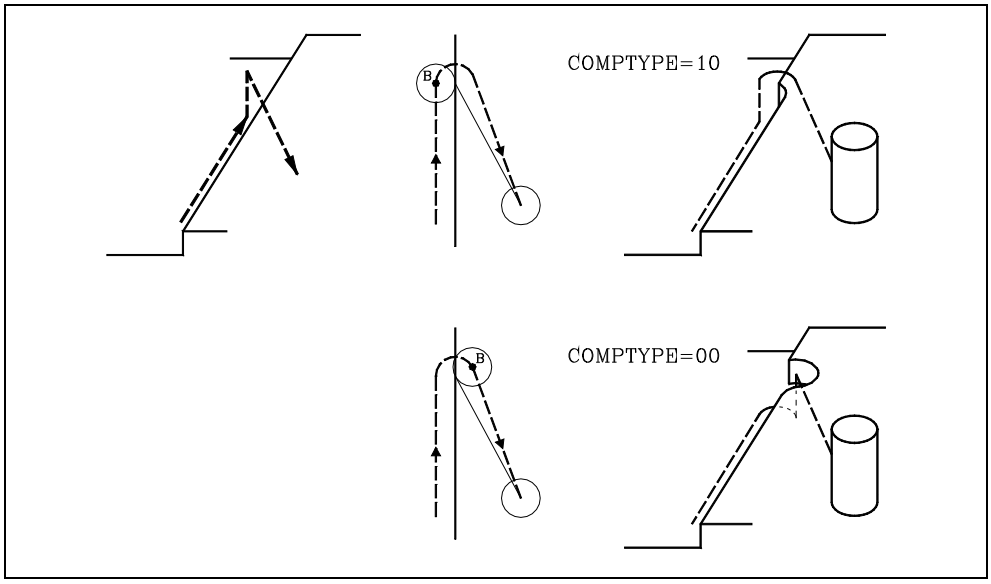
Executing block by block (single block mode), the first movement ends at point "A".

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.

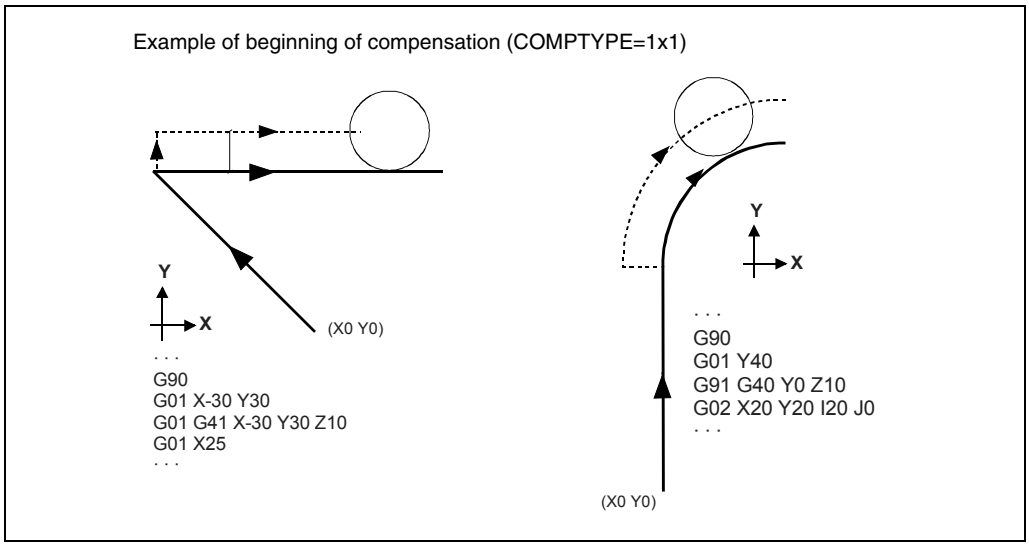
The hundreds indicate whether the compensation is activated in the first motion block or not, even if the plane axes are not involved. The same criteria also applies when turning the compensation off.

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.



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)

It is used on the lathe model to indicate whether there is tool compensation in all planes or just in the ZX plane.

Value	Meaning
0	Only in the ZX plane.
1	In all planes..

Default value: 0

When "PLACOM = 1", the CNC interprets the tool table as follows:

	ZX plane	WX plane
The Z and K parameters, with the abscissa axis	Z axis	W axis
The X and I parameters, with the ordinate axis	X axis	X axis

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 4.4 mm type display format, set MPGRES=2

   With 6.2 mm 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 and up to 3 handwheels.

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)

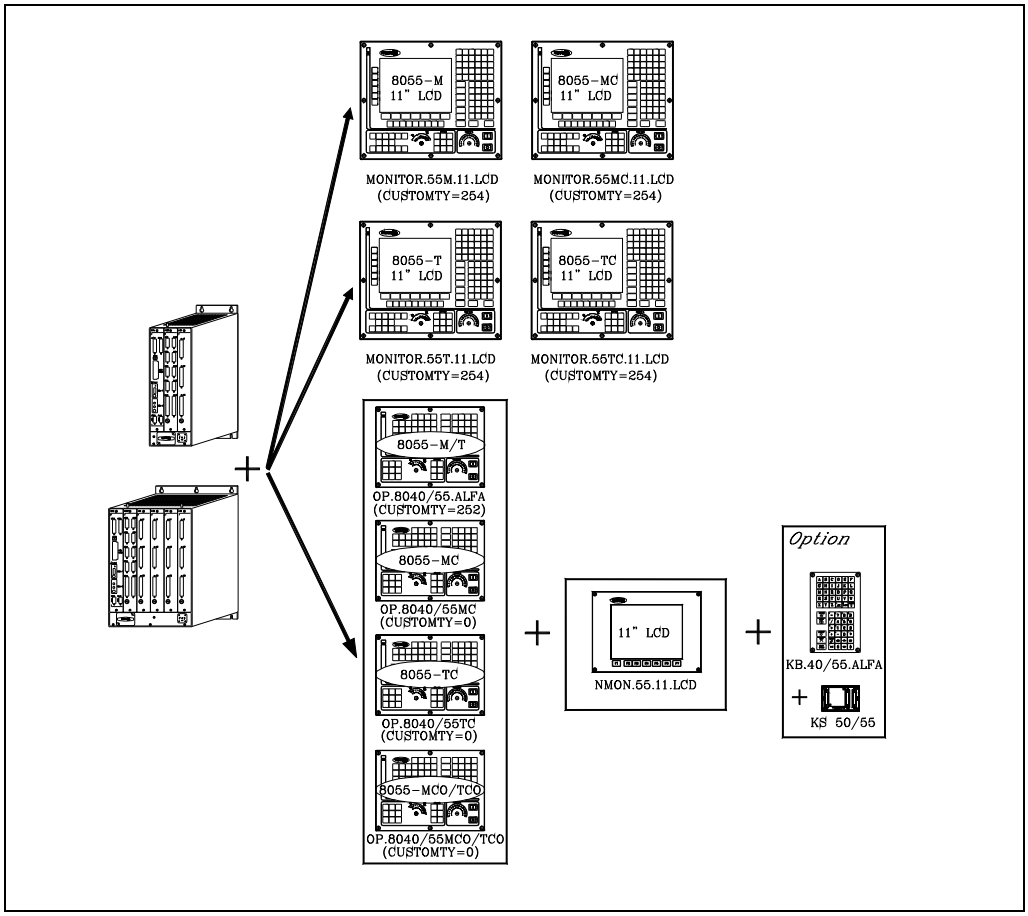
It indicates the configuration being used.

Keyboard auto-identification

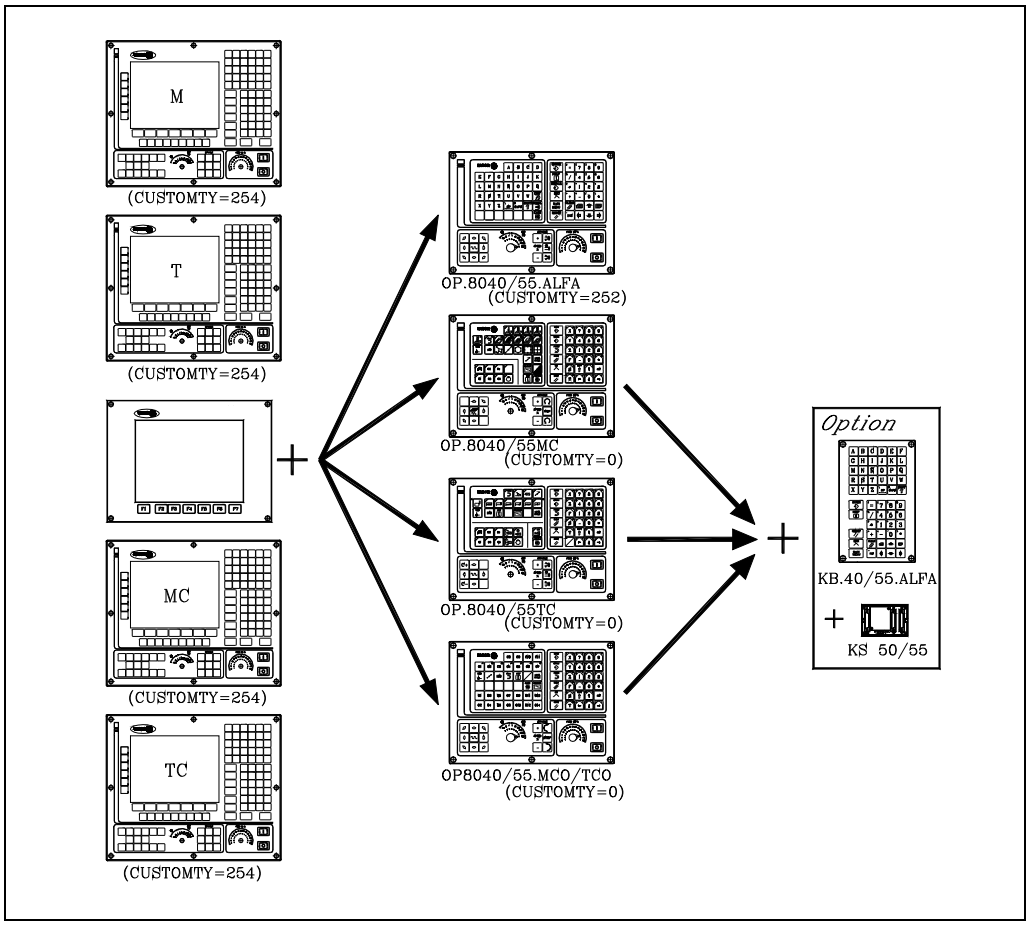
Certain keyboard models have a auto-identification system. With this type of keyboards, this parameter is updated automatically; for the rest of the keyboards, this parameter must be set manually.

If the keyboard does not match the CNC model, it shows the relevant error message and it loads the key codes that correspond to the CNC model. For example, if a mill model keyboard is connected to a lathe CNC, the keyboard is set for a lathe and the CNC shows the error message.

Possible configuration at an 8055 CNC.



Possible configuration at an 8055i CNC.

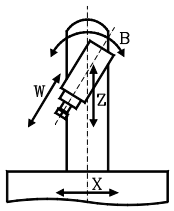


XFORM (P93)

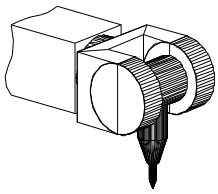
Type of kinematics.

Value	Meaning
0	Without kinematics.
1	Reserved.
2	Orthogonal, spherical or oscillating spindle.
3	Angular spindle.
4	Rotary table.

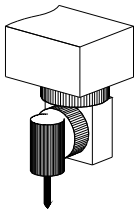
Default value: 0



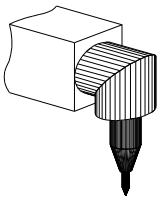
Swinging spindle.
XFORM = 2



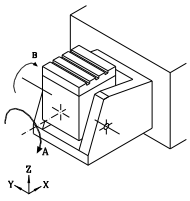
Orthogonal spindle.
XFORM = 2



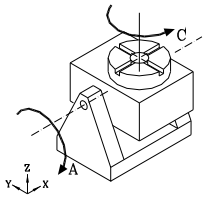
Spherical spindle.
XFORM = 2



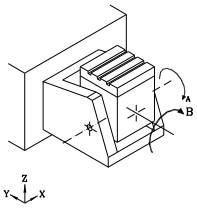
Angular spindle.
XFORM = 3



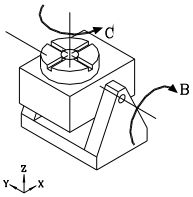
AB table.
XFORM = 4



AC table.
XFORM = 4



BA table.
XFORM = 4



BC table.
XFORM = 4

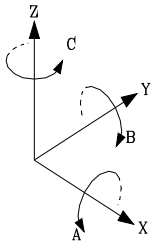
Definition of the kinematics of a swinging, orthogonal or spherical spindle (XFORM =2)

XFORM1 (P94)

It defines the axes of the kinematics and their order, indicating which is the main axis and which the secondary or dragged axis.

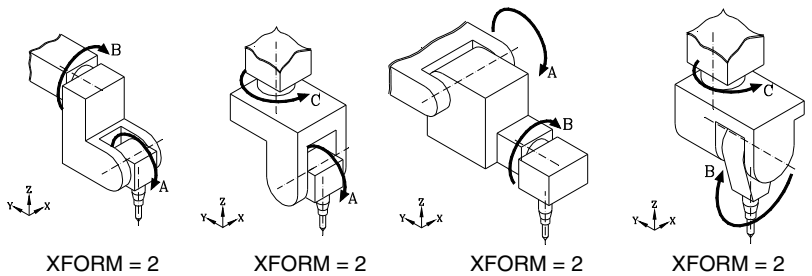
Value	Meaning
0	B is the main axis and A is the secondary axis.
1	C is the main axis and A is the secondary axis.
2	A is the main axis and B is the secondary axis.
3	C is the main axis and B is the secondary axis.

Default value: 0



The rotary axes are called A, B or C depending on whether the rotation axis is X, Y or Z respectively. The turning direction of the axes may be changed with parameter "XFORM2".

Orthogonal or spherical spindle.



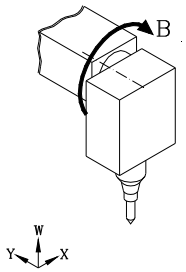
XFORM = 2
XFORM1 = 0

XFORM = 2
XFORM1 = 1

XFORM = 2
XFORM1 = 2

XFORM = 2
XFORM1 = 3

Swinging spindle.



XFORM = 2
XFORM1 = 0

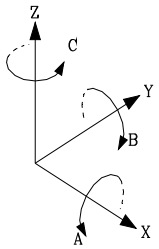
The swinging spindle head "XFORM=2" does not have a secondary or "dragged" axis but parameter "XFORM1" is defined like in the case of the 45º and spherical one.

XFORM2 (P95)

It defines the turning direction of the rotary axes.

Value	Meaning
0	The one indicated by the DIN 66217 standard (see figure).
1	It changes the turning direction of the main axis.
2	It changes the turning direction of the secondary axis.
3	It changes the turning direction of both axes (main and secondary).

Default value: 0



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

These parameters are used to define the dimensions of the spindle. All of them need not be defined.

The parameters to be defined for each model and their meanings are listed here below.

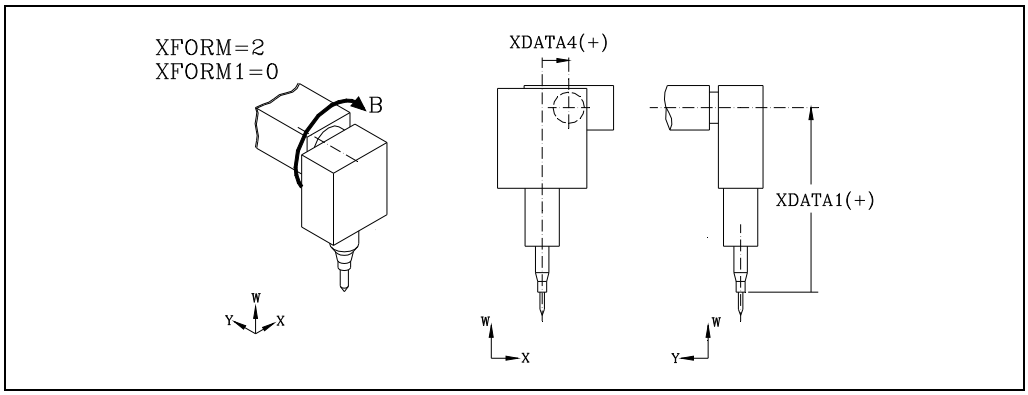
Swinging spindle.

XDATA1 Distance, when the quill is retracted, between the nose of the quill and the rotary axis along the tool axis (W).

XDATA2 Distance between the tool axis and the carried rotary axis. It is defined with ·0· value because there is no secondary axis.

XDATA3 Distance between both rotary axes. It is defined with ·0· value because there is no secondary axis.

XDATA4 Distance between the tool axis and the main (carrier) rotary axis.



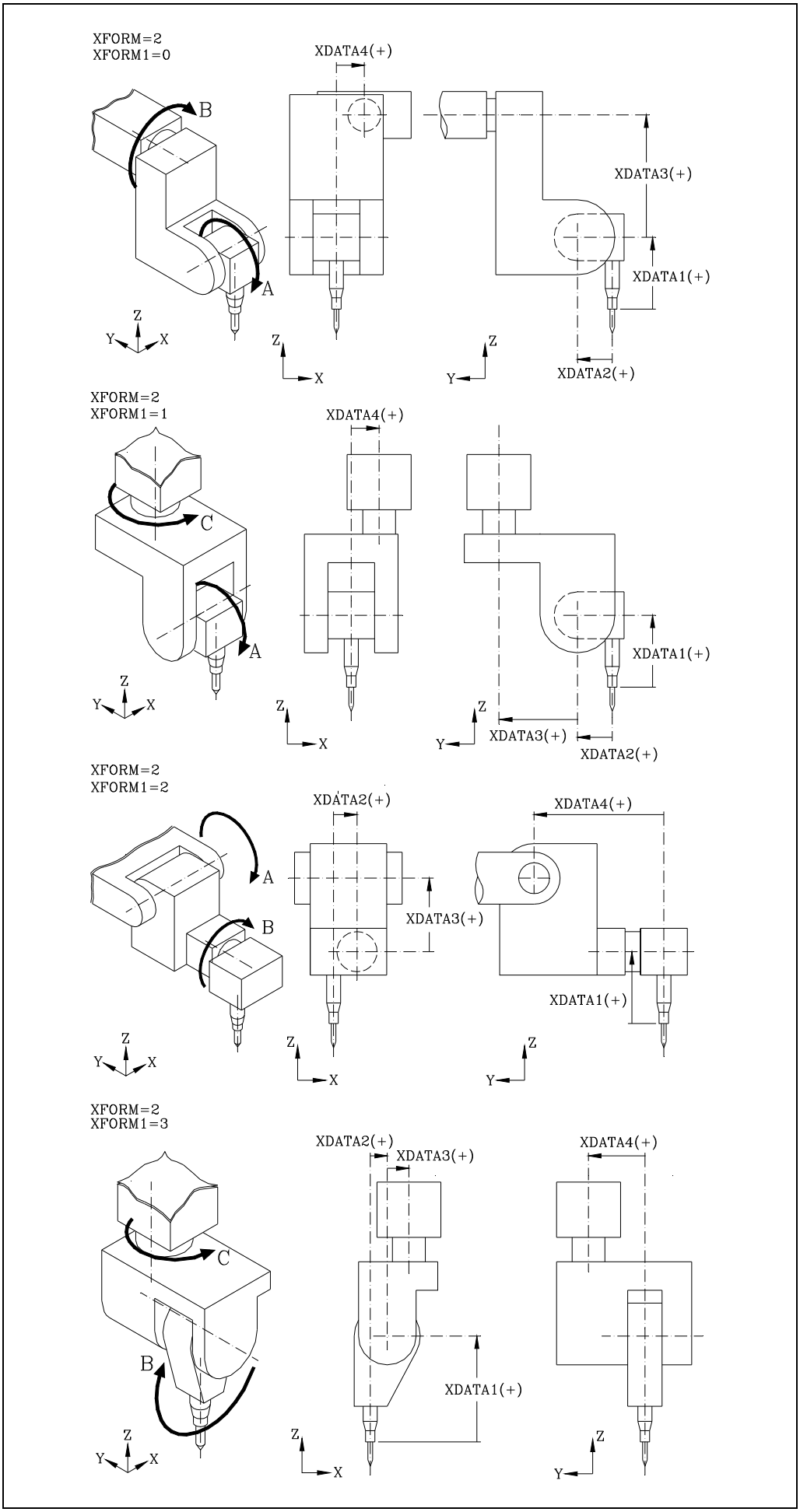
Dual swivel spindle head.

XDATA1 Distance between the spindle nose and the carried rotary axis (secondary)

XDATA2 Distance between the tool axis and the carried rotary axis.

XDATA3 Distance between both rotary axes.

XDATA4 Distance between the tool axis and the main (carrier) rotary axis. This distance must be measured in the direction of the carried rotary axis.

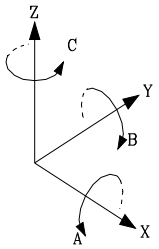

XFORM=2
XFORM1=0
XDATA4(+)
B
A
XDATA3(+)
XDATA1(+)
XDATA2(+)
Z
Y
X
XFORM=2
XFORM1=1
XDATA4(+)
C
A
XDATA1(+)
XDATA3(+)
XDATA2(+)
XFORM=2
XFORM1=2
XDATA2(+)
XDATA4(+)
A
B
XDATA3(+)
XDATA1(+)
XFORM=2
XFORM1=3
XDATA2(+)
XDATA3(+)
XDATA4(+)
C
B
XDATA1(+)

Definition of the kinematics of the angular spindle (XFORM = 3)

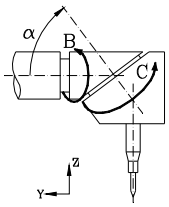
XFORM1 (P94)

It defines the axes of the kinematics and their order, indicating which is the main axis and which the secondary or dragged axis.

Value	Meaning
0	A is the main axis and C is the secondary axis.
1	B is the main axis and C is the secondary axis.
2	C is the main axis and A is the secondary axis.
3	C is the main axis and B is the secondary axis.

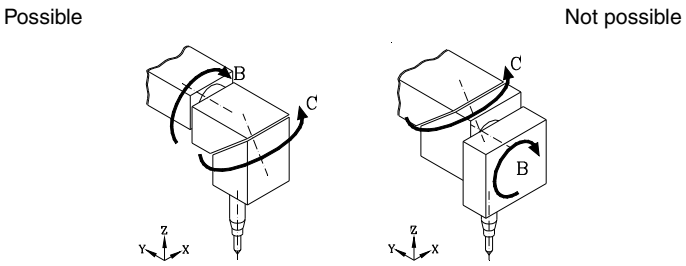


The rotary axes are called A, B or C depending on whether the rotation axis is X, Y or Z respectively. The turning direction of the axes may be changed with parameter "XFORM2".

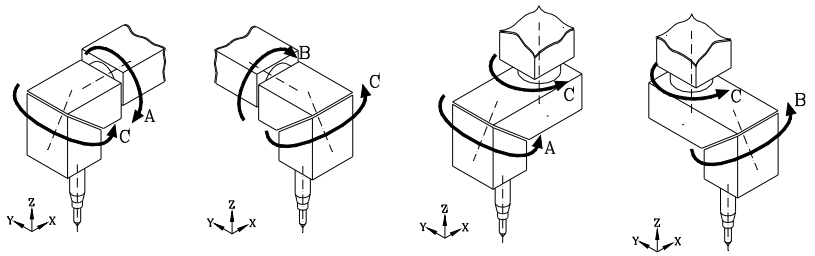


In an angular spindle, the main axis must be parallel to one of the axes X, Y, Z and the secondary axis will be at an angle with respect to it.

In the example, B is the main axis associated with the Y axis and C is the secondary axis associated with the Z axis.



Angular spindle.



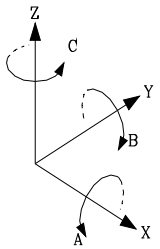
XFORM1 = 0 XFORM1 = 1 XFORM1 = 2 XFORM1 = 3

XFORM2 (P95)

It defines the turning direction of the rotary axes.

Value	Meaning
0	The one indicated by the DIN 66217 standard (see figure).
1	It changes the turning direction of the main axis.
2	It changes the turning direction of the secondary axis.
3	It changes the turning direction of both axes (main and secondary).

Default value: 0

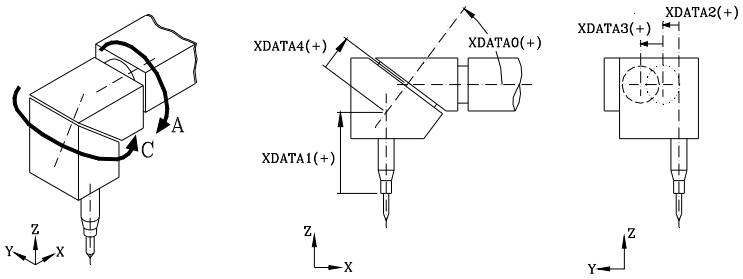


XDATA0 (P96) XDATA1 (P97) XDATA2 (P98) XDATA3 (P99) XDATA4 (P100) XDATA5 (P101) XDATA6 (P102) XDATA7 (P103) XDATA8 (P104) XDATA9 (P105)

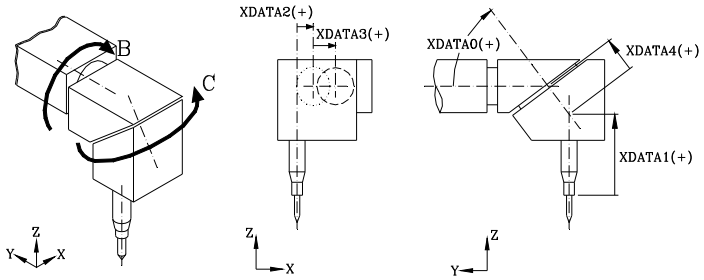
These parameters are used to define the dimensions of the spindle. All of them need not be defined.

XDATA0	Angle in degrees between both rotary axes.
XDATA1	Distance between the spindle nose and the carried rotary axis (secondary)
XDATA2	Distance between the tool axis and the carried rotary axis.
XDATA3	Distance between both rotary axes.
XDATA4	Distance between the tool axis and the main (carrier) rotary axis. This distance must be measured in the direction of the carried rotary axis.

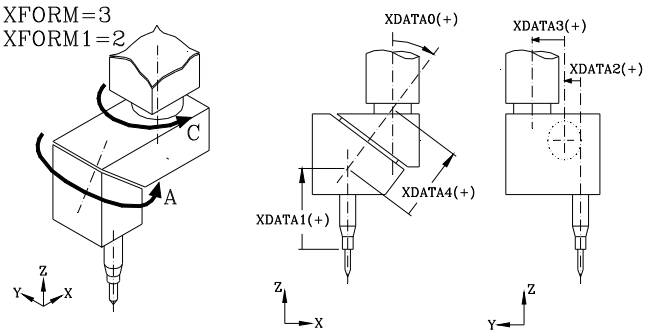
XFORM=3
XFORM1=0



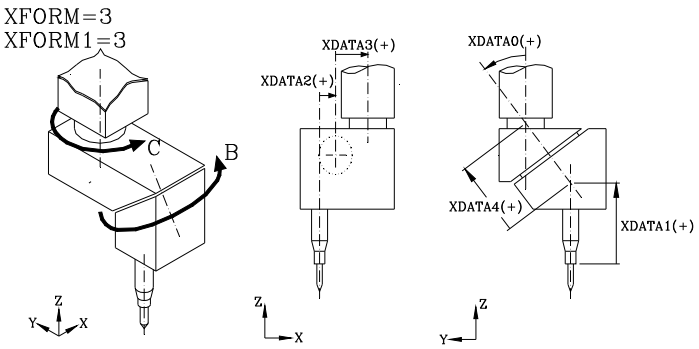
XFORM=3
XFORM1=1



XFORM=3
XFORM1=2



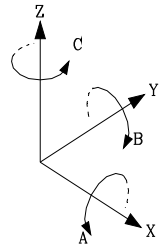
XFORM=3
XFORM1=3



Definition of the kinematics of the angular spindle (XFORM = 4)

XFORM1 (P94)

Sets the spindle axes and their order.

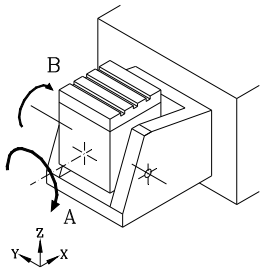


The rotary axes are called A, B or C depending on whether the rotation axis is X, Y or Z respectively. The turning direction of the axes may be changed with parameter "XFORM2".

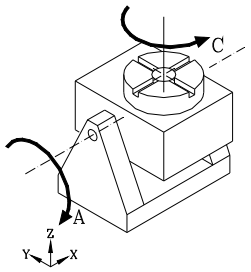
When having a rotary axis "XFORM=4", parameter "XFORM1" indicates which is the main axis and which one is the secondary or "dragged" axis.

Value	Meaning
0	A is the main axis and B is the secondary axis.
1	A is the main axis and C is the secondary axis.
2	B is the main axis and A is the secondary axis.
3	B is the main axis and C is the secondary axis.

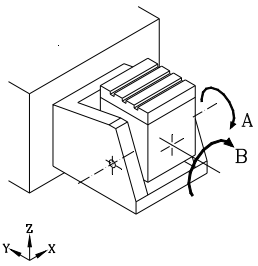
Default value: 0



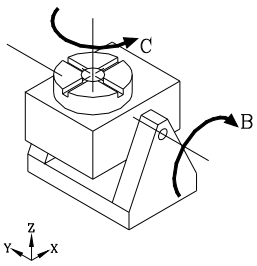
XFORM=4
XFORM1=0



XFORM=4
XFORM1=1



XFORM=4
XFORM1=2



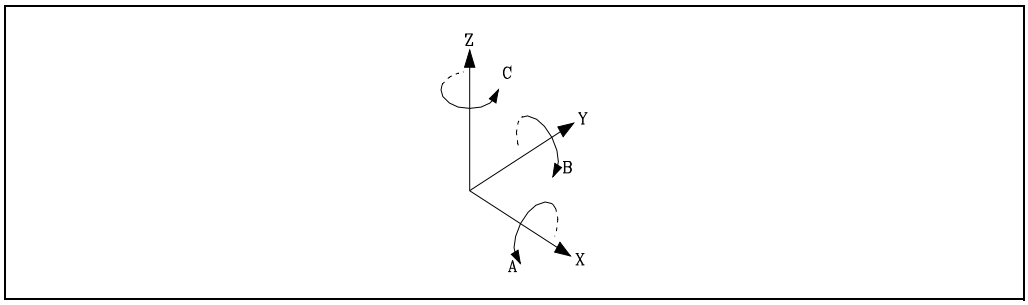
XFORM=4
XFORM1=3

XFORM2 (P95)

It defines the turning direction of the rotary axes.

Value	Meaning
0	The one indicated by the DIN 66217 standard (see figure).
1	It changes the turning direction of the main axis.
2	It changes the turning direction of the secondary axis.
3	It changes the turning direction of both axes (main and secondary).

Default value: 0



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

These parameters are used to define the dimensions of the spindle. All of them need not be defined.

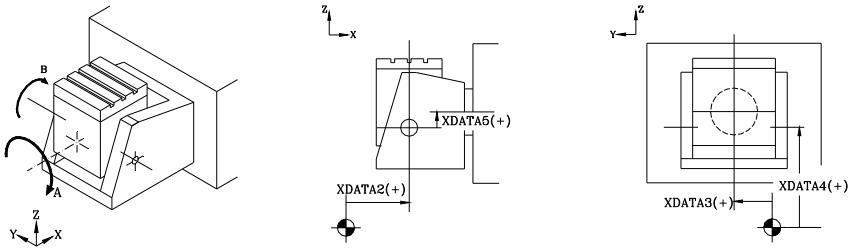
XDATA2 Position of the secondary rotary axis or of the intersection between the primary and secondary axes along the X axis, referred to machine zero.

XDATA3 Position of the secondary rotary axis or of the intersection between the primary and secondary axes along the Y axis, referred to machine zero.

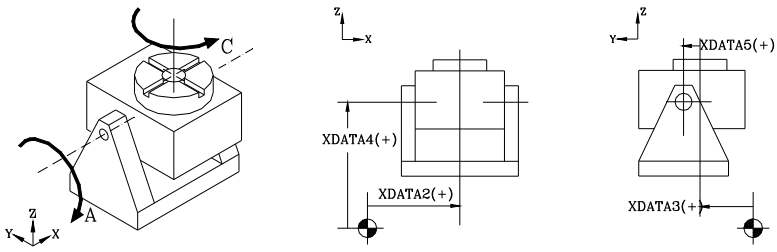
XDATA4 Position of the secondary rotary axis or of the intersection between the primary and secondary axes along the Z axis, referred to machine zero.

XDATA5 Distance from the secondary rotary axis to the main rotary axis.

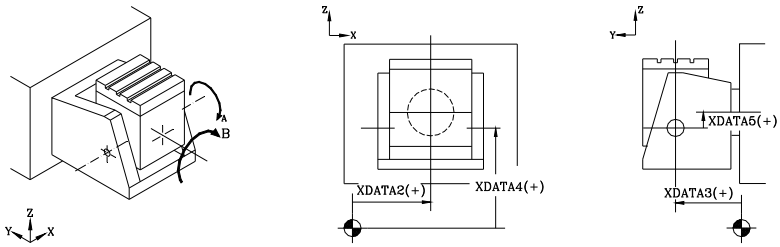
XFORM=4
XFORM1=0



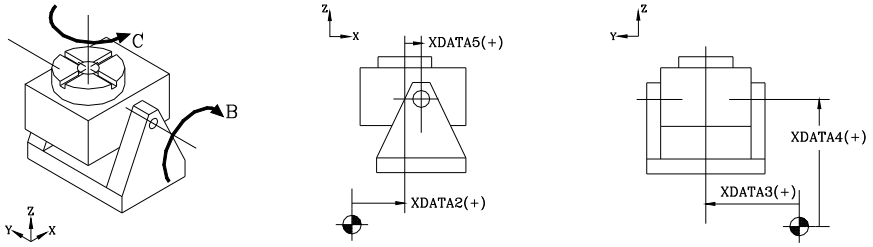
XFORM=4
XFORM1=1



XFORM=4
XFORM1=2



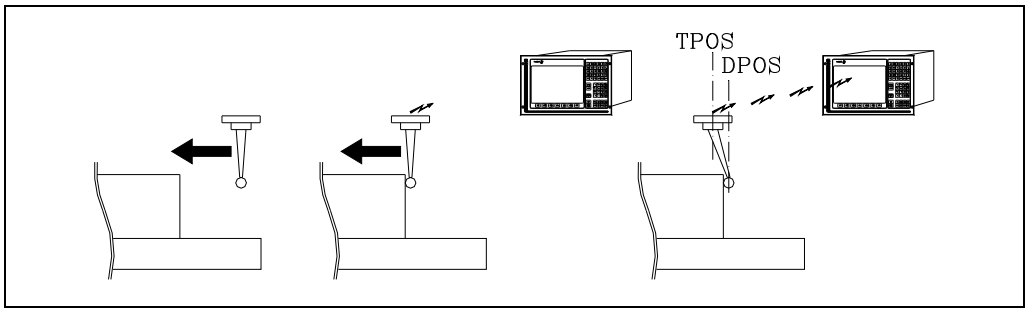
XFORM=4
XFORM1=3



PRODEL (P106)

The CNC takes this parameter into account when probing, functions G75, G76 and Probe cycles PROBE, DIGIT.

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.

To set this parameter, the PROBE2 probe calibration cycle can be used. After it is executed, global parameter P299 returns the best value to be assigned to parameter PRODEL.

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.

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	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 bit of ACTGAINT2 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)

The CNC has specific connectors to connect the handwheels.

Optionally, specific digital PLC inputs (I) may also be used to connect handwheels (signals A and B). In these cases, the handwheels must operate at 24 V.

HANDWIN (P111).

g.m.p. HANDWIN (P111) indicates which input group the electronic handwheels are associated with.

Possible values
0, 17, 33, 49, 65, 81, 97, 113, 129, 145, 161, 177, 193, 209, 225, etc.

HANDWIN = 0 There is no handwheel connected to the PLC inputs.

HANDWIN = 17 Handwheels connected to the input group I17 through I25.

HANDWIN = 33 Handwheels connected to the input group I33 through I41.

HANDWIN = 225 Handwheels connected to the input group I225 through I240.

HANDWIN = 241 Handwheels connected to the input group I241 through I256.

The meaning of these inputs is the following:

I17	I33...	I225	I241	button signal from the handwheel with selector button (only the first one)
I18	I34...	I226	I242	A signal from the first handwheel.
I19	I35...	I227	I243	B signal from the first handwheel.
I20	I36...	I228	I244	A signal from the second handwheel.
I21	I37...	I229	I245	B signal from the second handwheel.
I22	I38...	I230	I246	A signal from the third handwheel.
I23	I39...	I231	I247	B signal from the third handwheel.
I24	I40...	I232	I248	A signal from the fourth handwheel.
I25	I41...	I233	I249	B signal from the fourth handwheel.

HANDWHE1 (P112) / HANDWHE2 (P113) / HANDWHE3 (P114) /HANDWHE4 (P115).

To define the type of handwheel and its associated axis, use the following g.m.p.:

HANDWHE1 (P112) for the first handwheel.

HANDWHE2 (P113) for the second handwheel.

HANDWHE3 (P114) for the third handwheel.

HANDWHE4 (P115) for the fourth handwheel.

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.		

A general handwheel (11 or 12) and up to 3 handwheels associated with the axes can be used simultaneously. There cannot be 2 general handwheels.

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)

It sets the Sercos transmission speed. Regardless of the baudrate used, always respect the values recommended by SERPOWSE.

Value	Meaning
0	4 Mbit/s.
1	2 Mbit/s.
8	8 Mbit/s
16	16 Mbit/s
80	Sercos test. Continuous signal mode.
81	Sercos test. Zero bit stream mode at 2 Mbit/s.
91	Sercos test. Zero bit stream mode at 4 Mbit/s.

Default value: 0

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

SERPOWSE (P121)

It sets the Sercos power or the intensity of the light going through the optical fiber. Its value depends on the length of the cable used from the OUT connection of the Sercos board of the CNC to the IN connection of the Sercos board of the drive.

Possible values (Sercos board).

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

Default value: 0

Assigning other values, for example value 4 for 3 m, causes communications errors due to signal distortion in the optical fiber.

Possible values (Sercos816 board).

Value	Meaning	Recommended type of cable
1, 2, 3, 4	Less than 15 meters.	SFO / SFO-FLEX
5, 6	From 15 to 30 meters.	SFO-FLEX
7	From 30 to 40 meters.	SFO-FLEX
8	More than 40 meters.	SFO-V-FLEX

Default value: 2

Assigning other values, for example value 4 for 17 m, causes communications errors due to signal distortion in the optical fiber.

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)

It indicates whether the spindle is oriented in the canned cycles either using the M19 function or using the "C" axis.

Value	Meaning
0	The spindle positions by means of the M19 function.
1	The spindle is oriented by means of the "C" axis.

Default value: 0

When the machine uses a "C" axis, it is recommended to always orient the spindle using the "C" axis since better accuracy is achieved that way.

AUXSTYPE (P125)

It indicates whether the live tool is handled with the M45 function or like a second spindle (G28 function).

Value	Meaning
0	Using function M45.
1	As second spindle (function G28).

Default value: 0

When a live tool uses several work gear, it must be used like a second spindle. To do that:

- Set "AUXSTYPE (P125)=1".
- Define the machine parameters of the second spindle to set the live tool.
- Use the G28 function to select the live tool.

In the mill model, with "AUXSTYPE (P125)=1" and "STOPTAP (P116)=YES" it is possible to interrupt the execution of the drilling and tapping canned cycles by means of the general inputs /STOP (M5001), /FEEDHOL (M5002) and /XFERINH (M5003).

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)

Number of the file to configure the windows that may be customized.

STEODISP (P128)

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

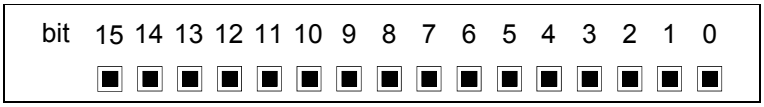
Value	Meaning
0	It displays the real RPM.
1	It displays the theoretical RPM.

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	Handwheel ·3·.	10	
3	Handwheel ·4·.	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, 1, 2 and 3 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 or the "rapid" key is pressed, the movements are executed in rapid.
2	When the mark is activated and the "rapid" key is pressed, the movements are executed in rapid.

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 error messages.
P999995	Texts and titles used by all the OEM screens.
P999994	Help texts of the OEM screens or cycles.

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

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 may 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)

It is used on the lathe model CNC. Indicates the number of the subroutine associated with function G15.

Possible values
Integer numbers between 0 and 9999.

Default value: 0 (no associated subroutine)

When there is an associated subroutine, the CNC acts as follows:

- If there is no other G15 inside the associated subroutine, the G15 will be executed after the subroutine.
- If there is another G15 inside the associated subroutine, it will execute this G15 without calling the subroutine and after executing the associated subroutine, it will not execute the G15 again.

This feature may be used to change the set of drive parameters when switching from spindle mode to C axis mode.

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.

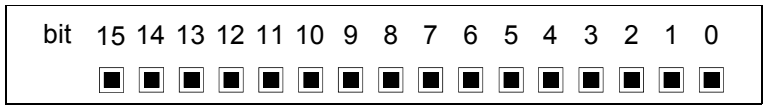
Define which connector is each handwheel associated with. Values 1 through 8 must be assigned to these parameters depending on the connector that the handwheel is associated with.

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

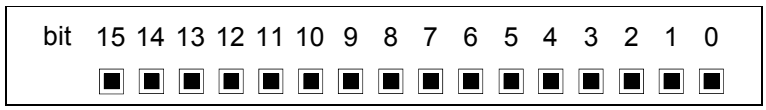
Bit 13. Backlash compensation on arcs G2/G3.

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.



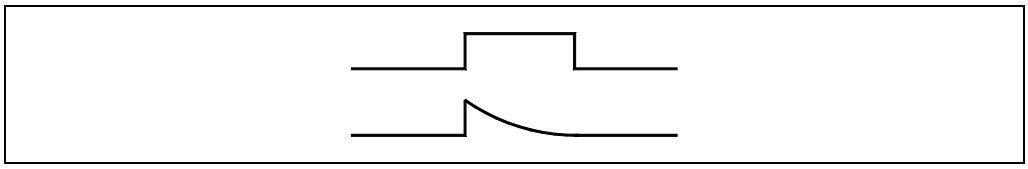
Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of ·0·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning	Bit	Meaning
0	Exponential backlash peak.	8	
1	Minimizes the inside backlash peaks when changing quadrants.	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.

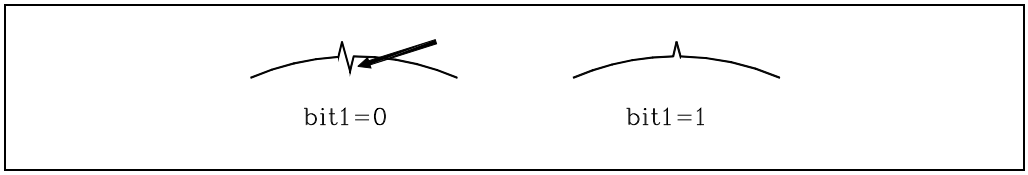
The additional command pulse used to make up for the possible leadscrew backlash in movement reversals may be rectangular or exponential. This bit indicates whether it applies a rectangular backlash peak (bit=0) or an exponential backlash peak (bit=1).



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 1. Eliminate inside backlash peaks when changing quadrants.

This bit indicates whether the inside backlash peaks are eliminated (bit=1) or not (bit=0) when the axis changes its moving direction.



In these conditions, the CNC eliminates the additional command peak in the second position loop after detecting that the moving direction has changed. If the internal peaks are not eliminated, adjust the leadscrew backlash compensation better.

A fine adjustment of the leadscrew backlash consists in testing the circle geometry and watch for internal peaks when changing quadrants (left figure).

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).

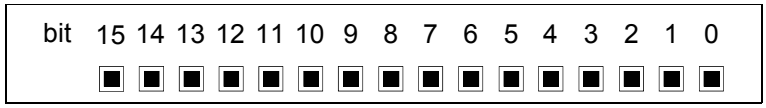
Possible values
Integer numbers between 0 and 65535.

Default value: 0

CODISET (P147)

This parameter is associated with the MC / TC work mode.

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	There are auxiliary M functions in the cycles.
1	It is not possible to access cycles or programs from the auxiliary screen.
2	The CNC is configured as having two and a half axes.
3 - 4	Not being used.
5	Plane change option.
6	It shows the coolant icon.
7	It enables the option to select roughing or finishing.
8	Enables the possibility of defining the thread pitch as number of threads per inch.
9	It enables the option to program variable-pitch threads.
10	Enables zig-zag penetration by the flanks (lathe only).
11	Enables the option for repairing part of a thread (lathe only).
12	Enables the option for repairing threads with multiple entries or starts (lathe only).
13 - 15	Not being used.

Default value in all the bits except bit 7: 0
Default value of bit 7: 1

Bit 0. There are auxiliary M functions in the cycles of the MC work mode.

This bit indicates whether the operations or cycles have (bit=1) or not (bit=0) auxiliary M functions associated with roughing and finishing operations.

Enabling the auxiliary M functions in the roughing and finishing operations of the cycles displays a window where up to 4 auxiliary M functions. These functions are executed at the beginning of the operation (roughing or finishing) where they have been defined.

Bit 1. It is not possible to access the cycles or the programs from the auxiliary screen.

This bit indicates whether it allows (bit=0) or not (bit=1) access the cycles and programs from the auxiliary screen. With restricted access, the program selected for execution cannot be removed either with [CLEAR].

Bit 2. The CNC is configured as having two and a half axes (MC model).

This bit indicates whether the CNC is configured as two and a half axes (bit=1) or not (bit=0).

A two-and-a-half-axis configuration is a milling machine where the X and Y axes are motorized and the Z axis is set as a DRO axis (display only). In this configuration, the Z axis is moved manually.

The interface and the machining cycles are adapted for this type of configuration. Since not all the cycles are applicable to a two-and-a-half-axis configuration, it will be necessary to set parameters COCYZ, COCYF1 and COCYF5 to hide them.

Bit 5. It enables the plane change option (MC model).

This bit indicates whether the plane change option of the programming assistance of the MC model is enabled (bit=0) or disabled (bit=1).

Bit 6. It shows the coolant icon (MC/TC models).

This bit indicates whether the coolant icon is displayed (bit1=0) or hidden (bit1=1) in the cycles of the MC/TC model.

Bit 7. It enables the option to select or unselect roughing or finishing.

This bit indicates whether the roughing or finishing operation may be selected (bit=1) or not (bit=0).

Bit 8. Enables the possibility of defining the thread pitch as number of threads per inch.

This bit enables (bit=0) or disables (bit01) the option to define the thread pitch as number of threads per inch.

The number of threads per inch can only be entered when not working with any particular thread defining system; i.e. when using a FREE system.

Bit 9. It enables the option to program variable-pitch threads.

This bit enables (bit=0) or disables (bit=1) the option to program variable-pitch threads.

Bit 10. Enables zig-zag penetration by the flanks (lathe only).

This bit enables (bit=0) or disables (bit01) the option to penetrate in zig-zag by the flanks in lathe type threading cycles.

Bit 11. Enables the option for repairing part of a thread (lathe only).

This bit enables (bit=0) or disables (bit01) the option for thread repair in lathe type threading cycles.

Bit 12. Enables the option for repairing threads with multiple entries or starts (lathe only).

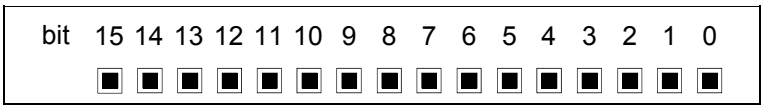
This bit enables (bit=0) or disables (bit01) the option for multiple-entry thread repair in lathe type threading cycles.

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

In the TC and MC work modes, it allows hiding the operations or cycles not being used showing only the desired ones.

Each parameter is associated with an operation or cycle and each one of their bits refers to each available level.

This parameter has 16 bits counted from right to left.



By default, all the bits will be assigned the value of ·0· (option available). Assigning the value of ·1· hides the level of the corresponding cycle.

Setting of a CNC configured as two-and-a-half axes.

Since not all the cycles are applicable to a two-and-a-half-axis configuration, it will be necessary to set parameters COCYZ, COCYF1 and COCYF5 as follows.

COCYZ (P155) 0000 0000 0100 0110

It hides the tapping, drilling 2 and drilling 3 cycles.

COCYF5 (P152) 0000 0000 0000 0010

It hides the 3D profile pocket cycle.

COCYF1 (P148) 0000 0000 0000 0010

It hides the PROBE 1 cycle.

Meaning of the parameters in the MC work mode.

Parameter	Operations or cycles			
COCYF1		Probe 1 (Bit 1)		Probe calibration (Bit 3)
COCYF2				
COCYF3	Profile milling 1 (Bit 0)	Profile milling 2 (Bit 1)		
COCYF4	Surface milling (Bit 0)	Slot milling (Bit 1)		
COCYF5	Pocket with 2D profile (Bit 0)	Pocket with 3D profile (Bit 1)		
COCYF6	Rectangular boss (Bit 0)	Circular boss (Bit 1)		
COCYF7	Rectangular pocket 1 (Bit 0)	Rectangular pocket 2 (Bit 1)	Circular pocket 1 (Bit 2)	Circular pocket 2 (Bit 3)
COCYZ	Drilling 1 (Bit 0) Thread milling (Bit 4) Boring 1 (Bit 11)	Drilling 2 (Bit 1) Tapping (Bit 6) Boring 2 (Bit 12)	Drilling 3 (Bit 2) Bore milling (Bit 7)	Drilling 4 (Bit 3) Reaming (Bit 9)
COCYPOS	Positioning 1 (Bit 0)	Positioning 2 (Bit 1)		
COCYPROF				
COCYGROO				

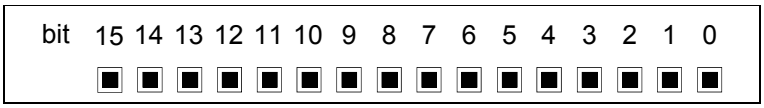
Parameter	Multiple positioning ...			
COCYZPOS	.... at several points (Bit 0) ... in grid pattern (Bit 11)	... in line (Bit 3) ... in parallelogram (Bit 12)	... in arc (Bit 6)	... in polar arc (Bit 7)

Meaning of the parameters in the TC work mode.

	Bit 0	Bit 1	Bit 2	Bit 3	Bit 4
COCYF1					
COCYF2	Turning 1	Turning 2			
COCYF3	Facing 1	Facing 2			
COCYF4	Taper turning 1	Taper turning 2	Taper turning 3		
COCYF5	Rounding 1	Rounding 2			
COCYF6	Threading 1	Threading 2	Threading 3	Threading 4	Threading 5
COCYF7	Grooving 1	Grooving 2	Grooving 3	Grooving 4	Cut-off
COCYZ	Drilling 1	Drilling 2	Drilling 3	Drilling 4	Drilling 5
COCYPOS	Positioning 1	Positioning 2			
COCYPROF	Profile 1	Profile 2	Profile 3	Profile 4	Profile 4
COCYGROO					
COCYZPOS					

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·, except bits 12, 14 and 15 that will be assigned a value of ·1·. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning
0	It may be used to apply jerk control in look-ahead.
1...11	Not being used.
12	It activates/deactivates the speed control in arcs with linear blocks in look-ahead.
13	It makes it possible to use Fagor filters with Look-ahead (standard look-ahead algorithm and with jerk control).
14	Activates/deactivates smoother machining.
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 12. Speed control in arcs with linear blocks in look-ahead.

0 Value: It deactivates the speed control in arcs with linear blocks in look-ahead.

1 Value: It enables the speed limit in arcs with linear blocks in look-ahead. This provides more accurate and smoother machining when machining arcs of a small radius programmed with linear blocks (G1).

This function could extend machining time slightly.

This speed control may be applied with the standard look-ahead algorithm, with the advanced look-ahead algorithm or with Jerk control. Plus, it takes into account the limitation set by a.m.p. JERKLIM (P67) in arcs formed by linear blocks when using the jerk control algorithm.

Default value: 1.

Bit 13. Use Fagor filters with look-ahead.

Fagor filters always work with the advanced look-ahead algorithm, but they can also be used in the standard algorithms and with jerk control using this bit 13, in which case, only those Fagor filters will be used.

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

1 Value: Fagor filters will be used in all movements. In look-ahead (standard and with jerk control), 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.

0 Value: Deactivates smoother machining.

1 Value: Activates smoother machining. This smoothing is especially noticeable when using the standard and advanced algorithms.

Activating this bit has the following effects:

–It activates a filter on the machining path.

–It does not limit the speed in very small blocks.

Default value: 1.

We recommend the value of bits 12 and 14 to be the same.

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

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

TLOOK (P161)

Not being used

MAINTASF (P162)

This parameter is associated with the MC and TC work modes.

This parameter indicates whether on CNC power-up, the F, S and Smax values are maintained or initialized to zero.

Value	Meaning
0	They are initialized with the values of F=0, S=0 and Smax=0.
1	F, S, Smax keep the values of the last machining operation.

Default value: 0

If the parameter is set to ·1· (maintain the values), the CNC acts as follows on power-up.

- The CNC assumes the G94/G95 feedrate set by g.m.p. IFEED, but it restores the F in mm/min (G94) and the F in mm/rev (G95) programmed last.
- It maintains the feedrate type G96/G97 used last, but it restores the S in rev/min (G97) and the S in m/min (G96) programmed last.

CAXGAIN (P163)

It permits maintaining the proportional gain (FFGAIN) and derivative gain while machining in the XC and ZC planes. By default, these gains are canceled automatically for smoother machining.

Value	Meaning
0	The gains are canceled.
1	The gains are not canceled.

Default value: 0

With "CAXGAIN (P163) = 1" and high values of both gains, the machine could be jerky depending on the type of certain parts. When that happens, we recommend to select a gain set with very small values or zero.

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)

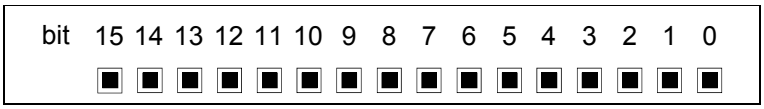
Tool wear offsets may be modified from tool inspection mode. This parameter indicates the maximum amount of wear that may be entered for "K" (it is programmed in mm or in inches)..

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.

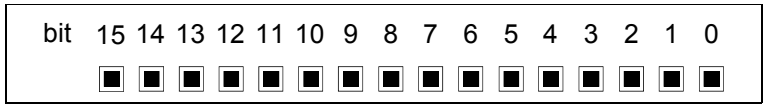
Remember that when having a subroutine associated with the "T" function, the tool change is carried out as follows:

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)

For a speed of 1Mbit/s, the number of elements available at the CAN bus, depending on loop time, will be:

LOOPTIME (P72)	Number of elements
4 ms	4 elements (axes + spindles).
5 ms	5 elements (axes + spindles).
6 ms	6 elements (axes + spindles).
2 or 3 ms	Not allowed.

Neither the fast channel nor the oscilloscope will be available in a configuration of LOOPTIME = 6ms and 6 elements at the bus.

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.

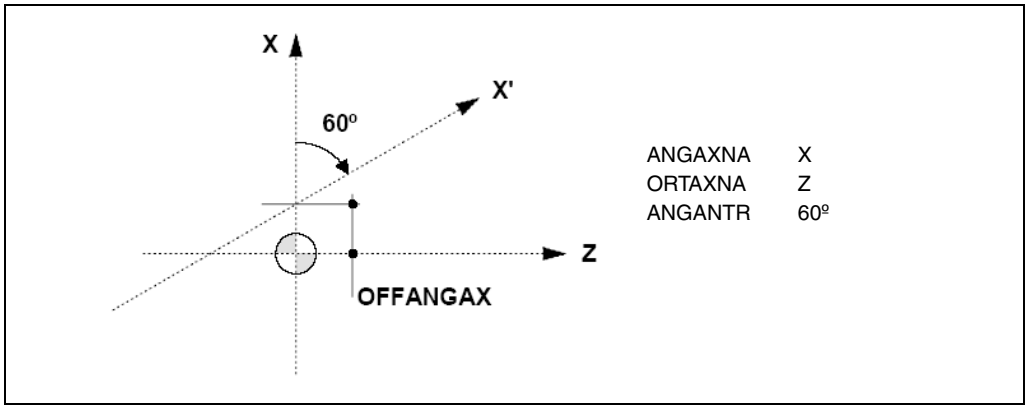
ANGAXNA (P171)

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. These axes may have associated with them Gantry axes, coupled axes or axes synchronized by PLC.

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.

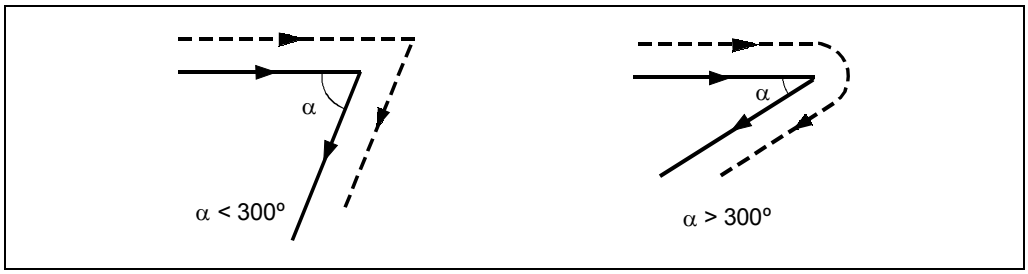
The compensation method depends on the angle between paths.

- 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.

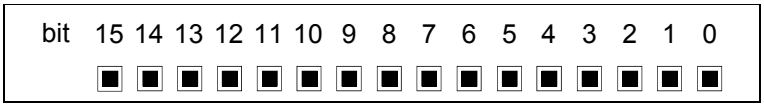


ADIMPG (P176)

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, slaved or synchronized by PLC, 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

When setting this bit to 1, even if there is a general handwheel, the additive handwheel will always be the handwheel 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)

It allows setting the Sercos transmission delay when working at 8 MHz or at 16 MHz. The default delay is 400µs and this parameter may be used to set a delay of 600µs.

Value	Meaning
0	It sets a delay of 400 µs.
400	It sets a delay of 400 µs.
600	It sets a delay of 600 µs.

Default value: 0 (400 µs.)

Setting a longer delay at the bus permits increasing the amount of data going through the fast channel. This may be necessary, for example, when having set many PLC parameters (SRR700 to SRR739) that define the fast channel or when accessing variables of several drives from the oscilloscope.

In any case, if the CNC detects a limit situation in the bus capacity, it will show a message suggesting to increase this parameter.

When changing the parameter value from 0 (same as 400 µs) to 600 on a machine using the fine adjustment, i.e. with "zero following error", we recommend to slightly increase the value of the DERGAIN parameter (when ACFGAIN = YES) to compensate for the additional 200 microsecond delay.

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: 1

REPOSTY (P181)

It allows selecting 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.

ISOSIMUL (P183)

In the conversational mode of the CNC, it is possible to generate an ISO-coded program with some basic G functions as well as M and T functions from an operation (cycle) or on a part-program.

This parameter identifies the number of the ISO program generated in user RAM memory.

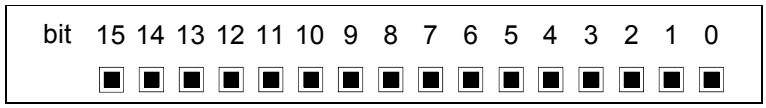
Value	Meaning
0	An ISO-coded program cannot be generated.
1 - 65535	Indicates the number of the ISO-coded program generated.

Default value: 0

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

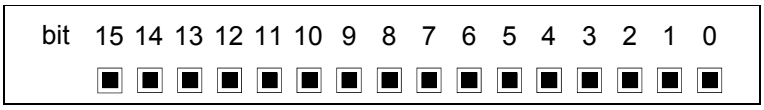
bit 10 = 1	RAPID [S0]
bit 11 = 1	RAPID
bit 12 = 1	MAIN PLANE
bit 13 = 1	G, M, S, T FUNCTIONS
bit 14 = 1	G FUNCTIONS
bit 15 = 1	THEORETICAL PATH

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 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	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 bit of ACTGAINT2 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 G1 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)

Enables or disables the various retracting options for drilling or 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)

Rigid tapping without sending M functions to the PLC.

If bit 0 of g.m.p. TAPTYPE (P188) = 1, the M3, M4 and M5 that are executed in rigid tapping will 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 will be faster.

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

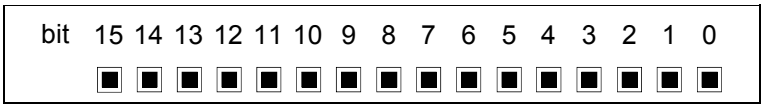
Default value: 1

MANTFCON (P189)

While executing in look-ahead (G51), some blocks of the program cause the feedrate to slow down almost to zero, hence 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.

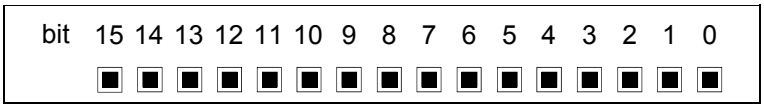
STARTDIS (P190)

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).

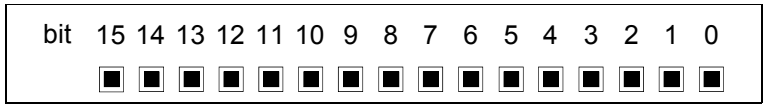
Value	Meaning
0	The program is sent to the CNC and is executed.
1	The program is sent to the CNC and is not executed until the START key is pressed.

Default value: 1

LCOMPTYP (P191)

It may be used to define whether to maintain or change the longitudinal axis 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

G16SUB (P192)

General machine parameter G16SUB (P192) indicates the number of the subroutine associated with function G16.

Possible values
Integer numbers between 0 and 9999.

Default value: 0 (no associated subroutine)

When there is a subroutine associated with G16, the CNC acts as follows:

- If there is no other G16 inside the associated subroutine, the G16 will be executed after the subroutine.
- If there is another G16 without XC or ZC in the subroutine, this G16 will be executed with the call parameters (XC, ZC) and without calling the subroutine. After executing the associated subroutine, the G16 will not be executed again.

The programmed value is given to this subroutine as call parameter in CALLP:

```
XC CALLP= $800004
ZC CALLP= $2000004

XCZ CALLP= $10800004
ZCX CALLP= $12000004
```

NEWLOOK (P193)

General machine parameter NEWLOOK (P193) may be used to select the desired interface for the CNC.

Value	Meaning
0	Interface A/Plus.
1	Interface FL/Power.

Default value: 1

6.4 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 "7.1 Axes and coordinate systems" on page 285.

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)

Several Gantry pairs are allowed. 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)

It is possible to couple or decouple each one of the axes by PLC program using the logic inputs of the CNC: "SYNCHRO1" through "SYNCHRO7". It is possible to couple CNC axes (main channel) or PLC axes.

This parameter, set for the axis coupled as slave, indicates which axis will be its master.

Value	Meaning	Value	Meaning
0	None.	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

Example:
To couple the V axis with the X axis, program as follows:
Parameter SYNCHRO (P3) of the X axis = 0
Parameter SYNCHRO (P3) of the V axis = 1 (associated with the X axis)
When the PLC activates the logic input "SYNCHRO" of the CNC corresponding to the V axis, this axis will be electronically coupled to the X axis.

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 or Sercos 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; it is sent via Sercos (DRIBUSLE = 0) or 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

On an 8055 CNC with –Vpp Axes– module, the feedback connectors of the first 4 axes are for differential TTL and Vpp signals. This parameter is ignored for these four axes. Use Fagor signal adapter "SA-TTL-TTLD" (from "non-differential TTL" to "differential TTL") to connect non-differential signals to these connectors.

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 1000.

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. 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

$$= \left(\xi \cdot PROGAIN + \frac{\xi \cdot DERGAIN}{10 \cdot t}\right)$$

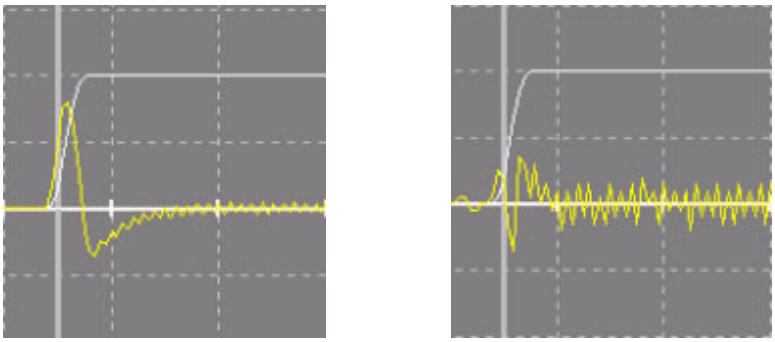
To apply this gain to an axis, that axis should be working with acc/dec [a.m.p. ACCTIME (P18) other than 0].

Possible values
Integer numbers between 0 and 65535.

Default value: 0 (derivative gain not applied)

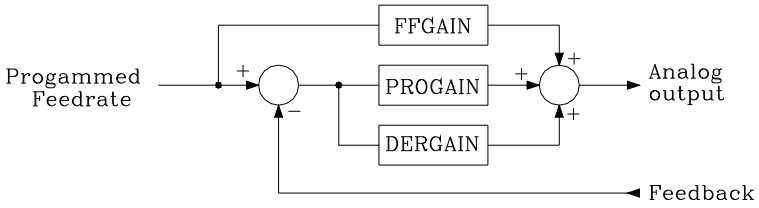
The best adjustment is achieved when minimizing the following error as much as possible but without inverting the peaks. The peaks of the right graph are inverted. Bad adjustment.

The graph on the left shows the system response without DERGAIN (10 μm per square) and the one on the right with DERGAIN (1 μm per square).



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

$$= \left(\xi \cdot PROGAIN + \frac{\dot{\xi} \cdot DERGAIN}{10 \cdot t} + \frac{FFGAIN \times Fprog \times MAXVOLT}{100 \cdot G00FEED} \right)$$

The feed-forward gain improves the position loop minimizing the following error, but it should only be used when working with acceleration/deceleration.

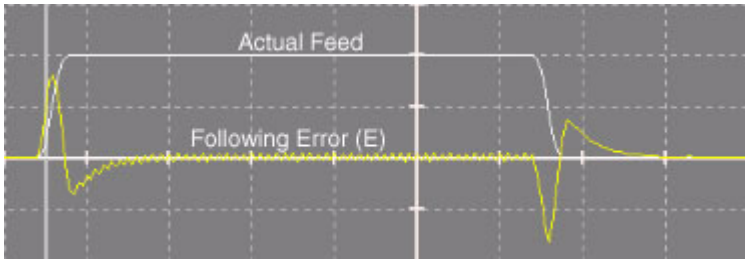
Possible values
Between 0 and 100.99 (numbers with two decimals).

Default value: 0 (feed-forward gain not applied)

The best adjustment is achieved when the following error is minimized as much as possible, but without changing its sign, maintaining the moving direction of the axis.

The scale for the following error is 10µm per square.

- Proper adjustment with Feed-forward.



- Wrong adjustment with Feed-forward.



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

When controlling two axes with a single servo drive, the direction of the analog command must be set on both axes.

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.

With Sercos connection, when the drive uses absolute feedback, instead of parameter "REFVALUE", it takes into account its equivalent "SERCOS PP177" of the drive.

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)

Indicates the maximum difference allowed between the following errors of the axes electronically coupled (by program, PLC or as 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).

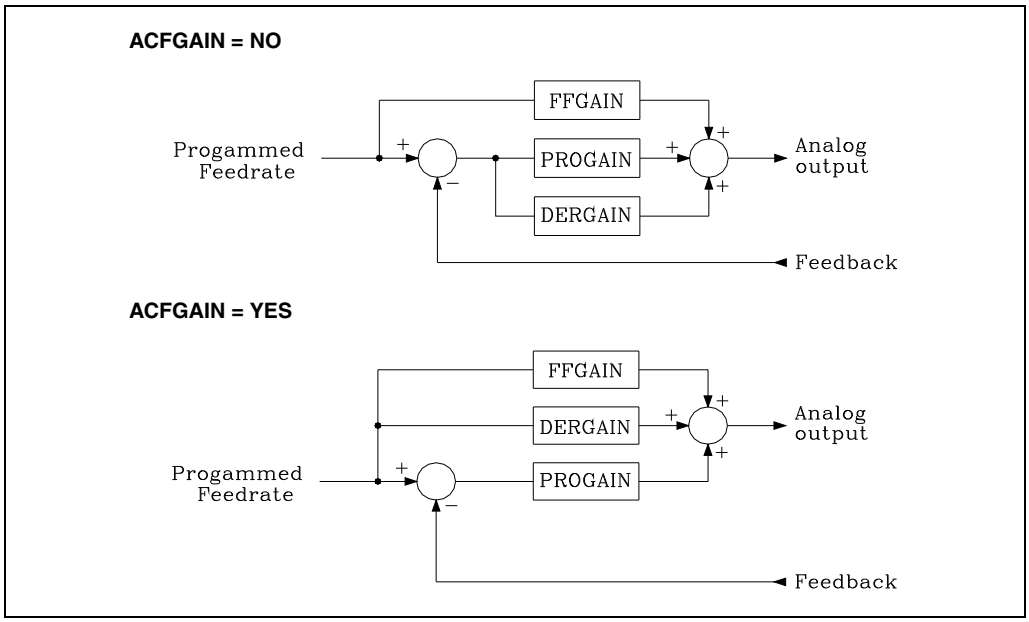
Value	Meaning
NO	It is applied on variations of following error (derivative gain).
YES	It is applied on the variations of the programmed feedrate that are due to acceleration/deceleration (AC-forward).
ADVANCED	It is used in a digital drive system and provides a smoother and more stable adjustment of the machine.

Default value: YES

If the value of a.m.p. ACFGAIN (P46) is changed, DERGAIN will have to be readjusted.

In the digital system a.m.p. ACFGAIN(P46) must be set to ADVANCED. This parameter setting allows a smoother and more stable adjustment of the machine using a lot lower DERGAIN values than those used when a.m.p. AFCGAIN(P46) =YES. This is valid for axes and spindles.

All axes of the channel, CNC or PLC, must have the same value for a.m.p. ACFGAIN(P46). When setting ACFGAIN=ADVANCED in an analog system, it will be treated as AFGAIN=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 ±838.8608 degrees or millimeters. Within ±33.026 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).

Tens.

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)

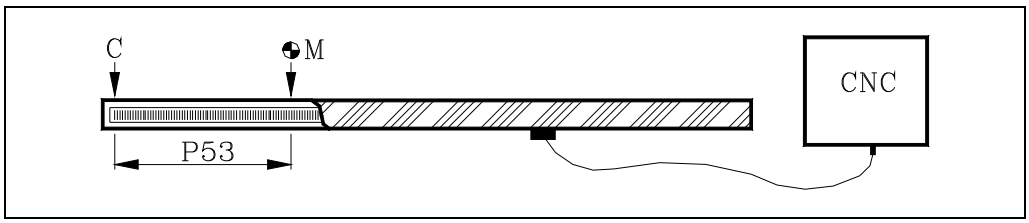
The CNC takes this parameter into account when a.m.p. I0TYPE (P52) has been set with a value other than 0.

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 "ANT6".

If the axis move is smaller than the value indicated by this a.m.p. MINMOVE (P54), the corresponding axis logic output "ANT1 through "ANT6" 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 (Sercos or 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 Sercos axes and 1 Sercos spindle, the values of this parameter must be 1, 2, 3 and 4. Proceed in the same way if it is a CAN connection.

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

$$\text{EXTMULT (P57)} = \frac{\text{Period of the graduation on the glass (mechanical period or pitch)}}{\text{Period of the feedback signal (electrical period)}}$$

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 GOP MOT COT FOP SVOP MOC COC MOP COP	1000	1001	1
SOX GOX MOX COX FOT SVOX	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. Movements with handwheel or when the CNC internally transforms the programmed coordinates (C axis, RTCP, 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 (Sercos or CAN). Axis parameter DRIBUSID (P56) other than 0.

Even when the data exchange between the CNC and the drive is done via digital bus (Sercos or CAN), 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	Feedback is handled via digital bus (Sercos or CAN). First feedback (motor feedback).
2	Feedback is handled via digital bus (Sercos). Second feedback (direct feedback).

When using a communication interface via Sercos bus.

DRIBUSLE = 0	The CNC controls the position loop. The axis feedback is done via connector. The command to the drive is sent out via Sercos.
DRIBUSLE = 1	The CNC controls the position loop. The axis feedback is done via Sercos. First feedback (motor feedback). The command to the drive is sent out via Sercos.
DRIBUSLE = 2	The CNC controls the position loop. The axis feedback is done via Sercos. Second feedback (direct feedback). The command to the drive is sent out via Sercos.

When using a communication interface via CAN bus.

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)

Usually when working with Sercos feedback, the motor-drive system has an absolute encoder. Thanks to this, the system knows at all times the relative position of the axis per each turn of the motor.

In these cases, when searching home on the axis, the CNC knows the position of the axis as soon as the home switch is pressed; thus not being necessary to move all the way to the machine reference point.

Parameter POSINREF indicates whether the axis has to move to a marker pulse or not after hitting the home switch.

Value	Meaning
NO	It does not move.
YES	It moves.

Default value: NO

When the distance-coded reference mark is managed through the drive's second feedback, it is recommended to set parameter POSINREF with "NO". Otherwise, the axis moves to the position set in REFVALUE.

The movement to the reference point is made at the feedrate set by a.m.p. REFEED1 (34). With P34=0, it moves at F0.

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. See *"7.13 Axes (2) controlled by a single drive"* on page 362.

When wishing to make an additive coupling between axes, parameter SWITCHAX of the dro axis indicates which is the main axis it is associated with. A typical application of additive coupling takes place on milling machines when the Z axis has a second axis W coupled to it that moves by hand. See *"7.14 Additive coupling between axes"* on page 368.

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

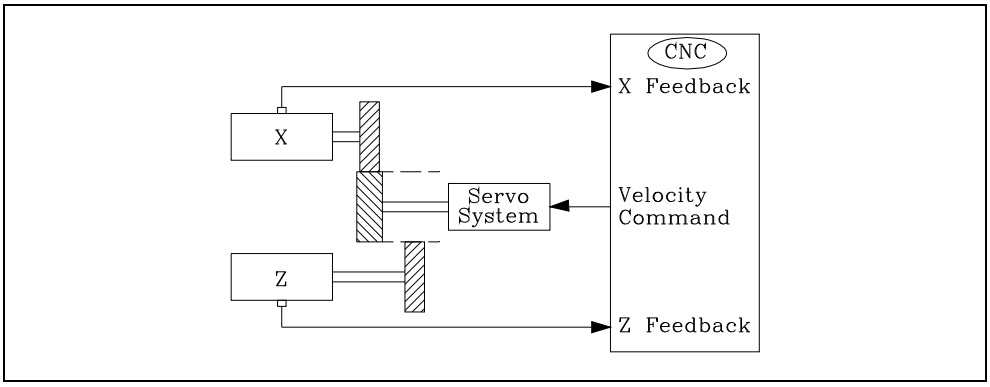
Parameter SWINBACK (P66) must also be set when having two axes controlled by a servo system or wishing to do an additive coupling,

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)

This parameter indicates whether an additive coupling is being applied between axes or there are two axes controlled by a single servo system. 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.
2	It uses the feedback of the main axis, but it has its own velocity command.
10	For an additive coupling.

Default value: 0

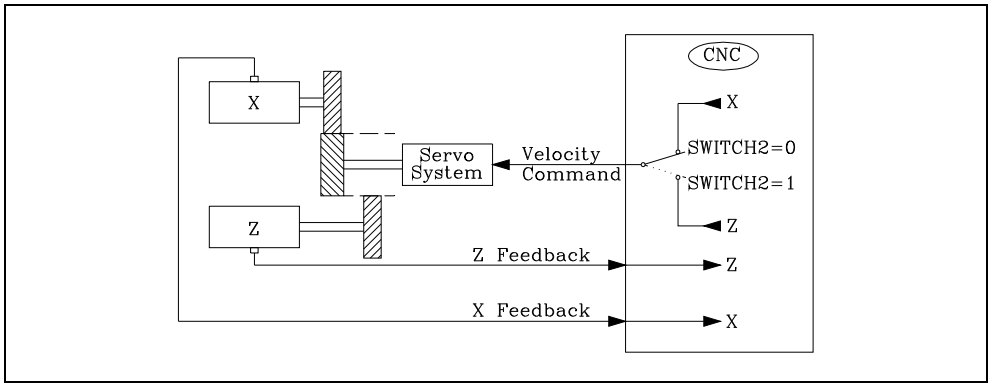
See *"7.13 Axes (2) controlled by a single drive"* on page 362.

The following examples show the various possibilities of two axes controlled by a single servo system. In all of them, the toggling of the velocity command must be done from the PLC using the SWITCH2 mark.

Each axis has its own feedback device.

X axis (main) SWINBACK for X axis = 0.

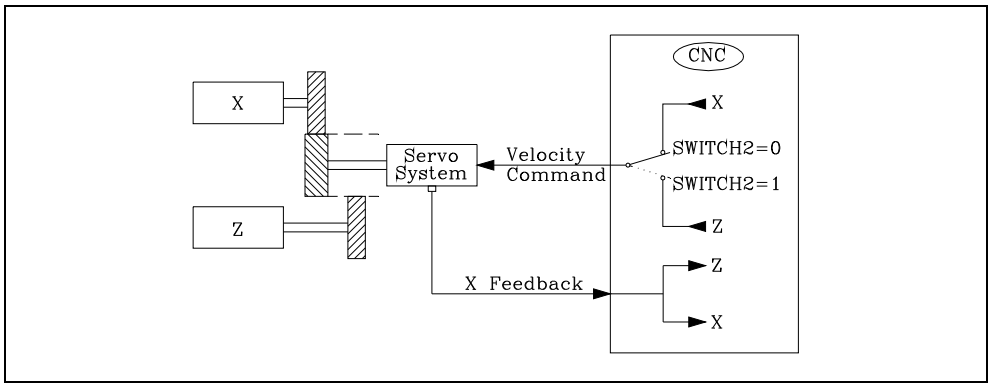
Z axis (secondary) SWINBACK for Z axis = 1.



The two axes share the same feedback device. It must be connected to the feedback connector of the main axis.

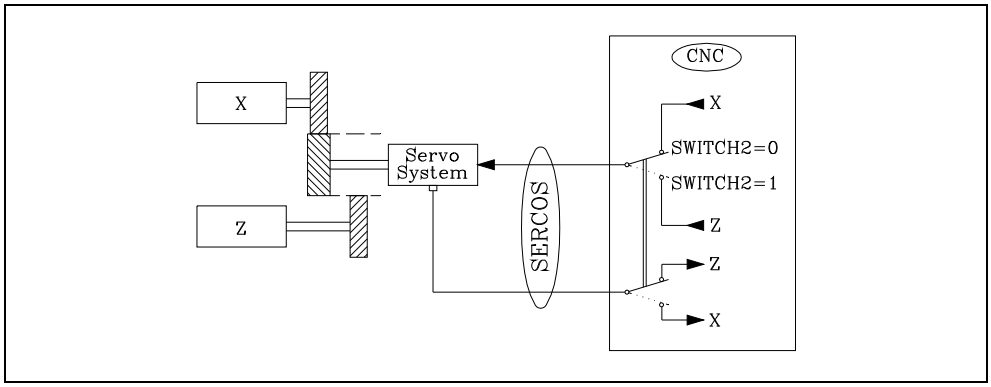
X axis (main) SWINBACK for X axis = 0.

Z axis (secondary) SWINBACK for Z axis = 0.



The communication with the drive is done through Sercos, feedback included.

X axis (main)	SWINBACK for X axis = 0.
Z axis (secondary)	SWINBACK for Z axis = 1.



The CNC internally switches the feedback it receives via Sercos and it supplies it to either axis depending on the status of the SWITCH2 mark.

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.
If a.m.p. TYPE (P71) = 0 or 1, the filters are not applied in electronic or mechanical handwheel movements.
It is recommended not to activate these filters on machines carrying out movements against a hard stop.

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.

TYPE (P71)

Filter type. There are three types of filter: "low passing", "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.

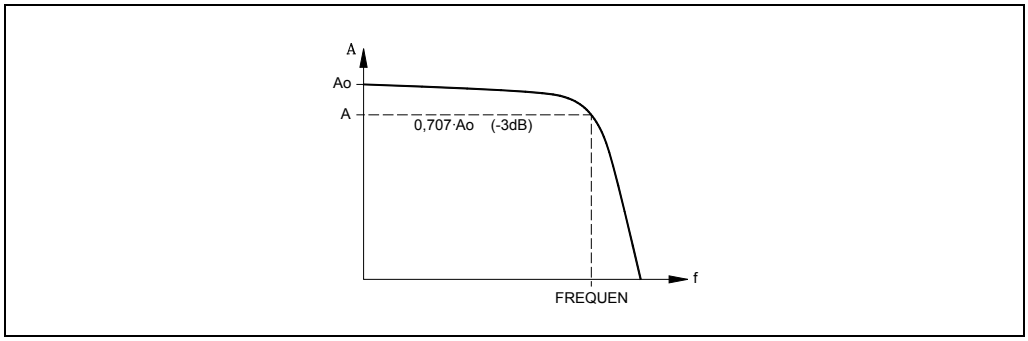
Value	Meaning
0	"Low passing" filter.
1	"Anti-resonance" (notch) filter.
2	FAGOR "low passing" filter.

Default value: 0

When defining anti-resonance filters, parameters NORBWIDTH and SHARE must also be set.

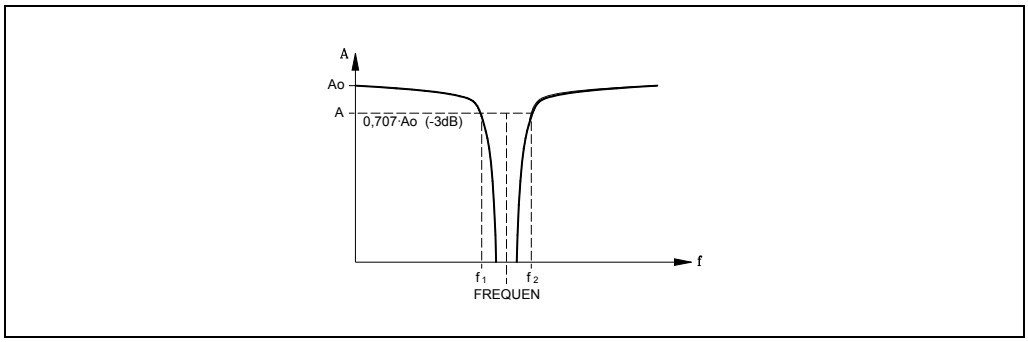
"Low passing" filter.

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.



CNC start-up with Fagor filters active.

If on CNC start-up, Fagor filters are active on any of the axes and a.m.p. SMOTIME (P58) is other than 0 on the same axis, the CNC displays the following error message:

- Parameter TYPE=2 is incompatible with general parameter SMOTIME.

After start-up, if the value of the indicated 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

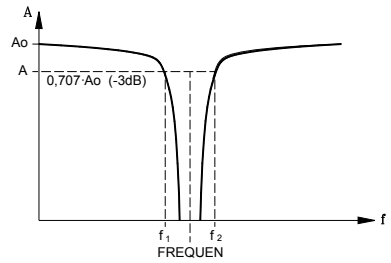
NORBWID (P73)

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)

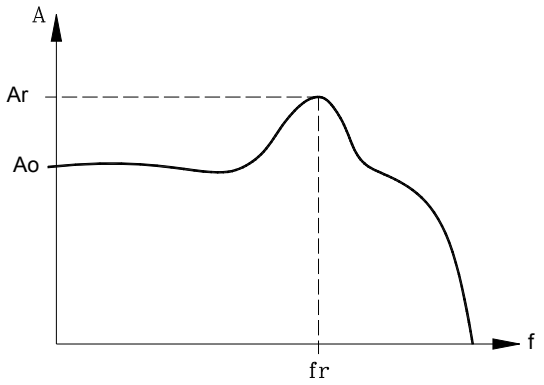
Signal percentage that passes through the filter. This value must be equivalent to the percentage overshooting of the resonance because it has to make up for it.

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.

In the following cases, the value set in axis parameter FLIMIT (P75) is not exceeded:

1. Electronic handwheel:

   In order to respect the limit set by a.m.p FLIMIT(P75), some of the pulses received from the handwheel must be ignored when exceeding FLIMIT; i.e. if FLIMIT is active, some pulses will be lost whether bit 15 of general parameter HDIFFBAC (P129) is set to 0 or to 1.
2. Handwheel associated with a mechanical handwheel.
3. Path-handwheel.

TANSLAID (P76)

Sercos identifier ID of the slave axis of the Tandem axis. This parameter of the master axis selects the axis that will generate the velocity command for the slave axis.

This parameter enables all the other parameters of the Tandem axis. If this parameter is zero, there is no Tandem axis and it ignores the rest of the Tandem control parameters. All the parameters of the Tandem axis are set in the parameter table of the master axis.

TANSLANA (P77)

Slave axis of the tandem axis. The axis whose table sets the parameters will be the master axis of the Tandem 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)

This parameter is used in the following cases:

- To identify the marks of the slave axis DRENA, SPENA, DRSTAF and DRSTAS. To refer to these marks, it is recommended to use as indexes the name of the axis (DRENAX, SPENAZ, etc.). Numerical indexes (DRENA1, SPENA2, etc.) follow a different criteria; they are the ones following the rest of non-slave axes.
- Identify at the CNC the errors of the Tandem slave axis.
- Identify the parameter tables of the Tandem slave axis drive.

TORQDIST (P78)

Torque distribution. It indicates the torque supplied by each motor in order to obtain the necessary total torque on the Tandem axis.

This parameter refers to the master axis. It is defined as the percentage of the total torque required from the master axis. The difference between the value of this parameter and 100% is the percentage applied to the slave axis.

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

Possible values
Between 0 and 100% (both included).

Default value: 50

PRELOAD (P79)

Pre-load between both motors. It is the torque difference to be applied between the master and the slave axes. The pre-load sets a traction between both motors in order to eliminate the backlash in resting position.

This parameter refers to the master axis. It is defined as the percentage of the rated torque to be applied as preload.

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

Possible values
Between -100% and 100%

Default value: 0 (pre-load is disabled)



Applying the preload necessarily implies mechanically joining the master and slave axes that make up the Tandem axis. Otherwise, the motors will move even without the control velocity command.

PRELFITI (P80)

Filter for pre-load. It sets the amount of time for gradually applying the pre-load. Setting it to zero disables the filter.

Possible values
Integers between 0 and 65535 ms.

Default value: 1000

It eliminates the torque steps (shoulders) at the input of the Tandem compensator when setting a pre-load value. This avoids a step (shoulder or sudden change) in the velocity commands of the master and slave axes of the Tandem.

TPROGAIN (P81)

Value of the proportional gain (Kp) for the Tandem axis. The proportional controller generates an output proportional to the torque error between the two motors.

$$k_P = \left(\frac{S_{max}}{T_{nom}}\right) \times TPROGAIN$$

$$T_{error} = (-T_{master} + T_{slave} + Preload)$$

$$Speed = k_P \bullet T_{error}$$

Possible values
Between 0 and 100%.

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

Example

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

Kp = (2000 rpm / 20 Nm) ? 0.1= 10 rpm / Nm.

TINTTIME (P82)

Value of the integral gain (Ki) for the Tandem axis. The integral controller generates an output proportional to the integral of the torque error between the two motors.

$$k_i = \frac{ControlTime}{IntegralTime} \times k_p$$

$$T_{error} = (-T_{master} + T_{slave} + Preload)$$

$$Speed = k_i \bullet \sum T_{error}$$

Possible values
Integers between 0 and 65535 ms.

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

TCOMPLIM (P83)

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

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

Possible values
Between 0 and 100%.

Default value: 0 (the Tandem axis is disabled).

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 be set when using SERCOS, CAN or analog servo drives. Depending on the type of servo drives, a.m.p. PITCHB (P86) must be set as follows:

Analog servo drives:

When using analog servo drives, if a.m.p. NPULSES (P8) =0, it indicates that it is a linear encoder and axis machine parameter PITCH (P7) indicates the pitch of the linear encoder.

If a.m.p. NPULSES (P8) is other than 0, it indicates that it is an encoder and axis machine parameter PITCHB (P86) indicates the pitch of the leadscrew.

SERCOS digital servo drives

Axis machine parameter PITCHB (P86) indicates the leadscrew pitch and it is written at the drive. If axis machine parameters PITCHB (P86)=0, INPREV (P87)=0 and OUTPREV (P88)=0, the leadscrew pitch is read from the drive.

Digital CAN servo

Axis machine parameter PITCHB (P86) sets the leadscrew pitch.



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 deal with gear ratios on the axis. 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 axes 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 are read.

Default value: 0

A.m.p. DRISET (P91) is only taken into account when two Sercos axes share the same drive with switch parameters. Otherwise, it reads the data of set 0.

Special cases:

When the C axis of a lathe shares a drive with the spindle, if a.m.p. DRISET (P91) = 0, it reads the values of the drive's set 7. This is because set 0 is reserved to the spindle.

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 or not the position difference between master and slave will be corrected so that difference is zero, once the two 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 master and slave, from where on to compensate for the difference after homing the two axes of a Gantry pair.

Value	Meaning
0	There is no maximum limit from which the position difference is not compensated.
0.0001 - 99999.9999	From this value on, the position difference is not compensated for.

Default value: 0

This axis machine parameter is taken into account when it is about correct the coordinate 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 velocity command is eliminated (peak compensation 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).

The 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.

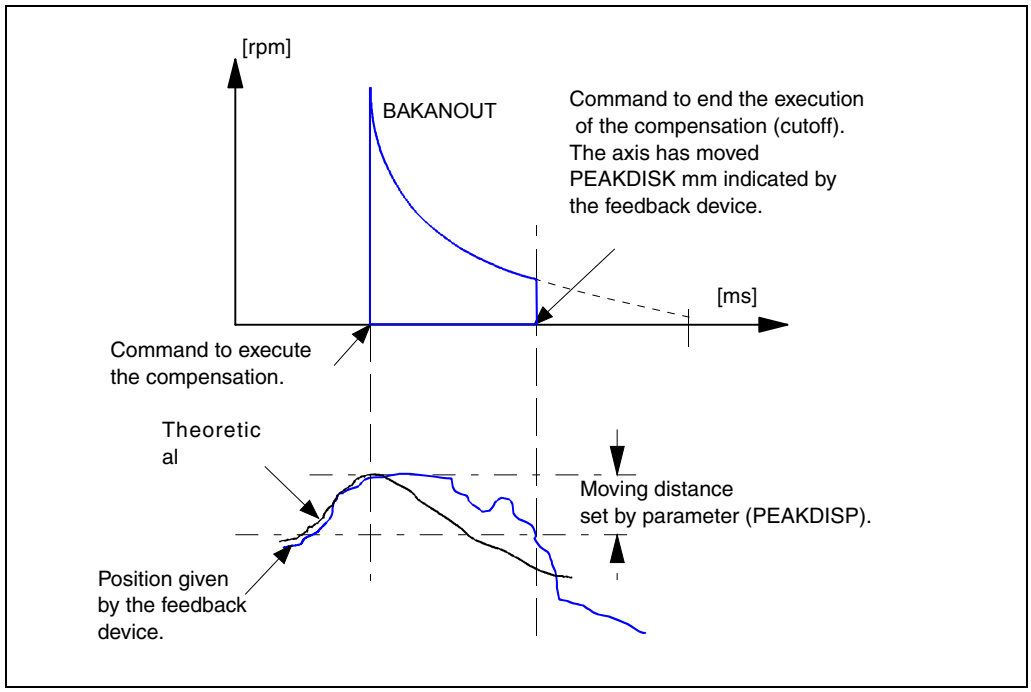
Default value: 0,005

This axis machine parameter will be taken into account when bit 1 of general machine parameter 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 will be 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.

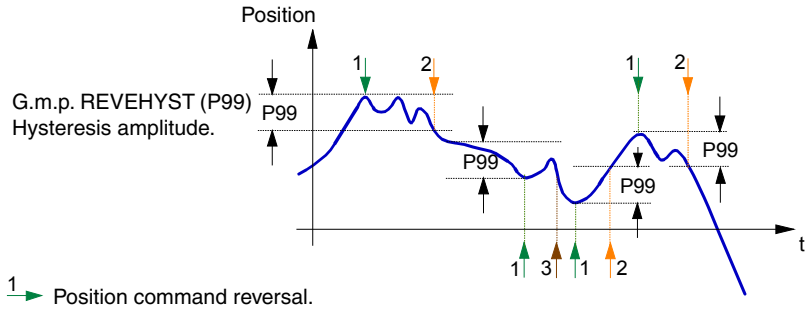
This a.m.p. should be set with the value that the position must vary after the first movement reversal (hysteresis) so it is considered that the command to compensate has been issued, hence prevent it from issuing the compensations every time it receives the command to reverse the moving direction if 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

- Setting a.m.p. REVEHYST (P99) =0, backlash error compensation using a reversal peak will be applied at every 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 second feedbacks:

Possible values
Between 0 and 99999.9999 degrees or millimeters. Between 0 and 3937.00787 inches.

Default value: 1mm (for linear axes)
Default value: 1º (for rotary axes)

This difference in position values may be monitored on the oscilloscope through the read-only 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) with a value other than 0.

If the feedback difference exceeds the value set in a.m.p. FBACKDIF (P100) the CNC will issue the corresponding error message.

MAXDIFAB (P101)

This parameter sets the maximum feedback difference between that of the CNC and the one indicated by the absolute encoder on power-up.

Possible values
Between 0 and 99999.9999 degrees or millimeters. Between 0 and 3937.00787 inches.

Default value: 1mm (for linear axes)
Default value: 1º (for rotary axes)

When using absolute feedback and a.m.p. MAXDIFAB (P101)=0, on power-up the CNC will issue a warning 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 a 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] key is pressed, the CNC will show the error message "Feedback error on the axis" and it will prevent the machine from moving. This error can only be eliminated by powering 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 will come up the first time an absolute feedback is connected or when the offsets of the feedback device are changed. In these cases, once the error has been removed as described earlier, that error will not come up again.

FBMIXTIM (P102)

Machine parameter that may be used to set the time constant to be used when combining feedbacks; i.e. it sets the delay between the position values of the first and the second feedback. This parameter only works for Sercos axes with external feedback, a.m.p. DRIBUSLE (P63) =0.

Possible values
Between 0 and 9999.9 ms.

Default value: 0

Operation of feedback combination depending on the value of a.m.p. FBMIXTIM:

- A value equal to or greater than g.m.p. LOOPTIME (P72) enables the use of feedback combination.
- A value smaller than g.m.p. LOOPTIME (P72) disables feedback combination and the external feedback will be used.

Activating feedback combination, if it is off on CNC power-up, after setting a.m.p. FBMIXTIM to a value equal to or greater than g.m.p. LOOPTIME, requires a Shift/Reset or turning the CNC off and back on.

Once feedback combination is activated on power-up, later changes to the value of parameter FBMIXTIM, even those involved in turning feedback combination on or off, are validated by simply pressing Reset or automatically if it was changed from the oscilloscope.

6.5 Spindle parameters

This CNC can control the main spindle, a second spindle and an auxiliary spindle They all have their own setup parameters. The main and secondary spindle have identical parameter tables.

In order to synchronize the main and secondary spindles, they both must have a feedback device, their s.m.p. M19TYPE (P43) must be set to "1" and their parameters defining the third set of gains and accelerations must be set for a similar behavior of both spindles.

The G77 function synchronizes the spindles in speed, so the secondary spindle turns at the same speed as the main spindle.

The G30 function synchronizes the spindles in position and it sets an angular offset between them so the secondary spindle follows the main spindle maintaining that offset.

6.5.1 Machine parameters for the main and 2nd spindles

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 653.
2	8-digit BCD coded "S" output.

Default value: 0

DFORMAT (P1)

Indicates the display format for the spindle. It is not used for the second 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. It is not used for the second spindle.

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. It is not used for the second spindle.

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. It is not used for the second spindle.

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; it is sent via Sercos (DRIBUSLE = 0) or 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

$$= \left(\xi \cdot PROGAIN + \frac{\xi \cdot DERGAIN}{10 \cdot t}\right)$$

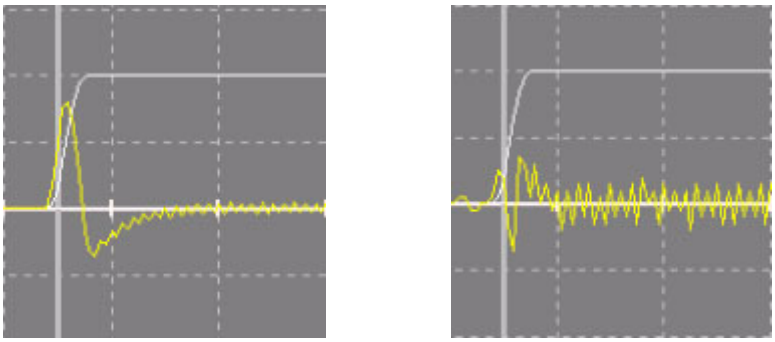
It is a good idea to also use the acc./dec. a.m.p. ACCTIME2 (P18) for this axis (with a value other than "0") if this gain is to be applied.

Possible values
Integer numbers between 0 and 65535.

Default value: 0 (derivative gain not applied)

The best adjustment is achieved when minimizing the following error as much as possible but without inverting the peaks. The peaks of the right graph are inverted. Bad adjustment.

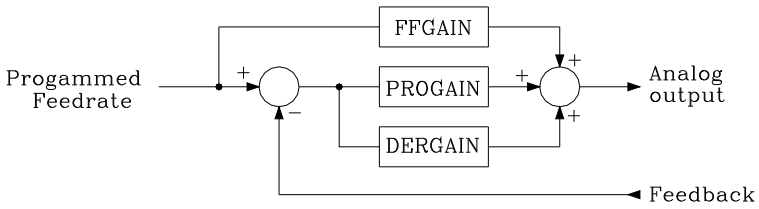
The graph on the left shows the system response without DERGAIN (10 μm per square) and the one on the right with DERGAIN (1 μm per square).



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

$$= \left(\xi \cdot PROGAIN + \frac{\dot{\xi} \cdot DERGAIN}{10 \cdot t} + \frac{FFGAIN \times Fprog \times MAXVOLT}{100 \cdot G00FEED}\right)$$

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)

The best adjustment is achieved when the following error is minimized as much as possible, but without changing its sign, maintaining the moving direction of the axis.

The scale for the following error is 10μm per square.

- Proper adjustment with Feed-forward.



- Wrong adjustment with Feed-forward.



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)

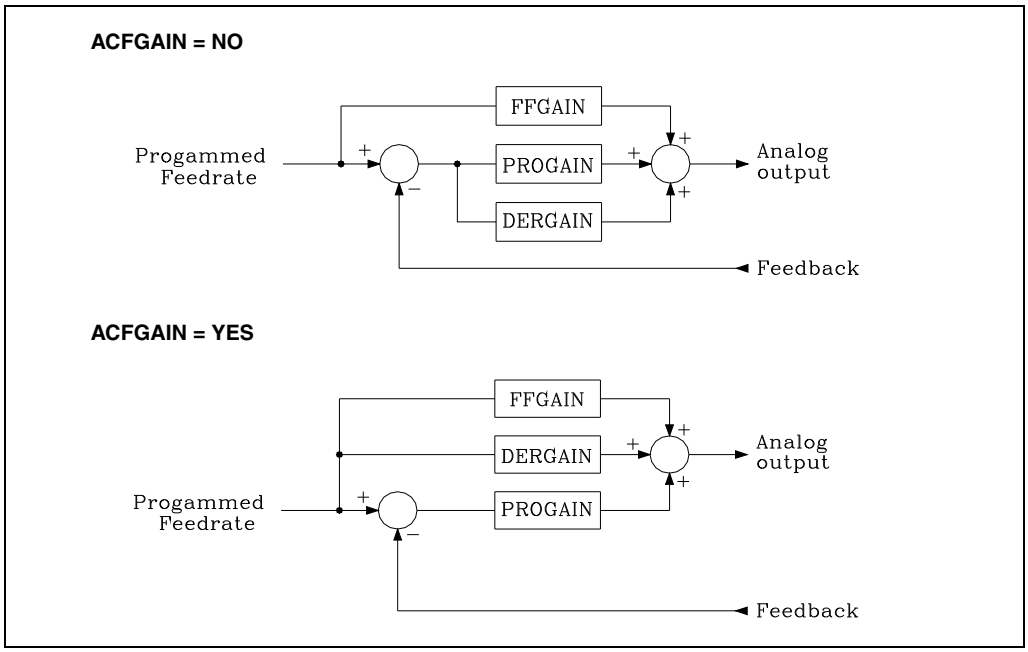
ACFGAIN (P42)

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



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 (Sercos or 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 Sercos axes and 1 Sercos spindle, the values of this parameter must be 1, 2, 3 and 4. Proceed in the same way if it is a CAN connection.

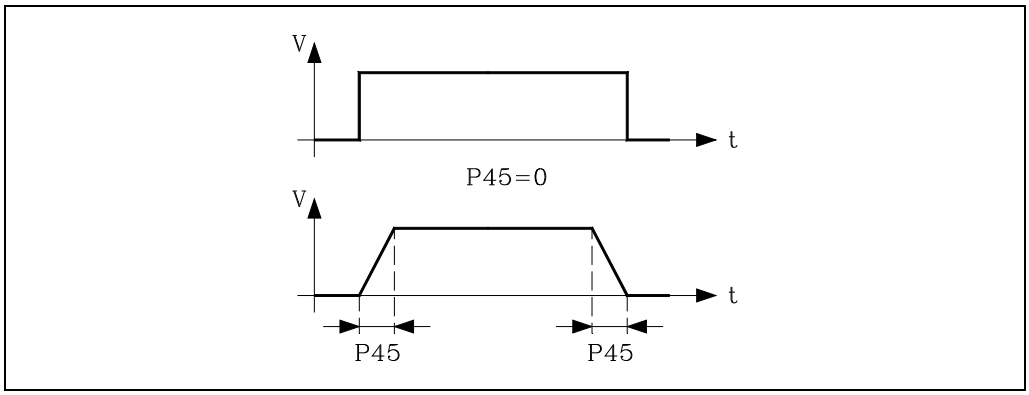
OPLACETI (P45)

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. Movements with handwheel or when the CNC internally transforms the programmed coordinates (C axis, etc.).

In these cases, the response of the spindle may be smoothed by applying a filter to the speed changes.

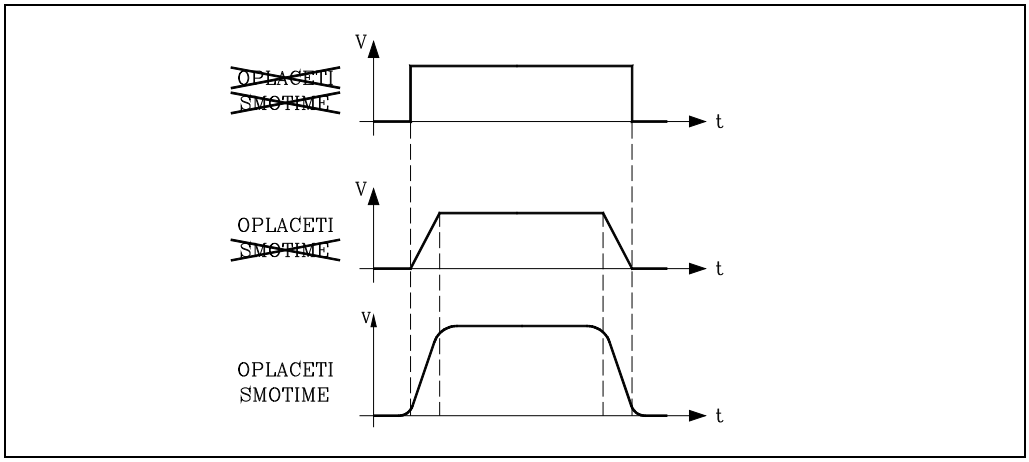
This filter is set by parameter SMOTIME that indicates the duration of the filter in milliseconds, value given by g.m.p. LOOPTIME (P72).

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 (Sercos or CAN). Spindle parameter DRIBUSID (P44) other than 0.

Even when the data exchange between the CNC and the drive is done via digital bus (Sercos or CAN), 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	Feedback is handled via digital bus (Sercos or CAN). First feedback (motor feedback).
2	Feedback is handled via digital bus (Sercos). Second feedback (direct feedback).

When using a communication interface via Sercos bus.

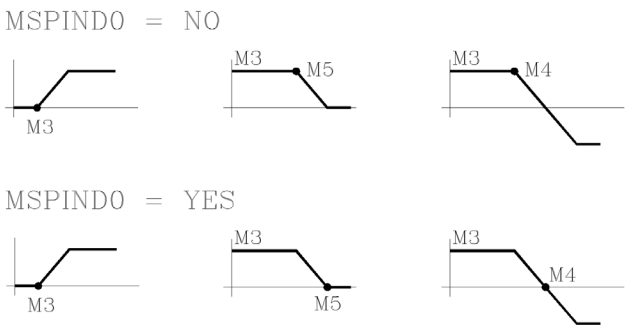
DRIBUSLE = 0	The CNC controls the position loop. The axis feedback is done via connector. The command to the drive is sent out via Sercos.
DRIBUSLE = 1	The CNC controls the position loop. The axis feedback is done via Sercos. First feedback (motor feedback). The command to the drive is sent out via Sercos.
DRIBUSLE = 2	The CNC controls the position loop. The axis feedback is done via Sercos. Second feedback (direct feedback). The command to the drive is sent out via Sercos.

When using a communication interface via CAN bus.

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)

When both spindles are synchronized in position, the second spindle must follow the main spindle maintaining the offset set by function G30.

The parameter of the main spindle sets the maximum error allowed. If this value is exceeded, no error message is displayed and the movement is not stopped. It only sets general output SYNCPOSI (M5559) low.

Possible values
Between 0 and 99999.9999 degrees.

Default value: 2 degrees.

SYNSPEOF (P54)

When both spindles are synchronized in speed, the second spindle must turn at the same speed as the main spindle.

The parameter of the main spindle sets the maximum error allowed. If this value is exceeded, no error message is displayed and the movement is not stopped. It only sets general output SYNSPEED (M5560) low.

Possible values
Integers between 0 and 65535 rpm.

Default value: 1 rpm.

ACCTIME3 (P55) PROGAIN3 (P56) DERGAIN3 (P57) FFGAIN3 (P58)

These parameters define the third set of gains and accelerations. The CNC uses the third set when working with synchronized spindles (G77).

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)

The spindles (main and second) must have their own feedback devices and their parameters must be set in such a way that their behaviors are similar.

Possible values
The same as for the first gear.

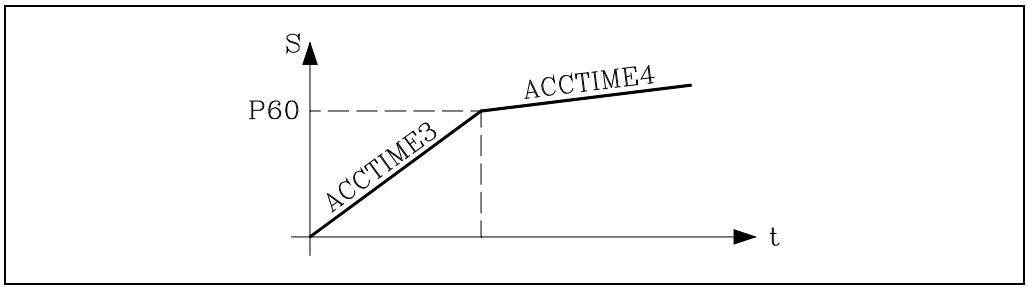
Default value: For ACCTIME3 (P55) = 4000 ms .
For PROGAIN3 (P56) = 50 mV/degree.
For DERGAIN3 (P57) = 0.
For FFGAIN3 (P58) = 100.

When working with FFGAIN3 (P58) = 100, set the MAXGEAR and MAXVOLT parameters properly.

ACCTIME4 (P59) SECACESP (P60)

In order to compensate for the lack of a linear response on some spindles, it is possible to use two accelerations: ACCTIME3 for low speeds [up to the one set by SECACESP (P60)] and ACCTIME4 for the rest of higher speeds.

Once the spindles are in synchronism, the CNC applies to both spindles the accelerations defined for the main spindle.



ACCTIME4 (P59).

Parameter ACCTIME4 is set like ACCTIME3.

Possible values
Integers between 0 and 65535 ms.

Default value: 8000.

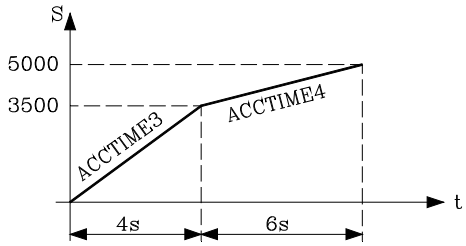
SECACESP (P60)

Parameter SECACESP (P60) indicates at what speed the acceleration change takes place. If P60=0, it always applies ACCTIME3.

Possible values
Integers between 0 and 65535 rpm.

Default value: 700.

Example:
Being the maximum speed for the selected gear MAXGEAR = 6000 rpm.



Maximum synch speed: 5000 rpm.
SYNMAXSP (P63) = 5000
Acceleration changing speed: 3500 rpm.
SECACESP (P60) = 3500
ACCTIME3(P55) = 6000 x 4 / 3500 = 6857 ms.
ACCTIME4(P59) = 6000 x 6 / 1500 = 24000 ms.

SYNCPOLA (P61)

It is defined at the second spindle. It indicates whether the spindles being synchronized are facing each other or not (opposite turning directions in M3 or M4) for the CNC to take it into consideration when synchronizing them.

Value	Meaning
NO	They are NOT facing each other; they both turn in the same direction.
YES	They are facing each other; they turn in opposite directions.

Default value: NO

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.

When working with synchronized spindles (G77), third set of gains and accelerations is used. Therefore, the CONCLOOP parameter of the spindle to be synchronized should be set to "YES"

SYNMAXSP (P63)

It is set for the main spindle. It indicates the maximum turning speed when the spindle are synchronized (G77).

Possible values
Integers between 0 and 65535 rpm. When set to 0, there is no limit.

Default value: 1000 rpm.

M3M4SIM (P64)

In TC mode, it indicates the turning direction of the corresponding spindle with each key for turning direction.

Possible values

M3M4SIM (P64) = 0  = M3  = M4

M3M4SIM (P64) = 1  = M3  = M4

Default value: 0

For example in a live tool cycle, it will take into account the value assigned to the spindle that has been defined as live tool whereas in a deep hole drilling cycle, it will consider the one assigned to the main spindle.

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. It is recommended not to activate these filters on machines carrying out movements against a hard stop.

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.

TYPE (P68)

Filter type. There are three types of filter: "low passing", "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.

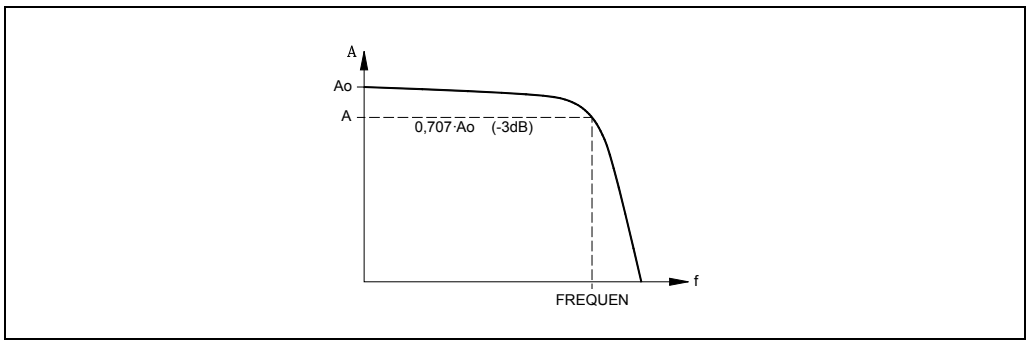
Value	Meaning
0	"Low passing" filter.
1	"Anti-resonance" (notch) filter.
2	FAGOR "low passing" filter.

Default value: 0

When defining anti-resonance filters, parameters NORBWIDTH and SHARE must also be set.

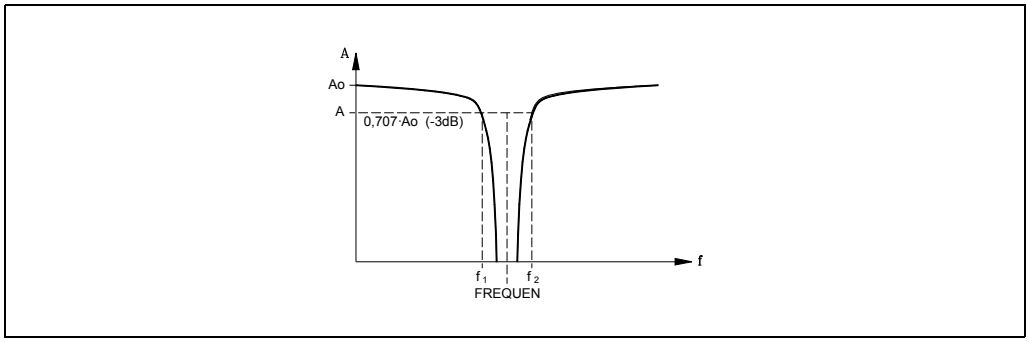
"Low passing" filter.

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.



CNC start-up with Fagor filters active.

If on CNC start-up, Fagor filters are active on any of the axes and a.m.p. SMOTIME (P58) is other than 0 on the same axis, the CNC displays the following error message:

- Parameter TYPE=2 is incompatible with general parameter SMOTIME.

After start-up, if the value of the indicated 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

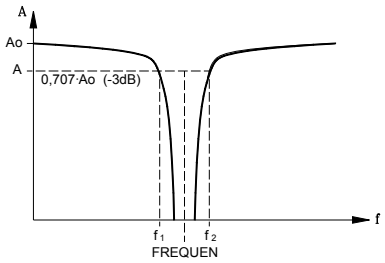
NORBWID (P70)

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)

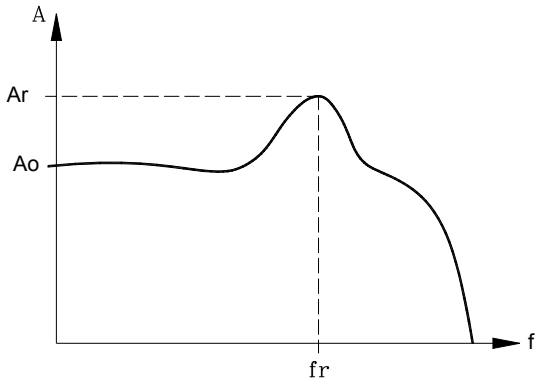
Signal percentage that passes through the filter. This value must be equivalent to the percentage overshooting of the resonance because it has to make up for it.

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 INPREV4 indicate the input speed for each gear.

Parameters OUTPREV1 through OUTPREV4 indicate the output speed for each gear.

Possible values
Integer numbers between 0 and 65535.

Default value: 0

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:

$$JERKLIM = 6000 \cdot MAXGEAR1 / OPLACETI^{2}$$

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% - 50%	Maximum increment allowed via override switch.

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.

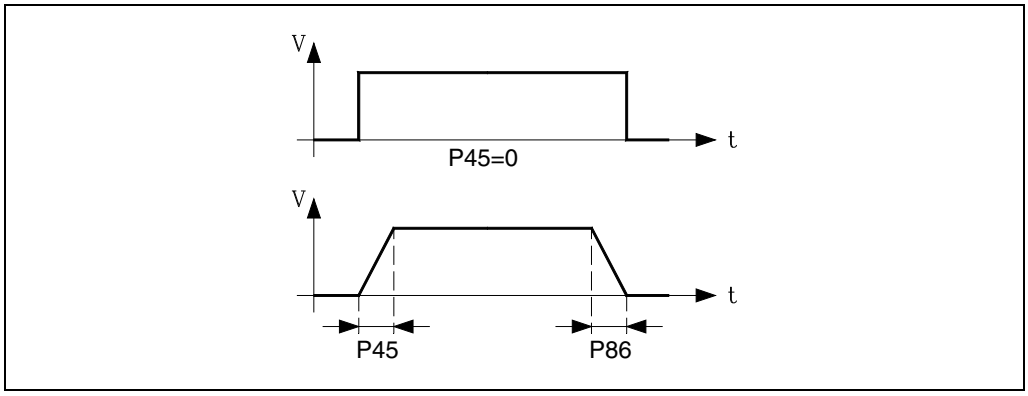
Possible values
Integers between 0 and 65535 ms.

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.



6.5.2 Machine parameters for auxiliary spindle

MAXSPEED (P0)

Indicates the maximum speed of the auxiliary spindle.

Possible values
Integers between 0 and 65535 rpm.

Default value: 1000 rpm.

SPDLOVR (P1)

It indicates whether the spindle override keys of the operator panel modify or not the speed of the auxiliary spindle when it is active.

Value	Meaning
NO	They have no effect.
YES	SINMAGNI = 360 / (3600 x 0.001) = 100 The CNC will apply the values set for main s.m.p. "MINSOVR" (P10), "MAXOVR" (P11) and "SOVRSTEP" (P12).

Default value: NO.

MINANOUT (P2)

It sets the minimum velocity command value.

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 (P3)

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.

MAXVOLT (P4)

Indicates the velocity command corresponding to the maximum speed defined by s.m.p. MAXSPEED (P0).

Possible values
Integer numbers between 0 mV and 9999 mV.

Default value: 9500 (9.5 V)

DRIBUSID (P5)

It indicates the address of the digital drive (Sercos or CAN) associated with the auxiliary spindle. 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 Sercos axes and 1 Sercos spindle, the values of this parameter must be 1, 2, 3 and 4. Proceed in the same way if it is a CAN connection.

6.6 Drive parameters

This option is available when the CNC uses digital servo drive system; i.e. the drives are connected to the CNC via Sercos or CAN. These types of servo drive systems are similar in the way they look and in the way they manage the parameters. In both cases, the generated files have the same names and the internal format of the parameters has the same structure (e.g. SP1.7 123).

Sercos interface

This option works fine with drive version V3.9 or newer. When using older versions, it might not be possible to access all variables and parameters and some data might not be shown such as the name of the associated motor.

If the communication through the Sercos ring is interrupted, a screen is displayed. Press [ENTER] to restore it.

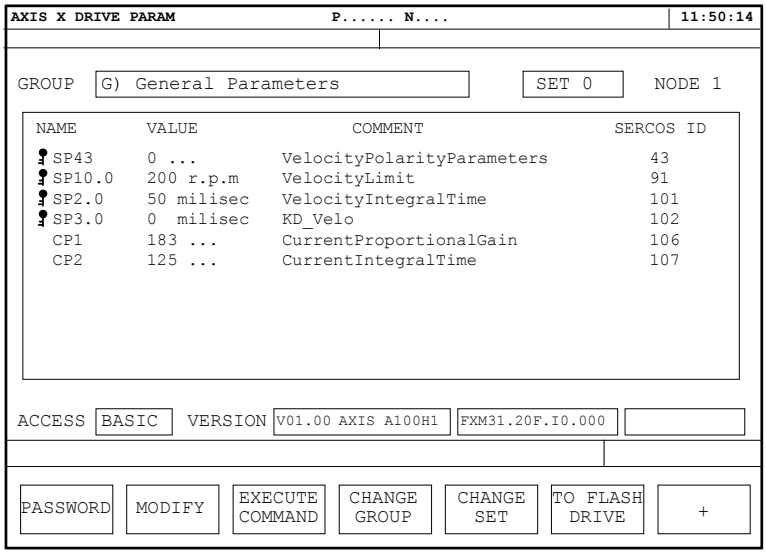
CAN interface

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 Sercos ring or in the CAN connection; i.e. the position of the 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 Sercos 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.

  Both the Sercos and the CAN servo systems share the same Sercos identifier; which offers full compatibility when accessing the fast and slow channels from the PLC, user programs and screen configuration (consumption bars), etc.

- 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; not a pre-established one as when using Sercos .

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 [⬆] [⬇].

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.

6.6.1 Friction compensation

From version V3.14 on, the drive offers parameters TP10, TP11, TP12, TP13, TP14 and TV4 for friction compensation. Refer to the drive manual.

It also has 2 more general purpose variables: XV10 and XV11 (ID SERCOS 34800 and 34801). These variables may be accessed from the CNC via Sercos.

The following example shows how to use variable XV10 to monitor or show on the oscilloscope the X axis following error using WinDDS.

Example

The following example shows how to use variable XV10 to monitor or show on the oscilloscope the X axis following error using WinDDS.

Assign the value of PLC register R800 to the "ID SERCOS 34800" of the drive that occupies the "Sercos 1" address.

```
Set plc.m.p. SWR800 = 1.34800"
```

The PLC program must set register R800 to the value of the X axis following error (FLWEX variable). A periodic module should be used to refresh that value at every position loop. (Note: In order for the Sercos variable to be actually updated, the NWR instruction must be written in the periodic module).

```
PE 4
()= CNCRD (FLWEX, R800, M1)
MWR
END
```

6.7 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.
3	Open communication.

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: YES

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

RCVMAXCAR (P11)

It indicates the maximum number of characters that may be received in the communication.

Possible values
Integer numbers between 0 and 255.

Default value: 0.

RCVENDTI (P12)

Indicates the maximum reception time. Reception ends when the time indicated in this parameter has elapsed, after reception began.

Possible values
Integers between 0 and 65535 ms.

Default value: 0.

6.8 Ethernet parameters

These parameters may be used to configure the CNC like any other node in the network, the DNC for Ethernet, the remote hard disk 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 hard disk is accessible via FTP.
Basic configuration and also: CNHDPAS1 (P7)	Protect the access to the hard disk with a password.
Basic configuration and also: DNCEACT (P22) IPWDNC (P27)	Configure the DNC for Ethernet.
Basic configuration and also: IPSNFS (P28) DIRNFS (P29)	Configure the remote hard disk.

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 hard disk 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 (HD).
- Edit, modify, delete, rename, etc., the programs stored on the hard disk (HD).
- Copy programs from the hard disk 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 hard disk 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)

DNC number to be used by Ethernet.

Value	Meaning
0	There is no active DNC associated with Ethernet.
1	DNC 1 associated with Ethernet.
2	DNC 2 associated with Ethernet. The RS-232 serial line is disabled.

Default value: 1

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)

IP address of the server acting as remote hard disk.

If other than 0, the remote hard disk is activated. This means that the local hard disk, if there is any, is disabled and can no longer be accessed.

Possible values
Four numbers between 0 and 255 separated by dots.

Default value: 0.0.0.0 (there is no remote hard disk)

DIRNFS (P29)

Directory of the server that is used as remote hard disk.

Possible values
It admits up to a maximum of 22 characters (without blank spaces).

Default value: Nameless

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 hard disk 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.

6.9 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	0.2 ms every 8 samplings.
1	0.2 ms every 4 samplings.
2	0.2 ms every 2 samplings.
3	0.2 ms every sampling.
4	0.4 ms every sampling. With LOOPTIME = 3, 4, 5 or 6
5	0.6 ms every sampling. With LOOPTIME = 3, 4, 5 or 6
6	0.8 ms every sampling. With LOOPTIME = 4, 5 or 6
7	1 ms every sampling. With LOOPTIME 5 or 6.
8	1.2 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 0.2 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)

They are used in the data exchange via Sercos between the CNC and the drives.

They indicate which drive and what type of information will be put in CNC registers R700 through R739.

P28 ⇒ R700 P29 ⇒ R701 P30 ⇒ R702

P31 ⇒ R703 P32 ⇒ R704 etc.

The setting format for plc.m.p. "P28" through "P67" is 1.5

The units digit identifies the Sercos node number to get information from.

The decimal part indicates the Sercos identifier number.

Example:
P32=1.00040
Indicates that PLC register R704 contains the "VelocityFeedback" supplied by the drive located in Sercos 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.

SWR800 (P68) . . . SWR819 (P87)

They are used in the data exchange via Sercos between the CNC and the drives.

They indicate what type of information is put in registers R800 through R819 and which drive will be assigned that value.

P68 ⇒ R800 P69 ⇒ R801 P70 ⇒ R802

P71 ⇒ R803 P72 ⇒ R804 etc.

The setting format for plc.m.p. "P68" through "P87" is 1.5

The units digit identifies the Sercos node number to send information to.

The decimal part indicates the Sercos identifier number.

Example:
P70=2.34178
Indicates that the value of PLC register R802 will be assigned to "DigitalOutputsValues" of the drive located in Sercos node 2.

To identify the units of the variables, see the drive manual.

IOCANSPE (P88)

When using CAN connection, the transmission speed depends on the length of the cable or total CAN connection distance.

Value	Meaning
0	1 Mbit/s. Up to 20 meters.
1	800 kbit/s. From 20 to 40 meters.
2	500 kbit/s. From 40 to 100 meters.
3	250 kbit/s. From 100 to 500 meters.
4	125 kbit/s. From 500 to 1000 meters.

Default value: 2 (500 kbit/s)

Fagor modules that do not have a baudrate selector can only work at 500 Kbit/s.

IOCAGEN (P89)

Not being used.

IOCANID1 (P90) IOCANID2 (P91) IOCANID3 (P92) IOCANID4 (P93)

They are used to set the remote modules.

They indicate which remote module each plc.m.p. (ICAN*, OCAN*, NUICAN*, NUOCAN*) refer to.

Assign the CAN bus address that occupies the node (the one indicated by the address selector switch).

ICAN1 (P94) OCAN1 (P95) ICAN2 (P96) OCAN2 (P97) ICAN3 (P98) OCAN3 (P99) ICAN4 (P100) OCAN4 (P101)

They are used to set the remote modules.

They indicate the configuration of each remote module, number of inputs (ICAN*) and outputs (OCAN*).

Example for a remote module located in node 1, with 48 inputs and 32 outputs:

IOCANID1=1 CAN1=48 OCAN1=32

NUICAN1 (P102) NUOCAN1 (P103) NUICAN2 (P104) NUOCAN2 (P105) NUICAN3 (P106) NUOCAN3 (P107) NUICAN4 (P108) NUOCAN4 (P109)

They are used to set the remote modules.

Parameter NUICAN* indicates the number of the first digital input and NUOCAN* the number of the first digital output for the group.

The first I/O module is always the CPU (8055i CNC without expansion board) or an axes board or an I/O board (modular 8055 CNC).

In either case, the first input number is always I1 and the first output number O1 and they cannot be set by parameters.

On remote module the inputs and outputs of the different elements are numbered sequentially. The inputs and outputs are defined in groups of 8 and the possible values of NUICAN* and NUOCAN* must be multiple of 8 plus 1 (8n +1).

If incoherent parameter settings are detected on power-up, an error message will be issued indicating it.

If NUICAN=0 or NUOCAN=0, the group following the one assigned to the previous node is assigned to the corresponding node.

Examples of remote module setting

The central unit does not have an I/O expansion board.

Remote module in node 1, with 48 inputs and 32 outputs.

Remote module in node 2, with 24 inputs and 16 outputs.

Case 1: We want the inputs and outputs to be sequential starting with the first ones available.

Node ·1·	Node ·2·
IOCANID1=1	IOCANID2=2
ICAN1=48	ICAN2=24
OCAN1=32	OCAN2=16
NUICAN1=0	NUICAN2=0
NUOCAN1=0	NUOCAN2=0

Node 1 is assigned inputs I65 through I112 and outputs O33 through O64.

Node 2 is assigned inputs I113 through I136 and outputs O65 through O80.

The inputs of the first remote module are numbered sequentially after the last input of the first module (I64+1 = I65). The inputs of the second remote module are numbered sequentially after the last available input of the first remote module (I112+1=I113). Follow the same procedure for the outputs.

Case 2: We want the inputs and outputs to be sequential; but starting with I129 and O65, getting it ready for the I/O expansion board.

Node ·1·	Node ·2·
IOCANID1=1	IOCANID2=2
ICAN1=48	ICAN2=24
OCAN1=32	OCAN2=16
NUICAN1=129	NUICAN2=0
NUOCAN1=65	NUOCAN2=0

Node 1 is assigned inputs I129 through I176 and outputs O65 through O96.

Node 2 is assigned inputs I177 through I200 and outputs O97 through O112.

The inputs of the first remote module are numbered sequentially after the value assigned to parameter NUICAN1(I129). The inputs of the second expansion module are numbered sequentially after the last available input of the first remote module (I176+1=I177). Follow the same procedure for the outputs.

Case 3: We expect an I/O and element expansion in node 1 (up to 72 inputs and 48 outputs).

Node ·1·	Node ·2·
IOCANID1=1	IOCANID2=2
ICAN1=48	ICAN2=24
OCAN1=32	OCAN2=16
NUICAN1=129	NUICAN2=201
NUOCAN1=65	NUOCAN2=113

Node 1 is assigned inputs I129 through I176 and outputs O65 through O96

Node 2 is assigned inputs I201 through I224 and outputs O113 through O128

The inputs of the first remote module are numbered sequentially after the value assigned to parameter NUICAN1(I129). The inputs of the second remote module are numbered after the value assigned to parameter NUICAN2(I201) Follow the same procedure for the outputs.

IANALOG1 (P110) IANALOG2 (P113) IANALOG3 (P116) IANALOG4 (P119)

They are used to set the remote modules.

They indicate the number of analog inputs of each of the 4 remote modules; a CAN node number identifies each one in the system. Its value will be 4 if it is a Fagor remote module.

OANALOG1 (P111) OANALOG2 (P114) OANALOG3 (P117) OANALOG4 (P120)

They are used to set the remote modules.

They indicate the number of analog outputs of each of the 4 remote modules; a CAN node number identifies each one in the system. Its value will be 4 if it is a Fagor remote module.

PT100_1 (P112) PT100_2 (P115) PT100_3 (P118) PT100_4 (P121)

They are used to set the remote modules.

They indicate the number of physical connections available for PT100 probes in each of the 4 remote modules and also indicate which ones are connected. All this information is reflected in a 16-bit string. Their value will be 0000 0000 00**00** 00**11** if it is a Fagor remote module (2 physical connections, bits 0 and 1 set to "1") and whether they are both connected (bits 4 and 5 set to "0") or not. See table below.

A remote module may have physical connections for PT100 probes, but they are not connected. That is why there is a bit indicating that there is a physical connection for the PT100 probe and another bit to indicated whether a probe is connected or not.

Therefore, for a 16-bit string:



PT100_1 = x x x x x x x x xx 0/1 0/1 xx 0/1 0/1		
bit 0	Does it have a physical connection for PT100_1 probe? no / yes	0/1
bit 1	Does it have a physical connection for PT100_2 probe? no / yes	0/1
bit 4	Is there a PT100_1 probe connected to it? yes / no	0/1
bit 5	Is there a PT100_2 probe connected to it? yes / no	0/1
... others	Reserved	

for Fagor remote modules

If the probe is not properly connected or the cable is defective, the CNC will display an error message that will be treated like the errors at the digital inputs/outputs.

NUIANA1 (P122) NUIANA2 (P124) NUIANA3 (P126) NUIANA4 (P128)

They are used to set the remote modules.

They indicate the number of the first analog input of each remote module. The rest of analog inputs of the same module will be numbered sequentially.

NUOANA1 (P123) NUOANA2 (P125) NUOANA3 (P127) NUOANA4 (P129)

They are used to set the remote modules.

They indicate the number of the first analog output of each remote module. The rest of analog outputs of the same module will be numbered sequentially.

If the value of all these parameters is zero (they are by default), the numbering of the analog inputs/outputs will be consecutive to the last local inputs/outputs.

Their maximum value will be 16, both for the analog inputs and analog outputs.

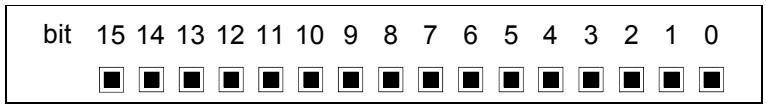
On power-up, the CNC verifies that the number of analog inputs/outputs detected at the module matches the value assigned to the relevant machine parameters.

If these values do not match, the CNC screen will display a message indicating it and then it will turn the remote module off.

IANA5V (P130)

On an 8055 CNC with –Vpp Axes– module, this parameter indicates the range to be used in each differential analog input of the module's connector X7. A range of ±5 V or ±10 V may be assigned for each analog input.

This parameter has 16 bits counted from right to left.



Each bit has a function or work mode associated with it. Assigning the value of ·1· activates the corresponding function. The default value for bits 13 to 16 is ·1· (±5 V range).

Bit	Meaning
0 - 11	Not being used.
12	Analog input 4 (±5 V range).
13	Analog input 3 (±5 V range).
14	Analog input 2 (±5 V range).
15	Analog input 1 (±5 V range).

Default value: 1111000000000000

Bit 12 - 15. Average range for the analog inputs.

These bits refer to the 4 analog inputs of the –Vpp Axes– module. Every bit corresponds to an analog input and indicates whether it uses a ±5 V range (bit=1) or a ±10 V range (bit=0).

NUILO1 (P131) NUOLO1 (P132) NUILO2 (P133) NUOLO2 (P134) NUILO3 (P135) NUOLO3 (P136) NUILO4 (P137) NUOLO4 (P138)

These PLC machine parameters may be used to redefine the numbering of the inputs/outputs of the local expansion modules without having to modify the PLC program.

Value	Meaning
NUILO1	Numbering of the first input of the first expansion module with inputs/outputs.
NUOLO1	Numbering of the first output of the first expansion module with inputs/outputs.
NUILO2	Numbering of the first input of the second expansion module with inputs/outputs.
NUOLO2	Numbering of the first output of the second expansion module with inputs/outputs.
NUILO3	Numbering of the first input of the third expansion module with inputs/outputs.
NUOLO3	Numbering of the first output of the third expansion module with inputs/outputs.
NUILO4	Numbering of the first input of the fourth expansion module with inputs/outputs.
NUOLO4	Numbering of the first output of the fourth expansion module with inputs/outputs.

The first I/O module is always the CPU (8055i CNC without expansion board) or an axes board or an I/O board (modular 8055 CNC). Note that this first module is not an expansion module.

In either case, the first input number is always I1 and the first output number O1 and they cannot be set by parameters.

IMPORTANT: The numbering of both the first local input and the first local output of any 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 inputs/outputs of the expansion modules will be numbered differently depending on the values entered in parameters NUILOn and NUOLOn (with n= 1, 2, 3, 4).

Expansion modules numbering examples

It is a system made up of an axes board with 40I/24O and two expansion modules with 64I/32O available in each one of them.

The inputs/outputs of the first module cannot be set by parameter; therefore, the inputs must be numbered from I1 on and the outputs from O1 on.

Case 1: We wish to number inputs and outputs by setting parameters NUILOn and NUOLOn to zero.

First expansion module	Second expansion module
NUILO1=0	NUILO2=0
NUOLO1=0	NUOLO2=0

The first expansion module assigned inputs I65 through I112 and outputs O33 through O64.

The second expansion module assigned inputs I129 through I192 and outputs O65 through O96.

The inputs of the first expansion module are numbered sequentially after the last input of the first module (I64+1 = I65). The inputs of the second expansion module are numbered sequentially after the last available input of the first expansion module (I128+1 = I129). Follow the same procedure for the outputs.

Case 2: We wish the inputs and outputs to be sequential, but starting with I41/O25, so they are numbered after the 40I/24O of the axis board.

First expansion module	Second expansion module
NUILO1=41	NUILO2=0
NUOLO1=25	NUOLO2=0

The first expansion module assigned inputs I41 through I104 and outputs O25 through O56.

The second expansion module assigned inputs I105 through I168 and outputs O57 through O88.

The inputs of the first expansion module are numbered sequentially after the value assigned to parameter NUILO1 (I41) chosen with the restriction (8n+1). The inputs of the second expansion module are numbered sequentially after the last input used in the first expansion module (I104+1 = I105). Follow the same procedure for the outputs.

Case 3: We wish the inputs and outputs start at I65/O33 the first expansion module and at I201/O113 the second expansion module.

First expansion module	Second expansion module
NUILO1=65	NUILO2=201
NUOLO1=33	NUOLO2=113

The first expansion module assigned inputs I65 through I112 and outputs O33 through O64.

The second expansion module assigned inputs I201through I264 and outputs O113 through O144.

The inputs of the first expansion module are numbered sequentially after the value assigned to parameter NUILO1 (I65) chosen at will with the restriction (8n+1). The inputs of the second expansion module are numbered sequentially after the value assigned to parameter NUILO2 (I113) chosen at will with the restriction (8n+1). Follow the same procedure for the outputs.

Expansion modules and remote modules numbering examples.

Let's suppose a system made up of an axes board with 40I/24O and two expansion modules with 64I/32O available in each one of them and the following remote modules.

Remote module in node 1, with 48 inputs and 32 outputs.

Remote module in node 2, with 24 inputs and 16 outputs.

The inputs/outputs of the first module (axis board) cannot be set by parameter; therefore, the inputs must be numbered from I1 on and the outputs from O1 on.

We wish the inputs and outputs to be sequential starting with the local ones, then the expansion modules and then the remote modules.

First expansion module	Second expansion module
NUILO1=41	NUILO2=0
NUOLO1=25	NUOLO2=0
Remote module (Node ·1·)	Remote module (Node ·2·)
IOCANID1=1	IOCANID2=2
ICAN1=48	ICAN2=24
OCAN1=32	OCAN2=16
NUICAN1=169	NUICAN2=0
NUOCAN1=89	NUOCAN2=0

The axis board is assigned inputs I1-40 and outputs O1-24.

The first expansion module assigned inputs I41 through I104 and outputs O25 through O56.

The second expansion module assigned inputs I105 through I168 and outputs O57 through O88.

The first remote module (node 1) is assigned inputs I169-216 and outputs O89-120.

The second remote module (node 2) is assigned inputs I217-240 and outputs O121-136.

The inputs of the first expansion module are numbered sequentially after the value assigned to parameter NUILO1 (I41) chosen with the restriction (8n+1). The inputs of the second expansion module are numbered sequentially after the last input used in the first expansion module (I104+1 = I105). Follow the same procedure for the outputs.

The inputs of the first remote module are numbered sequentially after the last input of the second expansion module (I168+1=I169) The inputs of the second remote module are numbered sequentially after the last available input of the first remote module (I216+1=I217). Follow the same procedure for the outputs.

In the previous example, the I/O will be ordered so expansion modules 1 and 2 come first and then remote modules 1 and 2. This order is not compulsory, these modules may be ordered as desired. This way, the remote modules could be numbered first and then the expansion ones or intercalating one of each.

The total limit of inputs (local + remote) is 512.

The total limit of outputs (local + remote) is 512.

WARNING: The order of local I/O modules corresponds with the numbering of their internal switch, not with the physical position of the modules.

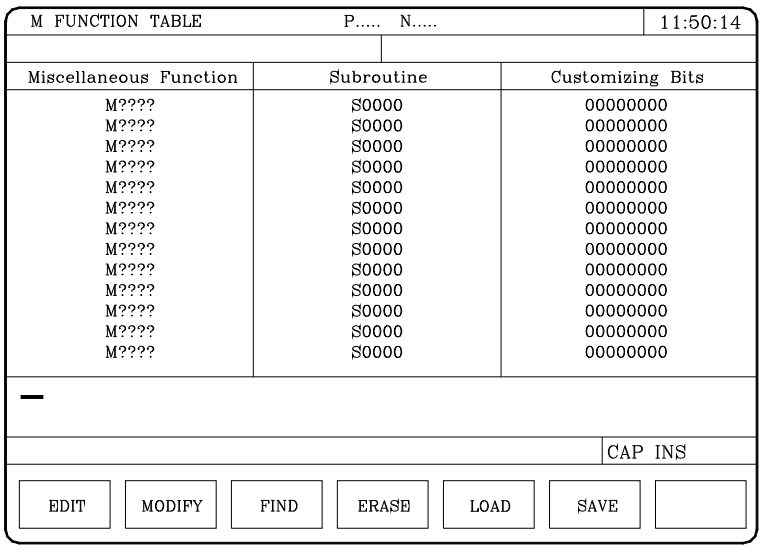
The values of the PLC machine parameters mentioned earlier should be multiple of 16 for best managing the inputs and outputs in time.

6.10 Tables

6.10.1 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, M44 and M45, 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

Not being used at this time.

bit 6

Not being used at this time.

bit 7

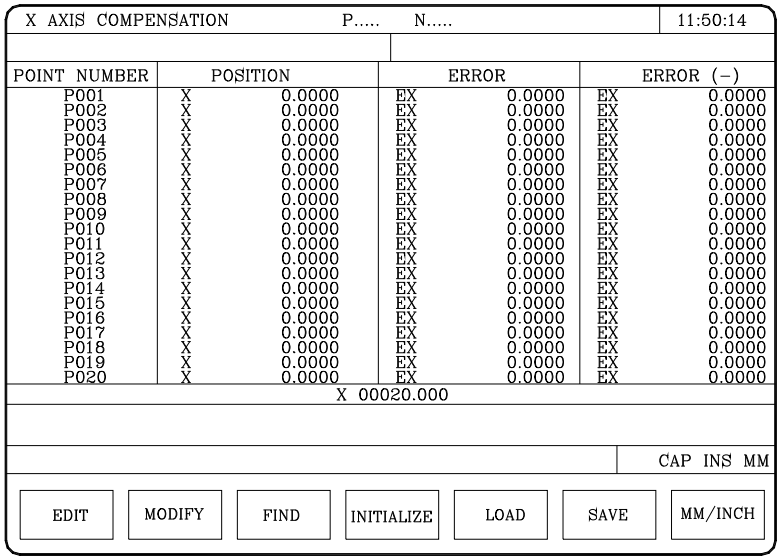
Not being used at this time.

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.

6.10.2 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 1000 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:

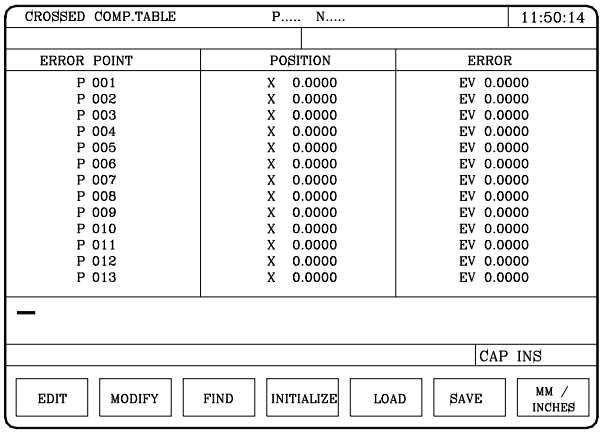
- The axis points must be in sequential order starting from the most negative (least positive) point to be compensated.
- For those points outside the compensation zone, the CNC will apply the compensation value corresponding to the table point closest to them.
- The amount of error of the machine reference point may have any value.
- The error difference between two consecutive points must not be greater than the distance between them (maximum slope= 100%).

6.10.3 Cross compensation parameter table

Up to 3 cross compensation tables may be used. To enable each one, set g.m.p.:

MOVAXIS (P32)	COMPAXIS (P33)	NPCROSS (P31)
MOVAXIS2 (P55)	COMPAXIS2 (P56)	NPCROSS2 (P54)
MOVAXIS3 (P58)	COMPAXIS3 (P59)	NPCROSS3 (P57)

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.

CONCEPTS

7

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.

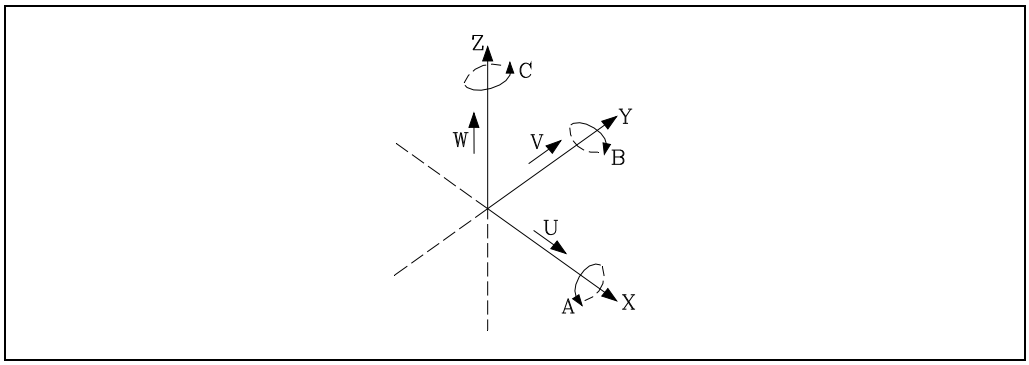
7.1 Axes and coordinate systems

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

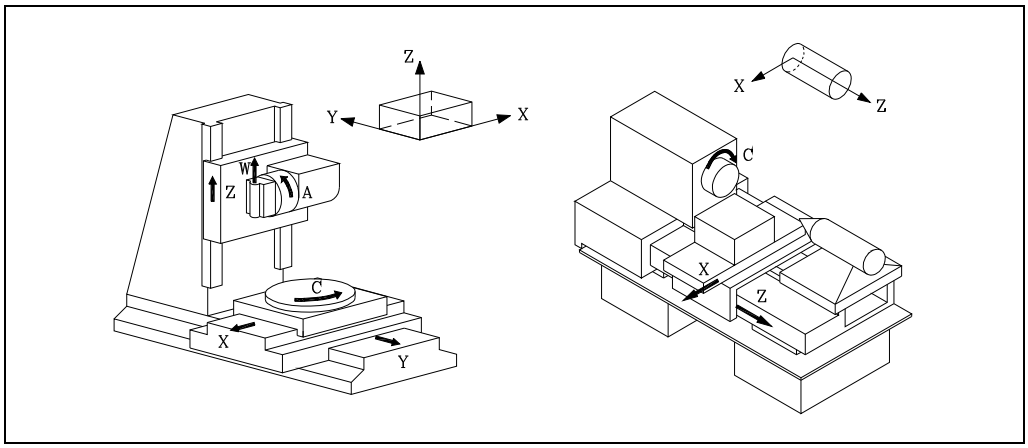
The axes are named according to DIN 66217.



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

Of the 9 possible axes that may exist, the CNC allows the manufacturer to select up to 7 of them.

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 7 axes may be interpolated at the same time.

Example of milling

The machine has three normal axes: X, Y and Z, one normal linear U axis controlled by the PLC, an analog spindle (S) and an electronic 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) = 4	U axis	associated with feedback X4 and output O4.
AXIS5 (P4) = 10	Spindle (S)	associated with feedback X5 (1-6) and output O5.
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
U axis	AXISTYPE (P0) = 5	Regular linear axis controlled by the PLC

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, a "C" axis, an analog spindle (S) and an auxiliary spindle (live tool).

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) = 9	"C" axis	associated with feedback X4 and output O4.
AXIS5 (P4) = 13	Auxiliary spindle	associated with feedback X5 (1-6) and output O5.
AXIS6 (P5) = 0		
AXIS7 (P6) = 0		
AXIS8 (P7) = 0		

The CNC activates a machine parameter table for each axis (X, Z, C), one for the main spindle (S) and another one for the auxiliary spindle.

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
"C" axis	AXISTYPE (P0) = 2	Normal rotary 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.

7.1.1 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 *"6.4 Axis parameters"* on page 206.

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 *"7.5.7 Leadscrew error compensation"* on page 322.

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. G91The 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. G91The 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. G91The 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. G91The 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 can take decimal coordinates.

More than one Hirth axis may be used, but they can only move one at a time.

Rotary Hirth axis **AXISTYPE=4**		
LIMIT+ = 8000 LIMIT- =-8000	ROLLOVER=YES	It counts between 0º and 360º. G90 does not admit negative values. G91The 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. G91The 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.

7.1.2 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".

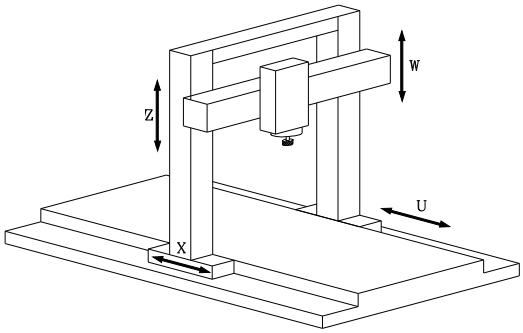
In order to operate this way, it is necessary to have the a.m.p. GANTRY (P2) corresponding to both axes set as follows:

- 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.

It is possible to have more than one pair of Gantry 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

7.1.3 Slaved axes and synchronized axes

Coupled or synchronized axes are two or more axes which are normally independent, but, sometimes need to be moved at the same time and in synchronism (temporarily slaved, versus permanently as by machine parameter). For example on multi-spindle milling machines.

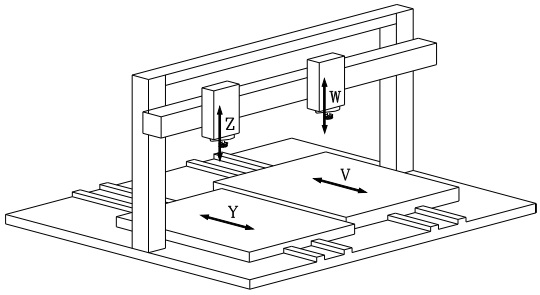
Slaved axes

- With function G77 it is possible to define which axes are to be coupled (temporarily slaved) by indicating the main axis and its subordinates or slave axes.
- It is possible to slave more than two axes to each other, to have several different electronic couplings (slaving), to add a new slave to the ones previously slaved, etc.
- With function G78, it is possible to decouple (unslave) one or all of the axes slaved temporarily; that is by means of G77, and not by machine parameter GANTRY (which would be "permanent" slaving).

Synchronized axes

- The axes are synchronized by the PLC, by activating the CNC input "SYNCHRO" of the axis to become the slave.
- To be able to do this, a.m.p. SYNCHRO (P3) of that axis must be set indicating which axis will be its master.
- It is possible to couple (slave) more than two axes to each other, to have several other axes slaved to each other, to add a new slave to existing ones, etc; but, they will always be slaved to the axes determined by the corresponding SYNCHRO parameters:
- To decouple (unslave) one of the slaved axes, the corresponding "SYNCRO" input of the CNC must be deactivated.

Example of a dual-spindle bridge type milling machine with two pairs of slaved axes (Y-V, Z-W).



Slaving (by program).	Synchronization (by external signal).	
G77 Y V	Y axis	SYNCHRO = 0
G77 Z W	V axis	SYNCHRO = 2
	Z axis	SYNCHRO = 0
	W axis	SYNCHRO = 3

If the machine has the X, Y, Z, V, W axes, the following signals must be activated (logic state "1") at the PLC:

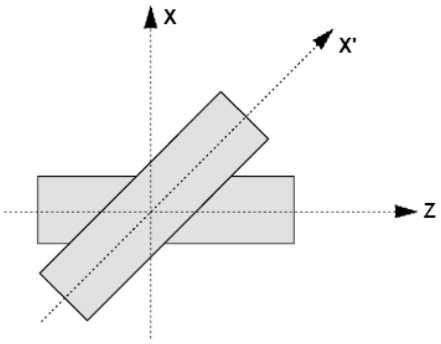
SYNCHRO4 to slave the V axis to the Y axis.

SYNCHRO5 to slave the W axis to the Z axis.

7.1.4 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.

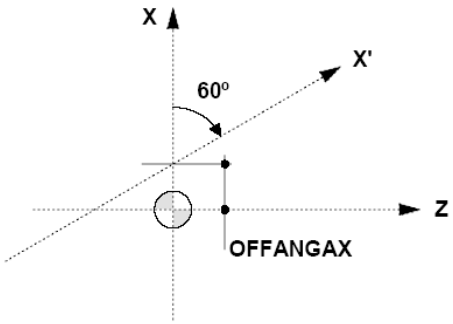


X Cartesian axis.
X' Angular axis.
Z Orthogonal axis.

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. These axes may have associated with them Gantry axes, coupled axes or axes synchronized by PLC.

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.

7.1.5 Tandem axes

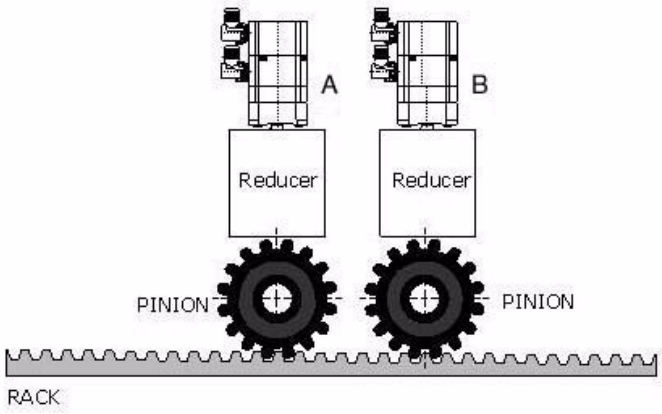
A tandem axis consists in two motors coupled to each other mechanically making up a single transmission system (axis). It is normally used to move the axes on large machines.

The CNC only displays one axis (referred to as master axis) whose movements are programmed. The other axis is called slave axis.

The following aspects may be pointed out in this configuration:

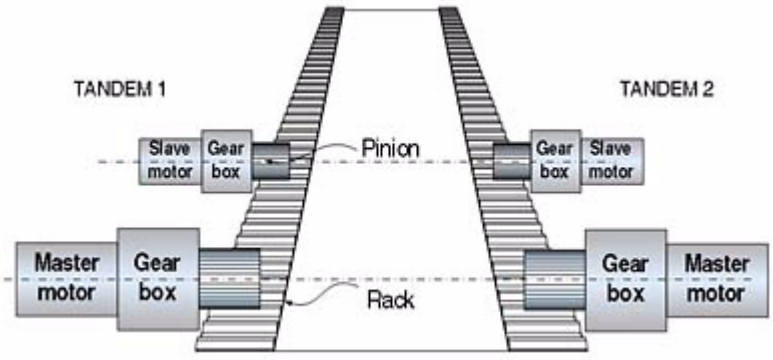
- A tandem axis helps provide the necessary torque to move an axis when a single motor is not capable of supplying enough torque to do it.
- Applying a pre-load torque between the main motor and the slave motor reduces the backlash of the axis.
- Greater rigidity is obtained on long leadscrews when applied to a rack-and-pinion system.

Example of a tandem axis made up of a rack-and-pinion system driven by two motors.



(A) Main or master motor. Besides generating torque, it is in charge of positioning.
(B) Slave motor. It only supplies torque.

One of the many applications for the control of a tandem axis has to do with Gantry machines.



Example diagram of a Gantry machine with two axes in tandem.

Configuration of the tandem axis

When configuring a tandem axis, bear in mind the following requirements.

- Each master tandem axis admits one single slave tandem axis.
- The axes must be "velocity Sercos".
- A preload may be applied between the two motors.
- Each motor may have a different rated torque.
- The turning direction of each motor may be different from the other's.
- The torque distribution between both motors may be different from 1:1 ratio. For example, on motors whose rated torque is different.
- It is not possible to share feedback or velocity commands using the switch parameters on axes having a tandem configuration.

A tandem axis is configured by the machine parameters of the master axis. The slave axis does not have an axis table at the CNC, although it is possible to access the parameter and variable tables of the slave axis drive. It is accessed from the machine parameters > drive.

A.m.p. TANSLAID (P76) enables the tandem parameters. If this parameter is zero, there is no tandem axis and it ignores the values of the rest of the tandem control parameters.

Selection of the master and slave axes

The axis whose table sets the parameters will be the master axis of the tandem. The slave axis of the tandem is set using parameters TANSLAID (P76) and TANSLANA (P77).

Parameter TANSLAID (P76) indicates the Sercos ID of the slave axis. This parameter determines the drive that will generate the velocity command of the slave axis.

Parameter TANSLANA (P77) determines the name of the slave axis. This parameter is used to identify the PLC marks, identify at the CNC the errors and identify the parameter tables of the drive.

Torque and pre-load distribution

The torque distribution determines the torque supplied by each motor in order to obtain the necessary total torque on the tandem axis. It is set with parameter TORQDIST (P78).

Pre-load is the torque difference to be applied between the two motors, so as to establish a traction between them in order to eliminate backlash. It is set with parameters PRELOAD (P79) and PRELFITI (P80).

Parameter PRELOAD (P79) sets the pre-load value and parameter PRELFITI (P80) sets the filter time to apply it. The pre-load is applied progressively during the time indicated in the filter, thus avoiding a step (shoulder) in the velocity command of the axes.



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

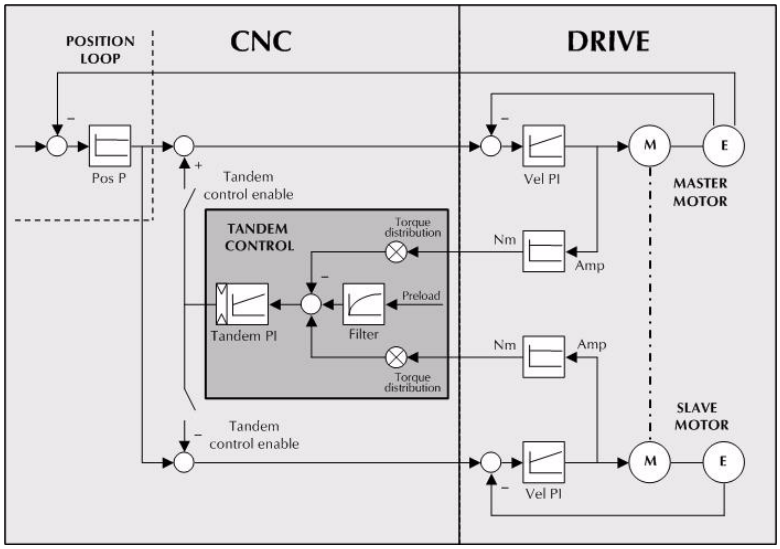
Gain setting

The proportional gain is set with parameter TPROGAIN (P81). The proportional controller generates an output proportional to the torque error between the two motors.

The integral gain is set with parameter TINTTIME (P82). The integral controller generates an output proportional to the integral of the torque error between the two motors.

Block diagram

The block diagram of the tandem control system shows the master tandem axis with its slave tandem axis. The block diagram for a Gantry machine consists of the same two diagrams as the one shown in the figure.

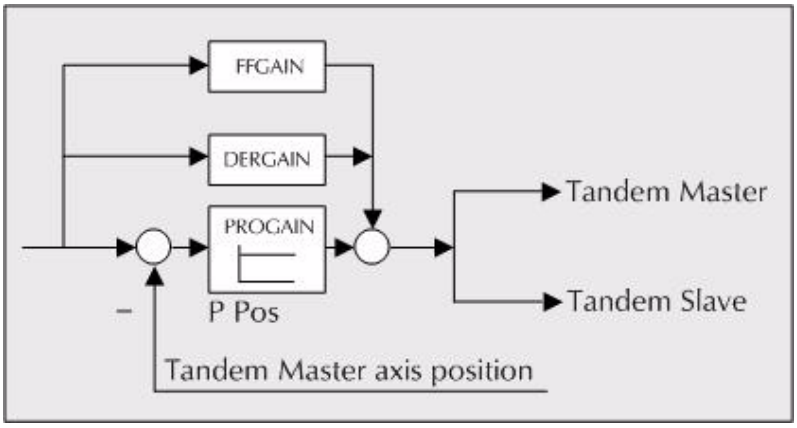


The block diagram has an area for the drive and another one for the CNC that comprises the position loop and the control of the tandem.

Position and velocity loops

The position loop is closed only with the position of the master axis of the tandem. The velocity command of the tandem master axis is also sent to the tandem slave axis closing the velocity loop.

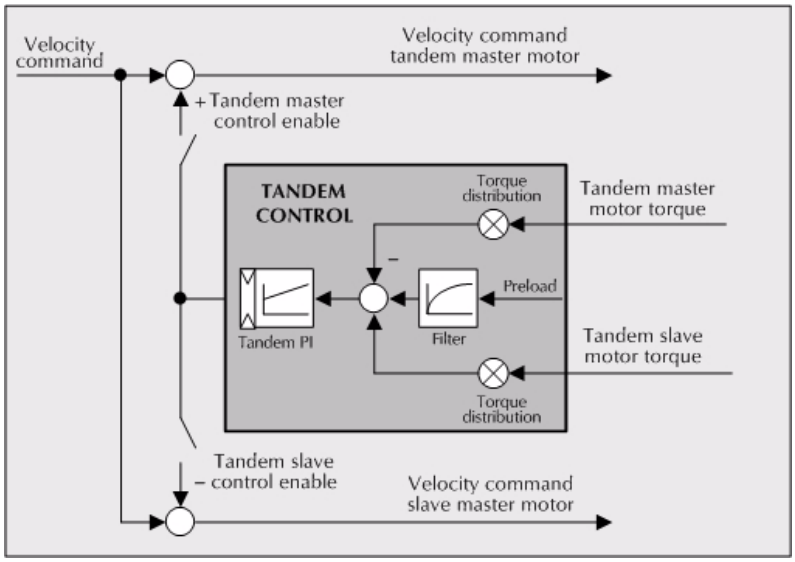
The tandem control changes the velocity command of the master axis and that of the slave axis according to the torque distribution and the selected preload.



The values of PROGAIN, FFGAIN, DERGAIN of the master axis are applied to the slave axis; therefore, the gear ratios must necessarily be the same.

Tandem axis control.

The block diagram for the application to control the tandem axis is:



Tandem master motor torque.

Is the percentage (%) of rated torque reflected by the Sercos variable TV2 of the drive that governs the master axis of the tandem. It is read in every loop.

Tandem slave motor torque.

Is the percentage (%) of rated torque reflected by the Sercos variable TV2 of the drive that governs the slave axis of the tandem. It is read in every loop.

Torque distribution.

Normalizing gain of the torque generated by the motors in order to distribute the torque in a ratio other than 1:1.

Pre-load.

Previous torque applied to both tandem axes in opposite direction. This sets a traction between them in order to eliminate the rack-and-pinion backlash when it is in rest position. It is determined as the torque difference supplied by each axis.

Pre-load filter.

First-order filter to prevent torque steps from coming in when configuring the preload.

Tandem PI.

PI for making each motor provide its corresponding torque. It increases its velocity command if the torque being supplied is too low and it decreases it if the torque being supplied is too high.

When defining a tandem axis, at each loop, the CNC reads via Sercos the torque supplied by the master and the slave axes. This may result in a smaller number of read-write variables in the fast Sercos channel, through PLC parameters.

This situation may generate a "Sercos ring error", even when using the same variables and number of axes as in a non-tandem configuration. This situation is especially critical for a loop time of 2 milliseconds.

Adjustment procedure

This procedure must bear in mind the type of machine. In general, a tandem machine has a low resonance frequency. Therefore, the CNC must generate position commands without frequency components higher than resonance frequency.

It is recommended to begin the process with low jerk values (lower than 10 m/sec3) and low Kv. They can always be increased in a later readjustment.

Steps for the adjustment

1. Move both axes independently.

   The first step is to ensure the perfect operation of both the master and the slave axes separately. Also verify that both axes move in the same direction with similar dynamics.

   To do this, the following methods may be used:

   – Define a master axis and a slave axis using a separate parameter table for each axis. When adjusting the master axis, set the slave axis as a dro axis and viceversa.

   – Cancel the "drive enable" of the slave axis and move the master axis. Change the parameter settings of the CNC to turn the slave axis into the master axis and repeat the process.

2. Move one of the motors at a constant low speed. Do not make jerky movements because the second motor is being dragged by the first one. In this situation, any acceleration or deceleration forces it to go from one side of the backlash to the other thus making it jerky.

   Verify that the turning direction of both motors are coherent once the movement has been carried out.

   Observe that reversing the turning direction of a motor reverses the direction of the torque and, consequently, it will be necessary to change the direction of its values monitored with drive parameters SP43 and TP85.

   Verify that the gear ratio in both motors is the same (same feed for same turning speed).

   Carry out a basic adjustment of the velocity loop so the machine can move. It will be readjusted later with both motors together.

   Do not set the friction parameters (there is already enough torque to move the machine).

3. Repeat the process with the second motor.

   When adjusting the loops, use the same parameters if the motors are identical and the torque distribution is 50%. If the motors are different, the axes must be adjusted so their dynamic response is the same or very similar.

   When using AC-Forward ("ACFGAIN" = YES), remember that each motor has half the inertia for a torque distribution of 50%.

4. Enable the tandem with both motors.

   Disable the tandem PI, supply power and verify that the system is resting. Enter a low proportional value and eliminate the integral value of the PI of the tandem.

   Then, without pre-load, verify that the machine moves and that each motor provides its corresponding torque according to parameter "TORQDIST" (e.g. for a 50% distribution, half the torque).

5. Enable the pre-load.

   Monitor the torque of each motor (Sercos variable TV2). While stopped, increase the preload gradually until the motors supply torque in opposite directions.

   Move slowly in both directions and verify that it works properly. Make sure that it is not jerky and that each motor supplies its corresponding torque according to parameters "TORQDIST" and "PRELOAD".

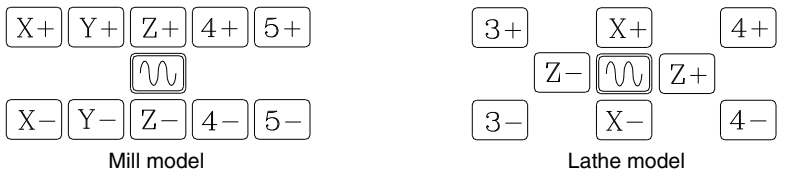
6. Readjust the velocity loop in both motors with the method used normally.

While changing the parameters of the velocity loop, the proper thing would be to change them on both drives at the same time. But, since this is not possible, it is recommended to make small changes in the values or make them while the motor is stopped.

7.2 Jog

7.2.1 Relationship between the axes and the JOG keys

The mill model CNC has 5 pairs of JOG keys and the lathe model has 4 pairs of keys to jog the axes of the machine.



Mill model — Lathe model

The X, Y and Z axes always use their own denomination; the "C" axis of a lathe model uses the [3+] and [3-] keys and the rest of the axes depend on the chosen name.

The logical order is: X Y Z U V W A B C.

Key configuration examples

A milling machine has the X Y Z U B axes.

The X axis keys are:	[X+]	[X-]
The Y axis keys are:	[Y+]	[Y-]
The Z axis keys are:	[Z+]	[Z-]
The U axis keys are:	[4+]	[4-]
The B axis keys are:	[5+]	[5-]

A laser machine has the X Y A B axes.

The X axis keys are:	[X+]	[X-]
The Y axis keys are:	[Y+]	[Y-]
The A axis keys are:	[Z+]	[Z-]
The B axis keys are:	[4+]	[4-]

A punch press has the X Y C axes.

The X axis keys are:	[X+]	[X-]
The Y axis keys are:	[Y+]	[Y-]
The C axis keys are:	[Z+]	[Z-]

A lathe has the X Z U A axes.

The X axis keys are:	[X+]	[X-]
The Z axis keys are:	[Z+]	[Z-]
The U axis keys are:	[3+]	[3-]
The A axis keys are:	[4+]	[4-]

A lathe has the X Z U C axes.

The X axis keys are:	[X+]	[X-]
The Z axis keys are:	[Z+]	[Z-]
The U axis keys are:	[4+]	[4-]
The C axis keys are:	[3+]	[3-]

7.2.2 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.

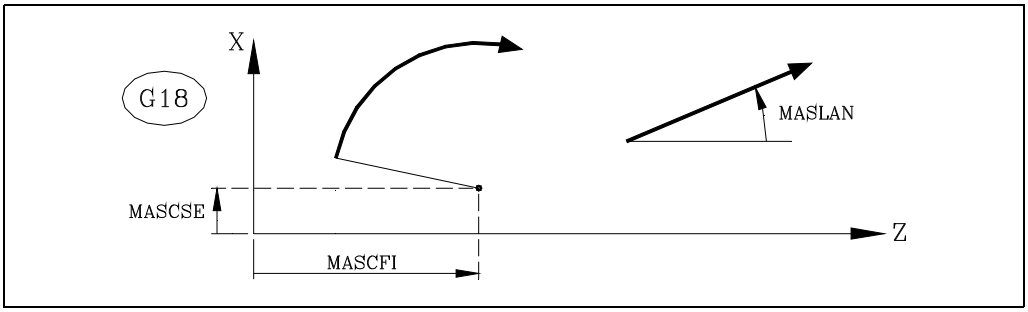
M5054 = 1 Path jog mode ON.

To indicate the type of movement, use CNC logic input "HNLINARC" M5053.

M5053 = 0 Linear Path.

M5053 = 1 Circular 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.

7.3 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.

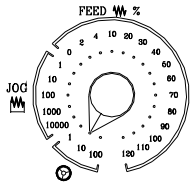
  It replaces the mechanical handwheels.

  Up to 3 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.*

7.3.1 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.

7.3.2 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.

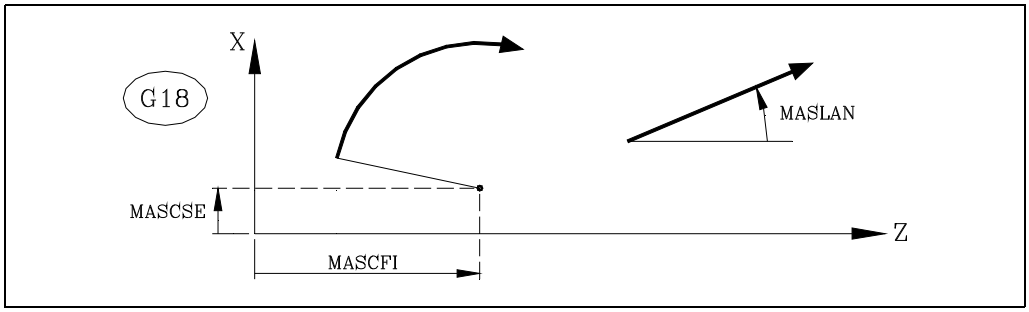
M5054 = 1 Path jog mode ON.

To indicate the type of movement, use CNC logic input "HNLINARC" M5053.

M5053 = 0 Linear Path.

M5053 = 1 Circular 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

7.3.3 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.

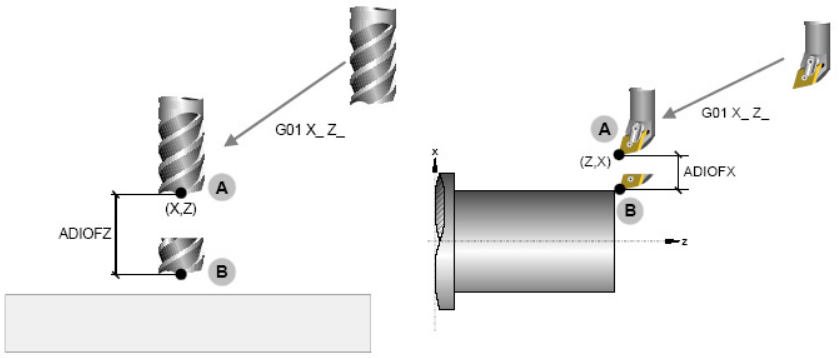
```
CY1
R101=0
    Resets the register containing the previous handwheel reading.
END

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
```

7.3.4 "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 cannot be enabled while TCP (G48) is active. However, the additive handwheel is enabled for a coordinate transformation G46 (inclined axis) or G49 (inclined plane) 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, slaved or synchronized by PLC, 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.

7.4 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.

- 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.

Next, the feedback counting speed (frequency) limitation is described as well as how to set these machine parameters for the axes.

7.4.1 Counting speed limitation

Sinusoidal signals

The maximum counting speed (frequency) for sinusoidal feedback is 50 kHz at the 8055 and 250 kHz at the 8055i.

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:

8055	20 µm/pulse x 50,000 pulses/s. = 60 m/min.
8055i	20 µm/pulse x 250,000 pulses/s. = 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:

8055	(360 degrees/turn / 3600 pulses/turn) x 50,000 pulses/s. = 5,000 degrees/sec.= 300,000 degrees/min.
8055i	(360 degrees/turn / 3600 pulses/turn) x 250,000 pulses/s. = 25,000 degrees/sec.= 1,500,000 degrees/min.

Since Fagor sine-wave encoders admit a frequency of up to 200 KHz, the maximum feedrate for the 8055i 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 squarewave differential feedback is 400 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).

7.4.2 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. Feed = 200,000 pulses/s. / 625 pulses./turn) x 5 mm/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

The feedback frequency of Fagor rotary encoders is limited to 200 KHz.

The 8055 CNC's maximum sinusoidal feedback input frequency is 50 KHz which means that the maximum feedrate for this axis will be:

Max. Feed = 50,000 pulses/s. / 250 pulses./turn) x 5 mm/turn

Max. feedrate = 1,000 mm/s = 60 m/min.

The 8055i CNC's maximum sinusoidal feedback input frequency is 250 KHz which means that the maximum feedrate for this axis will be:

Max. Feed = 200,000 pulses/s. / 250 pulses./turn) x 5 mm/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 PITCH=0.1000	NPULSES = 0	SINMAGNI=0

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/s.

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 8055 CNC's maximum sinusoidal feedback input frequency is 50 KHz which means that the maximum feedrate for this axis will be:

Max. Feed = 20 µm/pulse x 50.000 pulses/s.

Max. feedrate = 1,000 mm/s = 60 m/min.

The 8055i 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/s.

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. Feed = 200,000 pulses/s. / 625 pulses./turn) x 0.255 inch/turn

Max. Feed = 80 inches/s. = 4800 inches/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

The feedback frequency of Fagor rotary encoders is limited to 200 KHz.

The 8055 CNC's maximum sinusoidal feedback input frequency is 50 KHz which means that the maximum feedrate for this axis will be:

Max. Feed = 50,000 pulses/s. / 250 pulses./turn) x 0.2 inch/turn

Max. feed = 40 inches/s. = 2400 inches/min.

The 8055i CNC's maximum sinusoidal feedback input frequency is 250 KHz which means that the maximum feedrate for this axis will be:

Max. Feed = 200,000 pulses/s. / 250 pulses./turn) x 0.2 inch/turn

Max. Feed = 160 inches/s. = 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. Feed = 200,000 pulses/s. / 18,000 pulses/turn)

Max. Feed = 11.111 turns/s. = 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

The feedback frequency of Fagor rotary encoders is limited to 200 KHz.

The 8055 CNC's maximum sinusoidal feedback input frequency is 50 KHz which means that the maximum feedrate for this axis will be:

Max. Feed = 50,000 pulses/s. / 3,600 pulses/turn)

Max. Feed = 13.8889 turns/s. = 833.33 rpm

The 8055i CNC's maximum sinusoidal feedback input frequency is 250 KHz which means that the maximum feedrate for this axis will be:

Max. Feed = 200,000 pulses/s. / 3,600 pulses/turn)

Max. Feed = 55.5556 turns/s. = 3333.33 rpm

7.5 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).

7.5.1 Drive adjustment

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:

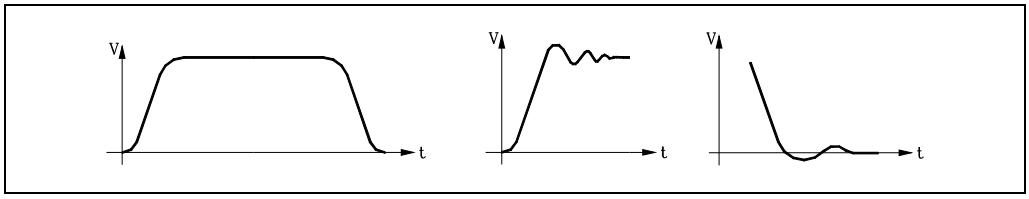
```
N10     G00 G90 X200
N20     X-200
        (RPT N10, N20)
```

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.

7.5.2 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.

7.5.3 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).



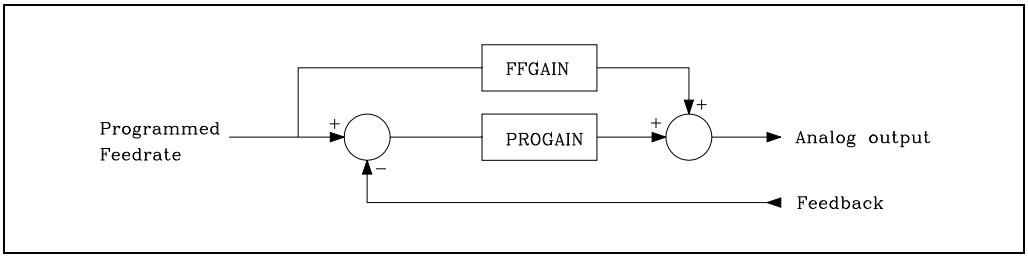
Once the axes have been adjusted separately, the ones being interpolated together should be further adjusted so their following errors are as identical as possible.
The more identical their following errors are, the more "round" the programmed circles will turn out.

7.5.4 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).

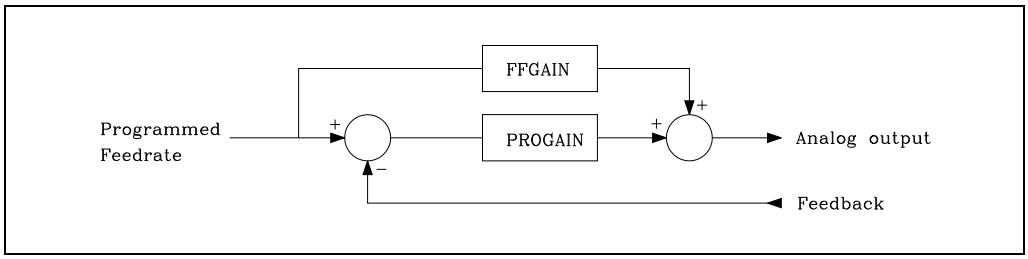
between 80 and 100%

7.5.5 Derivative (AC-forward) gain setting

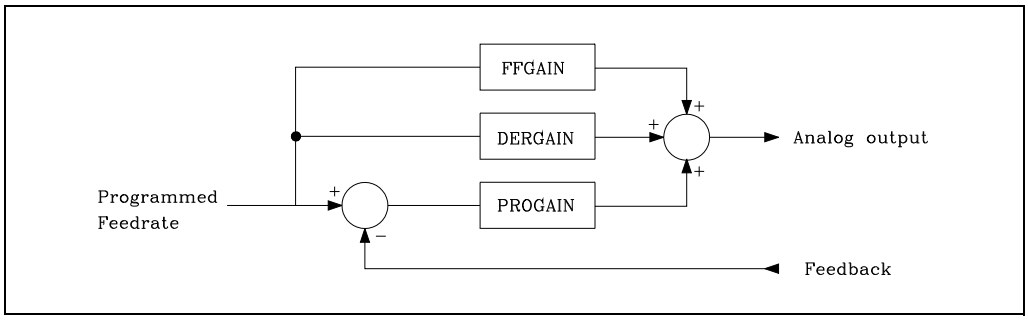
With the derivative gain, it is possible to reduce the following error during the acc./dec. stages.

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.



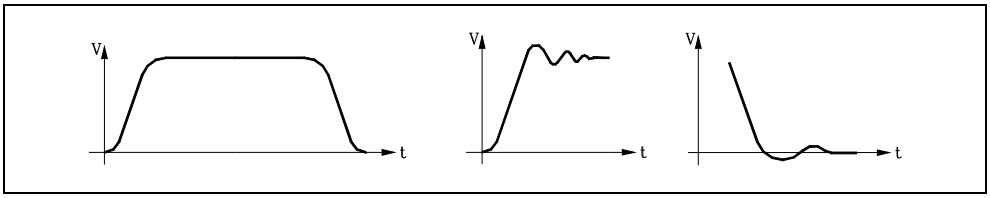
Best results are usually obtained when using it as AC-forward Gain, "ACFGAIN" (P46) = YES together with feed-forward gain.

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:

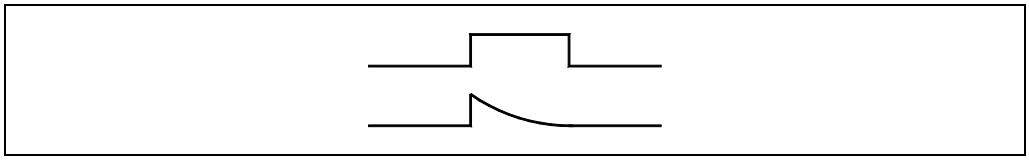
- Verify that there are no oscillations of following error, In other words, that it is not unstable.
- Check,with an oscilloscope, the tacho voltage or the command voltage at the drive (velocity command), verify that it is stable (left graph) and that there are no instabilities when starting up (center graph) or when braking (right graph).



7.5.6 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 velocity command is eliminated (peak compensation 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

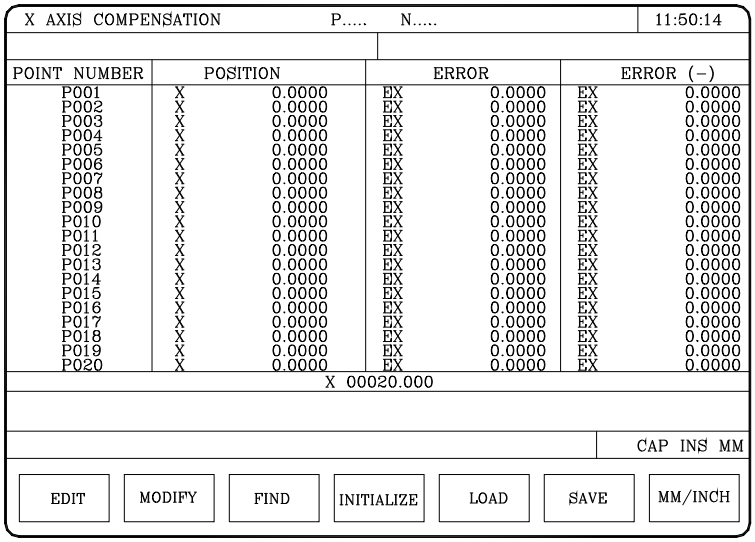
Axis machine parameter REVEHYST (P99) is used to be able to control when the compensation should really be applied after detecting a movement reversal and not applying it every time a reversal command is received.

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.

7.5.7 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 1000 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.

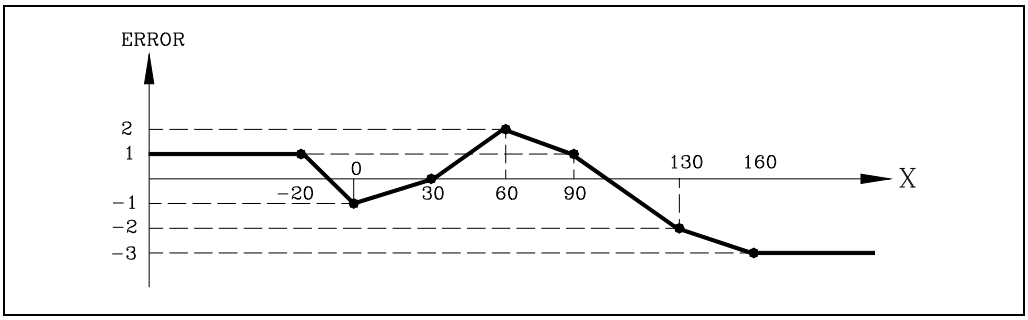
Considerations and limitations

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%).

Setting example

The X axis leadscrew 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

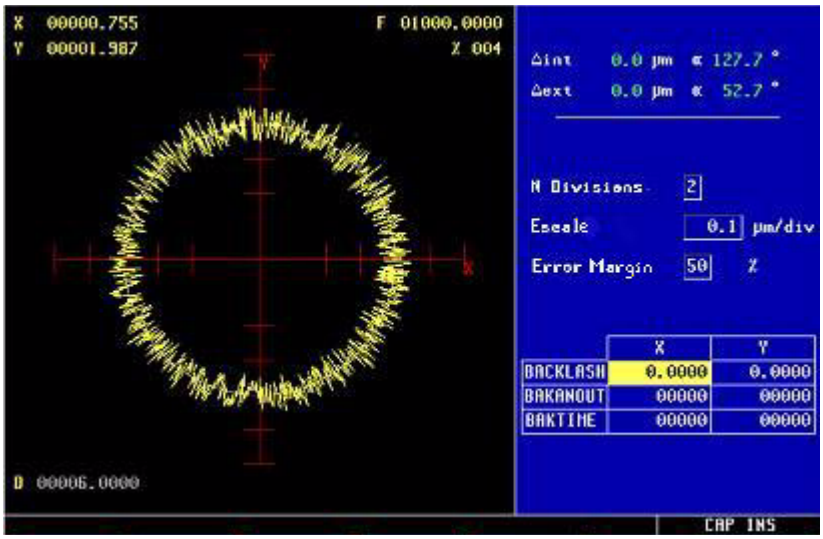
7.5.8 Circle geometry test

This adjustment improves the axis reversal peaks. It consists in machining a circle (without compensation) and verifying it on the graph displayed at the CNC.

The following example shows a program that machines repetitive circles.

```
        X0 Y0
        G5 G1 F1000
N10     G2 X0 Y0 I10 J0
        (RPT N10, N10) N50
        M30
```

After selecting this program in the Execution mode and starting it, access the "Diagnosis, Adjustments, Circle Geometry test" mode and the CNC will display the following screen:



If the machine parameters are protected, it will request the access password because the bottom right of the screen shows some of them. When not knowing the password, those values cannot be changed, but it will be possible to access the screen and the circle geometry test.

The left side of the CNC screen shows the result of the test.

The data at the top right is refreshed by the CNC after the test is completed.

The data at the center right must be defined before running the test.

The bottom right side of the screen shows the parameters associated with the plane axes and the values to set them.

The graph on the left must be defined before running the test. To do that, define the data of the center right:

- Number of divisions to the left and to the right of the theoretical circle.
- Scale or value in microns of each division.
- Error margin or % of circle radius occupied by the error margin (divisions area).

Knowing the password, the values shown at the bottom right may be modified. The CNC assigns the new values to the relevant machine parameters; therefore, it is recommended to jot the initial values down.

Once the graphic display area and the machine parameters have been defined, capture the data by pressing the following softkeys:

SINGLE

It deletes the current graph and draws, over the theoretical circle, the machining error enlarged according to the defined scale until a full circle is drawn or until the STOP softkey or ESC key is pressed.

CONTINUOUS

It deletes the current graph and draws, over the theoretical circle, a series of circles with the machining error enlarged according to the defined scale until the STOP softkey or ESC key is pressed.

DELETE

It may be pressed at any time, even while drawing the graph. It deletes the screen and resets the statistics shown at its right.

During continuous graphics, it is possible to modify the machine parameters and view the new graph over the previous one, or press the DELETE softkey to only display the new one.

The data shown by the CNC at the top right is updated while capturing data.

Δ internal	Maximum negative value of the error over the theoretical value, in microns or ten-thousandths of an inch and its angular position.
Δ external	Maximum positive value of the error over the theoretical value, in microns or ten-thousandths of an inch and its angular position.

Once data capture is done, it draws two lines indicating the angular positions of both errors on the graph. They appear in dashed lines when the error exceeds the value assigned to the display area in its quadrant and it goes on to the opposite quadrant.



While capturing points for the geometry test, the execution graphics stops drawing.

7.6 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

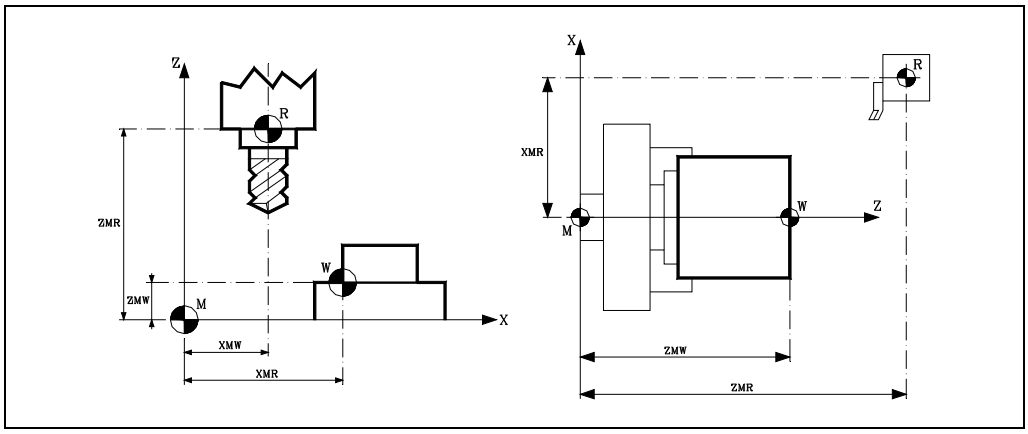
Part origin point. This is the origin point that is set for programming the measurements of the part. It can be freely selected by the programmer, and its machine reference zero can be set by the zero offset.

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

7.6.1 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).

Home search on SERCOS axes using absolute feedback

If the first feedback is absolute (position feedback obtained from the absolute encoder of the motor), the CNC knows the axis position at all times. This allows the CNC to do a home search without having to use the travel limit switches and home switches.

To do a home search this way, the home switch (PLC mark DECEL*) may be generated by PLC reading the axis position value (coordinate).

This home search requires servo drive version V6.17 or later.

If the second feedback uses distance-coded reference marks, the offset applied in the home search will be the value of a.m.p. ABSOFF (P53).

To remove travel limit switches and home switches, proceed as follows:

1. Get both feedbacks (variables PV51 and PV53) to have the same counting direction.
   - Drive: Bit 3 of parameter PP115: Direction of the second feedback
     Bit 5 of parameter PP115: Direction of distance-coded I0?'s
   - CNC: A.m.p. AXISCHG (P13) and a.m.p. LOOPCHG (P26)
2. Change the value of drive parameter PP177 (distance between the zero position of the drive and the theoretical zero) so the position value of the first feedback (PV51) is correct.
3. Restart the CNC and the drives.
4. Home the axis. Set the position value of the second feedback (PV53) right by changing one of the following axis parameters of the CNC:
   - If the linear encoder uses distance-coded reference marks: a.m.p. ABSOFF (P53).
   - If it is another type of linear encoder: a.m.p. REFVALUE (P36).
5. Restart the CNC and check that the coordinates after a home search are correct.

PLC programming example:

Use the following PLC program to generate the DECEL mark using absolute motor feedback and non-absolute second feedback. The example assumes that the home search is carried out in the positive direction, a.m.p. REFDIREC (P33) = + sign.

```
;
()
= CNCRD(POSX5,R100,M1000)        ; Read real machine position value of the X axis
= CNCRD(MPX5,R101,M1001)         ; Read X axis positive software travel limit
= SBS R101 200000 R101           ; Positive limit -20mm
;
CPS R600 GE R601 \               ; Real machine coordinate >= [Positive limit -20 mm]
= DECELX                         ; X axis home switch
;
```

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.

7.6.2 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:

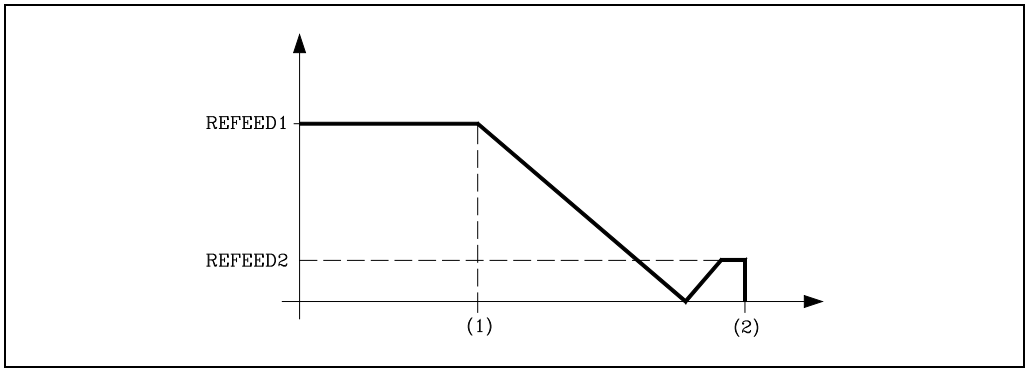
  "REFVALUE" = 230 - (-123.5) = 353.5 mm.
- After assigning this new value, press [RESET] so it is assumed by the CNC.
- The axis must be homed again in order for it to assume its right reference values.

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.

7.6.3 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. Then do 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.

7.6.4 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.

7.7 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.

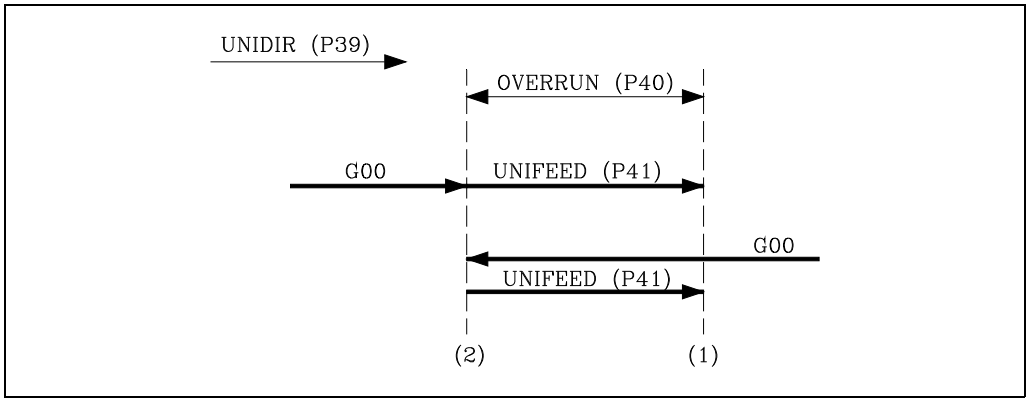
OVERRUN

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).

7.8 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 *"11.1 Auxiliary M, S, T functions"* on page 460. See *"12.7 General logic outputs"* on page 495. See *"6.10 Tables"* on page 280.

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.

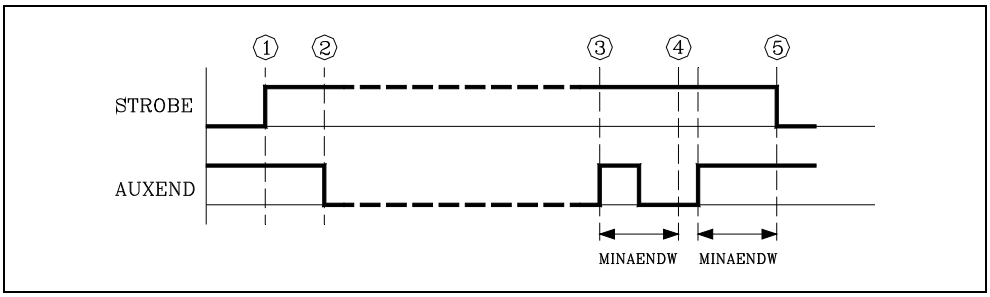
This indication will be made by means of the variable "T2BCD" (R559) and by activating the general logic output "T2STROBE" to tell the PLC that it must execute this. The CNC will wait for the general input AUXEND to be activated to consider the execution completed.



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.

7.8.1 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.

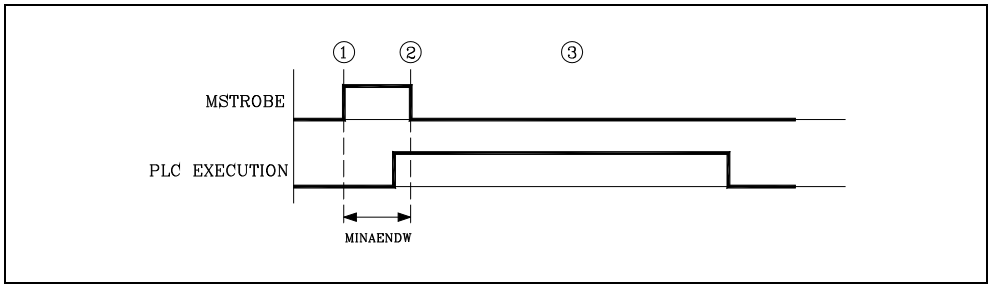
When executing 2 consective blocks which send information to the PLC and after finishing the execution of the first block, the CNC waits a MINAENDW period of time before starting to execute the second block.

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.

7.8.2 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".

7.9 Main and second spindle

This CNC can handle 2 spindles: a main spindle and a second spindle. They both can be operative simultaneously, but only one can be controlled at a time. This selection is made using functions G28 and G29. See programming manual.

Next, the steps to be followed when using two spindles are described.

Screen editor

Set g.m.p. AXIS1 (P0) through AXIS8 (P7) with the desired valures. A value of "10" for the main spindle and 14 for the second spindle.

Set the corresponding machine parameters for each spindle.

Spindle selection

On power-up, the CNC always selects the main spindle.

All the keyboard actions and by spindle related functions affect the main spindle.

Example: S1000 M3

Main spindle clockwise at 1000 rpm

To select the second spindle, execute function G28.

From then on, All the keyboard actions and spindle related functions affect the second spindle.

The main spindle remains in its previous status.

Example: S1500 M4

Second spindle turns counterclockwise at 1500 rpm.

The main spindle keeps turning at 1000 rpm

To select the main spindle again, execute function G29.

From then on, all the keyboard actions and spindle related functions affect the main spindle.

The second spindle stays in its previous status.

Example: S2000

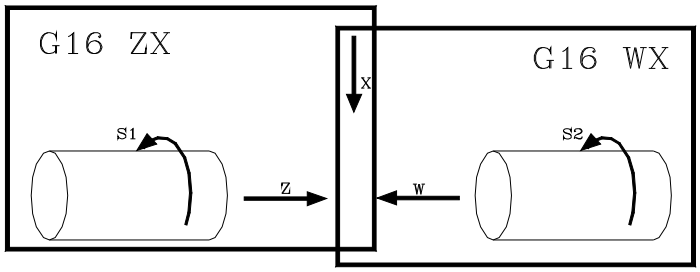
The main spindle keeps turning clockwise but at 2000 rpm.

The second spindle keeps turning at 1500 rpm.

Work plane selection

Use function G16 to select the work plane. See programming manual.

Example



Machining canned cycles

When working in a plane other than the ZX, for example: G16 WX, the CNC interprets the canned cycle parameters as follows:

- Parameter Z and all those related to it, with the abscissa axis (W in the example).
- Parameter X and all those related to it, with the ordinate axis (X in the example).

Tool compensation

When working in a plane other than ZX, for example G16 WX, the CNC allows associating the tool offset table to the work plane.

To do that, set g.m.p. PLACOMP (P78) to "1". See *"6.3 General machine parameters"* on page 148.

When g.m.p. PLACOM = 1, the CNC interprets the tool table as follows:

	ZX plane	WX plane
The Z and K parameters, with the abscissa axis	Z axis	W axis
The X and I parameters, with the ordinate axis	X axis	X axis

7.9.1 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.

7.9.2 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:

S Programmed	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

S Programmed	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

S Programmed	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.

7.9.3 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.

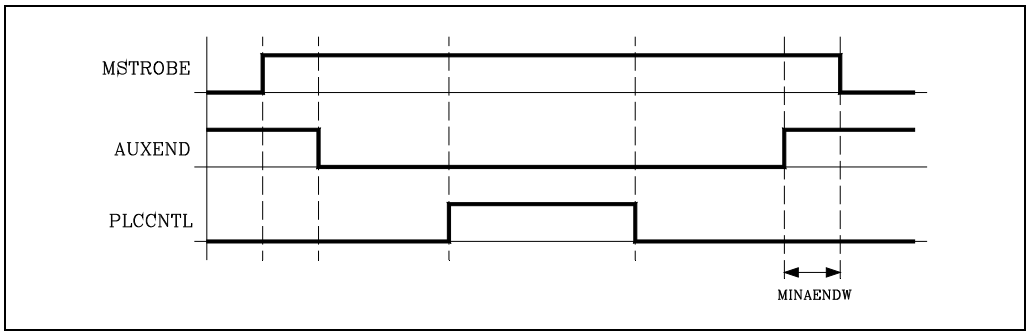
In turn, the PLC must "tell" the CNC the speed gear being selected. This will be indicated by means of the logic inputs for the spindle: "GEAR1 (M5458)", "GEAR2 (M5459)", "GEAR3 (M5460)" and "GEAR4 (M5461)".

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.

7.9.4 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 (SERCOS or 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:

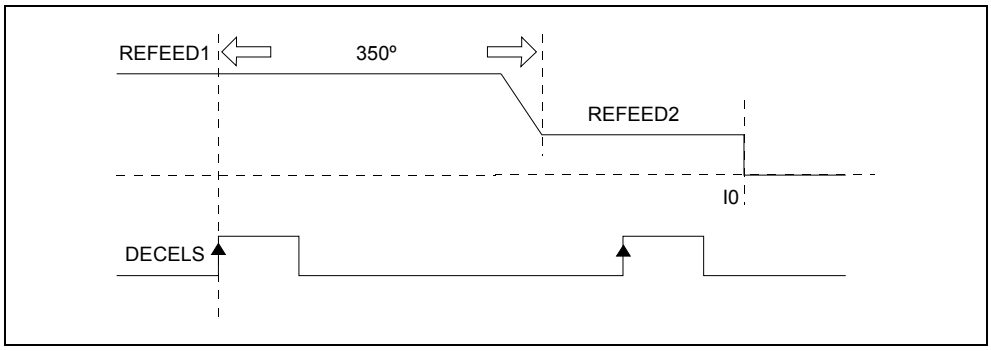
  In this case, it is possible to detect the right reference mark among the possible ones (having the encoder at the motor) originated especially by different gear ratios.

  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.

  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 SERCOS, analog or 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.

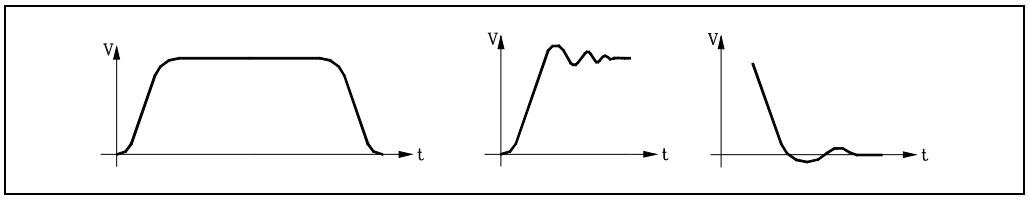
s.m.p. NPULSES (P13) must indicate the number of square pulses supplied by the spindle encoder.

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:

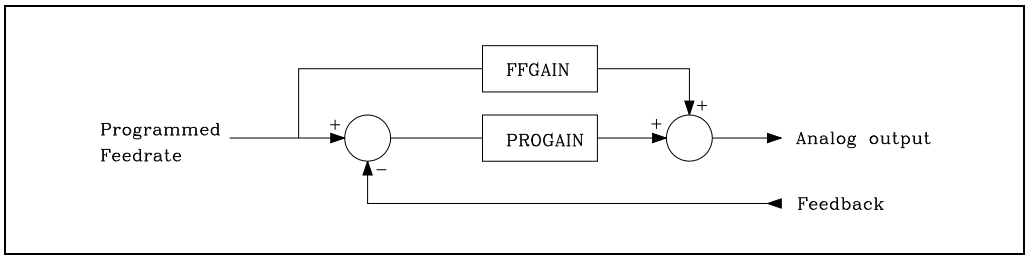
- The maximum amount of following error allowed by the CNC for the spindle is the value indicated by s.m.p. MAXFLWE1 (P21). When exceeded, the CNC issues the corresponding following error message.
- The amount of following error decreases as the gain increases, but it tends to make the system unstable.

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.

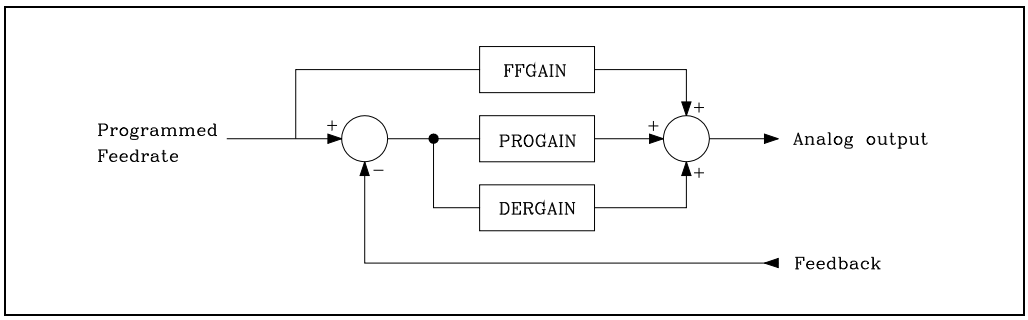
Next, set s.m.p. FFGAIN (P25) to the desired value.

Derivative (AC-forward) gain setting

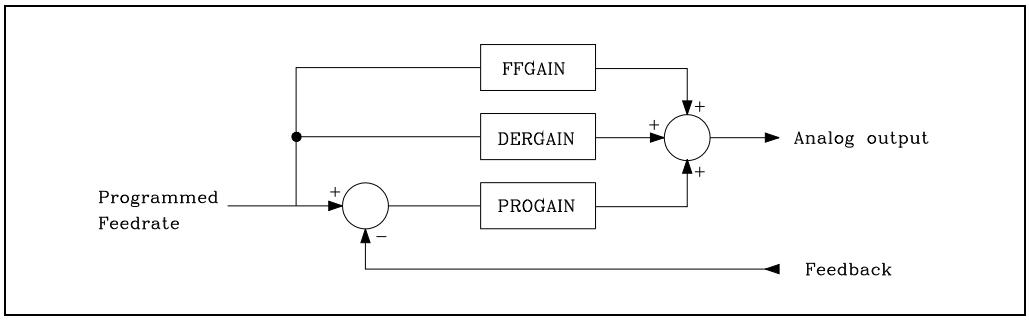
With the derivative gain, it is possible to reduce the following error during the acc./dec. stages.

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.



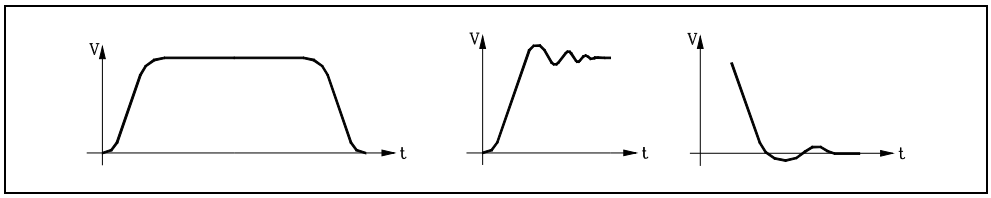
Best results are usually obtained when using it as AC-forward Gain, "ACFGAIN" (P42) = YES together with feed-forward gain.

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:

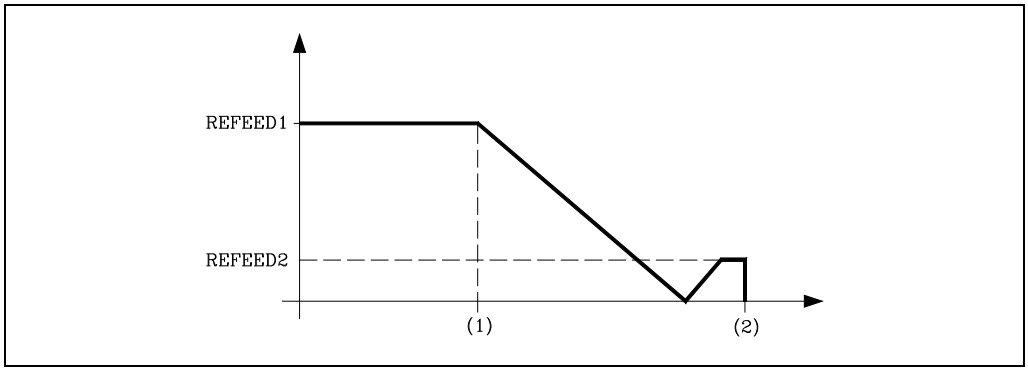
  If the point whose known position is located at 12º mm from Machine Reference Zero and the CNC reads -123.5º as the coordinate value for this point, the coordinate of the Machine Reference Point with respect to Machine Reference Zero will be:

  "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.

7.10 Auxiliary spindle controlled by PLC

With this feature, the PLC can temporarily control the auxiliary spindle.

To do that, follow these steps:

1. Indicate from the PLC at the logic CNC input "SANALOAS" (R509) the amount of velocity command to be applied to the drive for the auxiliary spindle.

   On the other hand, set logic CNC input "PLCCNTAS" (M5056) high to indicate to the CNC that from then on, it is going to be up to the PLC to control the velocity command output for the auxiliary spindle.
2. From then on, the CNC outputs the velocity command voltage indicated by the PLC for the auxiliary spindle as indicated at the CNC logic input "SANALOAS" (R509).

   If the PLC changes the value of the "SANALOAS" input, the CNC will update its velocity command output.
3. Once the operation has concluded, the control of the auxiliary spindle must be returned to the CNC. To do that, the logic CNC input "PLCCNTAS" (M5056) must be set low.

7.11 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 2 of connector X9 of the –Axes– module of the 8055 CNC

Pin 10 of connector X2 of the 8055i CNC

The emergency outputs of the CNC are:

/ALARM (M5507)

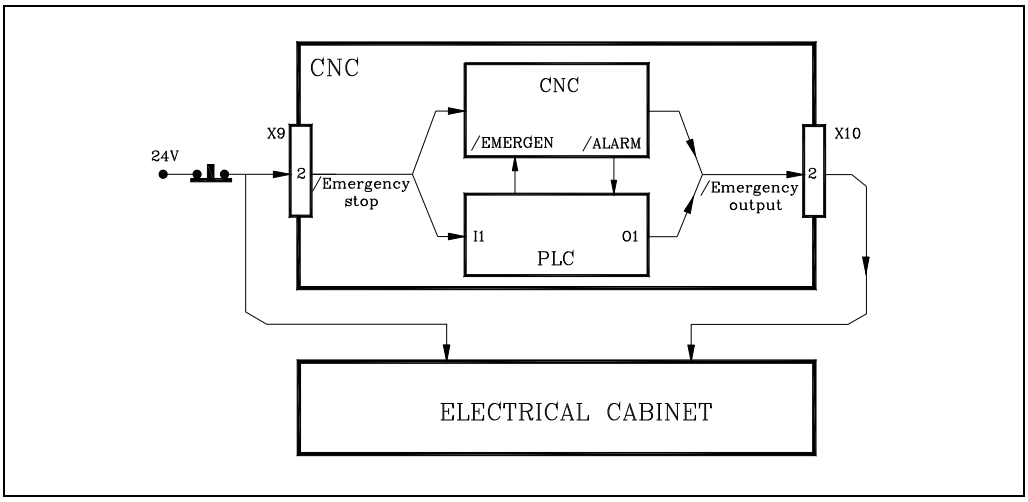
Physical output to the PLC.

/EMERGENCY OUTPUT

Physical output to the outside.

Pin 2 of connector X10 of the –Axes– module of the 8055 CNC

Pin 2 of connector X2 of the 8055i CNC.



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.

7.12 Digital servo (CAN or Sercos)

Sercos communication requires a drive version V3.01 or later. CAN communication requires a drive version V7.02 or later.

g.m.p. SERSPEED (P120) and SERPOWSE (P121) allow setting the communications speed and power for Sercos. G.m.p. CANSPEED (P169) may be used to set the CAN communication speed.

"C" axis and spindle with a single drive

When operating with Sercos or CAN and using a single drive for both the "C" axis and the spindle, proceed as follows:

The DRIBUSID parameters for the "C" axis and the spindle must be set with the same value (same Sercos address). Use two sets of parameters for the drive, one to work as "C" axis and the other one as spindle. The "C" axis must always be assigned the last set of parameters (7).

It is recommended to set C axis parameter SWITCHAX (P65) to ·10·.

The PLC must handle the change of parameter sets of the drive.

1. When switching over to operating as a C axis, the CNC lets the PLC know by activating the spindle logic output CAXIS. Communication takes place once the spindle speed is below the home searching speed.
2. The PLC, when detecting that the CAXIS signal has been activated (leading edge), must select, at the drive, the parameter set to work as "C" axis. This selection is made through the "Service Channel".
3. The PLC, once the change of parameter set at the drive has been confirmed, must let the CNC know. To do that, it must activate the logic input "CAXSEROK" indicating this way that the drive is ready to work as a "C" axis.
4. From then on, the CNC sends the velocity command to the "C" axis and receives the "C" axis position signals.
5. On the other hand, when quitting the "C" axis mode, the CNC deactivates the CAXIS signal. The PLC must select, at the drive, the parameter set to work as a spindle and let the CNC know by deactivating the logic input "CAXSEROK".

The errors that may be detected via Sercos or CAN will be identified as corresponding to the active axis, C axis or spindle.

If the "C" axis and the spindle do not share a drive, they will be assigned a different DRIBUSID identifier and no switching will be required via PLC.

7.12.1 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

Type of information	Identifier
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).

Service channel. Changing parameter sets and gear ratios via Sercos

It is recommended to use this feature when the feedback is handled via Sercos (SERCOSLE = 1 or 2).

The drive may have up to 8 gear ratios (0 through 7). Sercos identifier 218, "GearRatioPreselection".

It may also have up to 8 parameter sets (0 through 7). Sercos identifier 217, "ParameterSetPreselection".

The following write variables must be used to select these sets from the CNC:

SETGEX, SETGEY, SETGEZ	for the axes
SETGES	for the main spindle
SSETGS	for the second spindle

The 4 least significant bits of these variables indicate the gear ratio and the other 4 the parameter set to be selected.

To send this information to the drive, a high-level block must be executed in a part-program, PLC channel or user channel as mentioned earlier.

It takes time to the drive to change the parameter set and the gear ratios. That is why a new PLC mark has been defined SERPLCAC (M5562). This mark will be active from when the change is requested until the drive assumes the new values. No other SETGE* change may be requested while this mark is active because the command would be lost.

7.12.2 Drive's absolute feedback

If the drive has version V4.02 or later, absolute feedback is treated at the drive's first feedback.

The CNC checks the "RV5" variable of the drive (drive set with absolute encoder) and drive parameter PP177 (Absolute distance1) that indicates the distance between machine zero (home) and the encoder's absolute zero.

7.13 Axes (2) controlled by a single drive

To control 2 axes through a single servo drive:

- Set a.m.p. SWITCHAX (P65) and SWINBACK (P66).

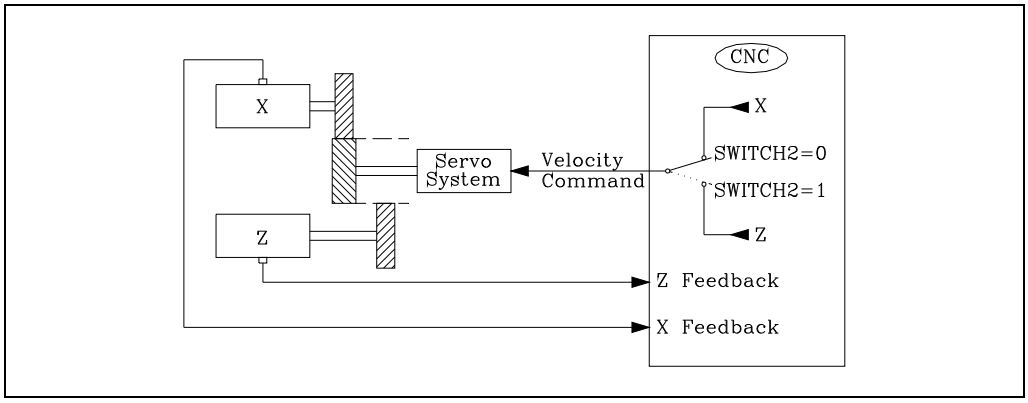
	Main axis	Associated axis
SWITCHAX	0	Code of the main axis.
SWINBACK	0	·0· If it assumes the feedback of the main axis. ·1· if it has its own feedback.

- Act upon marks SWITCH1 through 7 corresponding to the secondary axis for selecting the axis to be governed. "0" for the main axis and "1" for the secondary.
- Since the turning direction of the two axes may be different, indicate the sign of the velocity command in parameter LOOPCHG (P26) of both parameters.

When communication is via Sercos, proceed as follows:

- The a.m.p. SERCOSID for both axes must be set with the same value (same Sercos address).
- To govern one of the axes, enable the SERVOON, SPENA and DRENA signals of that axis and activate the DRO signal of the other axis so it works as a DRO (not controlled).

X and Z paraxial axes (moving one at a time) and independent feedback.



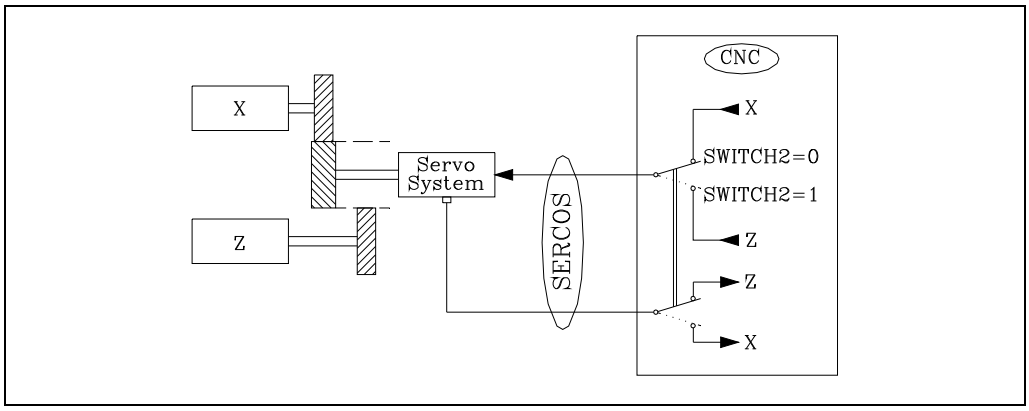
X axis (main)	Z axis (secondary)
SWITCHAX for X = 0	SWITCHAX for Z axis = 1 (X axis)
SWINBACK for X axis = 0	SWINBACK for Z axis = 1

The velocity command is always output through the X axis connector (main).

The mark for the secondary axis is SWITCH2 (M5155).

With SWITCH2=0 , velocity command of the X axis and with SWITCH2=1 that of the Z axis.

X and Z paraxial axes (moving one at a time) and communication via Sercos, feedback included



X axis (main)	Z axis (secondary)
SWITCHAX for X = 0	SWITCHAX for Z axis = 1 (X axis)
SWINBACK for X axis = 0	SWINBACK for Z axis = 0

Using the mark for the secondary axis, SWITCH2 (M5155), one can select which axis the velocity command and feedback data transmitted via SERCOS correspond to

With SWITCH2=0, velocity command and feedback data for the X axis.

With SWITCH2=1, velocity command and feedback data for the Z axis.

Cylindrical grinder (X and Z axes)

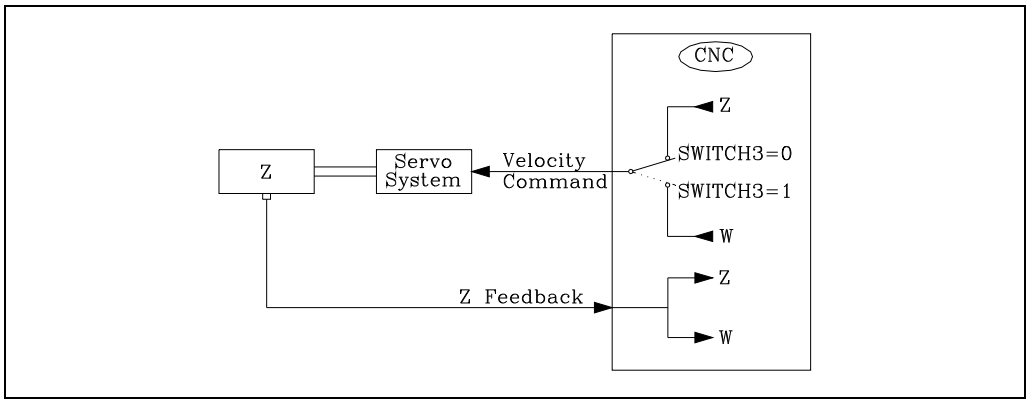
To make the reciprocating movement (back-and-forth table swing - Z axis) independent from the movement of the other axis (X), that movement should be controlled through the PLC execution channel.

When a cycle controls both axes or to move the Z axis manually (jog or handwheel), the Z axis must be controlled by the CNC.

Since an axis cannot be controlled through 2 execution channels, the CNC must be "cheated" by calling the axis with two different names.

Z	Main axis. Controlled by CNC
W	Secondary axis. Controlled by PLC

Although both axes may be displayed, only the Z axis (main) will displayed in this example.



Connect the Z axis feedback (main axis).

Since the two axes share the same feedback device, set the ungoverned axis as DRO so it does not trigger the following error alarm.

The velocity command is always output through the Z axis connector (main axis).

The mark for the secondary axis is SWITCH3 (M5205)

With SWITCH3=0 velocity command of the Z axis and with SWITCH3=1 that of the W axis.

PLC program

The M40 mark indicates that there is no external emergency (I1) and that the position loop of the axes are closed (NOT LOPEN).

I1 AND NOT LOPEN = M40

An external switch (I12) turns the reciprocating movement off, PLC execution channel, and to switch to the main execution channel (M41=1).

To switch from the PLC execution channel to the CNC channel, the PLC channel must be interrupted (PLCABORT) and one must make sure that the axis has stopped (INPOS3)

I12 AND (other conditions) = SET PLCABORT = SET M44

M44 AND INPOS3 = M41

With CNC channel selected (M41=1)

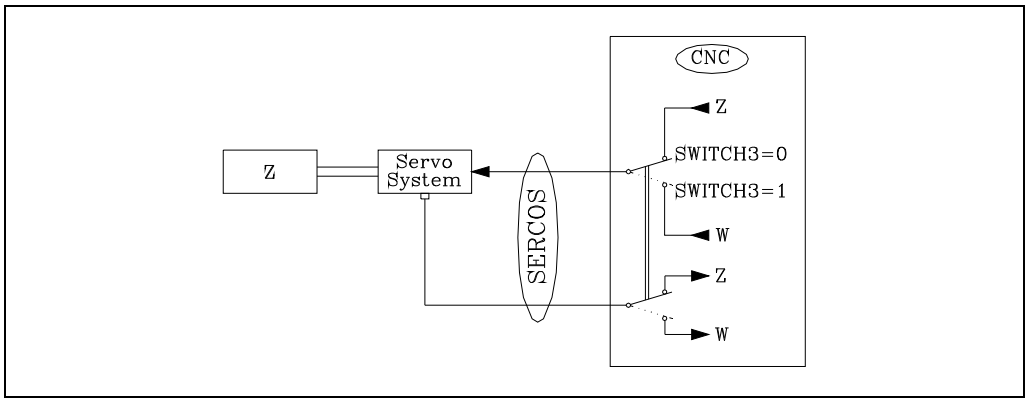
M40 AND M41	= DRO3	W axis as DRO
	= SERVO2ON	Z axis normal
	= RES SWITCH3	Z axis velocity command

With the PLC channel selected (M41=0)

M40 AND NOT M41	= DRO2	Z axis as DRO
	= SERVO3ON	W axis normal
	= SET SWITCH3	Z axis velocity command

Sercos

When using SERCOS communication with the servo drive, the axis being applied the velocity command and feedback data is selected by the mark of the secondary axis SWITCH3 (M5205).



a.m.p. SWINBACK (P66) of the secondary axis must be set to "0".

Z axis (main)	W axis (secondary)
	DFORMAT for W =0 (not displayed).
SWITCHAX for Z = 0	SWITCHAX for W = 3 (Z axis)
SWINBACK for Z axis = 0	SWINBACK for W = 0

7.13.1 "C" axis and spindle with a single feedback

At the lathe model, when using a single feedback for the spindle and "C" axis, set a.m.p. SWITCHAX (P65) and SWINBACK (P66) of the C axis.

SWITCHAX (P65)

It indicates with which main axis it is associated.

0 = with none	1 = with X	2 = with Y	3 = with Z
4 = with U	5 = with V	6 = with W	7 = with A
8 = with B	9 = with C	10 = with the spindle	

Set a.m.p. SWITCHAX=10 of the "C" axis

SWINBACK (P66)

It indicates the type of "C" axis feedback.

0	It uses the feedback of the main axis.
1	It has its own feedback device (external)
2	It uses the feedback of the main axis, but it has its own velocity command.
10	For an additive coupling.

When communication is via Sercos, proceed as follows:

- The a.m.p. DRIBUSID (P56) and the s.m.p. DRIBUSID (P44) must be set with the same value (same Sercos address).
- To govern the "C" axis, enable its SERVOON, SPENA and DRENA signals and activate the DRO signal of the spindle so it works as a DRO (not controlled); and vice versa.

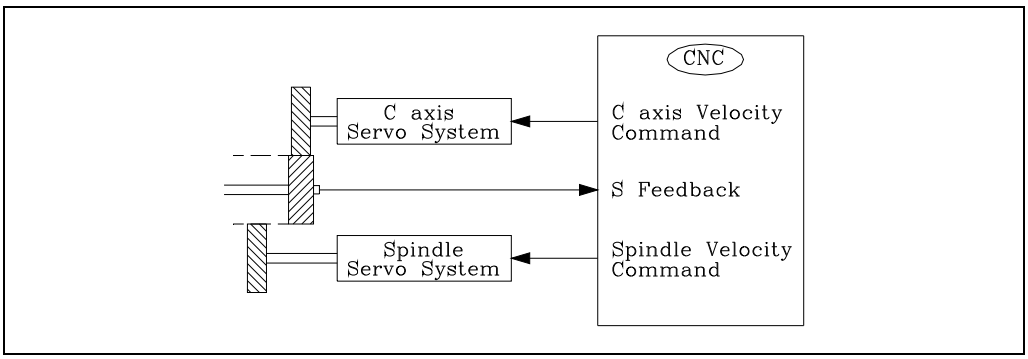
For digital servo system, the PLC mark CAXSEROK must be treated. See *"7.12 Digital servo (CAN or Sercos)"* on page 357.

The following examples show several connection possibilities.

In all of them, the velocity command is switched automatically ignoring the SWTCH2 mark.

2 motors and DRIBUSLE=0

Analog or Sercos connection with DRIBUSLE=0. Analog or Sercos connection with SERCOSLE=0. The feedback is taken to the CNC connector for the spindle and there are 2 motors (C axis and spindle).



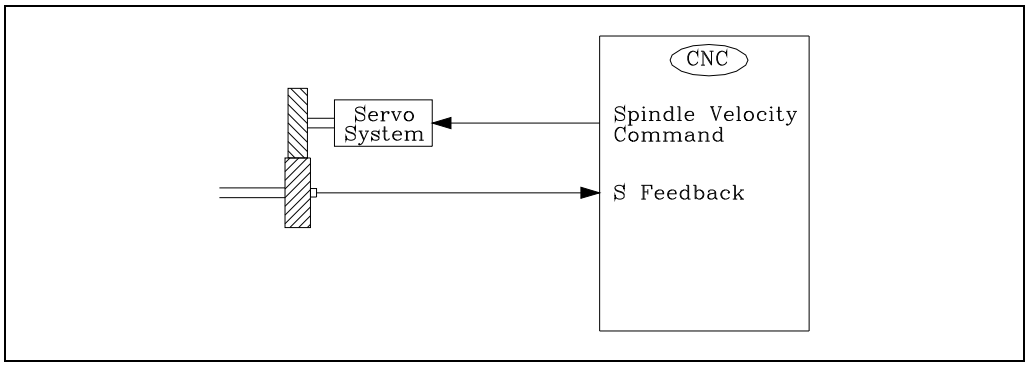
The feedback must be connected to the spindle (S) connector.

Spindle (S)		"C" axis	
DRIBUSID	(P44) = 1	DRIBUSID	(P56) = 5
DRIBUSLE	(P51) = 0	DRIBUSLE	(P63) = 0
		SWITCHAX	(P65) = 10
		SWINBACK	(P66) = 2

In this case, the drive works with SET 0 and GEAR RATIO 0.

1 motor and DRIBUSLE=0

Analog or Sercos connection with DRIBUSLE=0. The feedback is taken to the CNC connector for the spindle and there is 1 motor.



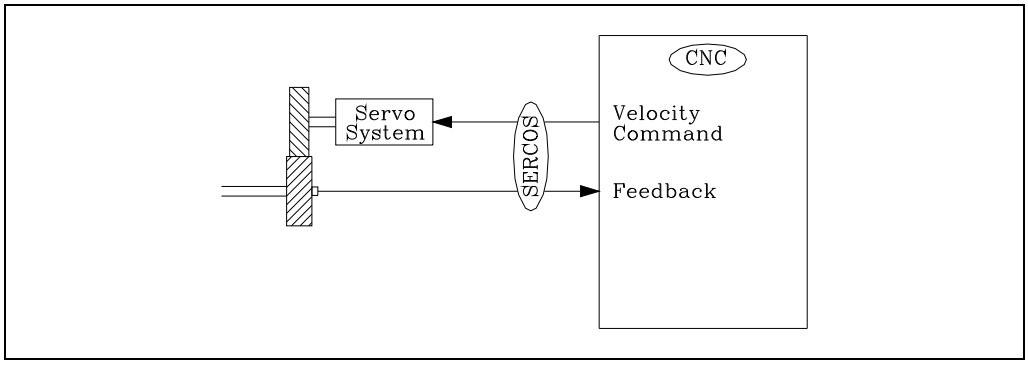
Spindle (S)		"C" axis	
DRIBUSLE	(P51) = 0	DRIBUSLE	(P63) = 0
		SWITCHAX	(P65) = 10
		SWINBACK	(P66) = 0

The feedback must be connected to the spindle (S) connector.

Use the velocity command for the spindle (S).

DRIBUSLE=1

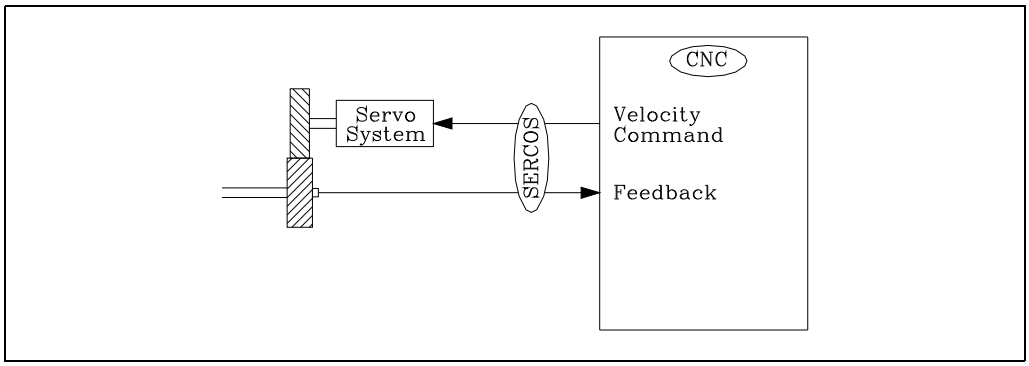
Sercos connection with DRIBUSLE=1. The axis feedback is handled via Sercos, first feedback (motor feedback) and the command for the drive is sent via Sercos.



Spindle (S)		"C" axis	
DRIBUSLE	(P51) = 1	DRIBUSLE	(P63) = 1
		SWITCHAX	(P65) = 10
		SWINBACK	(P66) = 0

DRIBUSLE=2

Sercos connection with DRIBUSLE=2. The axis feedback is handled via Sercos, second feedback (direct feedback) and the command for the drive is sent via Sercos.



Spindle (S)		"C" axis	
DRIBUSLE	(P51) = 2	DRIBUSLE	(P63) = 2
		SWITCHAX	(P65) = 10
		SWINBACK	(P66) = 0

Notes

When selecting the number of encoder pulses, bear in mind the maximum spindle rpm so they do not exceed the maximum feedback reading frequency.

When working with Sercos, the motor-drive system is supposed to know the spindle-C-axis position at all times, even after turning it off and back on. If it is an analog system, after turning the CNC on, it performs a home search automatically before moving the spindle or the C axis for the first time.

7.14 Additive coupling between axes

The additive coupling may be applied on any pair of axes, but only one pair of axes may be defined in an additive coupling. An additive coupling is not possible with the spindle.

A typical application of additive coupling is on milling machines where the Z axis has a second axis W coupled to it that moves by hand. This allows operating manually on the Z component through that axis.

In this case, the Z axis movements are defined by two axes. One motorized, whose parameters are defined in the Z axis parameters and the other one (dro) whose parameters are defined in the W axis.

The CNC screen shows, in the Z axis coordinates, the sum of both feedbacks Z+W. The coordinates of the W axis may be displayed either as a separate axis or may be set to be hidden (not displayed). Likewise, the graphics show the Z axis as the sum of Z+W.

Operation in jog mode

When displaying the coordinates of the Z axis, it shows the sum of both axes. The W axis may be moved manually without affecting the Z axis. When moving the Z axis, it moves up to the software limits of the Z axis taking into account the W axis position at the time.

Operation in the other modes (execution, etc.)

The CNC always monitors the software limits of both axes separately. If any of the limits is exceeded, it will issue the corresponding error on the Z or W axis. It also monitors the software limits of the Z axis when programming a movement either in execution or in JOG mode.

Home search

To carry out a full home search, do a home search on both axes separately.

- Z axis home search.

  When detecting the marker pulse, the Z axis display shows the "REVALUE" of the Z axis plus the position of the W axis. The PLCFZ variable is set to zero. The W axis is not involved in the home search.

- W axis home search.

  When detecting the marker pulse, the W axis display shows the "REVALUE" of the W axis. The Z axis display shows the Z axis position plus the "REVALUE" of the W axis plus W axis position.

Configuration

The two axes have separate machine parameters. The additive coupling is configured with the axis machine parameters "SWITCHAX" and "SWINBACK" of the DRO axis. The SWITCH* mark is not involved in the additive coupling.

Likewise, we recommend to set general machine parameter "DIPLCOF=2" so the motorized axis position display shows the PLCOF* component.

SWITCHAX (P65)

This DRO axis parameter indicates which is the main axis it is associated with.

SWINBACK (P66)

This DRO axis parameter indicates that an additive coupling is to be carried out on the axis indicated by parameter "SWITCHAX". For additive slaving (coupling), this parameter is set to ·10·.

PLC program

The W axis movements may be compensated on the Z axis via PLC. This is done by adding, in the PLC, the feedback increments of the W axis to the PLCOFZ variable. We recommend to manage the compensation through PLCOFZ with a periodic subroutine whose period is the same as the time set in LOOPTIME.

The following example assumes that the Z axis is motorized and that the W axis is a DRO axis. The W axis is coupled with the Z axis.

```
;
;* COMPENSATION OF THE W AXIS MOVEMENT WITH PLCOFZ *
;
;R73 - Operation mode
;R74 - previous POSW
;R75 - current POSW
;R76 - Resulting PLCOFZ
;R77 - previous PLCOFZ
;
; ### FIRST CYCLE ###
CY1
() = CNCRD(POSW,R74,M9)
END
;
; ### MAIN PROGRAM ###
PRG
REA
() = CNCRD(OPMODA,R73,M9)
NOT B0R73 = JMP L17
 ; The W axis is only compensated in execution
() = CNCRD(POSW,R75,M9)
= CNCRD(PLCOFZ,R77,M9)
= SBS R75 R74 R76
= SBS R77 R76 R76
= MOV R75 R74
= CNCWR(R76,PLCOFZ,M9)
= JMP L18
L17
() = CNCRD(POSW,R74,M9)
L18
END
```

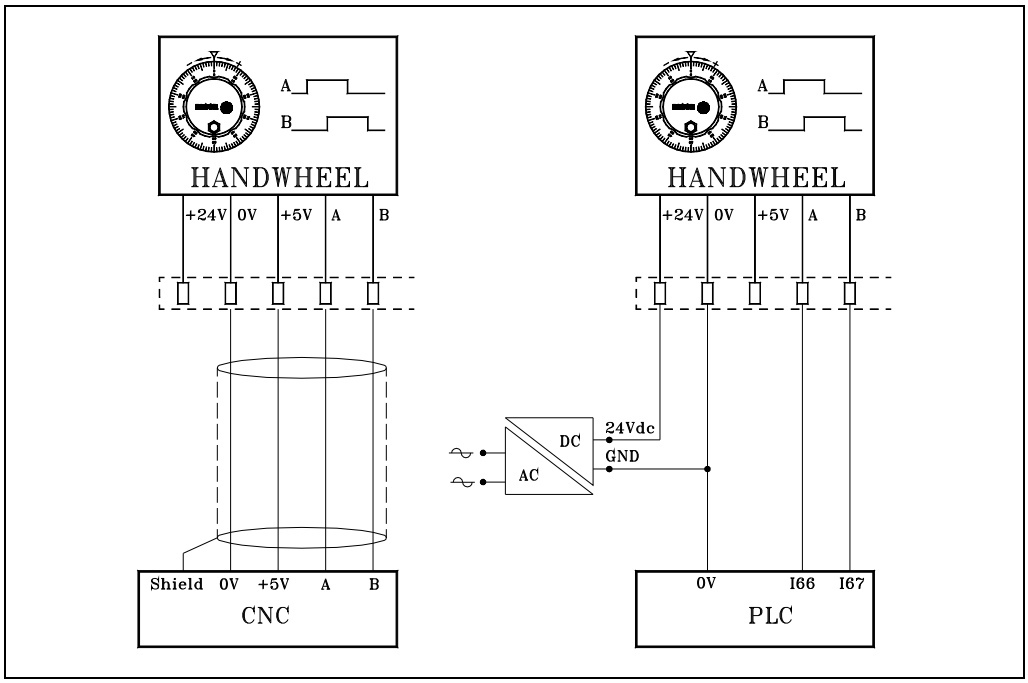
7.15 Fagor handwheels: HBA, HBE and LGB

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.

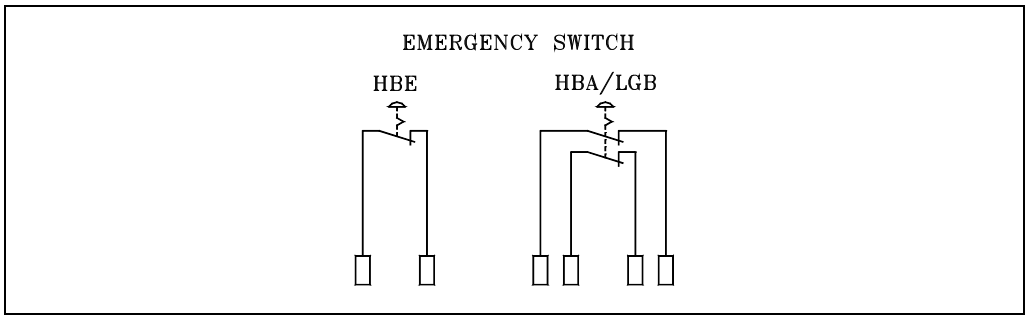
The signals from the 24V HBA and HBE models may also be taken to the PLC digital inputs located at the central unit (not remote).



In the example on the left, the handwheel signals are taken to the feedback input (connector). Set the corresponding g.m.p. AXIS , for example: AXIS4(P3)=11.

In the example on the right, the handwheel signals are taken to the digital PLC input. Set g.m.p. HANDWIN (P111) = 65 and HANDWHE1 (P112) = 11.

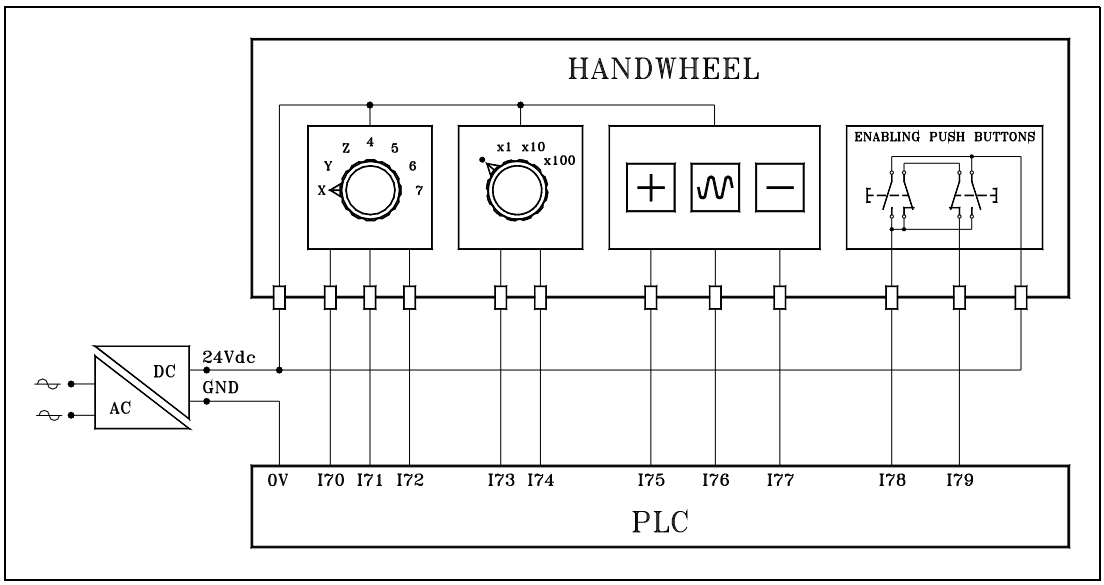
The emergency button must be used in the safety chain of the electrical cabinet.



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

							I70	I71	I72
NOT I70	AND	NOT I71	AND	NOT I72	= XSEL	XSEL	0	0	0
NOT I70	AND	NOT I71	AND	I72	= YSEL	YSEL	0	0	1
NOT I70	AND	I71	AND	I72	= ZSEL	ZSEL	0	1	1
NOT I70	AND	I71	AND	NOT I72	= 4SEL	4SEL	0	1	0
I70	AND	I71	AND	NOT I72	= 5SEL	5SEL	1	1	0
I70	AND	I71	AND	I72	= 6SEL	6SEL	1	1	1
I70	AND	NOT I71	AND	I72	= 7SEL	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	I74	c	b	a
			x1	0	1	0	0	1
I73 AND	I74	= RL1 R60 1 R60	x10	1	1	0	1	0
I73 AND	NOT I74	= RL1 R60 2 R60	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

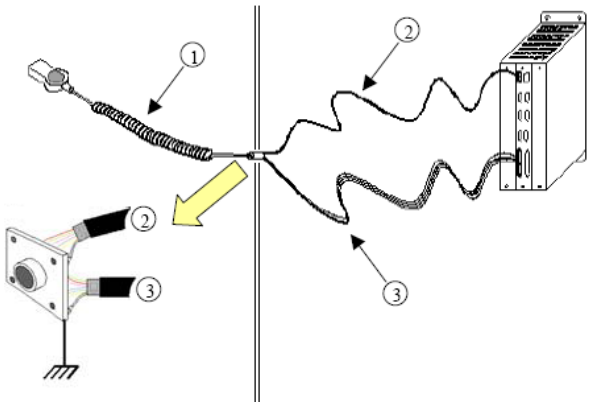
Safety. When releasing the "Enable Push Button", the STOP command is sent out to the CNC (100 ms pulse) to stop the possible movement active at the time (for example: 10 mm in incremental). Only if the JOG mode is selected and NOT MDI

DFD I79 = TG1 17 100

MANUAL AND NOT MDI AND T17 = NOT /STOP

END

In order to comply with the EN61000 - 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.



With the 8055 CNC, use a 7x1x0.14 PVC shielded cable for the feedback cable (2) of 5 V. The shield of both cables must be connected to the connector and to ground.

7.16 Machine safety related functions

7.16.1 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
```

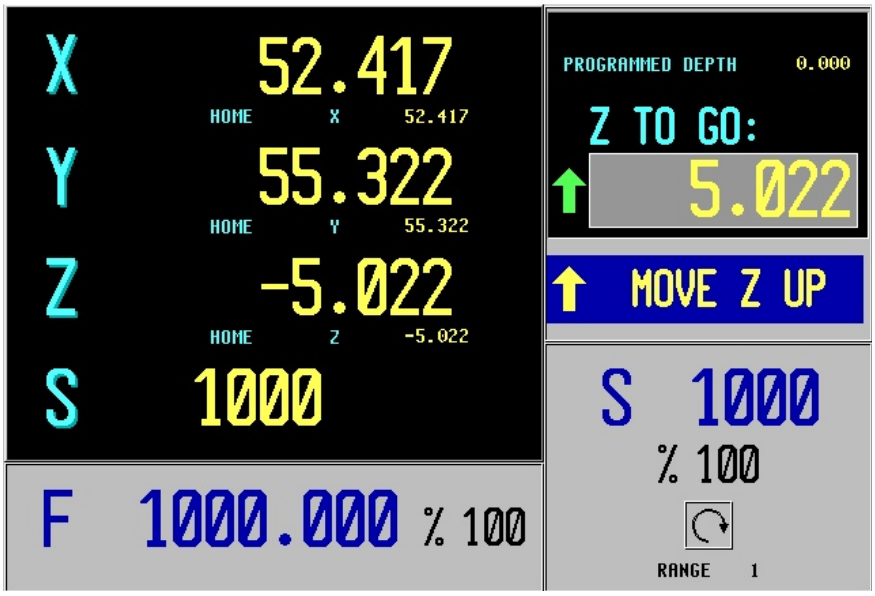
7.16.2 Cycle start disabled when hardware errors occur.

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.

7.17 Configuring a CNC as two and a half axes

A two-and-a-half-axis configuration is a milling machine where the X and Y axes are motorized and the Z axis is set as a DRO axis (display only). In this configuration, the Z axis is moved manually.

The CNC cycles and interface have been adapted for this type of configuration.



Cycle editing and execution

The cycles are edited, stored and simulated just like a 3-axis configuration.

The most significant different lays in the execution because the operator must move the Z axis by hand. The standard screen shows the operations to be carried out by the operator. In each case, it shows the status of the Z axis and the various actions to be executed by the operator.

- Move Z up (it shows an icon next to the final Z coordinate).

  The operator must move the axis up manually. When the Z axis is in position, the message will change.

- Move Z down (it shows an icon next to the final Z coordinate).

  The operator must move the axis down manually. When the Z axis is in position, the message will change.

- Press [CYCLE START].

  The operator must press [CYCLE START] to begin the automatic movement in the X-Y plane.

- Moving in X-Y.

  The machine is moving in X-Y. When a Z axis move is required, the machine will stop and it will request the operator's intervention.

- Tool inspection.

  It went into tool inspection.

Canned cycles

Not all the cycles can be applied to a two-and-a-half-axis configuration. The forbidden cycles may be hidden by machine parameters.

In some of the permitted cycles, some data has been eliminated to adapt the cycle to the two-and-a-half-axis configuration. The eliminated data is the one referring to the Z axis operations.

7.17.1 Machine parameter setting

The following machine parameters must be modified to configure the CNC as having two and a half axes:

CODISET (P147)

This parameter has 16 bits counted from right to left. Bit ·2· indicates whether the CNC is configured as two and a half axes (bit=1) or not (bit=0).

COCYF1 (P148) COCYF5 (P152) COCYZ (P155)

These parameters allow hiding operations or cycles that are not used showing only the desired ones. Each parameter is associated with an operation or cycle and each one of their bits refers to each available level.

Each parameter has 16 bits counted from right to left. By default, all the bits will be assigned the value of ·0· (option available). Assigning the value of ·1· hides the level of the corresponding cycle.

COCYZ (P155) 0000 0000 0100 0110

Bit	Meaning
6	It hides the threading cycle.
2	It hides the drilling 3 cycle.
1	It hides the drilling 2 cycle.

COCYF5 (P152) 0000 0000 0000 0010

Bit	Meaning
1	It hides the 3D profile pocket cycle.

COCYF1 (P148) 0000 0000 0000 0010

Bit	Meaning
1	It hides the PROBE 1 cycle.

The PROBE1 cycle can also be hidden by setting parameters "PRBXMIN (P40)" through "PRBZMAX (P45)" to zero.

7.17.2 PLC program

The PLC program must be modified as follows in order for the machine to work properly with a two-and-a-half-axis configuration:

- Stop the machine when the Z axis gets in position and the X-Y axes are going to start moving.
- Disabled the X-Y axes when the axis to be moved is the Z axis.
- Distinguish between the execution status of programs, cycles, jog mode and tool inspection in order to set the Z axis as a dro axis (display only) or not.
- The PLC mark "TOOLINSP"must always be active. Otherwise, when a STOP is generated with the PLC, it goes directly into tool inspection mode.

The PLC program must contain a routine for the two-and-a-half-axis configuration and for the 3-axis configuration. Switching from one to the other must be automatic depending on the value of parameter CODISET.

PLC programming example

```
;
;* * * * * * * * 2+1/2-AXIS APPLICATION * * * * * * * *
;
() = SET TOOLINSP = CNCRD(MPG147,R131,M1000)
;
NOT B2R131 = JMP L99
    If it is not a 2+1/2-AXIS configuration, jump to label L99.
() = CNCRD(OPMODA,R130,M1000)
;
NOT B0R130 AND NOT B4R130 AND NOT B7R130 = DRO3
    If the program is not in execution (B0R130), it is not interrupted (B4R130) and it is not in tool inspection
    mode (B7R130), then Z is a DRO axis.
;
(NOT INPOSX OR NOT INPOSY) AND DFD B3R130 = TG1 124 100
    If the X axis or the Y axis is not in position and we it is coming out of tool inspection (B3R130), then
    generate /STOP.
;
ENABLEZ AND INPOSX AND INPOSY = SET M200
    If wished to move the Z axis and the X-Y axes are in position, then the Z axis may be moved.
;
M200 AND B0R130 AND NOT B3R130 AND NOT B7R130 = SET INHIBITX = SET INHIBITY = MOV 100
R131 = CNCWR(R131,PLCFRO,M9)
    If the Z axis may be moved and the program is not in execution (B0R130), it is not repositioning
    (B3R130) and it is not in tool inspection (B7R130), then inhibit the X-Y axes, set the feedrate override
    of the PLC to 100% and disable the feedrate override.
;
M200 AND B0R130 AND NOT B4R130 = M412
    If the Z axis may be moved, there is a program in execution (B0R130) and it is not interrupted
    (B4R130), then the Z axis is moving.
;
M412 AND (ENABLEX OR ENABLEY) = TG1 123 100
    If the Z axis is moving and you wish to move the X or Y axis, then generate /STOP.
;
DFD INCYCLE = RES M200
    If the program execution is finished or a /STOP has been generated, then the Z axis movement has
    finished.
;
(START AND NOT M412) OR DFD B0R130 OR DFU B7R130 = RES INHIBITX = RES INHIBITY = MOV
0 R131 = CNCWR(R131,PLCFRO,M9)
    If [CYCLE START] is pressed and the Z axis does not move or the execution of the program ends
    (B0R130) or it goes into tool inspection (B7R130), then enable the X-Y axes, set the PLC feedrate
    override to 0% and enable the feedrate override.
;
L99
;
;* * * * Emergency, feedhold, stop, auxend  * * * *
;
NOT T123 AND NOT T124 = /STOP
I1 = /EMERGEN
    E-stop button.
/ALARM = O1
() = /FEEDHOL = /XFERINH
START AND NOT M412 = CYSTART
    [CYCLE START] is pressed and the Z axis does not move.
NOT T1 = AUXEND
;
```

7.18 Tool magazine

7.18.1 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 the 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.

7.18.2 Tool magazine management

The following must be borne in mind when doing a tool change:

- The tool change is not validated until the execution of the T function or M06 is completed correctly. If it is not completed correctly, the tool magazine table is not refreshed.
- If while executing a T or M06 function something unexpected happens, (CNC error, PLC error; e-stop button pressed, CNC reset, etc.) a mark (TMINEM) will be activated that will set 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 memorizes this error until it is canceled by a PLC mark (RESTMEM) or by the option [REMOVE ERROR] that appears in the error message.

  If this error is not canceled as mentioned earlier, it will stay memorized even if the CNC is turned off and back on repeatedly. Even if the magazine error is memorized, the machine can keep working.

  The CNC will only show this error message if the user requests a new tool while the error situation is unsolved (PLC mark TMINEM active). The magazine error only prevents a new tool change.

  In the conversational mode, it will show the error: "Tool magazine in error state".



  In ISO mode, in MANUAL (JOG), MDI or AUTO, it shows the error: "Tool magazine in emergency".

- During the state error, 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 be disabled.

  This management is only possible when a tool magazine has been defined.

Example of a PLC program to manage the emergencies at 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_POWER        ;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
;
```

7.19 Gear ratio management on axes and spindle

The gear ratios on axes and spindle, depending on whether they are analog, CAN or Sercos, are managed as follows:

Sercos

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 axis machine parameters PITCHB (P86)=0, INPREV (P87)=0 and OUTPREV (P88)=0, it will read these parameters from the drive. The equivalences are:

- a.m.p. PITCHB (P86) = NP123 (drive)
- a.m.p. INPREV (P87) = NP121 (drive)
- a.m.p. OUTPREV (P88) = NP122 (drive)

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.

When using SERCOS spindles, the velocity command may be output based on spindle parameters INPREV1..4 and OUTPREV1..4 (P72..79) as it is done with CAN spindles. To do that, some of spindle parameters INPREV1..4 or OUTPREV1..4 must be set to a value other than 0. In this case, parameters INPREV and OUTPREV are written at the drives.

Another option is to set spindle machine parameters MAXVOLT1..4 (P37..40). For the CNC to work taking MAXVOLT1..4 (P37..40) into account, all INPREV1..4 and OUTPTREV1..4 parameters must be set to 0. In this case, it will read the gear ratios of the drive system. The CNC applies the values of NP121 and NP122 of the first 4 gears, SP20 (Volts) and SP21 (rpm) of the drive and MAXGEAR1..4 and MAXVOLT1..4 of the CNC.

CAN

If axis machine parameters INPREV (P87)=0 and OUTPREV (P88)=0, they will be taken as if they were 1. There is no need to put anything in a.m.p. PITCH (P7), except in the following case:

If a.m.p. DRIBUSLE (P63) = 0 and a.m.p. INPREV (P87) and OUTPREV (P88) are 0, it will attend to a.m.p. PITCH (P7).

the way to set gear ratios in an axis is the following:

- PITCHB (P86) = leadscrew pitch.
- INPREV (P87) = input revolutions.
- OUTPREV (P88) = output revolutions.

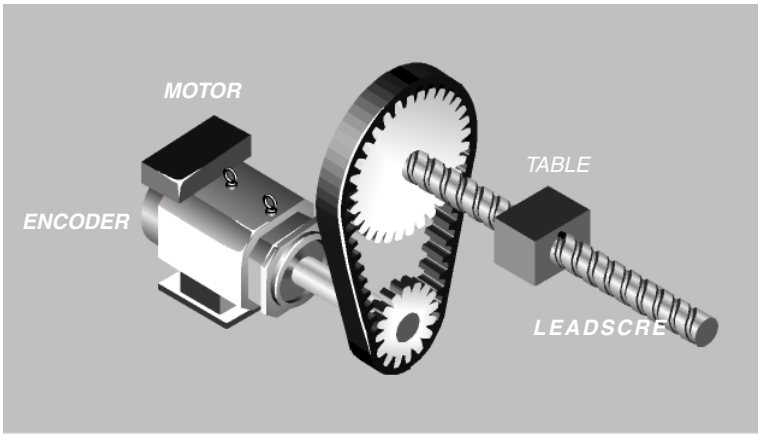
Analog

the way to set gear ratios in an axis is the following:

- 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 gears, 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 INPREV or OUTPREV values are indivisible, the I0 (home) signal will be generated from the home switch (DECEL).*
The feature to detect the right I0 among several possible ones by different gear ratios only works for spindles and Sercos rotary axes.

7.19.1 Axis example: Encoder at the motor



We have an axis with a maximum feedrate of 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.

Sercos / CAN axes

– a.m.p. DRIBUSLE (P63) = 1.

Parameters involved in the calculation of the 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 maximum motor speed for a feedrate of G00FEED:

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 the velocity command:

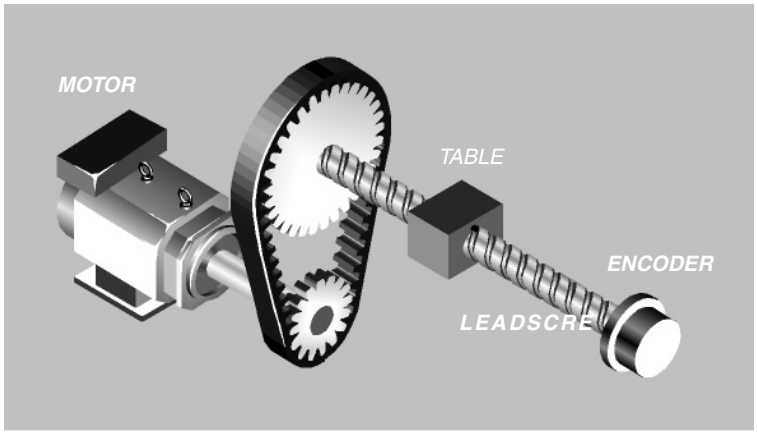
– a.m.p. G00FEED (P38) = Maximum feedrate of the axis = 20000.

Parameters involved in the calculation of the 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.

7.19.2 Axis example: external feedback device without a gear box

In this case, on linear axes the encoder is connected directly to the leadscrew and on rotary axes, it is connected directly to the center of rotation. If it is a rotary axis, the leadscrew pitch will be 360.



We have an axis with a maximum feedrate of 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.

Sercos axes

1. External feedback device connected to the drive (second feedback)

– a.m.p. DRIBUSLE (P63) = 2.

Parameters involved in the calculation of the 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 the position feedback:

– GP10 (drive) = feedback type of the external encoder = 2.

– NP131 (drive) = input revolutions of the external encoder = 1 (default value).

– NP132 (drive) = output revolutions of the external encoder = 1 (default value).

– NP133 (drive) = Leadscrew pitch = 20.

With rotary encoder:

•PP115 (drive) = bit 0 = 0 (direct external rotary feedback).

•NP117 (drive) = number of pulses per turn of the external encoder = 18000.

•NP165 (drive) = 1001 (see table of the drive manual).

•NP166 (drive) = 1000 (see table of the drive manual).

With linear encoder:

- •PP115 (drive) = bit 0 = 1 (direct external linear feedback).
- •NP117 (drive) = glass/tape graduation pitch of the linear encoder = 20.
- •NP118 (drive) = actual (real) counting pitch of the linear encoder = 4. If there is no multiplier circuit (EXE) incorporated into the feedback, the value is the same as NP117.
- •NP165 (drive) = 1001 (see table of the drive manual).
- •NP166 (drive) = 1000 (see table of the drive manual).

2. External feedback connected to the CNC

- A.m.p. DRIBUSLE (P63) = 0.

Parameters involved in the calculation of the 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 the position feedback.

With rotary 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 = 200.
- •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 feedrate of G00FEED:

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

CAN axes

1. External feedback connected to the CNC

– A.m.p. DRIBUSLE (P63) = 0.

Parameters involved in the calculation of the 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 the position feedback.

With rotary 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 = 200.

•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 feedrate of G00FEED:

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 the velocity command:

– a.m.p. G00FEED (P38) = Maximum feedrate of the axis = 20000.

Parameters involved in the calculation of the position feedback.

With rotary 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 = 200.
- 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:

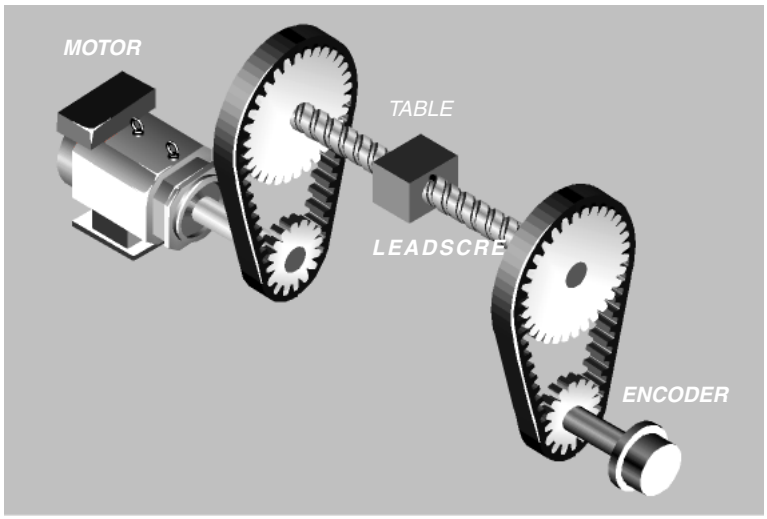
- a.m.p. PITCH (P7) = linear encoder pitch = 20.
- a.m.p. NPULSES (P8) = 0.
- a.m.p. SINMAGNI (P10) = Multiplying factor if the encoder is sinusoidal = 0.
- A.m.p. EXTMULT (P57) = Multiplying factor of the feedback = 20µ/4µ = 5.

Calculation of the motor speed with a command of MAXVOLT for a feedrate of G00FEED:

Motor speed = (G00FEED x INPREV) / (PITCHB x OUTPREV) = (20000 x 3) / (20 x 1) = 3000 rpm.

7.19.3 Axis example: external feedback device with gear box

In this case, on linear axes the encoder is connected to the leadscrew through a gear box and on rotary axes, it is connected to the center of rotation through a gear box.



We have an axis with a maximum feedrate of 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.

Sercos axes

1. External feedback device connected to the drive (second feedback)

- a.m.p. DRIBUSLE (P63) = 2.

Parameters involved in the calculation of the 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 the position feedback:

- GP10 (drive) = feedback type of the external encoder = 2.
- PP115 (drive) = bit 0 = 0 (direct external rotary feedback).
- NP117 (drive) = number of pulses per turn of the external encoder = 18000.
- NP131 (drive) = input revolutions of the external encoder = 2.
- NP132 (drive) = output revolutions of the external encoder = 3.
- NP133 (drive) = Leadscrew pitch = 20.
- NP165 (drive) = 1001 (see table of the drive manual).
- NP166 (drive) = 1000 (see table of the drive manual).

2. External feedback connected to the CNC

– A.m.p. DRIBUSLE (P63) = 0.

Parameters involved in the calculation of the 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 the position feedback.

– a.m.p. NPULSES (P8) = number of pulses per turn of the encoder
= 18000 / (3 / 2) = 12000. (only integer values are allowed).

– a.m.p. SINMAGNI (P10) = Multiplying factor if the encoder is sinusoidal = 200.

– a.m.p. EXTMULT (P57) = Multiplying factor of the feedback = 1.

– a.m.p. PITCHB (P86) = Leadscrew pitch = 20.

CAN axes

1. External feedback connected to the CNC

– A.m.p. DRIBUSLE (P63) = 0.

Parameters involved in the calculation of the 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 the position feedback.

– a.m.p. NPULSES (P8) = number of pulses per turn of the encoder
= 18000 / (3 / 2) = 12000. (only integer values are allowed).

– a.m.p. SINMAGNI (P10) = Multiplying factor if the encoder is sinusoidal = 200.

– 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 the velocity command:

– a.m.p. G00FEED (P38) = Maximum feedrate of the axis = 20000.

Parameters involved in the calculation of the 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 = 200.

– 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.

7.19.4 Spindle example: Encoder at the motor

Having a spindle with 4 gears. The maximum rpm and the gear ratio for each gear are the following:

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.

Sercos / CAN spindle

- s.m.p. DRIBUSLE (P51) = 1.

Parameters involved in the calculation of the 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. INPREV1 (P72) = Input revolutions of the first gear =4.
- s.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- s.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
- s.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- s.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- s.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- s.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- s.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.

Parameters involved in the calculation of the position feedback.

- s.m.p. INPREV1 (P72) = Input revolutions of the first gear =4.
- s.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- s.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
- s.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- s.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- s.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- s.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- s.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.

Analog spindle

Parameters involved in the calculation of the 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 the 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 = 200.
- s.m.p. INPREV1 (P72) = Input revolutions of the first gear =4.
- s.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- s.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
- s.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- s.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- s.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- s.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- s.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.

7.19.5 Spindle example: external encoder without gear box

Having a spindle with 4 gears. The maximum rpm and the gear ratio for each gear are the following:

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.

Sercos spindle

1. External encoder device connected to the drive (second feedback)

- s.m.p. DRIBUSLE (P51) = 2.

Parameters involved in the calculation of the 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. INPREV1 (P72) = Input revolutions of the first gear =4.
- s.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- s.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
- s.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- s.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- s.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- s.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- s.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.

Parameters involved in the calculation of the position feedback:

- GP10 (drive) = feedback type of the external encoder = 2.
- PP115 (drive) = bit 0 = 0 (direct external rotary feedback).
- NP117 (drive) = number of pulses per turn of the external encoder = 18000.
- NP131 (drive) = input revolutions of the external encoder = 1 (default value).
- NP132 (drive) = output revolutions of the external encoder = 1 (default value).
- NP133 (drive) = Leadscrew pitch = 360.
- NP165 (drive) = 1001 (see table of the drive manual).
- NP166 (drive) = 1000 (see table of the drive manual).

2. External encoder connected to the CNC

- s.m.p. DRIBUSLE (P51) = 0.

Parameters involved in the calculation of the 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. INPREV1 (P72) = Input revolutions of the first gear =4.
- s.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- s.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
- s.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- s.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- s.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- s.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- s.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.

Parameters involved in the calculation of the 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 = 200.

CAN spindle

1. External encoder connected to the CNC

- s.m.p. DRIBUSLE (P51) = 0.

Parameters involved in the calculation of the 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. INPREV1 (P72) = Input revolutions of the first gear =4.
- s.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- s.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
- s.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- s.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- s.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- s.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- s.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.

Parameters involved in the calculation of the 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 = 200.

Analog spindle

Parameters involved in the calculation of the 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. INPREV1 (P72) = Input revolutions of the first gear =1.
- s.m.p. INPREV2 (P74) = Input revolutions of the second gear =1.
- s.m.p. INPREV3 (P76) = Input revolutions of the third gear =1.
- s.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- s.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- s.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- s.m.p. OUTPREV3 (P77) = output revolutions of the third gear =1.
- s.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 the 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 = 200.

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.

7.19.6 Spindle example: external encoder with gear box

Having a spindle with 4 gears. The maximum rpm and the gear ratio for each gear are the following:

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.

Sercos spindle

1. External encoder device connected to the drive (second feedback)

- s.m.p. DRIBUSLE (P51) = 2.

Parameters involved in the calculation of the 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. INPREV1 (P72) = Input revolutions of the first gear =4.
- s.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- s.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
- s.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
- s.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
- s.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
- s.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- s.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.

Parameters involved in the calculation of the position feedback:

- GP10 (drive) = feedback type of the external encoder = 2.
- PP115 (drive) = bit 0 = 0 (direct external rotary feedback).
- NP117 (drive) = number of pulses per turn of the external encoder = 18000.
- NP131 (drive) = input revolutions of the external encoder = 2.
- NP132 (drive) = output revolutions of the external encoder = 3.
- NP133 (drive) = Leadscrew pitch = 360.
- NP165 (drive) = 1001 (see table of the drive manual).
- NP166 (drive) = 1000 (see table of the drive manual).

2. External encoder connected to the CNC

– s.m.p. DRIBUSLE (P51) = 0.

Parameters involved in the calculation of the 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. INPREV1 (P72) = Input revolutions of the first gear =4.
– s.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
– s.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
– s.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
– s.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
– s.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
– s.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
– s.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.

Parameters involved in the calculation of the position feedback.

– S.m.p. NPULSES (P16) = number of pulses per turn of the encoder
= 18000 / (3 / 2) = 12000. (only integer values are allowed).
– S.m.p. SINMAGNI (P65) = Multiplying factor if the encoder is sinusoidal = 200.

CAN spindle

1. External encoder connected to the CNC

– s.m.p. DRIBUSLE (P51) = 0.

Parameters involved in the calculation of the 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. INPREV1 (P72) = Input revolutions of the first gear =4.
– s.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
– s.m.p. INPREV3 (P76) = Input revolutions of the third gear =4.
– s.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =1.
– s.m.p. OUTPREV1 (P73) = output revolutions of the first gear =1.
– s.m.p. OUTPREV2 (P75) = output revolutions of the second gear =1.
– s.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
– s.m.p. OUTPREV4 (P79) = output revolutions of the fourth gear =1.

Parameters involved in the calculation of the position feedback.

– S.m.p. NPULSES (P13) = number of pulses per turn of the encoder
= 18000 / (3 / 2) = 12000. (only integer values are allowed).
– S.m.p. SINMAGNI (P65) = Multiplying factor if the encoder is sinusoidal = 200.

Analog spindle

Parameters involved in the calculation of the 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 the 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 = 200.
- s.m.p. INPREV1 (P72) = Input revolutions of the first gear =2.
- s.m.p. INPREV2 (P74) = Input revolutions of the second gear =2.
- s.m.p. INPREV3 (P76) = Input revolutions of the third gear =2.
- s.m.p. INPREV4 (P78) = Input revolutions of the fourth gear =2.
- s.m.p. OUTPREV1 (P73) = output revolutions of the first gear =3.
- s.m.p. OUTPREV2 (P75) = output revolutions of the second gear =3.
- s.m.p. OUTPREV3 (P77) = output revolutions of the third gear =3.
- s.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.

7.20 Feedback combination for SERCOS axes with external feedback to the CNC

On large machines with a lot of backlash that use external feedback to achieve greater accuracy, some instability may occur. This kind of machines run smoothly with internal feedback, but it may lose precision; with external feedback, however, the accuracy improves but the machine movements may be jerky. Combining both feedbacks both precision and smoothness may be achieved.

The feedback combination is handled with axis machine parameter FBMIXTIM (P102) and is valid for linear and rotary axes.

The CNC uses the feedback combination to calculate the speed. The CNC uses the external feedback (direct feedback) to calculate the compensations, the circularity (roundness) test, etc.

The feedback combination can only be used when the external feedback goes to the CNC. If the external feedback goes to the drive, the feedback combination is disabled.

Cases where feedback combination is not supported:

- Spindles.
- CAN axes.



In order for the feedback combination to work properly, the counting direction of the internal and external feedbacks must be the same. If the counting direction of both feedbacks is not the same, it must be changed using a.m.p. AXISCHG (P13). Otherwise, the feedback combination will not work.

The feedback combination is compatible with the monitoring of the difference between the first and second feedbacks. Even if monitoring is not active, if the accumulated feedback difference between the first (internal) feedback and the second (external) feedback is too big, the CNC issues an error message and opens the loop.

Use variables MIXPO(X..C) and FLWAC(C..C) to know the axis position with combined feedback and the real error considering the second feedback.

7.21 Open communication

Open communication lets the CNC communicate with any device through the RS232 serial line without having to use any particular protocol.

Open communication uses the machine parameters, variables and PLC instructions described next.

Machine parameters for serial line 2

Using open communication requires setting the following parameters for serial line 2:

- BAUDRATE (P0): Communication speed.
- NBITSCHR (P1): Number of data bits per transmitted character.
- PARITY (P2): Type of parity check used.
- STOPBITS (P3): Number of stop bits at the end of each transmitted word.
- PROTOCOL (P4): Type of protocol to be used.
- RCVMAXCAR (P11): Maximum number of characters that may be received.
- RCVENDTI (P12): Indicates the maximum reception time.

Open communication is enabled with parameter PROTOCOL (P4). To enable it, set parameter PROTOCOL (P4) = 3.

CNC/PLC/DNC variables

There are 2 communication buffers that may be set indistinctively as data input or data output.

These buffers may be accessed with the instructions CNCRD/CNCWR indicating the buffer number and the position. Each buffer may have up to 255 Bytes and it may be accessed in Byte, Word or Long mode.

- Access in BYTE mode will be done with COMBUFB.
- Access in WORD mode will be done with COMBUFW.
- Access in LONG mode will be done with COMBUFL.

The accesses to the buffers are defined as: COMBUFB/W/L nBuffer posBuffer.

- nBuffer indicates the buffer number (1 or 2).
- posBuffer gives access de byte indicated inside the buffer.

The first 2 bytes of the buffer indicate the number of bytes to be sent in transmission When receiving, the first 2 bytes will indicate the number of bytes received.

Example:

() = CNCRD(COMBUFB 1 R3,R4,M1) = CNCWR(R4, COMBUFB 1 8,M1)

It reads from buffer 1 the position BYTE of index R3 leaving the result in R4 and writes it in position 8 of buffer 1.

In direct writing mode, the buffer can also be accessed in Byte/Word/Long mode directly without having to go through register.

Example:

() = CNCWR($28, COMBUFB 1 4,M1)

Writes the hexadecimal character 28 in the position of the BYTE 4 of buffer 1.

Using open communication requires considering the following variables:

RCVSTAT: Variable that indicates reception status.

RCVCTRL: Variable used to control reception.

PLC instruction.

PLC instructions to control communication via RS232. These READ and WRITE instructions may be executed from a periodic execution module (PE).

= WRITE nBuffer.

= READ nBuffer.

PLC MARKS

There are the following PLC marks:

READEND (M5507): CNC output that indicates that the message received is in the buffer.

READRDY (M5571): CNC output that indicates whether the READ commands may be received or not.

WRITERDY (M5572): CNC output that indicates whether the WRITE commands may be sent or not.

7.21.1 Reception setting

Reception of a message begins and ends as follows:

Beginning of the reception of a message

As soon as the READ character is executed, it puts the characters in the message as they come in. The variable RCVSTARTTI is set to 0, this causes the RCV instruction to receive characters as soon as it is executed.

With this mode, parameter RCVENDTI (P12) of serial line 2 may be used as message reception timeout.

End of the reception of a message

The READ instruction has several modes to end the reading of a message :

1. Message time :

   The message ends after the time indicated in parameter RCVENDTI (P12) of serial line 2 has elapsed since the beginning of the message. It starts counting as soon as the conditions to receive the first character of the message are met.

   RCVCTRL: bit2=1

   RCVENDTI (P12) = time

2. Maximum number of characters:

   Indicates the maximum number of characters to be received using parameter RCVMAXCAR (P11) of serial line 2. When this number is reached, the message is considered completed. It is used for protocols where the number of bytes received is always the same. It may be used in combination with other systems.

3. Parity errors:

   If the hardware detects a parity error in the characters, the messge ends.

4. Via PLC program :

   The message ends if bit 7 of variable RCVCTRL is set to 1.

7.21.2 Incompatibility with WINDNC via RS232

Having set an open communication via RS232 serial line, that line cannot be used to communicate with the Windnc. To use Windnc at the same time, it must be used via the Ethernet line.

7.21.3 Trace of characters sent and received

To help debug the open communication at a CNC, it is possible to do a trace of all characters that are sent and received through the RS232 serial line.

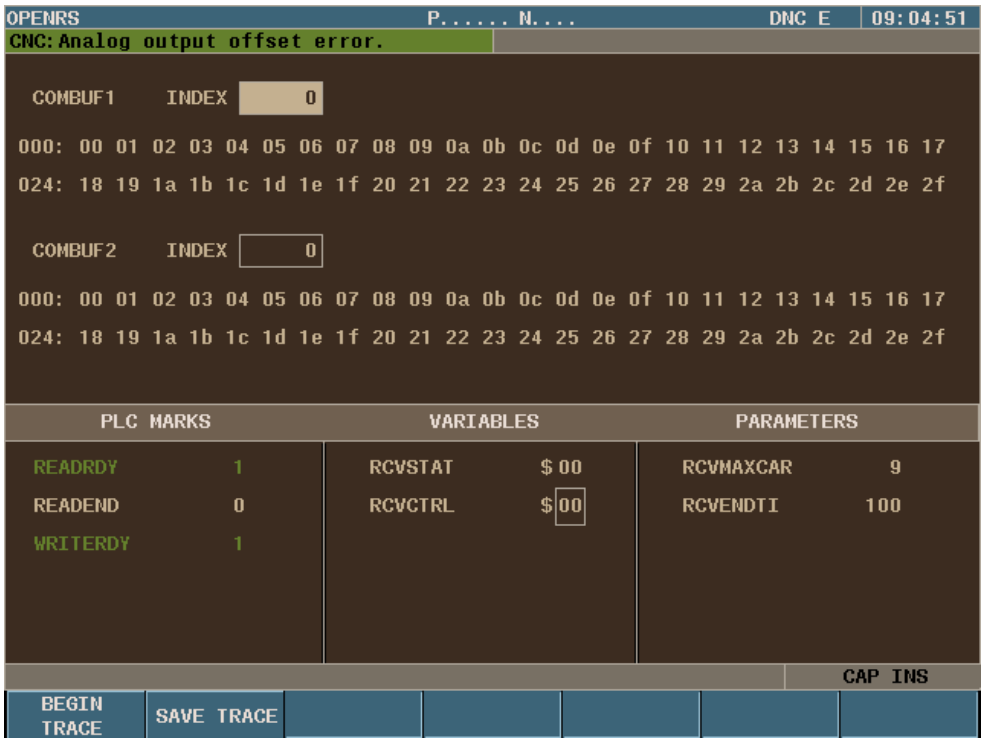
The trace will be triggered with two PLC marks:

- RSTRAEN: PLC mark that enables the trace. From this instant on, it will capture the traces all the characters sent/received through the serial line.
- RSTRATRG: PLC mark that triggers the trace. In order to pay attention to this signal, the trace must've been enabled with the RSTRAEN signal. It captures 15 lines sent/received before and after triggering this trace.

  Once the trace has been triggered, it won't be triggered until the CNC is powered up again, until it is saved into the file and/or until another ENABLE is done again.

The trace data may be displayed on the STATUS-DNC screen that offers the [OPENRS] softkey. Pressing this softkey gives access to the open communication status screen.

As shown on the screen below, besides displaying the two communication buffers (COMBUF1 and COMBUF2), there are two softkeys to activate and manage traces without having to use PLC instructions.



PLC instructions are used to begin a trace, the [SAVE TRACE] softkey must be pressed to save the trace into the KeyCF.

7.21.4 Examples of open communication

Here are some examples of open communication:

Send 1 message

Pressing the F key sends the ?"HELLO" message through the serial line.

```
DFU B0R560= CNCWR($48,COMBUFB 1 2,M1); 'H'
          = CNCWR($45,COMBUFB 1 3,M1); 'E'
          = CNCWR($4C,COMBUFB 1,4,M1); 'L'
          = CNCWR($4C,COMBUFB 1 5,M1); 'L'
          = CNCWR($4F,COMBUFB 1 6.M1) ; 'O'
          = CNCWR(5,COMBUFW 1 0,M1)   ; Write the length of the string to sen in the
                                        first word
          = WRITE 1                   ; Send through the serial line.
```

Send 1 message and wait for the response

Pressing the F key sends 2 integers with the numbers $10000000 and $20000000 and waits for the answer that must be the same data that was sent. It does not have a wait timeout.

Set the following serial line parameters as follows:

RCVMAXCAR (P11) = 8

RCVENDTI (P12) = 0

```
DFU B0R560   = CNCWR($10000000,COMBUFL 1 2,M1)
             = CNCWR($20000000,COMBUFL 1 6,M1)
             = CNCWR(8,COMBUFW 1 0,M1)    ;2 long are 8 bytes of message length
             = WRITE 1
             = READ 2

; Read the data from the buffer and leave it in registers at the end of reception.
DFU READEND= CNCRD(COMBUFW 2 0,R100,M1)
           = CNCRD(COMBUFL 2 2,R101,M1)
           = CNCRD(COMBUFL 2 6,R102,M1)

; Verify (compare) that the data received has the same value as the ones sent.
DFU READEND AND CPS R100 EQ 8
AND CPS R101 EQ $10000000
AND CPS R102 EQ $20000000 = NOT ERR7
```

Send 1 message and wait for the response with timeout

Pressing the F key sends 2 integers with the numbers $10000000 and $20000000 and waits for the answer that must be the same data that was sent.

Wait a maximum of 1 second since it sends the message until it receives the answer.

Set the following serial line parameters as follows:

RCVMAXCAR (P11) = 8

RCVENDTI (P12) = 1000

```
DFU B0R560   = CNCWR($10000000,COMBUFL 1 2,M1)
             = CNCWR($20000000,COMBUFL 1 6,M1)
             = CNCWR(8,COMBUFW 1 0,M1)    ;2 long are 8 bytes of message length
             = WRITE 1
             = READ 2

; Read the data from the buffer and leave it in registers at the end of reception.
DFU READEND= NOT JMP L1000
( )          = CNCRD(COMBUFW 2 0,R100,M1)
             = CNCRD(COMBUFL 2 2,R101,M1)
             = CNCRD(COMBUFL 2 6,R102,M1)
             = CNCRD(RCVSTAT,R110,M1)      ; Reception status

; Verify (compare) that the data received has the same value as the ones sent.
CPS R100 EQ 8
AND CPS R101 EQ $10000000
AND CPS R102 EQ $20000000 = NOT SET ERR7

B0R110 = ERR100                            ; Parity error
B2R110 = ERR101                            ; Timeout error
B6R110 = ERR102                            ; Reception error
L1000
```

Continuous communication between 2 CNC's where one is the master and the other the slave

Two CNC's continuously exchange 64 PLC marks when pressing the F key at the master.

Master CNC programming:

Set the following serial line parameters as follows:

RCVMAXCAR (P11) = 8

RCVENDTI (P12) = 500

It must receive the answer in 500 ms.

```
PRG
IMA
; Beginning of the process when pressing the F key
DFU B0R560  = SET M1000

; Stage 0
; Send message
M1000 AND WRITERDY  = CAL L1000
                    = WRITE 2
                    = RES M1000 = SET M1001     ; Go to stage 1

; Stage 1
; Wait for end of transmission
M1001 AND WRITERDY = RES M1001 = SET M1002      ; Go to stage 2

; Stage 2
; Command to read 1 message
M1002 AND READRDY = READ 1
                  = RES M1003 = SET M1000       ; Go to stage 3

; Stage 3
; Wait for end of message reception
M1003 AND READEND   = CAL L1001
                    = RES M1003 = SET M1000     ; Go to stage 0

REA
END                                             ; end of PRG

; Copy marks M3100..M3163 into the buffer to send them
L1000
()          = MOV M3100 R300
            = MOV M3132 R301
            = CNCWR(8,COMBUFW 2 0,M1)
            = CNCWR(R300,COMBUFL 2 2,M1)
            = CNCWR(R301,COMBUFL 2 6,M1)
 END

; Copy the received data into marks M3000.. M3063
L1001
()          = CNCRD(COMBUFL 1 2,R300,M1)
            = CNCRD(COMBUFL 1 6,R301,M1)
            = MOV R300 M3000
            = MOV R301 M3032
            = CNCRD(RCVSTAT,R310,M1)            ; Reception status
B6R310 = SET ERR102                             ; Reception error
END
```

Slave CNC programming:

Set the following serial line parameters as follows:

RCVMAXCAR (P11) = 8

RCVENDTI (P12) = 0

```
CY1
() = SET M1000
END

PRG
IMA

; Stage 0
; Command to read 1 message
M1000 AND READRDY = READ 1
                  = RES M1000 = SET M1001      ; Go to stage 1

; Stage 1
; Wait for end of message reception
M1001 AND READEND  = CAL L1000
                   = RES M1001 = SET M1002     ; Go to stage 2

; Stage 2
; Send message
M1002 AND WRITERDY = CAL L1001
                   = WRITE 2
                   = RES M1002 = SET M1003     ; Go to stage 3

; Stage 3
; Wait for end of transmission
M1003 AND WRITERDY = RES M1003 = SET M1000     ; Go to stage 0

REA
END                                            ; end of PRG

; Copy the received data into marks M3000.. M3063
L1000
()           = CNCRD(COMBUFL 1 2,R300,M1)
             = CNCRD(COMBUFL 1 6,R301,M1)
             = MOV R300 M3000
             = MOV R301 M3032
             = CNCRD(RCVSTAT,R310,M1)          ; Reception status
B6R310 = SET ERR102                            ; Reception error
END

; Copy marks M3100..M3163 into the buffer to send them
L1001
()           = MOV M3100 R300
             = MOV M3132 R301
             = CNCWR(8,COMBUFW 2 0,M1)
             = CNCWR(R300,COMBUFL 2 2,M1)
             = CNCWR(R301,COMBUFL 2 6,M1)
 END
```

7.22 Auto-adjustment of axis machine parameter DERGAIN

The auto-adjustment of DERGAIN allows setting the axis machine parameters DERGAIN of the three sets so the following error of the axis is close to 0.

- First set of gains and accelerations a.m.p. DERGAIN (P24).
- Second set of gains and accelerations: a.m.p. DERGAIN2 (P61).
- Third set of gains and accelerations: a.m.p. DERGAINT (P94).

The CNC makes this adjustments automatically. For that, the CNC creates a part program that is executed by pressing the "START" key. The movement limits of the program are taken from the current position (JOG).

When doing the DERGAIN auto-adjustment, it saves a .log file with the auto-adjustment process in a part program.

While executing the program, it will be possible to access the oscilloscope by keying in 71 to monitor the adjustment as it is carried out. One must check that there is no position overshooting.

The value of the OPMODE variable when going into the DERGAIN auto-adjustment will be 120.

Going into the DERGAIN auto-adjustment

To auto-adjust the DERGAIN, go into [DIAGNOSIS / ADJUSTEMENTS / AUTOADJUSTMENT] and follow these steps:

1. Select axis and parameter set:

   First select the axis whose DERGAIN is to be adjusted and the set of gain parameters. Then, confirm with the START softkey.

2. Generating the adjustment program:

   Once the axis and the parameter set have been selected, the CNC generates the program that will be executed to auto-adjust the DERGAIN. Configure the oscilloscope and initiate the trace for the selected axis. Then, it remains waiting for the axis to position correctly and for confirmation.

3. Program execution:

   Once the position has been confirmed, the CNC waits for the START key to be pressed. Pressing START runs the auto-adjustment program and the screen shows the execution status warnings.

   For safety reasons, the first movement is made slowly giving the user enough time to cancel the auto-adjustment by pressing STOP if necessary and restarted by pressing RESET.

   The value of the DERGAIN parameter for each axis and set is continuously updated on the auto-adjustment screen.

   When the execution is over, it displays the confirmation request for the adjusted value. Pressing RESET returns to the initial state.



The auto-adjustment of DERGAIN can only be done when a.m.p.. ACFGAIN (P46) is YES or ADVANCED.
While executing the auto-adjustment, the CNC ignores the position of the feedrate override switch.
If the value of a.m.p. ACFGAIN (P46) is changed, DERGAIN will have to be readjusted.
The best results of the auto-adjustment are obtained by setting SERCOS drive parameter SERCOS SP51=2.

Feature limitations

The area for the back-and-forth axis movement must be long enough to reach G0.

DERGAIN can only be auto-adjusted on linear and rotary axes. Auto-adjustment is not possible on switched axes, gantry, tandem axes, dead axes or spindles.

The DERGAIN auto-adjustment requires properly setting parameter FFGAIN of the corresponding set. If parameter FFGAIN is not properly set, it will issue the relevant error message.

- First set of gains and accelerations a.m.p. FFGAIN (P25).
- Second set of gains and accelerations: a.m.p. FFGAIN2 (P62).
- Third set of gains and accelerations: a.m.p. FFGAINT (P95).

If the CNC has the OEM password, going into auto-adjustment will require that password.

7.

CONCEPTS

Auto-adjustment of axis machine parameter DERGAIN

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

INTRODUCTION TO THE PLC

8

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.

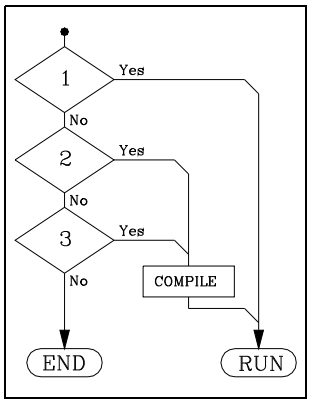
The PLC program (PLC_PRG) may be edited at the front panel or copied from the hard disk (KeyCF) or from a peripheral device or PC.

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.

8.1 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 3999 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 512 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.

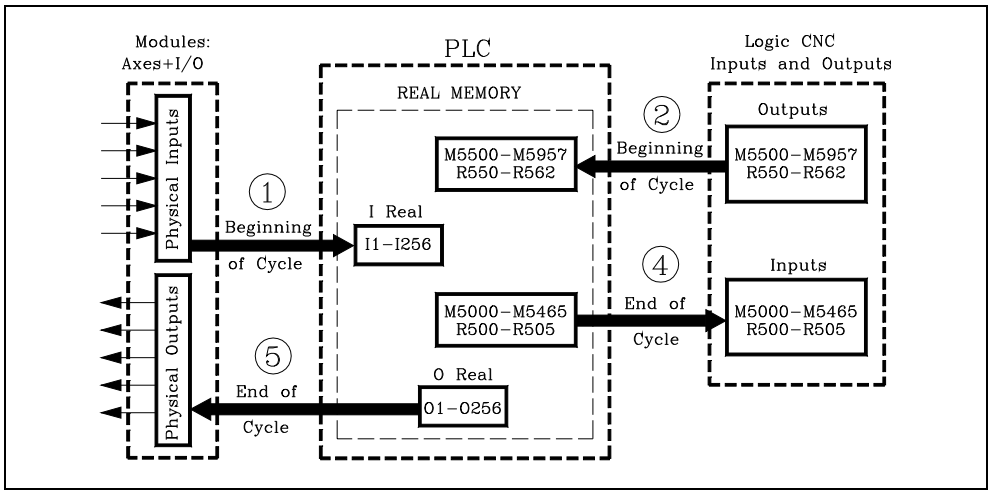
8.2 PLC program execution

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 *"8.4 Modular structure of the program"* on page 417.
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

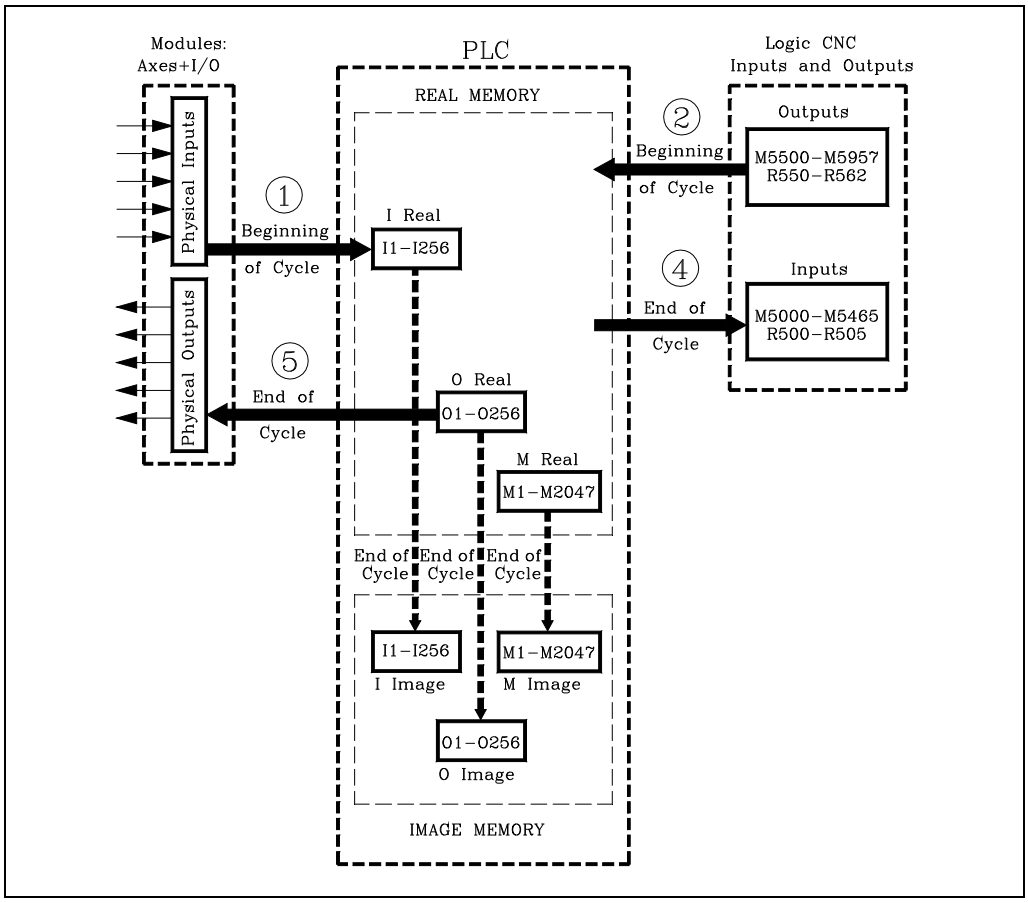
The value of M100 depends on the previous instruction.

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.

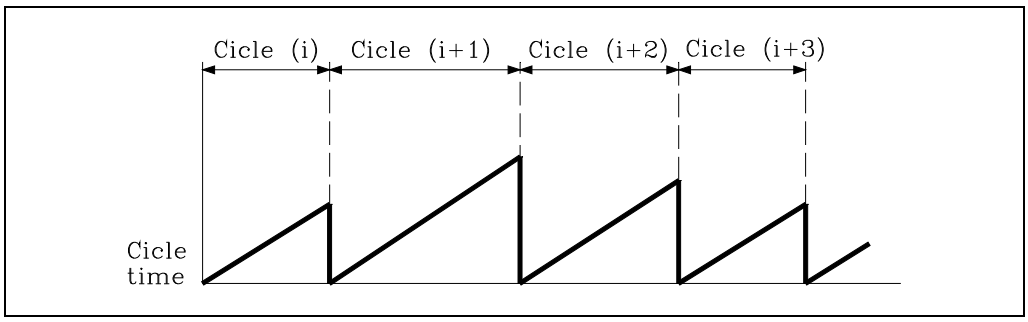
In the 2nd cycle (scan), the image value of M1 is "1" and the real value of M2 is set to "1"; but the image value of M2 will not be set to "1" until the end of the cycle.

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.

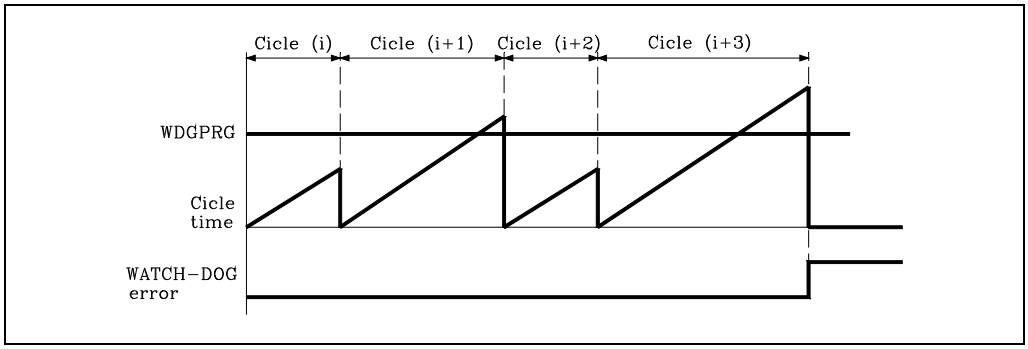
In the 4th cycle (scan), the image value of M3 is "1" and the real value of O5 is set to "1".

8.3 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.

8.4 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.

8.4.1 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.

8.4.2 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.

8.4.3 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:

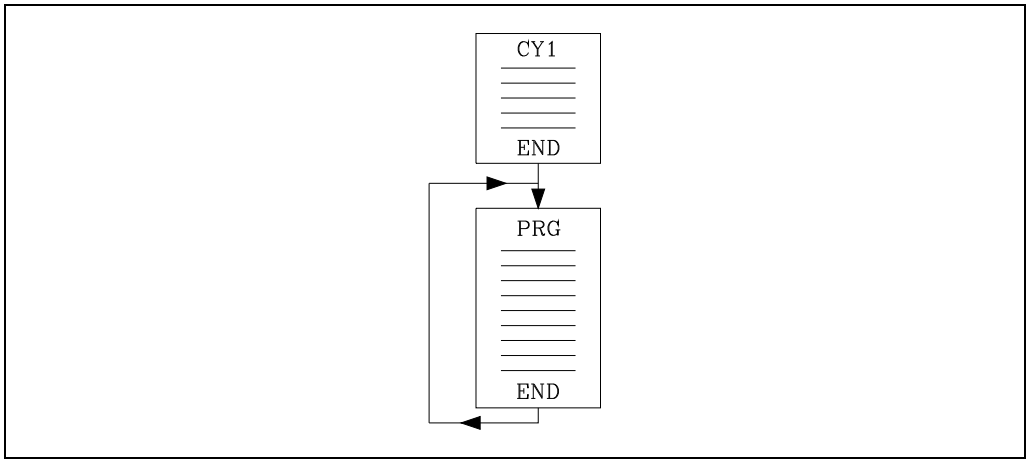
PE 10	Defines the beginning of the periodic module PE which will be executed every 10 milliseconds.

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.

8.4.4 Priority of execution of the PLC modules

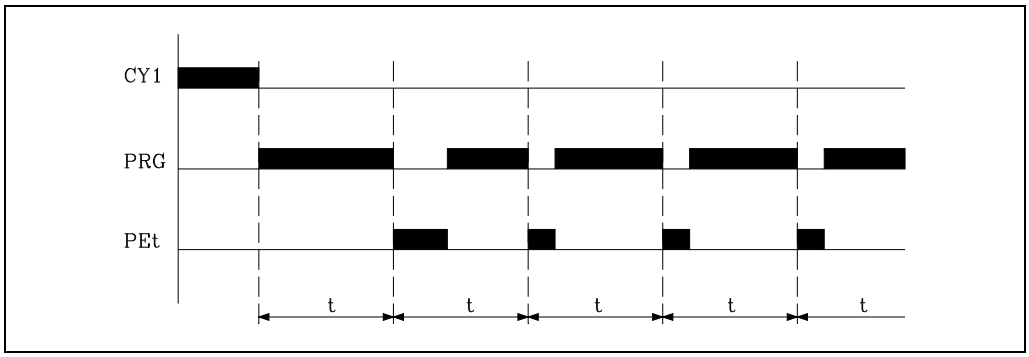
Every time the PLC program is started (command RUN) the first module to be executed is the first cycle module (CY1). Once execution has been completed, it will continue with the main module (PRG).

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.



8.

INTRODUCTION TO THE PLC

Modular structure of the program

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

PLC RESOURCES

9

9.1 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.

Remote physical inputs are the ones corresponding to the remote modules

9.2 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.

Remote physical outputs are the ones corresponding to the remote modules.

Output O1 coincides with the emergency output of the CNC (connector); thus, it must be kept high (logic level 1).

9.3 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 and M2049 - M3999
Arithmetic flag marks	M2003
Clock marks	M2009 - M2024
Fixed status marks	M2046 and M2047
Marks associated with messages	M4000 - M4254
Marks associated with errors	M4500 - M4627
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.

By activating each mark M4700-M4955, it is possible to activate at the CNC user pages (screens) 0-255. 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 *"12 Logic CNC inputs and outputs"*.

9.4 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

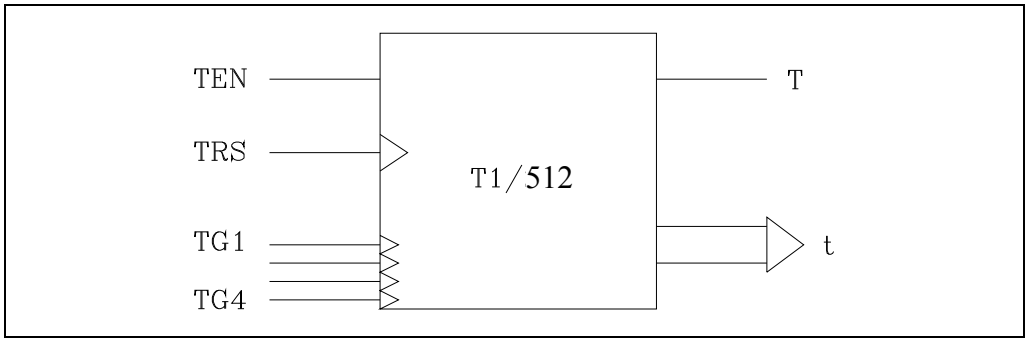
9.5 Timers

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 512 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.

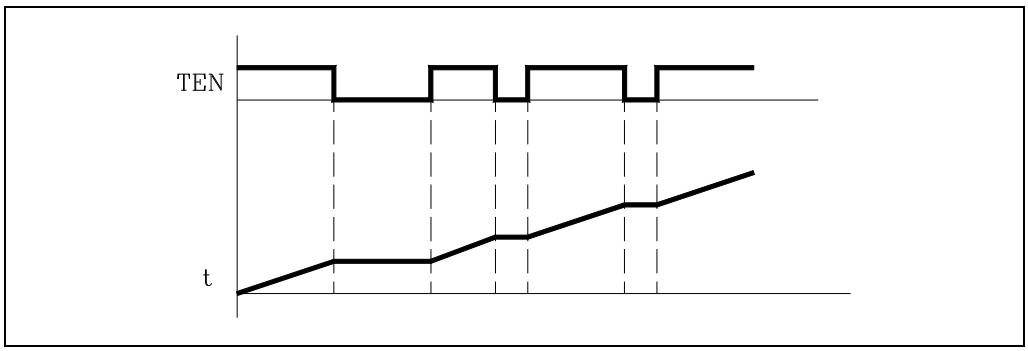


Enable input (TEN)

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:

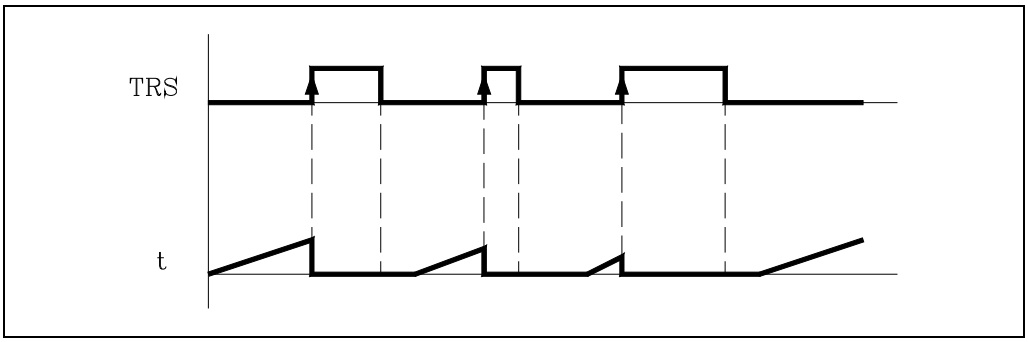
I2 = TEN 10 Input I2 controls the Enable input of timer T10.

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

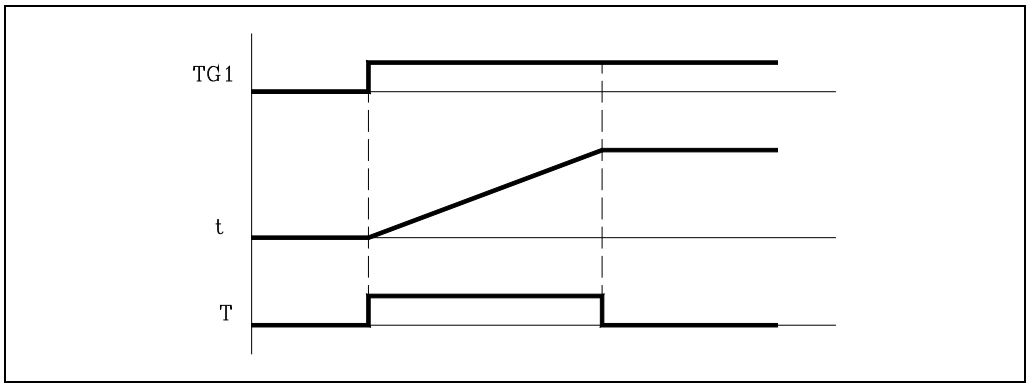
In arithmetic and comparison functions T123 makes reference to the time elapsed in the timer from the moment it was activated.

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.

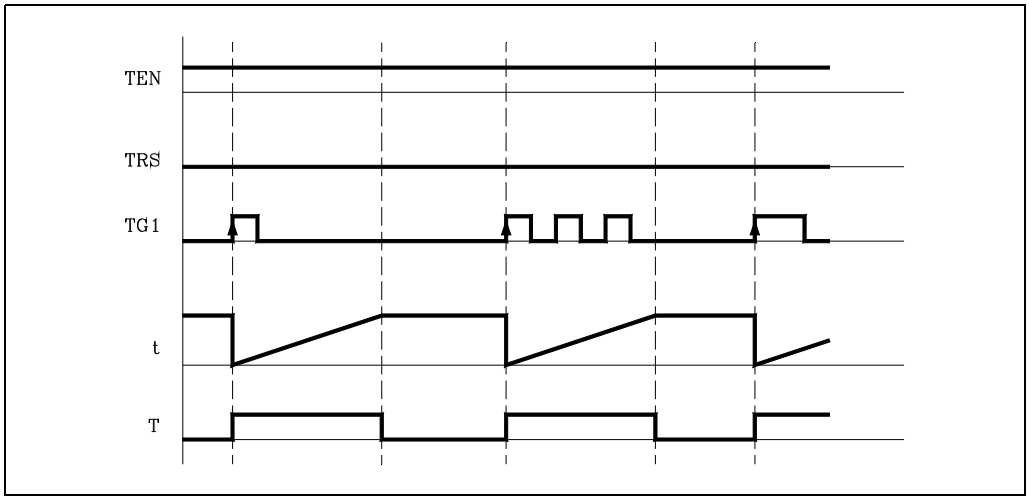
The PLC has a 32-bit variable to store the time of each timer.

9.5.1 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".



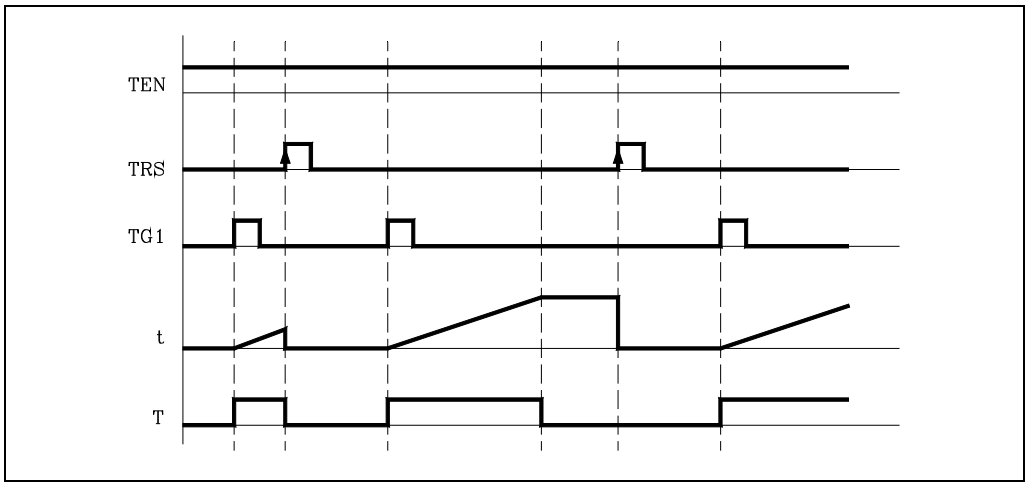
Once the time specified by the time constant has elapsed, timing will be considered as having finished. The timer status output (T) changes status (T=0) and the elapsed time will be maintained with the time value of the timer (T).

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.

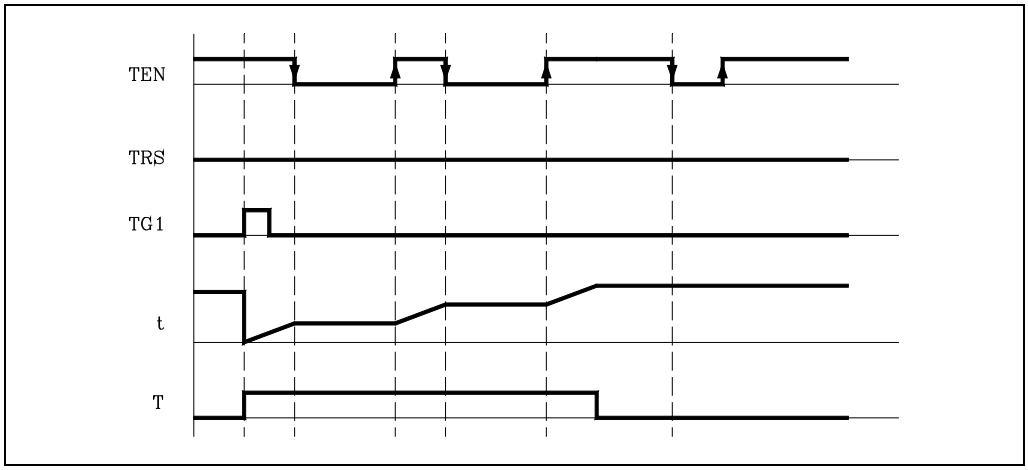
Operation of the TRS input in this mode

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

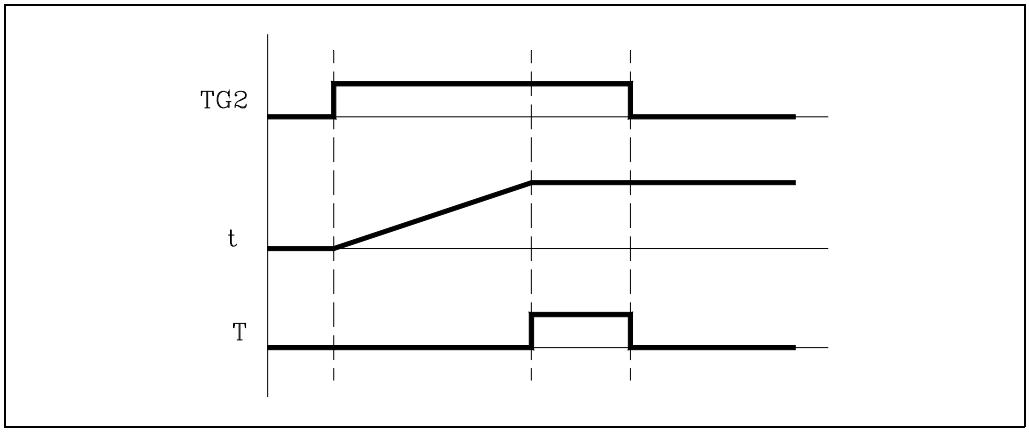
If once the timer has been activated, TEN = 0, the PLC interrupts the timing and TEN must be set to "1" to resume timing.



9.5.2 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".

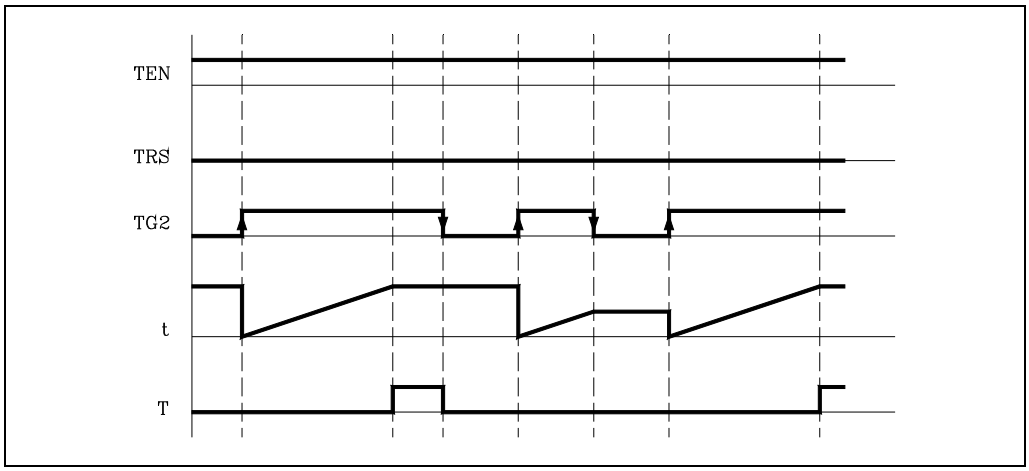
The time delay is set by the time constant.



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.

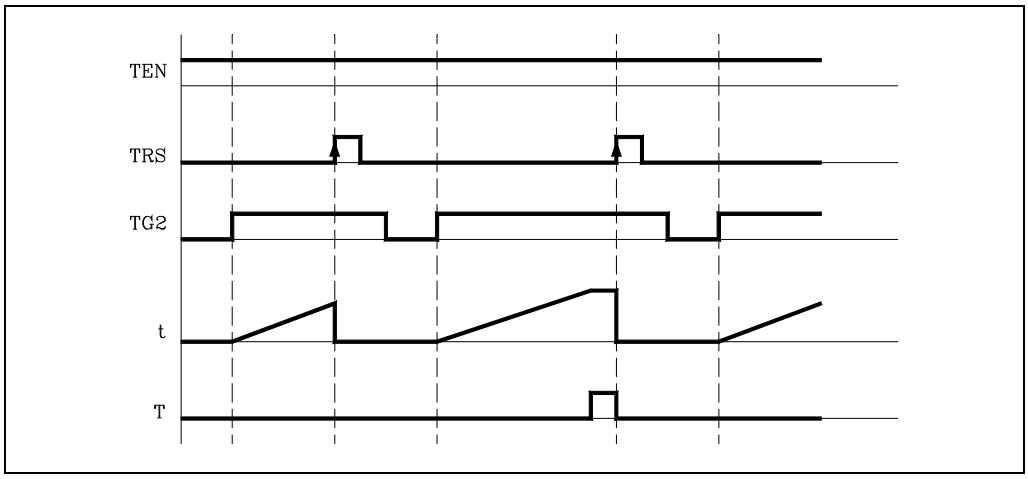


If, once the timing has finished, it is required to activate the timer again, another leading edge must be produced in the TG2 input.

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).

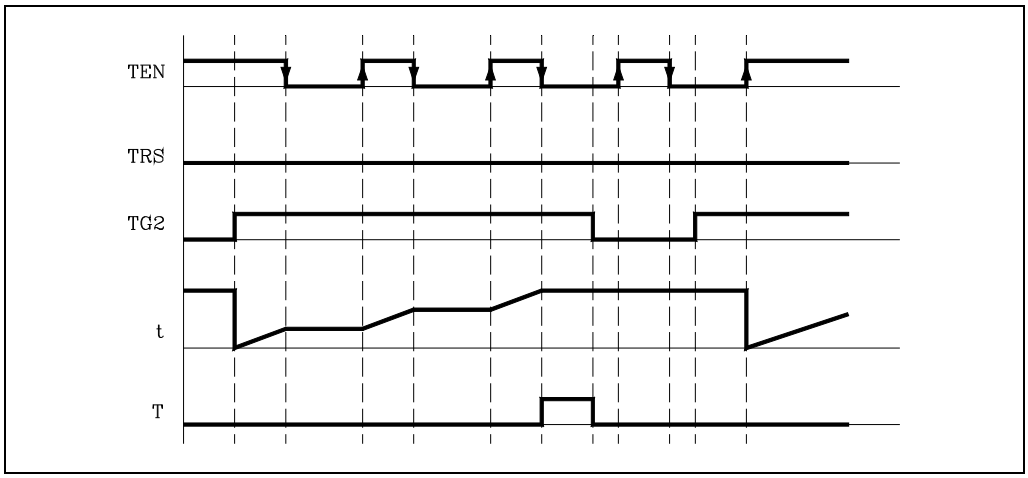
Operation of the TRS input in this mode

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

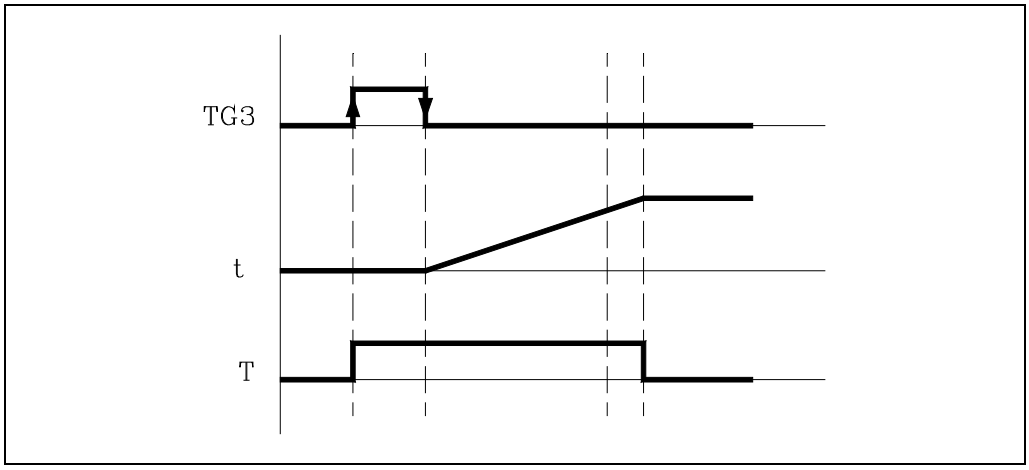
If once the timer has been activated, TEN = 0, the PLC interrupts the timing and TEN must be set to "1" to resume timing.



9.5.3 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.

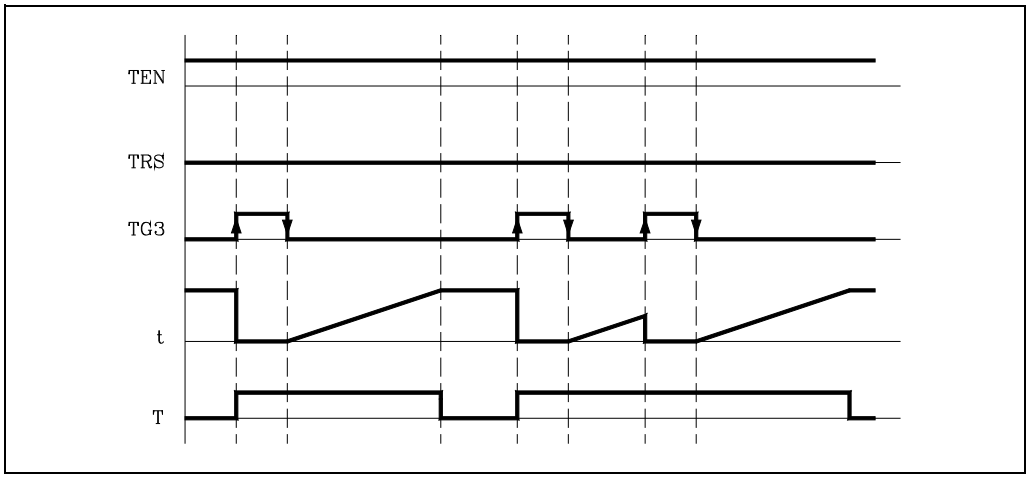


If the timer is initialized with values TEN=1 and TRS=0, the timer will be activated when a leading edge is produced at the TG3 input. At that moment, the timer status output will have a value of T=1.

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.

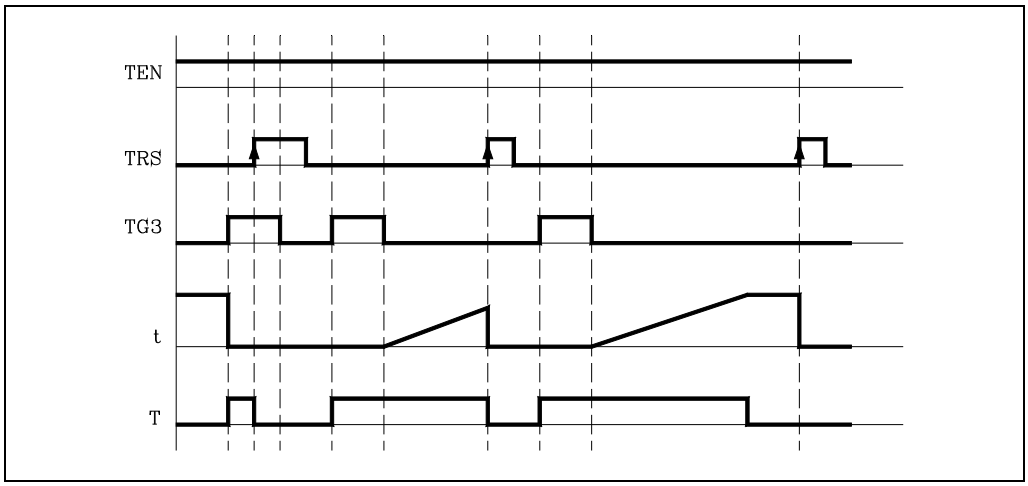


If, once the timing has finished, it is required to activate the timer again, another leading edge must be produced at the TG3 input.

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.

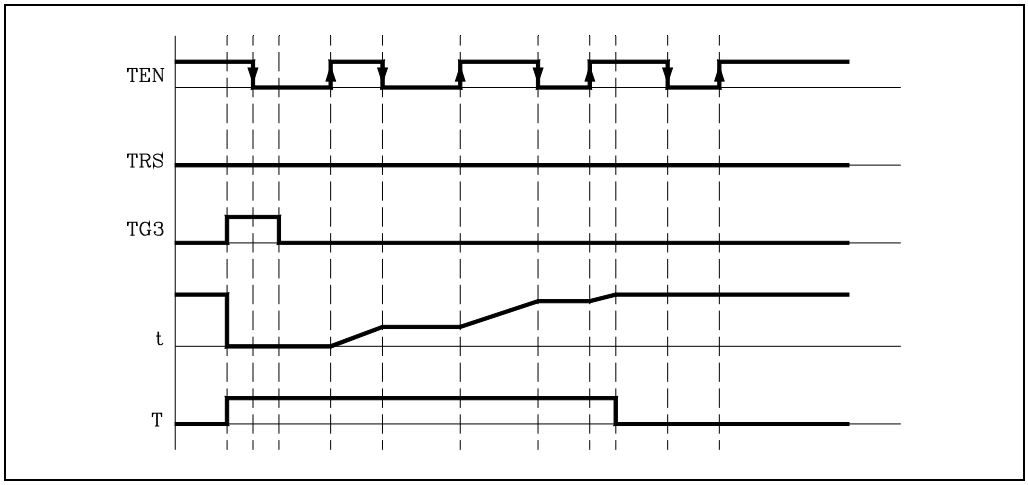
Operation of the TRS input in this mode

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.



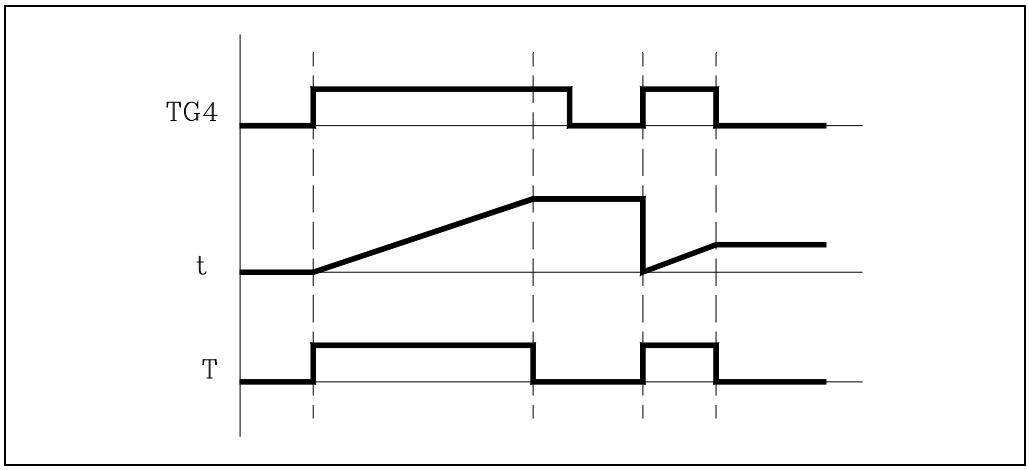
Operation of the TEN input in this mode

If once the timer has been activated, TEN = 0, the PLC interrupts the timing and TEN must be set to "1" to resume timing.

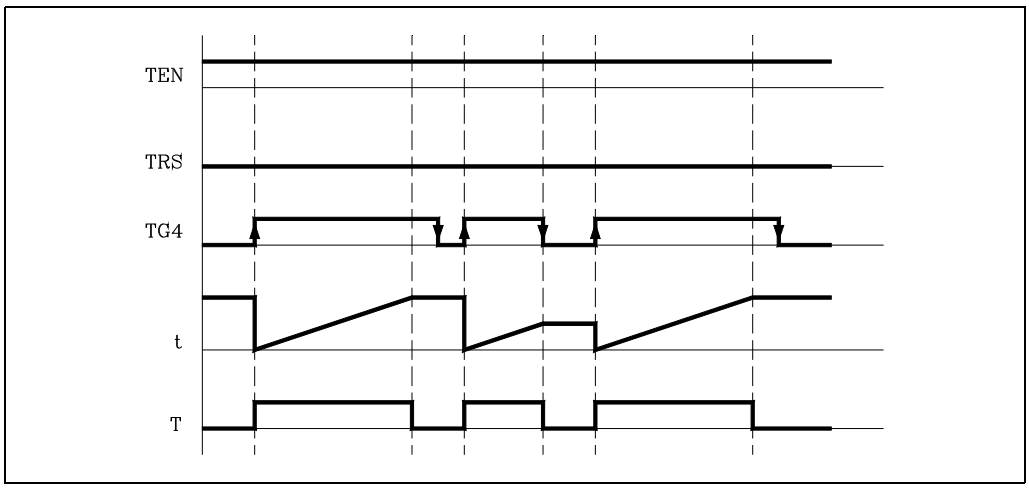


9.5.4 Signal limiting mode. TG4 Input

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".



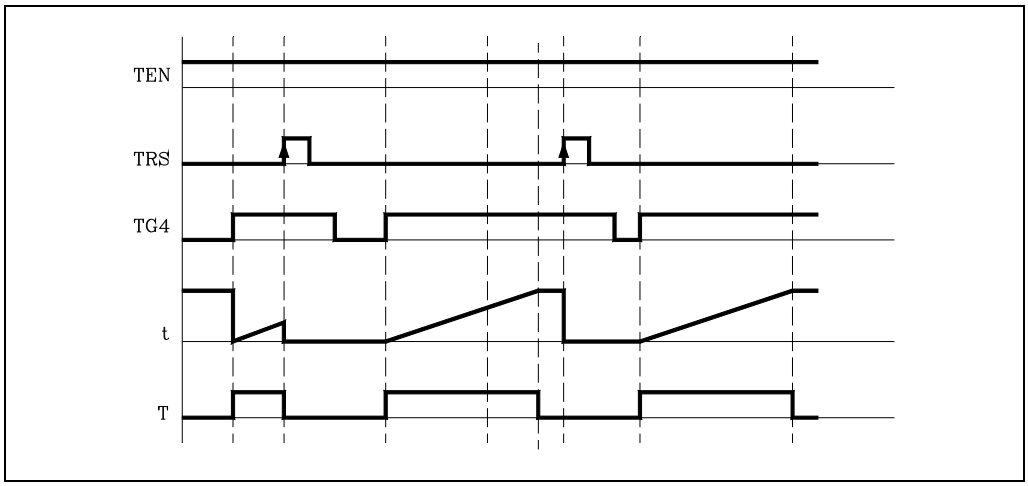
Once the time specified by the time constant has elapsed, timing will be considered as having finished. The time status output (T) changes status (T=0) and the elapsed time will be kept as a timer time value (T).

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.

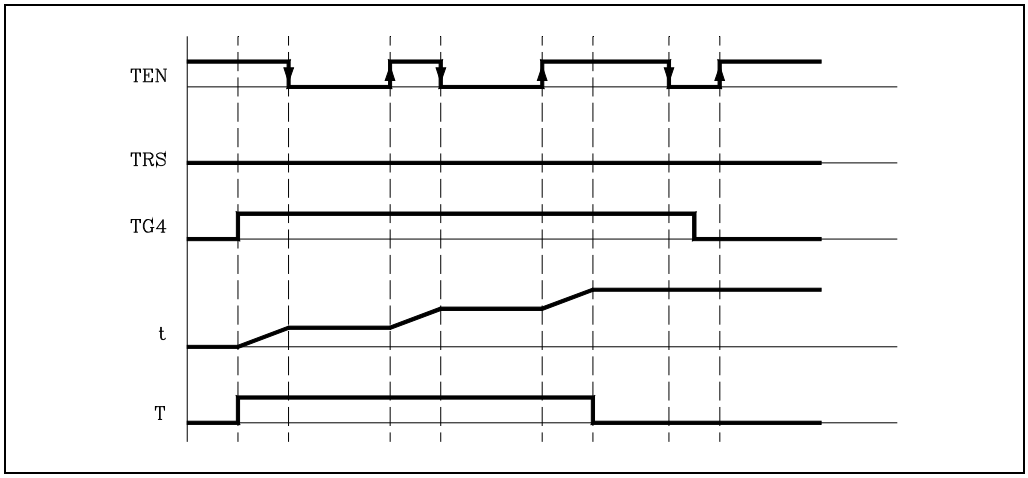
Operation of the TRS input in this mode

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.

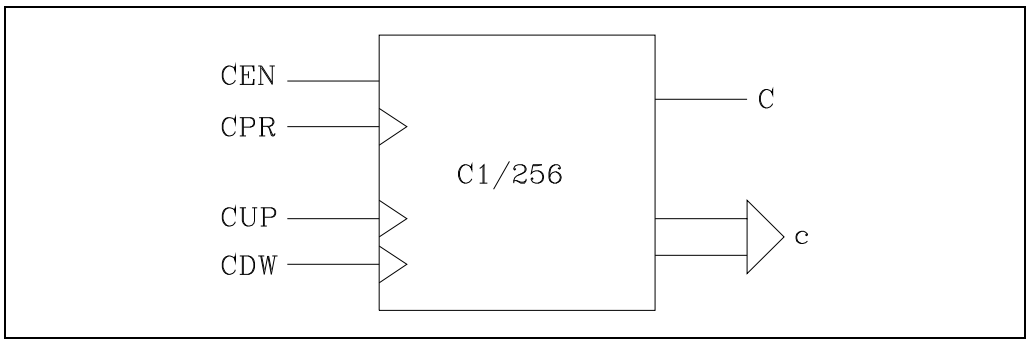


9.6 Counters

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:

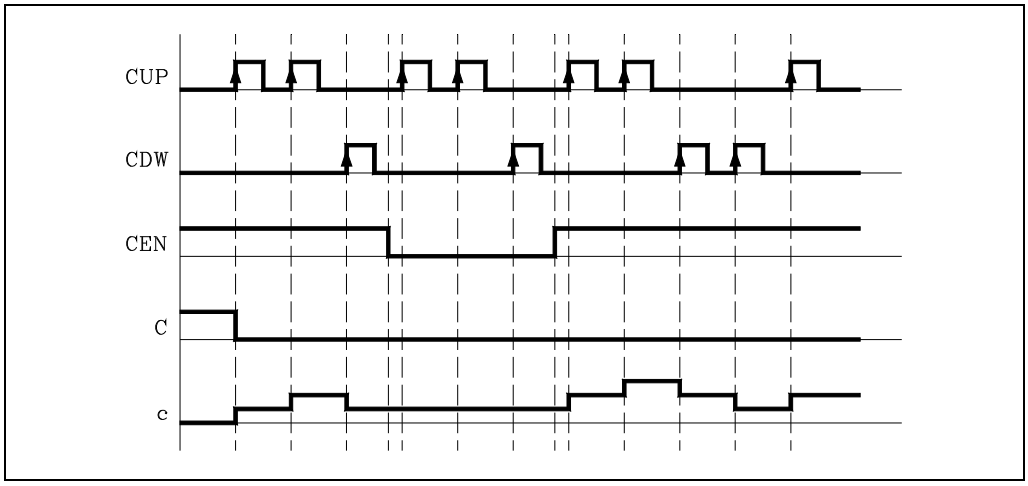
I3 = CDW 20	Every time a leading edge is produced at input I3 the counter count C20 will be decreased.

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 counter C20 with a value of 100.
CPR 22 R200	Presets the C22 counter with the value of the register 200 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 (0/1) status 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.

9.6.1 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.

PLC PROGRAMMING

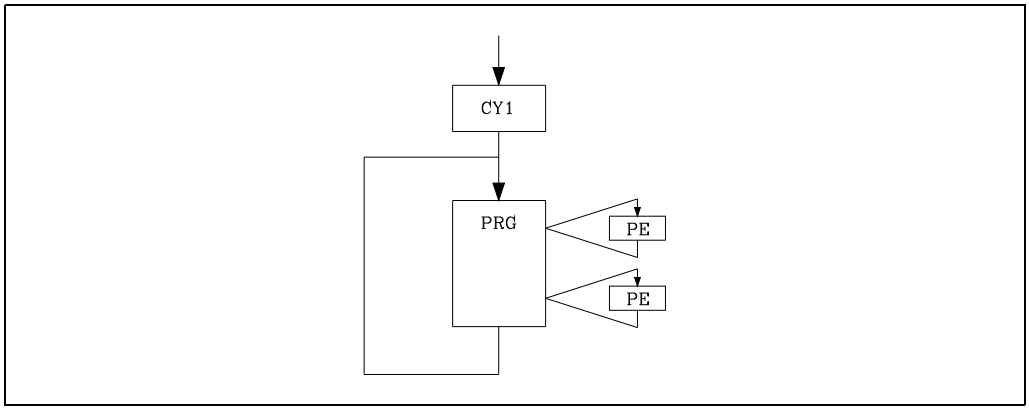
10

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 *"8.2 PLC program execution"* on page 413.

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.

To work with remote physical inputs and outputs, use the IREMRD and OREMWR instructions.

10.1 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 instruction.
I28 AND I30         ; Logic expression.
= O25               ; Action instruction.
I32 \               ; Consulting instruction (1st part of expression).
AND I36             ; Consulting instruction (2nd part of expression).
= M300              ; Action instruction.
END                 ; Directing instruction.
```

See *"Summary of PLC commands"* on page 643.

Empty lines are not allowed, they must contain at least one comment.

10.2 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 is represented with the letter L followed by up to 4 digits (1-2000), no order needs to be followed and numbers may even be skipped.

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 I1, 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 2000 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 and 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, IREMRD

Update the real values of the local inputs (IRD) and the remote ones (IREMRD) after reading the relevant physical inputs.

Care must be taken when using these instructions since the current real values of the inputs will be lost.

OWR, OREMWR

Update the local physical outputs (OWR) and the remote ones (OREMWR) 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.

10.3 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	1/512	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:

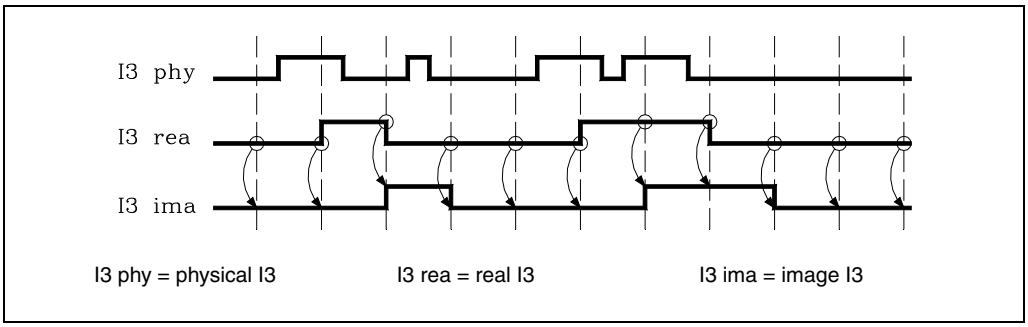
DFU (Up flank detection.)	I 1/512
DFD (Down flank detection)	O 1/512 M 1/5957

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.

10.4 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).

10.5 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.

10.5.1 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)) .

10.5.2 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 "1", it assigns a "0" 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.

10.5.3 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 1/2000).

A subroutine is any part of the program that starts with a label (L1/2000) 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.

10.5.4 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 SBS MLS DVS MDS	R1/559 #	R1/559 #	R1/559

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.

10.5.5 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 OR XOR	R1/559 #	R1/559 #	R1/559

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

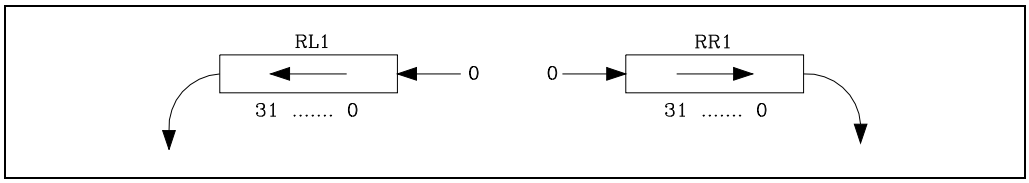
()=AND	R200	R201	R202	R202=B0	M2003=1
()=OR	R200	R201	R203	R203=B11010111	M2003=0
()=XOR	R200	R201	R204	R204=B11010111	M2003=0
()=AND	B1111	R201	R205	R205=B00000101	M2003=0
()=OR	R200	B1111	R206	R206=B10011111	M2003=0
()=XOR	B1010	B1110	R207	R207=B00000100	M2003=0

= 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).

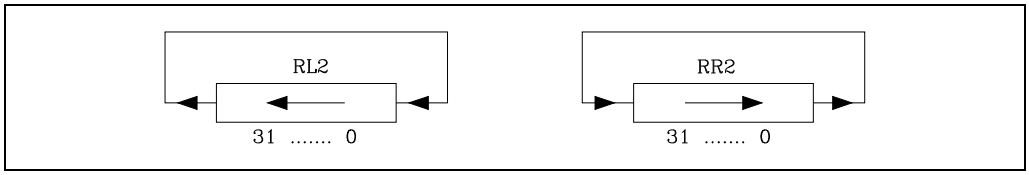
Rotation type 1 (RL1 or RR1):

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 RR2 RL1 RL2	R1/559	R1/559 0/31	R1/559

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

10.5.6 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.

= WRITE, = READ

Instructions to control communication via RS232. These READ and WRITE instructions may be executed from a periodic execution module (PE).

= WRITE nBuffer.

If the line is not busy, the data of the indicated buffer is sent through the RS232 serial line. If the line is busy, the data is not sent.

= READ nBuffer.

The RS232 serial line remains "listening" and fills up the buffer nBuffer with the data received according to the setting of RCVCTRL. Only one READ instruction may be executed at a time.

CNC-PLC COMMUNICATION

11

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.

11.1 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:

S Programmed	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

S Programmed	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

S Programmed	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

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 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.

11.2 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.

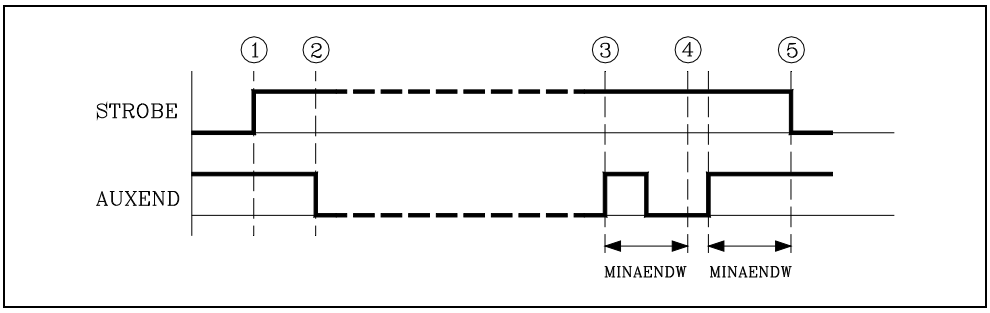
This indication will be made by means of the variable "T2BCD" (R559) and by activating the general logic output "T2STROBE" to tell the PLC that it must execute it. The CNC will wait for the general input AUXEND to be activated to consider the execution completed.



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.

11.2.1 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).

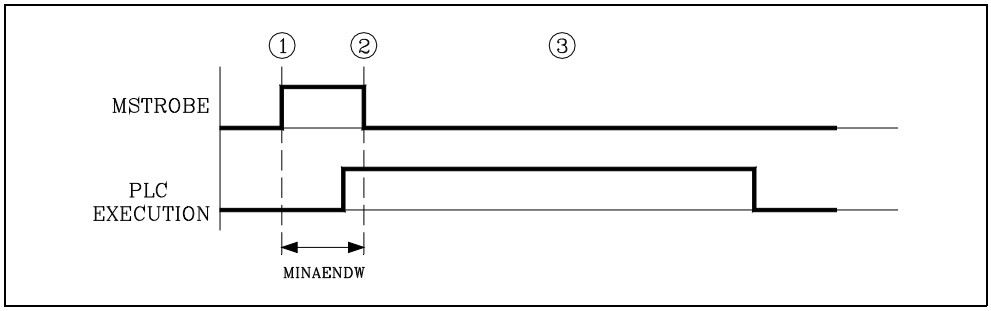
   This way, erroneous interpretations of this signal by the CNC due to an improper PLC program logic are avoided .
5. Once the period of time MINAENDW has elapsed with the general input "AUXEND" at a high logic level, the CNC will deactivate the general logic outputs "MSTROBE", "SSTROBE", "TSTROBE", "T2STROBE" 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.

11.2.2 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.

11.3 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.

11.4 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.

11.5 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.

11.

CNC-PLC COMMUNICATION

Access to the PLC from a PC, via DNC

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

LOGIC CNC INPUTS AND OUTPUTS

12

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.
- Logic inputs of the auxiliary spindle
- Key inhibiting logic inputs.
- Logic inputs of the PLC channel
- General logic outputs.
- Axis logic outputs.
- Spindle logic outputs.
- Logic outputs of the auxiliary spindle
- Logic outputs of key status
- Logic inputs of the PLC channel

12.1 General logic inputs

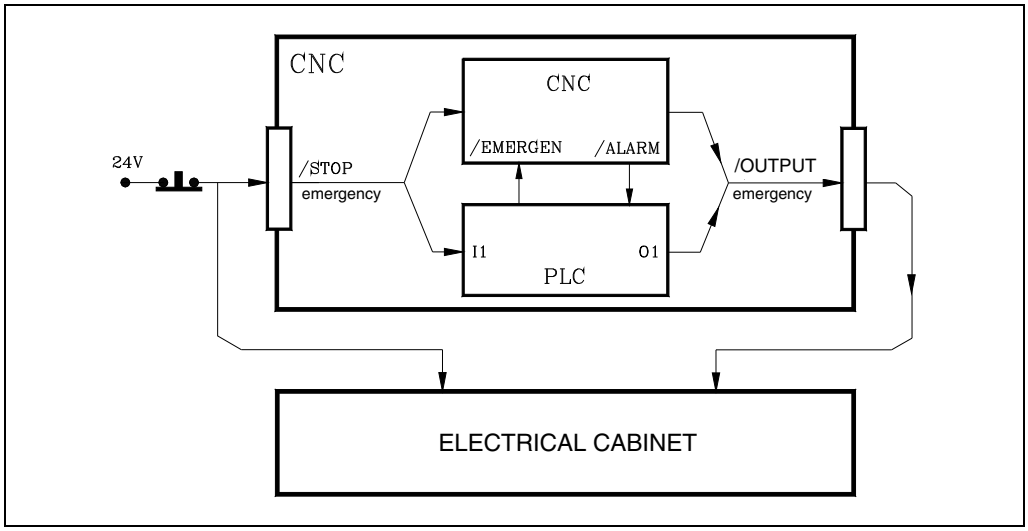
These inputs must always be defined in the PLC program.
/EMERGEN (M5000) /STOP (M5001)
/FEEDHOL (M5002) /XFERINH (M5003)

/EMERGEN (M5000)

There are to ways to cause an emergency at the CNC, by activating the physical input /Emergency stop or the general logic input "/EMERGEN" from the PLC.

8055 CNC Pin 2 of connector X9 of the –Axes– module or –Axes Vpp– module.

8055i CNC. Pin 10 of connector X2



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 from 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.

MACHMOVE (M5012)

When working with coordinate transformation of inclined planes, the axes movements are made with respect to the axes of the part

To make the jogging movements by handwheel or keyboard along the axes of the machine, use G53 or activate the general CNC input "MACHMOVE (M5012) at the PLC.

MACHMOVE = 0	The movements coincide with the axes of the part.
MACHMOVE = 1	The movements coincide with the axes of the machine.

See the chapter on "Coordinate Transformation" in the programming manual.

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 2nd 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.

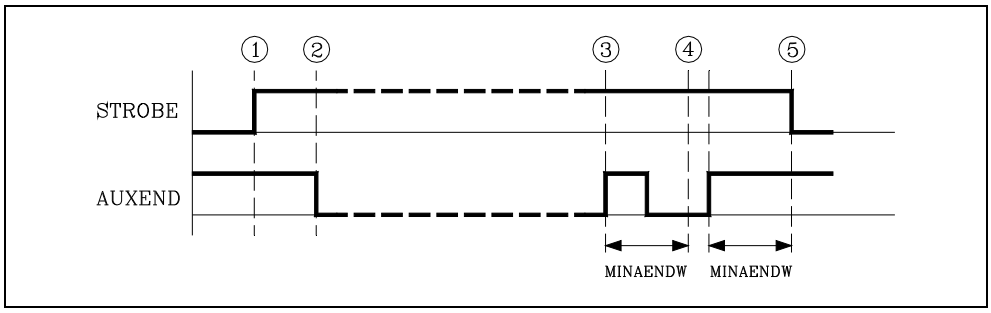
The treatment received by this signal is similar to that given to the RESET key on the front panel.

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. An application of this timer is to monitor tool life.

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 in order to tell the CNC that the front panel keyboard (monitor with keyboard) and the keyboard of the operator panel of the CNC are 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.

TOOLMOVE (M5021)

When working with coordinate transformation of inclined planes, the axes movements are made with respect to the axes of the part

To make the jogging movements by handwheel or keyboard along the axes of the machine, use G47 or activate the general CNC input "TOOLMOVE (M5021) at the PLC.

TOOLMOVE = 0 The movements coincide with the axes of the part.

TOOLMOVE = 1 The movements coincide with the axes of the tool.

See the chapter on "Coordinate Transformation" in the programming manual.

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.

The following example shows how the axes controlled by the PLC may be moved by means of external push-buttons.

Example
The PLC will order to move the "C" axis by 1 meter every time the "C+" button is pressed, but stopping it when this key is released.
DEF CPLUS I2
Symbol to define the "C+" push button.
DFU CPLUS =CNCEX (G91 G1 C1000 F3000, M1)
Pressing the button commands it to move 1000 mm.
DFD CPLUS = SET PLCABORT
Releasing the button aborts the movement.

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 in execution (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.

TOOLINSP (M5050)

The CNC takes into consideration this input in MC, MCO, TC and TCO modes.

It indicates whether the "T" key must be pressed or not after executing the operation or the part in order to inspect the tool.

TOOLINSP = 0	The tool inspection mode is available when interrupting the execution.
TOOLINSP = 1	To access the tool inspection mode, first interrupt the execution and then press the "T" key.

RETRACE (M5051)

The CNC takes into account this input when retracing is available, g.m.p. RETRACAC other than zero. The retracing function may 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 the last blocks that were already executed.

The retracing function ends in the following cases:

- When retracing the 100 previous blocks on the FL model or the 200 previous blocks on the Power model.
- When retraced all the way to the beginning of the program.
- When retracing up to block G51 (activating the look-ahead function).
- 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 several blocks may go by from the moment the RETRACE mark is activated until the machine starts retracing. Besides,the look-ahead calculations will be different when going in one direction and returning; therefore, both paths might not be exactly the same.

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	Circular 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.

CAXSEROK (M5055)

It must be used on the lathe model, when the "C" axis and the spindle share the same drive. See *"7.12 Digital servo (CAN or Sercos)"* on page 357.

The PLC sets this signal high to let the CNC know that the drive is ready to work as a "C" axis.

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.

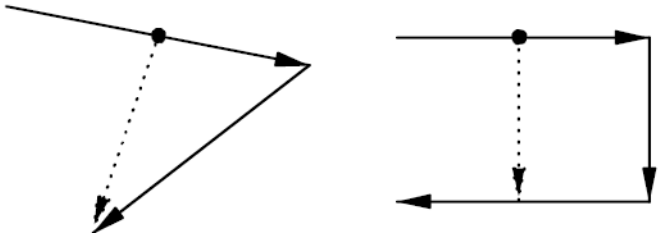
These marks do not affect the following functions.

- 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 (home search).
- It has no effect when tangential control is active.

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 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. g.m.p. ACTGAINT (P185) indicates with which functions or in which mode the third set is applied, the one set by a.m.p. ACCTIMET (P92), PROGAINT (P93), DERGAINT (P94) and FFGAINT (P95) or 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)

During the 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 withdrawal is completed, the user may finish the hole, go to the next hole or go into tool inspection.

General logic input SKIPCYCL (M5064) is used to go to the next hole, once the withdrawal is completed.

RETRACYC (M5065)

During the 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 withdrawal is completed, the user may finish the hole, go to the next hole 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 message during tool change. When this mark is activated, the CNC activates the TMINEM mark.

RESTMEM (5067)

PLC mark that may be used to cancel the error state of the CNC. This mark is activated when the user confirms that the tool magazine has been inspected and that everything is OK to continue working.

12.2 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 7 (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.

Mnemonics using numbers 1 through 7.

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, Y, Z, B, C and U axis, the order will be: X, Y, Z, U, B, C 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:
LIMIT+4, LIMIT-4, DECEL4, etc.	for the U axis:
LIMIT+5, LIMIT-5, DECEL5, etc.	for the B axis
LIMIT+6, LIMIT-6, DECEL6, etc.	for the C 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)**	**LIMIT+4 (M5250)**	**LIMIT-4 (M5251)**
LIMIT+5 (M5300)	**LIMIT-5 (M5301)**	**LIMIT+6 (M5350)**	**LIMIT-6 (M5351)**
LIMIT+7 (M5400)	**LIMIT-7 (M5401)**		

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)**	**DECEL4 (M5252)**
DECEL5 (M5302)	**DECEL6 (M5352)**	**DECEL7 (M5402)**	

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)** **INHIBIT4 (M5253)**
INHIBIT5 (M5303) **INHIBIT6 (M5353)** **INHIBIT7 (M5403)**

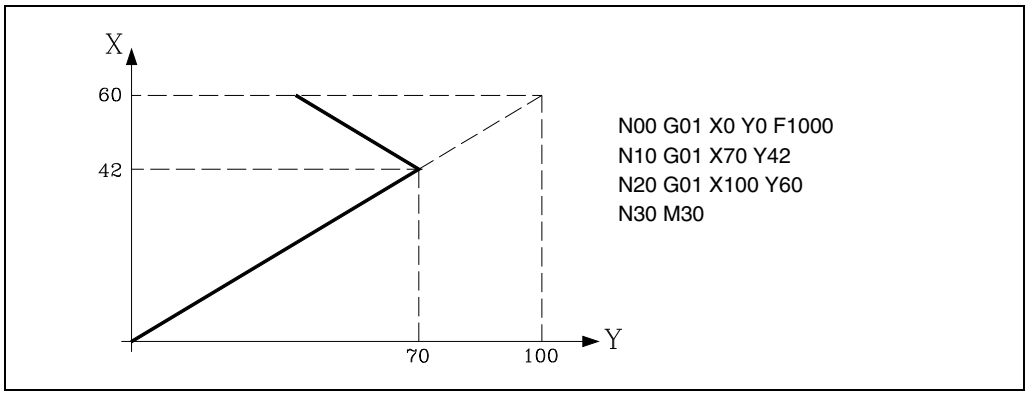
The PLC sets one of these signals at a high logic level in order to tell the CNC to prevent any movement of the corresponding axis. This movement will continue when the PLC sets this signal at the low logic level once more.

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)** **MIRROR4 (M5254)**
MIRROR5 (M5304) **MIRROR6 (M5354)** **MIRROR7 (M5404)**

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.



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)** **SWITCH4 (M5255)**
SWITCH5 (M5305) **SWITCH6 (M5355)** **SWITCH7 (M5405)**

When having 2 axes controlled by the same servo drive, this mark may be used to toggle the velocity commands between the two axes.

See *"7.13 Axes (2) controlled by a single drive"* on page 362.

DRO1 (M5106) **DRO2 (M5156)** **DRO3 (M5206)** **DRO4 (M5256)**
DRO5 (M5306) **DRO6 (M5356)** **DRO7 (M5406)**

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)** **SERVO4ON (M5257)**
SERVO5ON (M5307) **SERVO6ON (M5357)** **SERVO7ON (M5407)**

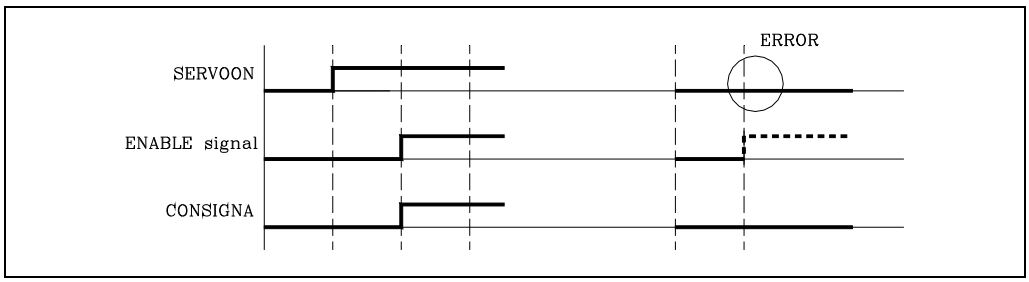
When one of these signals is set high, the CNC closes the positioning loop of the corresponding axis.

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.

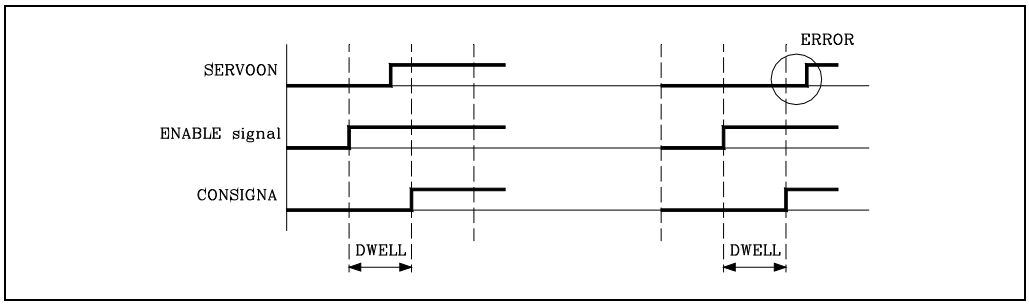
On the other hand, if the SERVOON signal is low or if it changes during the movement of the axes, the CNC stops the axes feed and the spindle rotation displaying the corresponding error message.

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)**	**AXIS+4 (M5258)**	**AXIS-4 (M5259)**
AXIS+5 (M5308)	**AXIS-5 (M5309)**	**AXIS+6 (M5358)**	**AXIS-6 (M5359)**
AXIS+7 (M5408)	**AXIS-7 (M5409)**		

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)**	**SPENA4 (M5260)**	**DRENA4 (M5261)**
SPENA5 (M5310)	**DRENA5 (M5311)**	**SPENA6 (M5360)**	**DRENA6 (M5361)**
SPENA7 (M5410)	**DRENA7 (M5411)**	**SPENA8 (M6110)**	**DRENA8 (M6111)**
SPENA9 (M6160)	**DRENA9 (M6161)**		

The CNC uses these signals when communicating with the drive via Sercos or 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.

When using Sercos, when an emergency occurs at the CNC, the SPENA signals of the axes and the spindle are disabled and the drive brakes using the emergency ramps.

SYNCHRO1 (M5112)	**SYNCHRO2 (M5162)**	**SYNCHRO3 (M5212)**	**SYNCHRO4 (M5262)**
SYNCHRO5 (M5312)	**SYNCHRO6 (M5362)**	**SYNCHRO7 (M5412)**	

The PLC sets one of these signals high to synchronize the corresponding axis to the axis defined by the a.m.p. "SYNCHRO".

ELIMINA1 (M5113)	**ELIMINA2 (M5163)**	**ELIMINA3 (M5213)**	**ELIMINA4 (M5263)**
ELIMINA5 (M5313)	**ELIMINA6 (M5363)**	**ELIMINA7 (M5413)**	

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.

When the axis is controlled via Sercos and the PLC sets the relevant ELIMINA signal high, the commands to park that axis are generated automatically via Sercos. The drive of that axis will not issue, for example, when removing the feedback, the CNC will not issue a feedback error.

SMOTOF1 (M5114)	**SMOTOF2 (M5154)**	**SMOTOF3 (M5214)**	**SMOTOF4 (M5254)**
SMOTOF5 (M5314)	**SMOTOF6 (M5354)**	**SMOTOF7 (M5414)**	

The SMOTIME (P58) filter set for each axis with parameter P58 can be canceled from the PLC a.m.p. SMOTIME (P58).

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)**	**LIM4OFF (M5265)**
LIM5OFF (M5315)	**LIM6OFF (M5365)**	**LIM7OFF (M5415)**	

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)** **MANINT4 (M5266)**
MANINT5 (M5316) **MANINT6 (M5366)** **MANINT7 (M5416)**

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)** **DIFFCOM4 (M5267)**
DIFFCOM5 (M5317) **DIFFCOM6 (M5367)** **DIFFCOM7 (M5417)**

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 of axes.

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.

12.3 Spindle logic inputs.

This CNC can handle 2 spindles: a main spindle and a second spindle. They both can be operative simultaneously, but only one can be controlled at a time. This selection can be made via part-program by means of functions G28 and G29.

LIMIT+S (M5450) **LIMIT-S (M5451)** **Main spindle**
LIMIT+S2 (M5475) **LIMIT-S2 (M5476)** **Second 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**
DECELS2 (M5477) **Second 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**
SPDLEIN2 (M5478) **Second 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**
SPDLERE2 (M5479) **Second 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**
SMOTOFS2 (M5480) **Second spindle**

The SMOTIME (P46) filter set for the main and second spindles with parameter P46 can be canceled from the PLC. SMOTIME (P46).

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.

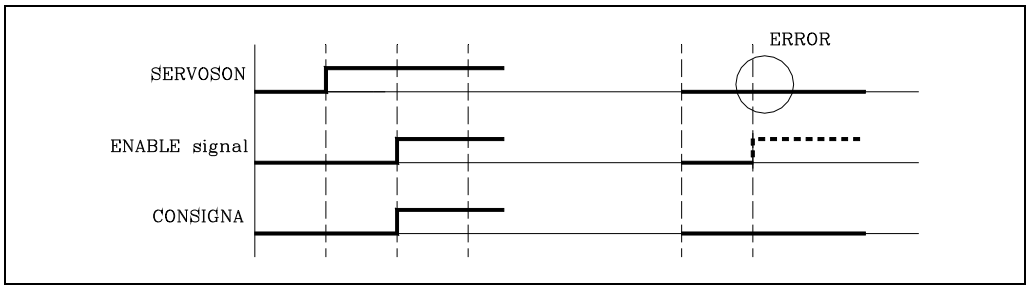
SERVOSON (M5457) **Main spindle**
SERVOSO2 (M5482) **Second spindle**

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.



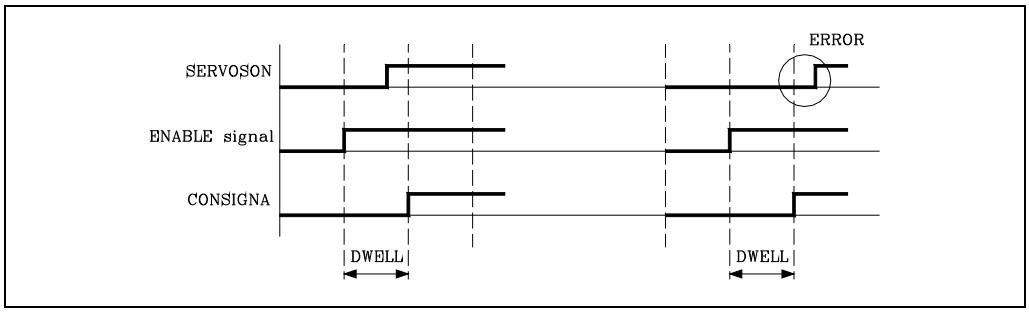
On the other hand, if the SERVOSON signal is low or if it changes to low during the rotation of the spindle, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.

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
GEAR12 (M5483) GEAR22 (M5484) GEAR32 (M5485) GEAR42 (M5486) Second 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:

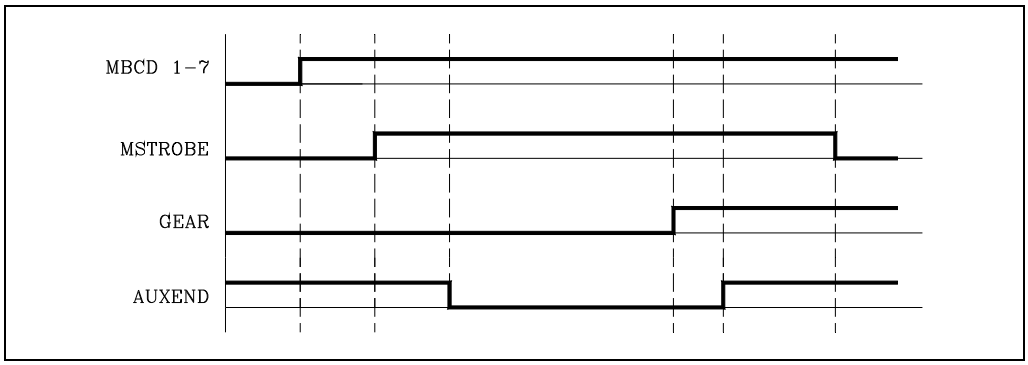
The M41, M42, M43 or M44 function uses the "AUXEND" signal:

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 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 .

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 general logic 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)	**DRENAS (M5463)**	**Main spindle**
SPENAS2 (M5487)	**DRENAS2 (M5488)**	**Second spindle**

The CNC uses these signals when communicating with the drive via Sercos or 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.

When using Sercos, when an emergency occurs at the CNC, the SPENA signals of the axes and the spindle are disabled and the drive brakes using the emergency ramps.

PLCFM19 (M5464)	**M19FEED (R505)**	**Main spindle**
PLCFM192 (M5489)	**M19FEED2 (R507)**	**Second 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).

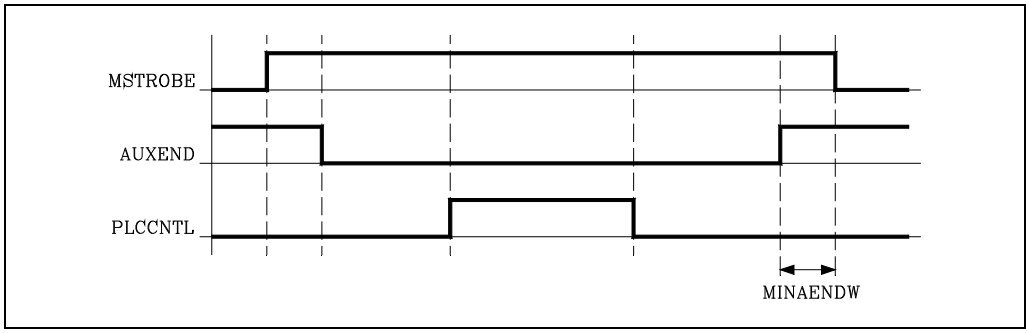
The "M19FEED" value is given in 0.0001º/min.

PLCCNTL (M5465) **Main spindle**
PLCCNTL2 (M5490) **Second 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**
SANALOG2 (R506) **Second 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

When working with SERCOS, the values of drive parameters SP20 and SP21 are taken into account. This way, the following value is assigned to the velocity command (corresponding FTEO): SP21 * 10 / SP20 rpm.

ELIMIS (M5456) **Main spindle**
ELIMIS2 (M5481) **Second spindle**

If the PLC sets one these signals 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.

When the spindle is controlled via Sercos and the PLC sets this signal high, the commands to park the axis are generated automatically via Sercos. The drive of that spindle will not issue, for example, when removing the feedback, the CNC will not issue a feedback error.

PLCM3 (M5070) **PLCM4 (M5071)** **PLCM5 (M5072)** **Main spindle**
PLCM3SP2 (M5073) **PLCM4SP2 (M5074)** **PLCM5SP2 (M5075)** **Second spindle**

These marks may be used to execute functions M3, M4 and M5 from the PLC. Executing these M functions from the PLC is similar to doing it from the main channel; i.e. besides acting upon the spindle command, it changes the execution history.

The same PLC operation that handles the execution from the main channel also serves for this execution mode, that does it through PLC marks.

The names of the marks of the first spindle will be as follows:

- To execute function M3: PLCM3.
- To execute function M4: PLCM4.
- To execute function M5: PLCM5.

When having two spindles, the marks of the first spindle will be called in one of the following ways:

- To execute function M3: PLCM3 or PLCM3SP1.
- To execute function M4: PLCM4 or PLCM4SP1.
- To execute function M5: PLCM5 or PLCM5SP1.

The marks of the second spindle will be called as follows:

- To execute function M3: PLCM3SP2.
- To execute function M4: PLCM4SP2.
- To execute function M5: PLCM5SP2.

12.4 Logic inputs of the auxiliary spindle

SPENAAS (M5449) DRENAAS (M5448)

The CNC uses these signals when communicating with the drive via Sercos or 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.

When using Sercos, when an emergency occurs at the CNC, the SPENA signals of the axes and the spindle are disabled and the drive brakes using the emergency ramps.

PLCCNTAS (M5056)

It is used to "tell" the CNC that the auxiliary spindle is controlled directly by the PLC (active high).

SANALOAS (R509)

With this 32-bit register, the PLC will "tell" the CNC what spindle velocity command to output when the auxiliary spindle is controlled by the PLC or via Sercos.

A 10 V of analog command corresponds to SANALOAS=32767.

SANALOAS=1 corresponds (10/32767) 0.305185 millivolts of analog command.

This way, for 4V of velocity command, the following must be programmed:

SANALOAS = (4x32767)/10 = 13107

For -4V of analog command, the following must be programmed:

SANALOAS = (-4x32767)/10 = -13107

ELIMIAS (M5062)

If the PLC sets this signal high, the CNC does not display the corresponding spindle but keeps controlling it. This mark may be activated and deactivated any time and it also cancels the feedback alarms.

PLCM45 (M5076)

This PLC mark may be used to stop the auxiliary spindle.

PLCM45S (M5077)

This PLC mark may be used to start the auxiliary spindle.

12.5 Key inhibiting logic inputs.

KEYDIS1 (R500) **KEYDIS2 (R501)** **KEYDIS3 (R502)** **KEYDIS4 (R503)**
KEYDIS5 (R508)

The PLC can individually inhibit the operation of the panel keys, setting the corresponding bit of these 32-bit registers high. The appendix of this manual shows, for each keyboard, which key each bit corresponds to. See *"Key inhibiting codes"* on page 675.

Register KEYDIS4 disables the positions of the feedrate override switch (feedrate % selector). Register KEYDIS5 disables the specific keys for the conversational models.

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%.

12.6 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.

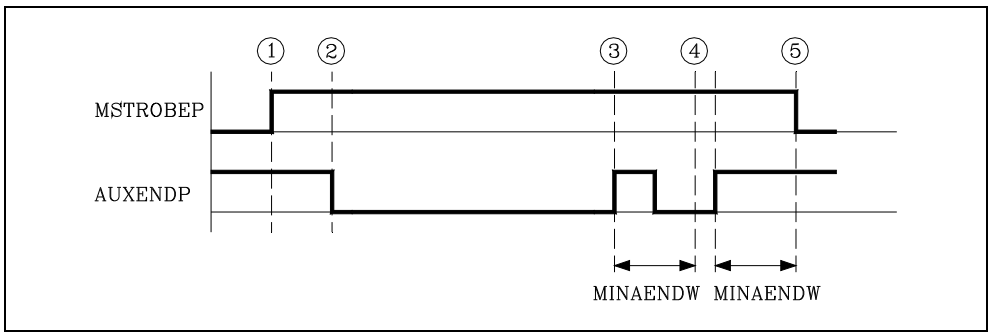
AUXENDP (M5006)

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.

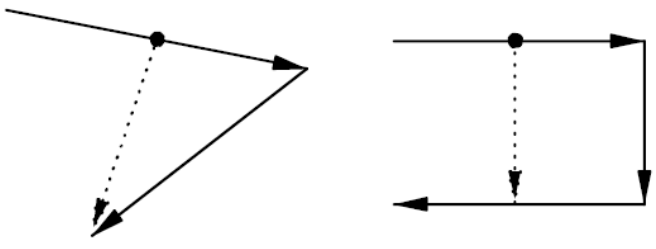
These marks do not affect the following functions.

- 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.
- It has no effect when tangential control is active.

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.

12.7 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)

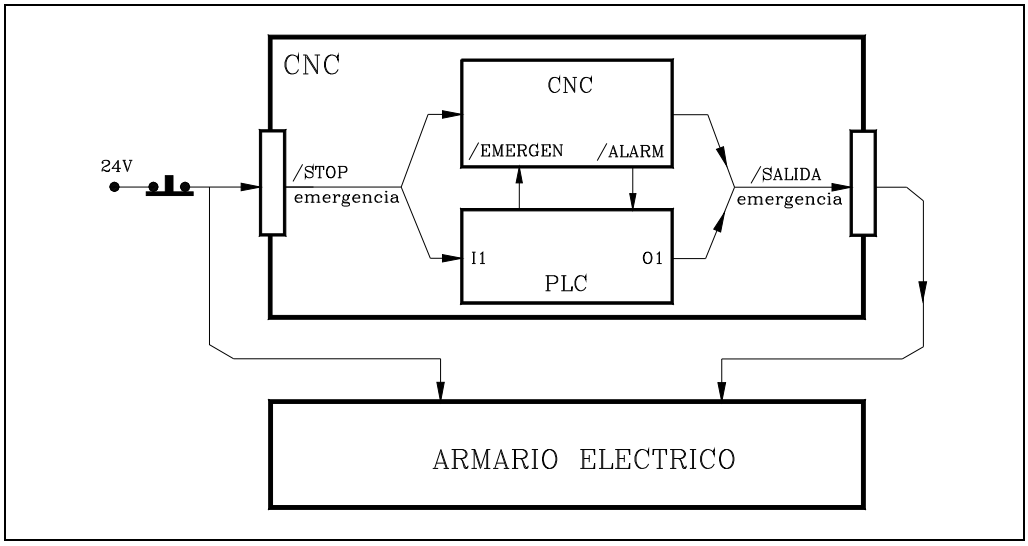
The CNC sets this signal high for 100 milliseconds, in order to tell the PLC that it is under initial conditions because the Reset key on the front panel has been pressed or because the general logic input RESETIN has been activated.

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 active (low).

8055 CNC Pin 2 of connector X10 of the –Axes– module or –Axes Vpp– module.

8055i CNC. Pin 2 of connector X2

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.

CUSTOM (M5512)

It indicates to the CNC which work mode is currently selected:

CUSTOM = 0 M or T work mode.

CUSTOM = 1 MC, MCO, TC or TCO work mode.

When having 2 keyboards, this variable may be used in the PLC in the following cases:

- To govern the keyboard switching board.
- To know the source of the keys and inhibit the desired keys.

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).

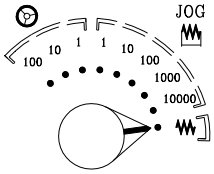
SELECT0 (M5524) **SELECT1 (M5525)** **SELECT2 (M5526)** **SELECT3 (M5527)**
SELECT4 (M5528) **SELECT5 (M5529)** **SELECT6 (M5530)** **SELECT7 (M5531)**
SELECTOR (R564)

The CNC uses these signals to indicate to the PLC the position selected at each keyboard switch.

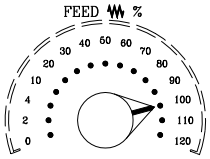
SELECTOR It indicates the position currently selected.

SELECT It 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(3) SELECT3	SELECTOR(2) SELECT2	SELECTOR(1) SELECT1	SELECTOR(0) SELECT0
Handwheel x100	0	0	0	0
Handwheel x10	0	0	0	1
Handwheel x1	0	0	1	0
JOG 10000	0	0	1	1
JOG 1000	0	1	0	0
JOG 100	0	1	0	1
JOG 10	0	1	1	0
JOG 1	0	1	1	1
Continuous JOG	1	0	0	0



	SELECTOR(7) SELECT7	SELECTOR(6) SELECT6	SELECTOR(5) SELECT5	SELECTOR(4) SELECT4
Feedrate override 0%	0	0	0	0
Feedrate override 2%	0	0	0	1
Feedrate override 4%	0	0	1	0
Feedrate override10%	0	0	1	1
Feedrate override 20%	0	1	0	0
Feedrate override 30%	0	1	0	1
Feedrate override 40%	0	1	1	0
Feedrate override 50%	0	1	1	1
Feedrate override 60%	1	0	0	0
Feedrate override 70%	1	0	0	1
Feedrate override 80%	1	0	1	0
Feedrate override 90%	1	0	1	1
Feedrate override 100%	1	1	0	0
Feedrate override 110%	1	1	0	1
Feedrate override 120%	1	1	1	0

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 is used when using a spindle operating 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 g.m.p. RANDOMTC (P25) has been set so it is not a random tool 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). 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.

S2MAIN (M5536)

It indicates which spindle is controlled by the CNC. This selection can be made via part-program by means of functions G28 and G29.

If the CNC controls the main spindle S2MAIN is low.

If the CNC controls the second spindle S2MAIN is high.

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) 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.

DM45 (M5548)

The CNC sets this signal high to tell the PLC that the miscellaneous function M45 has been programmed and, therefore, the "auxiliary spindle or live tool" is active.

TANGACT (M5558)

It indicates that the tangential control function, G45, is active.

SYNCPOSI (M5559)

Indicates that the spindles are synchronized in position (set high). In other words, that the second spindle follows behind the main spindle at an angular distance set by G30.

It goes low when the following error between them exceeds the maximum allowed by s.m.p. "SYNPOSOF (P53)".

SYNSPEED (M5560)

Indicates that the spindles are synchronized in speed (set high). In other words, that the second spindle turns at the same speed as the main spindle.

It goes low when the following error between them exceeds the maximum allowed by s.m.p. SYNSPEOF (P54).

SYNCHRON (M5561)

Indicates that the G77S function is currently selected (spindle synchronization).

SERPLCAC (M5562)

It is used in the data exchange, via Sercos between the CNC and the drives.

The CNC sets this signal high to "tell" the PLC that the requested change of parameter sets and gear ratios is in progress.

While this mark is on, no other change may be requested because the command would be lost.

RETRACT (M5567)

During the 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 is completed, the user may finish the hole, go to the next hole or go into tool inspection.

General logic output RETRACT (M5567) is activated when the stop is completed and it is canceled when completing the retraction of the drilling or mill type threading operation.

When retracting the axes on the lathe model, general logic output RETRACT (M5567) is activated when pressing [STOP] and the CNC starts retracting. This mark will stay active until the withdrawal distances set in G233 are reached.

TMINEM (M5569)

Mark activated when the CNC detects an error during tool change. This mark stays memorized until it is canceled by the RESTMEM mark or with the option [REMOVE ERROR] shown in the error message.

READEND (M5507)

CNC output that indicates that the message received is in the buffer.

Value	Meaning
0	The execution of the RCV instruction begins.
1	It indicates that the process started by the RCV instruction has finished and that the message is in the buffer.

Default value 0.

READRDY (M5571)

CNC output that indicates whether the READ commands may be received or not.

Value	Meaning
0	It indicates that no READ commands may be received.
1	It indicates that READ commands may be received.

WRITERDY (M5572)

CNC output that indicates whether the WRITE commands may be sent or not.

Value	Meaning
0	It indicates that no WRITE commands may be sent.
1	It indicates that WRITE commands may be sent.

12.

LOGIC CNC INPUTS AND OUTPUTS

General logic outputs

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

12.8 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 7 (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.

Mnemonics using numbers 1 through 7.

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, Y, Z, B, C and U axis, the order will be: X, Y, Z, U, B, C 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:
ENABLE4, DIR4, REFPOIN4, INPOS4	for the U axis:
ENABLE5, DIR5, REFPOIN5, INPOS5	for the B axis
ENABLE6, DIR6, REFPOIN6, INPOS6	for the C 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)** **ENABLE4 (M5750)**
ENABLE5 (M5800) **ENABLE6 (M5850)** **ENABLE7 (M5900)**

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)** **DIR4 (M5751)**
DIR5 (M5801) **DIR6 (M5851)** **DIR7 (M5901)**

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)** **REFPOIN4 (M5752)**
REFPOIN5 (M5802) **REFPOIN6 (M5852)** **REFPOIN7 (M5902)**

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 losing feedback count via Sercos due to broken communication. Difference greater than 10 microns (0.00039 inches) or 0.01º
- 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)** **DRSTAF4 (M5753)** **DRSTAS4 (M5754)**
DRSTAF5 (M5803) **DRSTAS5 (M5804)** **DRSTAF6 (M5853)** **DRSTAS6 (M5854)**
DRSTAF7 (M5903) **DRSTAS7 (M5904)** **DRSTAF8 (M6603)** **DRSTAS8 (M6604)**
DRSTAF9 (M6653) **DRSTAS9 (M6654)**

The CNC uses these signals when communicating with the drive via Sercos or via Can and indicate the status of the drive.

	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).

MAXDIFF1 (M5605) **MAXDIFF2 (M5655)** **MAXDIFF3 (M5705)** **MAXDIFF4 (M5755)**
MAXDIFF5 (M5805) **MAXDIFF6 (M5855)** **MAXDIFF7 (M5905)**

These marks are activated when 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 both axes of a Gantry pair.

This way, the PLC can issue a warning indicating that the position difference between the master and the slave has not been compensated for.

ANT1 (M5606) **ANT2 (M5656)** **ANT3 (M5706)** **ANT4 (M5756)**
ANT5 (M5806) **ANT6 (M5856)** **ANT7 (M5906)**

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)** **INPOS4 (M5757)**
INPOS5 (M5807) **INPOS6 (M5857)** **INPOS7 (M5907)**

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.

12.9 Spindle logic outputs.

This CNC can handle 2 spindles: a main spindle and a second spindle. They both can be operative simultaneously, but only one can be controlled at a time. This selection can be made via part-program by means of functions G28 and G29.

ENABLES (M5950) **Main spindle**
ENABLES2 (M5975) **Second 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**
DIRS2 (M5976) **Second 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**
REFPOIS2 (M5977) **Second 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) **DRSTASS (M5954)** **Main spindle**
DRSTAFS2 (M5978) **DRSTASS2 (M579)** **Second spindle**

The CNC uses these signals when communicating with the drive via Sercos or via Can and indicate the status of the drive.

	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).

CAXIS (M5955) **Main spindle**
CAXIS2 (M5980) **Second spindle**

This signal is used when working with the spindle as "C" axis (G15). The CNC only considers the signals for the currently selected spindle.

The CNC sets this signal high to tell the PLC that the C axis is active.

REVOK (M5956) **Main spindle**
REVOK2 (M5981) **Second 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**
INPOSS2 (M5982) **Second 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.

12.10 Logic outputs of the auxiliary spindle

DRSTAFAS (M5557) DRSTASAS (M5556)

The CNC uses these signals when communicating with the drive via Sercos or via Can and indicate the status of the drive.

	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).

12.11 Logic outputs of key status

KEYDIS1 (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.

The appendix of this manual shows, for each keyboard, the code for each key. See *"Logic outputs of key status"* on page 665.

ACCESS TO INTERNAL CNC VARIABLES

13

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

13.1 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.

13.2 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, G59 and G159.

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 axes X, Y, Z, U and B, it only allows the variables ORGX, ORGY, ORGZ,. ORGU and ORGB 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.

13.3 Variables associated with function G49

With function G49, it is possible to define a coordinate transformation or, in other words, the inclined plane resulting from that transformation.

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 associated with the definition of function G49

ORGROX **ORGROY** **ORGROZ**

X coordinate of the new part zero with respect to home.

Y coordinate of the new part zero with respect to home.

Z coordinate of the new part zero with respect to home.

ORGROA **ORGROB** **ORGROC** **ORGROI**
ORGROJ **ORGROK** **ORGROQ** **ORGROR**
ORGROS

Value assigned to parameter A.

Value assigned to parameter B.

Value assigned to parameter C.

Value assigned to parameter I.

Value assigned to parameter J.

Value assigned to parameter K.

Value assigned to parameter Q.

Value assigned to parameter R.

Value assigned to parameter S.

GTRATY

Type of G49 programmed.

0 = no G49 has been defined.

1 = Type G49 X Y Z A B C

2 = Type G49 X Y Z Q R S

3 = Type G49 T X Y Z S

4 = Type G49 X Y Z I J K R S

Every time G49 is programmed, the CNC updates the values of the parameters that have been defined.

For example when programming G49 XYZ ABC, the CNC updates the variables.

ORGROX, ORGROY, ORGROZ

ORGROA, ORGROB, ORGROC

The rest of variables keep their previous values.

Read-write variables updated by the CNC once function G49 is executed

Accessing the variables TOOROF or TOOROS interrupts block preparation and the CNC waits for that command to be executed before resuming block preparation.

When having a swivel or angular spindle, general machine parameter XFORM (P93) with a value of 2 or 3, the CNC shows the following data:

TOOROF

It indicates the position to be occupied by the spindle's main rotary axis in order to orient the spindle perpendicular to the indicated inclined plane.

TOOROS

It indicates the position to be occupied by the spindle's secondary rotary axis in order to orient the spindle perpendicular to the indicated inclined plane.

13.4 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.

MPSSn

Returns the value assigned to the indicated machine parameter (n) of the second spindle.

MPASn

Returns the value of the indicated machine parameter (n) of the auxiliary spindle.

MPLCn

Returns the value assigned to the indicated machine parameter (n) of the PLC.

13.5 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).

13.6 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.

13.7 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), FLWE(X-C), DEFLEX, DEFLEY or DEFLEZ 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.

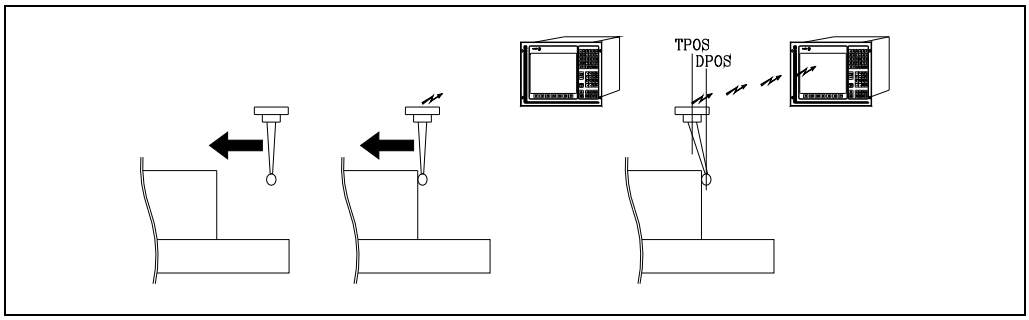
At the lathe model CNC, the coordinates of each axis are given as follows:

- 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 whenever probing operations G75, G76 and probing cycles PROBE and DIGIT are carried out.

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.

DEFLEX **DEFLEY** **DEFLEZ**

These variables can only be used at the mill model. They return the amount of deflection obtained at the time by the Renishaw probe SP2 on each axis X, Y, Z.

DPLY(X-C)

It returns the position value (coordinate) shown on the screen for the selected axis.

DRPO(X-C)

It returns the position indicated by the Sercos drive of the selected axis (variable PV51 or PV53 of the drive).

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).

13.8 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. It is irrelevant to have the handwheel connected to the feedback inputs or to the PLC inputs.

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

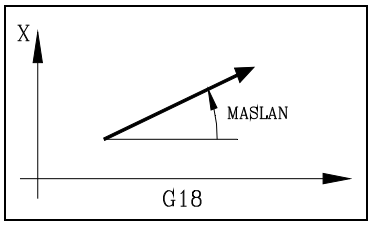
The HBE handwheel has priority. That is, regardless of the mode selected at the CNC switch (continuous or incremental JOG, handwheel), HBEVAR is set to other than "0", the CNC goes into handwheel mode.

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 *"7.15 Fagor handwheels: HBA, HBE and LGB"* on page 370.

MASLAN

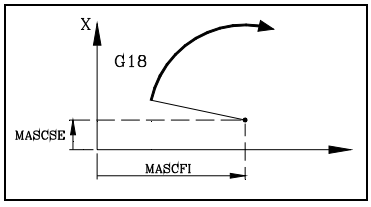
It must be used when the path-handwheel or the path-jog is selected.



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.

13.9 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.

SASINS

"A" signal of the CNC sinusoidal feedback for the second spindle.

SBSINS

"B" signal of the CNC sinusoidal feedback for the second spindle.

13.10 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

Indicates the spindle travel considering the ±214748.3647 limit of the variable. Once the maximum is exceeded, it starts adding from the minimum.

RPOSS

It returns the real position of the main spindle. Its value is given in 0.0001 degree units (between -360º 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.

DRPOS

Position indicated by the Sercos drive of the main spindle.

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.

SYNCER

Amount of lag of the second spindle following the main spindle when they are both synchronized. Its value is given in 0.0001 degree units within ±999999999º.

If the error is smaller than the maximum allowed by s.m.p. SYNPOSOF (P53) for the main spindle and the general output SYNCPOSI (M5559) is set to "1".

13.11 Variables associated with the second spindle

Variables associated with the real speed

SSREAL

It returns the actual (real) turning speed of the second spindle. Its value is given in 0.0001 rpm units.

SFTEOS

It returns the theoretical turning speed of the second spindle.

Variables associated with spindle speed

The variable SPLCS is a read-write variable, the rest are read-only.

SSPEED

It returns the turning speed of the second 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.

SDNCS

It returns the spindle speed selected via DNC. If it has a value of 0 it means that it is not selected.

SPLCS

It returns the spindle speed limit selected via PLC. If it has a value of 0 it means that it is not selected.

SPRGS

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)

The variable SPLCCSS is a read-write variable, the rest are read-only.

SCSS

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).

SDNCCS

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.

SPLCCS

It returns the constant surface speed selected by PLC. Its value is given in m/min or ft/min and it is 0 it means that it is not currently selected.

SPRGCS

It returns the constant surface speed selected by program. Its value is given in m/min or ft/min and it is 0 it means that it is not currently selected.

Variables associated with the spindle override

The variable SPLCSSO is a read-write variable, the rest are read-only.

SSSO

It returns the turning speed override (%) of the second spindle currently selected at the CNC. It is given in integer values between 0 and "MAXFOVR" (maximum 255).

This turning speed percentage of the second 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.

SPRGSO

This variable may be used to read or modify the speed override percentage of the second spindle currently selected by program. If it has a value of 0 it means that it is not selected.

SDNCSO

It returns the turning speed override % of the second spindle currently selected via DNC. If it has a value of 0 it means that it is not selected.

SPLCSO

It returns the turning speed override % of the second spindle currently selected by PLC. If it has a value of 0 it means that it is not selected.

SCNCSO

It returns the turning speed override % of the second spindle currently selected from the front panel.

Speed limit related variables

The variable PLCSL is a read-write variable, the rest are read-only.

SSLIMI

It returns the value set in rpm at the CNC for the turning speed limit of the second 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.

SDNCSL

It returns the speed limit of the second spindle in rpm currently selected via DNC. If it has a value of 0 it means that it is not selected.

SPLCSL

It returns the speed limit of the second spindle in rpm currently selected by PLC. If it has a value of 0 it means that it is not selected.

SPRGSL

It returns the speed limit of the second spindle in rpm currently selected by program.

Position related read-only variables

SPOSS

It returns the real position of the second spindle. Its value is given in 0.0001 degree units within ±999999999º.

SRPOSS

It returns the real position of the second spindle. Its value is given in 0.0001 degree units (between -360º and 360º).

STPOSS

It returns the theoretical position of the second spindle (real position + lag). Its value is given in 0.0001 degree units within ±999999999º.

SRTPOS

It returns the theoretical position of the second spindle (real position + lag). Its value is given in 0.0001 degree units within 0 and 360º.

SDRPOS

Position indicated by the Sercos drive of the second spindle.

SPRGSP

Position programmed in M19 via program for the second 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)

SFLWES

Following error (lag) of the second spindle. Its value is given in 0.0001 degree units within ±999999999º.

13.12 Variables associated with the live tool

Read-only variables

ASPROG

It must be used inside the subroutine associated with function M45.

It returns the rpm programmed by M45 S. When programming M45 alone, the variable takes the value of "0".

The ASPROG variable is updated just before executing the M45 function so it is already updated when executing its associated subroutine.

LIVRPM

It must be used when operating in TC mode.

It returns the rpm selected by the user for the live tool when in TC mode.

13.13 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.
```

13.14 Sercos variables

They are used in the data exchange via Sercos between the CNC and the drives.

Write variables

SETGE(X-C) **SETGES** **SSETGS**

The drive may have up to 8 gear ratios (0 through 7). Sercos identifier 218, GearRatioPreselection.

It may also have up to 8 parameter sets (0 through 7). Sercos identifier 217, ParameterSetPreselection.

With these variables the work range or gear ratio and the parameter set of each drive may be modified.

SETGE(X-C) ... for the axes,

SETGES ... for the main spindle.

SSETGS ... for the second spindle.

The 4 least significant bits of these variables indicate the gear ratio and the other 4 the parameter set to be selected.

Since it takes the drive some time to change the parameter set and the gear ratios, mark SERPLCAC (M5562) will remain active from when the change is requested until the drive assumes the new values. While this mark is on, no other change may be requested because the command would be lost.

13.15 Software & hardware configuration variables

Read-only variables

HARCON

It indicates, with bits, the CNC's hardware configuration. The bit will be "1" when the relevant configuration is available.

CNC8055 model:

Bit	Meaning	
4,3,2,1	0000 0010	8055 FL model. 8055 Power model.
5	Sercos integrated into CPU board.	
6	Sercos module on manager board.	
7	Axes module.	
10,9,8	001 010 011 100	One I/O module. Two I/O modules. Three I/O modules. Four I/O modules.
14	It has analog video.	
15	It has CAN integrated into the CPU board.	
18,17,16	Keyboard type (technical service department).	
20.19	CPU type (technical service department).	
23,22,21	1xx	CPU PPC5200.
26,25,24	000 001	Color LCD Monitor. Monochrome LCD monitor.
30	Ethernet connector integrated in the CPU.	
31	Compact flash memory (KeyCF).	

CNC8055i model:

Bit	Meaning	
4, 3, 2, 1	0100 0110	8055i FL model. 8055i Power model.
5	Sercos (digital model).	
6	Reserved.	
9, 8, 7	000 001 010 011	Expansion board missing. "Feedback + I/O" expansion board. Feedback-only expansion board. I/O-only expansion board.
	101 110 111	"Axes 2" board for expansion of "feedback + I/O". "Axes 2" board for expansion of feedback only. "Axes 2" board for expansion of I/O only.
10	Axis board with 12-bit (=0) or 16-bit (=1) Digital/Analog converter.	
12, 11	Reserved.	
14, 13	Reserved.	
15	It has CAN (digital module).	
18,17,16	Keyboard type (technical service department).	
20.19	CPU type (technical service department).	
23,22,21	1xx	CPU PPC5200.
26,25,24	000 001	Color LCD Monitor. Monochrome LCD monitor.
30	Ethernet..	
31	Compact flash memory (KeyCF).	

HARCOA

It indicates, with bits, the CNC's hardware configuration. The bit will be "1" when the relevant configuration is available.

CNC8055 model:

Bit	Meaning
0	Axis module 2.
1	It has a connector for compact flash.
10	The axis board is "Axis module SB" Note: Bit 0 of HARCOA must be set to 0.

Bit ·1· only indicates whether the hardware has a connector for the compact flash or not, it does not indicate whether the compact flash is inserted or not.

CNC8055i model:

Bit	Meaning
0	"Axes 2" board.
1	It has a connector for compact flash.
10	The axis board is "Axis module SB" Note: Bit 0 of HARCOA must be set to 0.

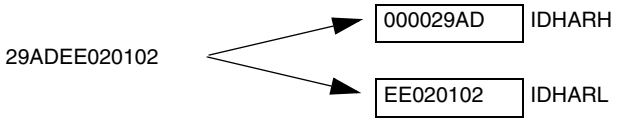
Bit ·1· only indicates whether the hardware has a connector for the compact flash or not, it does not indicate whether the compact flash is inserted or not.

IDHARH IDHARL

They return, in BCD code, the hardware identification number corresponding to the KeyCF. It is the number appearing on the software diagnosis screen.

Since the identification number has 12 digits, the IDHARL variable shows the 8 least significant bits and the IDHARH the 4 most significant bits.

Example:

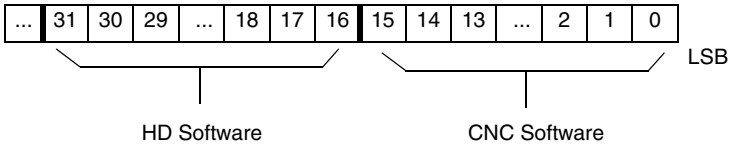


SOFCON

They return the software version numbers for the CNC and the hard disk.

Bits 15-0 return the CNC software version (4 digits)

Bits 31-16 return the software version of the hard disk (HD) (4 digits)



For example, SOFCON 01010311 indicates:

Hard disk (HD) software version	0101
CNC software version	0311

HDMEGA

Hard disk size (in megabytes).

KEYIDE

Keyboard code, according to the auto-identification system.

KEYIDE	CUSTOMTY (P92)	Keyboard
0	- - -	Keyboard without auto-identification.
130	254	Mill model keyboard.
131	254	Lathe model keyboard.
132	254	Conversational mill model keyboard.
133	254	Conversational lathe model keyboard.
134	254	Educational model keyboard.

MODEL

It identifies the CNC model, mill or lather.

Value	Meaning
0	Mill model
1	Lathe model

This variable may be read from the CNC, from the PLC and from DNC.

13.16 Variables associated with telediagnosis

Read-only variables

HARSWA HARSWB

They return, in 4 bits, the central unit configuration, a value of "1" if it is present and "0" if not. Logic address (device select) set on each board with the dip-switches (see installation manual).

HARSWA

Bits	Board
31 - 28	Large sercos
27 - 24	I/O 4
23 - 20	I/O 3
19 - 16	I/O 2
15 - 12	I/O 1
11- 8	Axes:
7 -4	
3 - 0 (LSB)	CPU

HARSWB

Bits	Board
31 - 28	
27 - 24	
23 - 20	Type of CAN at COM1.
19 - 16	Type of CAN at COM2.
15 - 12	0 CAN board missing. 1 CAN board in COM1. 2 CAN board in COM2. 3 Board in both COM.
11- 8	Small sercos
7 -4	
3 - 0 (LSB)	HD

The CPU board must be present in all the configurations and set to "0". In the rest of cases, it returns a "0" if the board is missing.

There could be a large Sercos card (that takes up the whole module) or the small board that is installed in the CPU module.

There may be two types of CAN boards (value of ·0001· if it is an SJ1000 type and value of ·0010· if it is an OKI9225 type).

HARTST

It returns the result of the hardware test. The data comes at the least significant bits with a "1" if it failed and with a "0" if OK or if the relevant board is missing.

Bits		
14	24V test of IO4 module	
13	Inside temperature	
12	I/O 3	(Board voltage)
11	I/O 2	(Board voltage)
10	I/O 1	(Board voltage)
8	Axes:	(Board voltage)
7	+3.3 V	(Power supply)
6	GND	(Power supply)
5	GNDA	(Power supply)
4	- 15 V	(Power supply)
3	+ 15 V	(Power supply)
2	Battery	(Power supply)
1	- 5 V	(Power supply)
0 (LSB)	+ 5 V	(Power supply)

MEMTST

It returns the result of the memory test. Each data uses 4 bits. The text is successful if the least significant bit of each data is set to 1. There is an error if the least significant bit of any data is other than 1.

Bits	Test
30	Test status
...	...
...	...
19 - 16	Caché

Bits	Test
15 - 12	Sdram
11- 8	HD
7 -4	Flash
3 - 0 (LSB)	Ram

Bit 30 stays at "1" during the test.

NODE

It returns the number of the node used to configure the CNC in the Sercos ring.

VCHECK

It returns the checksum of the code for the software version installed. It is the value appearing on the code test.

IONODE

It returns in 16 bits the position of the "ADDRESS" switch of the CAN of the I/O. If it is not connected, it returns the value 0xFFFF.

IOSLOC

They may be used to read the number of local digital I/O available.

Bit	Meaning
0 - 15	Number of inputs.
16 - 31	Number of outputs.

IOSREM

They may be used to read the number of remote digital I/O available.

Bit	Meaning
0 - 15	Number of inputs.
16 - 31	Number of outputs.

13.17 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.
34 = Profile 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 port machine parameter tables.
104 = PLC machine parameter table.
105 = M function table.
106 = Leadscrew error compensation tables and cross compensation tables.
107 = Machine parameter table for Ethernet.

110 = Diagnosis: configuration.
111 = Diagnosis: hardware test.
112 = Diagnosis: RAM memory test.
113 = Diagnosis: Flash memory test.
114 = User diagnosis.
115 = Hard disk diagnosis (HD).
116 = Circle geometry test.
117 = Oscilloscope.
120 = DERGAIN auto-adjustment.

Read-only variables related to the conversational mode (MC, TC, MCO, TCO) and configurable mode M, T ([SHIFT]-[ESC])

In these work modes, it is recommended to use variables OPMODA, OPMODB and OPMODC. The OPMODE variable is generic and contains different values to those of the standard mode.

OPMODE

It returns the code corresponding to the selected operating mode.

0 = CNC starting up.

10 = In execution mode.

In execution or waiting for the [CYCLE START] key (drawing of the [CYCLE START] on top).

12 = Indicates one of the following situations:

- In MDI mode, when pressing the ISO key from the jog mode or tool inspection mode.

- One of the following fields of the main screen that admits the START key has been selected: Axes, T, F or S.

21 = In graphic simulation mode.

30 = Cycle editing.

40 = In Jog mode (standard screen).

43 = Doing a home search.

45 = In tool calibration mode.

60 = Managing parts. PPROG mode.

OPMODA

Indicates the operating mode currently selected when working with the main channel.

Use the OPMODE variable to know at any time the selected operating mode (main channel, user channel, PLC channel).

This information is given at the least significant bits with a "1" when active and with a "0" when not active or when it is not available in the current version.

Bit 0	Program in execution.
Bit 1	Program in simulation.
Bit 2	Block in execution via MDI, JOG.
Bit 3	Repositioning in progress.
Bit 4	Program interrupted, by CYCLE STOP.
Bit 5	MDI, JOG Block interrupted.
Bit 6	Repositioning interrupted.
Bit 7	In tool inspection.
Bit 8	Block in execution via CNCEX1.
Bit 9	Block via CNCEX1 interrupted.
Bit 10	CNC ready to accept JOG movements: jog, handwheel, teach-in, inspection.
Bit 11	CNC ready to receive the CYCLE START command: execution, simulation and MDI modes.
Bit 12	The CNC is not ready to execute anything involving axis or spindle movement.
Bit 13	It identifies the block search.

OPMODB

Indicates the type of simulation currently selected. This information is given at the least significant bits indicating with a "1" the one currently selected.

Bit 0	Theoretical path.
Bit 1	G functions.
Bit 2	G M S T functions.
Bit 3	Main plane.
Bit 4	Rapid.
Bit 5	Rapid (S=0).

OPMODC

Indicates the axes selected by handwheel. This information is given at the least significant bits indicating with a "1" the one currently selected.

Bit 0	Axis 1.
Bit 1	Axis 2.
Bit 2	Axis 3.
Bit 3	Axis 4.
Bit 4	Axis 5.
Bit 5	Axis 6.
Bit 6	Axis 7.
Bit 7	
Bit 8	

The axis name corresponds to the number according to the programming order for them.

Example: If the CNC controls axes X, Y, Z, U, B, C, axis 1=X, axis2=Y, axis3=Z, axis4=U, axis5=B, axis6=C.

13.18 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 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.

G74	G73	G72	G71	G70	...	G54	G53	G52	G51	G50

GGSD

It returns the status of functions G5 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 G75 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

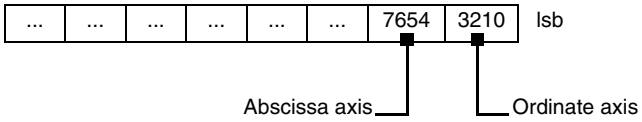
GGSM

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,Z,U,B,C 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) => 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, Z, U, B, C axes and the U axis is selected.

(CNCRD LONGAX, R22, M34) assigns the value of 4 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
		Axis 7	Axis 6	Axis 5	Axis 4	Axis 3	Axis 2	Axis 1

LSB

The axis name corresponds to the number according to the programming order for them.

Example: If the CNC controls axes X, Y, Z, U, B, C, axis 1=X, axis2=Y, axis3=Z, axis4=U, axis5=B, axis6=C.

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 of the indicated analog input (n). The value is given in tenth-thousandths of a volt.

- At the –Axes– module, it is possible to select one of the 8 analog inputs (1··8) available. The values returned will be within the ±5 V range.
- At the –Vpp Axes– module, it is possible to select one of the 4 analog inputs (1··4) available. The returned values will be either within the ±5 V or ±10 V range depending on how plc.m.p. IANA5V (P130) has been set.
- In the case of analog CAN I/O, the value assigned will be expressed in 0.0001 volt units and within ±10 V. It is possible to select one of the 16 analog inputs (1··16) available. See chapter *"4 Remote modules (bus CAN with CanOPEN protocol)"*.

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".

AXICOM

It returns in the 3 least significant bits the axis pairs switched with function G28.

		Pair 3		Pair 2		Pair 1		
		Axis 2	Axis 1	Axis 2	Axis 1	Axis 2	Axis 1	LSB

The axes are coded in 4 bits and indicate the axis number (1 to 7) according to their programming order.

If the CNC controls the X, Y, Z, B, C axes and G28BC has been programmed, the AXICOM variable will show:

		Pair 3		Pair 2		Pair 1		
						C	B	
		0000	0000	0000	0000	0101	0100	LSB

TANGAN

Variable associated with the tangential control function, G45. It indicates the programmed angular position.

TPIOUT(X-C)

Output of the PI of the tandem master axis in rpm.

DNCSTA

DNC transmission status, even when not having this option.

There is on bit for each DNC that will be set to ·1· when a transmission is in progress.

Bit	Meaning
1	Transmission in progress through DNC1.
8	Transmission in progress through DNC2.

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.

TIPPRB

It indicates the PROBE cycle being executed at the CNC.

If PROBE1 is running, the TPPRB variable takes the value of 1, if PROBE2 is running, it takes the value of 2, etc. if PROBE12 is running, it takes the value of 12.

PANEDI

WINDRAW55 application. Number of the screen created by the user or the manufacturer and is being consulted.

DATEDI

WINDRAW55 application. Number of the element (item) being consulted.

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.

TEMPIn

It returns the temperature in tenths of a degree detected by the PT100. It is possible to select one of the 4 temperature inputs (1··4) available.

FBDIF(X-C)

Variable that can be read from the CNC, PLC and DNC for monitoring the difference between the position value of the first and second feedbacks on the oscilloscope with Fagor digital servo system.

If the feedback difference exceeds the value set in a.m.p. FBACKDIF (P100) the CNC will issue the corresponding 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. If both feedbacks are connected to the drive, the CNC does not show the difference between them, but is issues the corresponding error message.

CYCLEV

Variable that can be read from CNC, PLC and DNC and indicates (on the conversational model) the tab number that is being displayed at all times, beginning from 0, when navigating (browsing) through the cycles. It takes the value of -1 when not navigating (browsing) through the cycles.

CYCEDI

Variable that may be read from CNC, PLC and DNC that indicates (on the conversational model) the cycle or screen number according to the following list:

Screen	CYCEDI
Standard screen	100
Auxiliary screen	101

Cycle (milling model ·MC·)	CYCEDI
Multiple positioning in several locations	1
Multiple positioning in a straight line	2
Multiple positioning in an arc 1	3
Multiple positioning in a rectangular pattern	4
Multiple positioning in a grid pattern	5
Multiple positioning in an arc 2	6
Surface milling cycle	7
Rectangular boss cycle	8
Circular boss cycle	9
Profile milling cycle 1	10
Profile milling cycle 2	11
Circular pocket cycle 1	12
Positioning cycle 1	13
Drilling cycle 1	14
Drilling cycle 2	15
Simple pocket cycle	16
Positioning cycle 2	17
Pocket cycle with a 2D profile	18
Pocket cycle with a 3D profile	19
Center punching cycle	20
Threading cycle	21

Cycle (lathe model ·TC·)	CYCEDI
Rounding cycle 1	1
Rounding cycle 2	2
Turning cycle 1	3
Facing cycle 1	4
Taper turning cycle 1	5
Taper turning cycle 2	6
Threading cycle 1	7
Threading cycle 2	8
Threading cycle 3	9
Profiling cycle 1	10
Profiling cycle 2	11
Grooving cycle 1	12
Positioning cycle 1	13
Drilling cycle	14
Tapping cycle	15
Grooving cycle 2	16
Positioning cycle 2	17
Grooving cycle 3	18
Grooving cycle 4	19
Multiple drilling cycle	20
Multiple tapping cycle	21

Cycle (milling model ·MC·)	CYCEDI
Reaming cycle	22
Boring cycle 1	23
ISO block editing cycle	24
Rectangular pocket cycle	25
Circular pocket cycle 2	26
Drilling cycle 3	27
Grooving cycle	28
Boring cycle 2	29
Probe 1 cycle	30
Zero offsets and presets cycle	31
Work zones cycle	32
Labels and jumps cycle	33
Mirror image cycle	34
Scaling factor cycle	35
Pattern rotation	36
Part centering cycle	37
Plane changing cycle	38
Probe calibration cycle	39
Thread milling cycle	40
Bore milling cycle	41
Tool measuring cycle	-301
Manual part centering cycle	-302

Cycle (lathe model ·TC·)	CYCEDI
Multiple slot milling cycle	22
Thread repair cycle	23
ISO block editing cycle	24
ZC profile cycle	25
XC profile cycle	26
Taper turning cycle 3	27
Turning cycle 2	28
Cut-off cycle	29
Multi-start (entry) threading cycle	30
Facing cycle 2	31
Probe 1 cycle	32
Probe 2 cycle	33
Part zero offset cycle	34
Work zones cycle	35
Labels and jumps cycle	36
Mirror image cycle	37
Scaling factor cycle	38
ZC / YZ rectangular pocket cycle	39
ZC / YZ circular pocket cycle	40
XC / XY rectangular pocket cycle	41
XC / XY circular pocket cycle	42
Pocket cycle with a ZC / YZ profile	43
Pocket cycle with an XC / XY profile	44
Tool measuring cycle	-301

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.

MIXPO(X..C)

Variable that indicates the axis position with the combined feedback. This variable may be read from the CNC, PLC and DNC and may be viewed at the oscilloscope.

FLWAC(X..C)

Variable that indicates the real error considering the second feedback. This variable may be used to control the INPOS(X..C) signals, but the position loop is not closed with it when feedback combination is active.

When feedback combination is not active, the value of the FLWAC(X..C) variable is the same as that of the FLWE(X..C) variable.

This variable may be read from the CNC, PLC and DNC and may be viewed at the oscilloscope.

RADIOC

Variable that indicates the programmed radius value when executing a G15 R. This variable can only be read from the CNC.

RCVSTAT

Variable that indicates reception status of an open communication. It has 7 bits that are counted from right (LSB) to left (MSB). This variable may be read from the CNC, from the PLC and from DNC.

The following table shows the meaning of each bit:

Bit	Meaning
0	Not being used.
1	(Value) End of reception Buffer full.
2	(Value 1) Timer RCVENDTI (P12) time exceeded, if this mode has been chosen for receiving messages.
3 ... 5	Not being used.
6	(Value 1) End of reception with error.
7	(Value 1) End of message reception if the PLC has sent a disable with bit 7 of variable RCVCTRL.

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 sixteen (1 through 16) available at the CNC may be modified, the corresponding error being displayed if an attempt is made to write in one which is occupied.

When setting an axis as a DRO axis from the PLC, the analog output becomes free so it can be written from the CNC or from the PLC.

In the case of analog CAN I/O: See chapter *"4 Remote modules (bus CAN with CanOPEN protocol)"*.

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.

PRBMOD=0	It issues an error message (default value).
PRBMOD=1	No error message is issued.

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 adjust the following error of the longitudinal axis so the displayed value of the "RIGIER" variable is as small as possible.

RCVCTRL

Variable used to control reception. It has 7 bits that are counted from right (LSB) to left (MSB). This variable may be read and written from the CNC, from the PLC and from DNC.

The following table shows the meaning of each bit:

Bit	Meaning
0, 1	Not being used.
2	(Value 0) End reception if RCVENDTI (P12) is exceeded. (Value 1) Ignore the timer. This bit checks this bit every time the = READ instruction is executed.
3 ... 6	Not being used.
7	(Valor 0) Message reception enabled. (Valor 1) Message reception disabled. It is tested at the beginning of the PLC cycle.

PLCAS

This variable may read and written from the PLC and it may be used to change the value of the auxiliary spindle speed. The sign indicates the turning direction.

AXES CONTROLLED FROM THE PLC

14

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.

14.1 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.

14.1.1 Considerations

Screen editor

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.

14.1.2 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), 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, M44 and M45.

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.

14.1.3 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.
```

14.2 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.

14.3 Synchronize a PLC axis with a CNC axis

The synchronization is carried out from the PLC. Previously, to synchronize an axis of the PLC channel with another one of the CNC channel (main channel), set a.m.p. SYNCHRO (P3) of the PLC axis indicating which axis it must synchronize with.

Axis synchronization is carried out from the PLC by activating the general signal SYNCHRO of the axis to be coupled as slave (PLC axis). To end synchronization, cancel the general signal SYNCHRO of the PLC axis.

If an error occurs canceling the ENABLE logic outputs of all the axes, it also cancels the synchronization.

How to carry out and end the synchronization

Both axes must be stopped in order to couple them. To ensure this condition, it is recommended to execute an M function at the CNC commanding the PLC to execute another M function in the PLC channel that actives the SYNCHRO signal. The M function of the main channel must not end until the PLC's M function execution is completed and the ENABLE signal of the slave axis is set high.

To assure that the PLC axis recovers its position after the synchronization, it is recommended to follow the same procedure as for coupling using other 2 special M functions, one at the CNC and another one at the PLC.

Considerations for the synchronization

Once both axes are synchronized, it won't be possible to program movements of the PLC axis.

During synchronization, it does not check whether the PLC axis gets in position or not.

- The logic output ENABLE of the PLC axis is activated (allowing motion).
- The logic output INPOS of the PLC axis is deactivated (the axis is NOT in position).
- General input INHIBIT of the PLC axis is ignored, thus not being possible to prevent it from moving.
- The execution of the movement of the synchronized slave axis cannot be aborted even by activating the general input PLCABORT.

CUSTOMIZABLE SCREENS

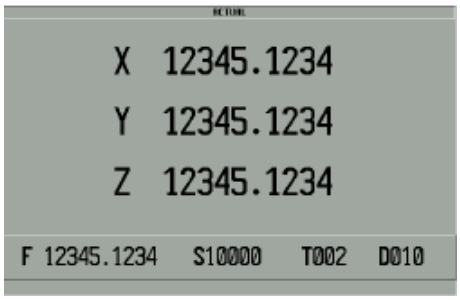
15

The machine manufacturer may customize some of the CNC screens to:

(1) display more information.



(2) display the same information but in a different way.



(3) display a completely different screen in contents and looks.



All of them use OEM screens that have been created on a PC using the Fagor WINDRAW55 application software and have been sent out to the CNC using the Fagor WinDNC application software.

- In (1) the OEM screen (consumption led bar graph) is superimposed on the standard CNC screen.
- In (2) the upper area corresponds to the standard screen and the lower area shows the part of the machine manufacturer (OEM).
- In (3) the OEM screen replaces completely the standard CNC screen.

Use the configuration file at the CNC to define how the screens are to be laid out and which values must be displayed on the OEM screen.

15.1 Configuration file

It is a program which describes the operating characteristics of the graphic elements of the screen.

Set g.m.p. "CFGFILE (127)" with the number of the program for the configuration file:

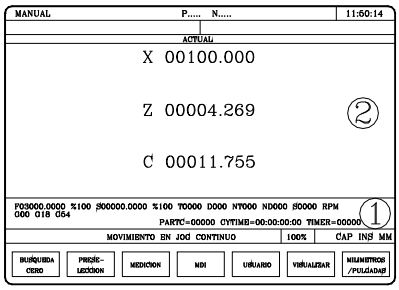
The configuration file is a CNC program edited in high level language (configuration language) which is described later on. It may be edited both at the CNC and at a PC.

It may be in the user RAM and in the hard disk (KeyCF). If it is in both places, the one located in user RAM will be used. It is recommended to store it only in the hard disk (KeyCF) once it has been fully debugged.

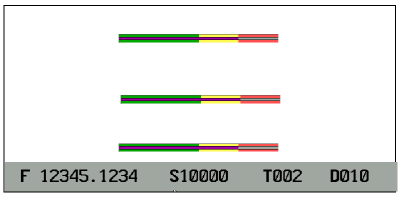
The configuration file must contain all the information regarding all the screens being customized.

When editing a screen, the CNC overlaps the OEM screen over the standard CNC screen. The DISABLE instruction of the configuration file serves to indicate which of the standard screen areas are eliminated.

Example: Standard screen + OEM screen + Disable 1

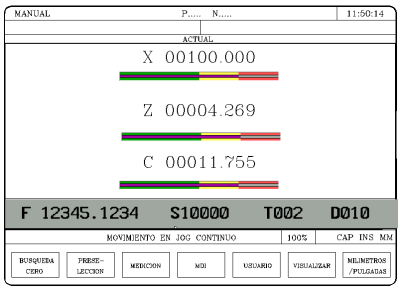


Standard screen

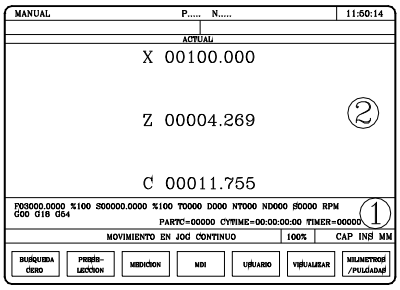


OEM screen

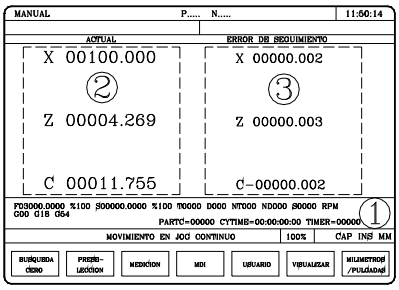
Both screens are overlapped; but "Disable 1" indicates that the area 1 of the standard screen is not displayed. Therefore:



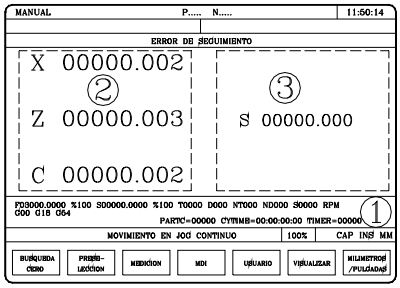
Next, all the screens that may be customized will be shown and the nomenclature to be used in the configuration file. Numbers 1, 2, 3, 4 and 5 indicate the areas each screen is divided into.

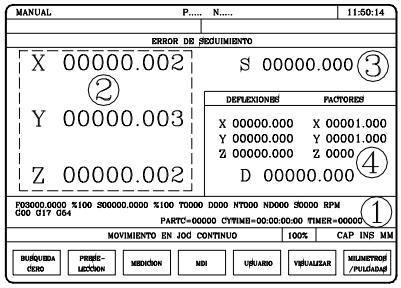


[JOG]
JOG mode - Actual

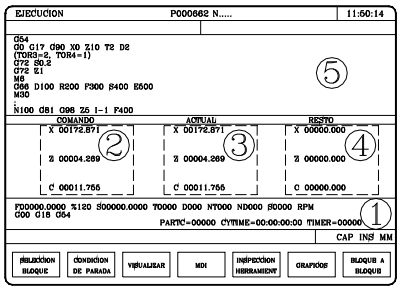


[JOGAFL] JOG mode - Actual and Following error

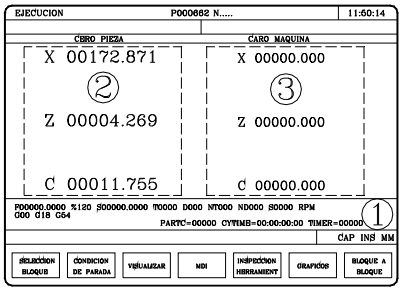




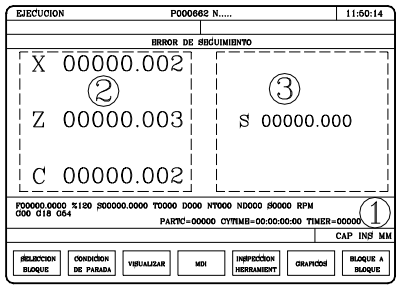
[JOGFLW] JOG mode - Following error

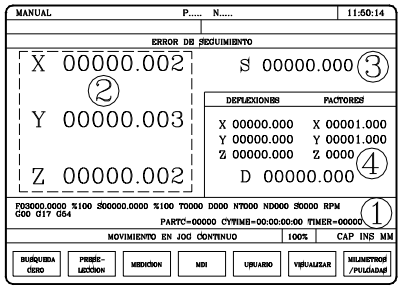


[STD]
Execution mode - Standard

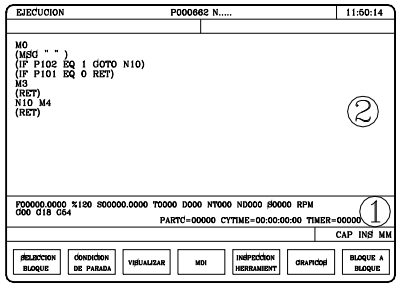


[POS]
Execution mode - Position

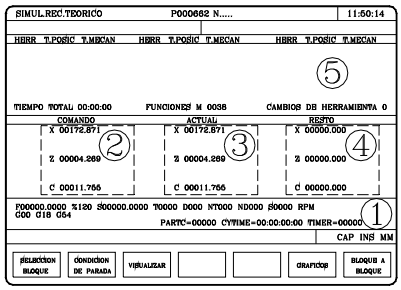




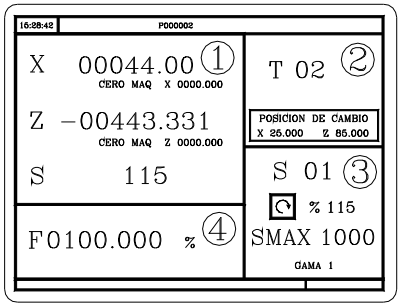
[FLW] Execution mode - Following error



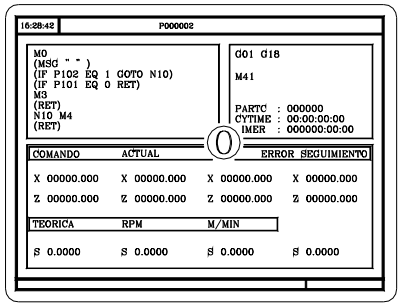
[PRG]
Execution mode - Program



[SUB]
Execution mode - Subroutines



[STDCONV]
Conversational mode - Standard



[AUXCONV]
Conversational mode - Auxiliary for execution

15.2 Configuration language

The general characteristics of the configuration language are:

- All instructions are preceded by ";" and enclosed in parenthesis.
- The comments must be alone and preceded by ";;"
- The configuration file must begin with the line ;(PRGSCRIPT 1)
- It indicates that it is a configuration file corresponding to the version being used (in this case "1")
- The configuration file should end with the line ;(END)
- While debugging the program, the ;(DEBUG) instruction should be used
- If an error occurs while checking the configuration file, the CNC will inform about it in program 999500.

The configuration language consists of:

- A series of key words or tokens.
- The names of the internal CNC variables.
- Numbers that may be associated with the previous two items.
- Various punctuation signs.

Example of a configuration file:

```
;(PRGSCRIPT 1)
;;==================================
;;   SCREEN LAYOUT
;;   Screen in JOG mode - Actual
;;==================================
;[JOG],PLCM1125
;(DEBUG)
;(DISABLE 0)
;(WGDWIN 201)
;;---------------------- X axis, coordinate, error and consumption
;(AUTOREFRESH W1=POSX)
;(AUTOREFRESH W2=FLWEX)
;(FORMAT W3,LEDBARDEC)
;(AUTOREFRESH W3=ANAI1)
;;---------------------- Z axis, position, error and consumption
;(AUTOREFRESH W4=POSZ)
;(AUTOREFRESH W5=FLWEZ)
;(FORMAT W6,LEDBARDEC)
;(AUTOREFRESH W6=ANAI2)
;;---------------------- Feedrate, F, %
;(AUTOREFRESH W7=FEED)
;(AUTOREFRESH W8=FRO)
;;---------------------- Spindle, S, Smax, %
;(AUTOREFRESH W9=SPEED)
;(AUTOREFRESH W10=SLIMIT)
;(AUTOREFRESH W11=SSO)
;;---------------------- Tool and offset (T, D)
;(AUTOREFRESH W12=TOOL)
;(AUTOREFRESH W13=TOD)
;(END)
```

15.3 Key words

;(PRGSCRIPT 1)
;(PRGSCRIPT 2)

Header of the configuration file and version used to edit it. It must always be defined.

;(PRGSCRIPT 1)	Version 1.
;(PRGSCRIPT 2)	Version 2. Includes arithmetic operations and conditional expressions.

Version 1 of the configuration language does not offer arithmetic operations or conditional expressions. In this case, the error file (999500) will show an error. The error file will only be generated if the configuration file shows the instruction ;(DEBUG).

In programs identified as ";(PRGSCRIPT 1)", the CPU handling has been optimized, so if conditional instructions or arithmetic operations are not used, we recommend to use version 1.

;[JOG],PLCM1125

Screen to be customized and condition. The screens that may be customized are:

[JOG]	JOG mode - Actual
[JOGFLW]	JOG mode - Following Error
[JOGAFL]	JOG mode - Actual and Following Error
[STD]	Execution mode - Standard
[FLW]	Execution mode - Following error
[POS]	Execution mode - Position
[PRG]	Execution mode - Program
[SUB]	Execution mode - Subroutines
[STDCONV]	Conversational mode - Standard
[AUXCONV]	Conversational mode - Auxiliary for execution

The screens may be active at all times or only when the set condition is met, Thus: Thus:

;[JOG]	Is always active
;[PRG],PLCM1000	Active with M1000=1. If M1000=0, standard screen

;(END)

End of the screen definition.

The configuration file must contain all the screens to be customized. Each screen starts with the [xxx] instruction and ends with the (END) instruction .

;(DEBUG)

It is optional. It indicates on which line program 999500 starts giving out information of the errors that have come up when debugging the configuration file.

The configuration file only debugs the portion of the selected screen. It starts with the [xxx] instruction and end with the (END) instruction.

It is recommended to program a (DEBUG) in the definition of each screen.

;(DISABLE x)

Indicates the area of the standard screen to be eliminated.

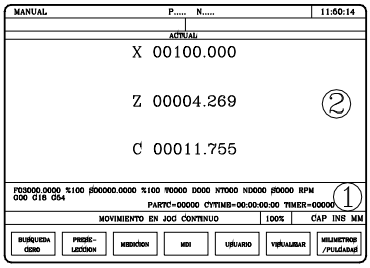
When editing a screen, the CNC overlaps the OEM screen over the standard CNC screen. The DISABLE instruction serves to indicate which areas of the standard display are to be eliminated (not displayed)

(DISABLE 1)	Eliminates area 1 of the standard display.
(DISABLE 2)	Eliminates area 2 of the standard display.

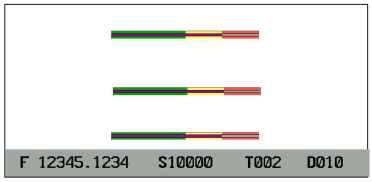
It is possible to define as many DISABLE instructions as screen areas are to be eliminated.

To eliminate the whole standard screen, program (DISABLE 0). In this case, only the OEM screen will be displayed.

Examples:



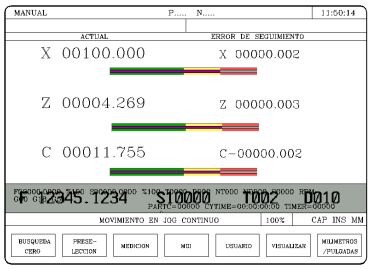
Standard screen



OEM screen

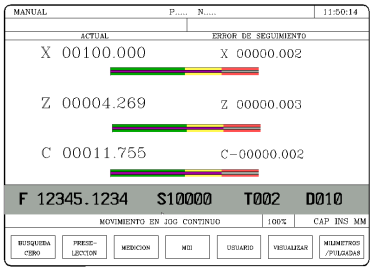
Without "Disable"

Both screens overlap.
There are areas with information overlapped.
In this case, area 1.



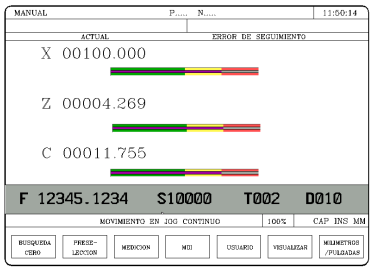
With (DISABLE 1)

Area 1 of the standard screen is not displayed.



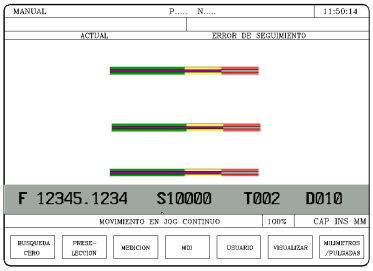
With (DISABLE 1) and (DISABLE 3)

Areas 1 and 3 of the standard display are not displayed.



With (DISABLE 0)

All areas of the standard screen are turned off
Only the OEM screen is displayed.



;(WGDWIN 201)

It must ALWAYS be defined. It indicates the number of the OEM screen to be overlapped, edited with the Fagor WINDRAW55 application software.

;(W1=GUP100)

Associates the value of a global parameter with the (W) data.

;(W2=PLCFRO)

Associates the value of a variable with the (W) data.

;(W3=PLCR127)

Associates the value of a PLC resource with the (W) data.

A register	;(W6=PLCR127)	
A mark	;(W6=PLCM1000,1)	first and how many
A group of inputs	;(W6=PLCI8,4)	first and how many
A group of outputs	;(W6=PLCO10,3)	first and how many

Associate only resources that are defined in the PLC program.

For marks, inputs and outputs, one must indicate how many of them, if none is indicated, 32 are assigned.

;(W6=PLCO11,4) Assigns the value of O11, O12, O13, O14

;(W6=PLCO11) Assigns the value of O11, O12 ... O41, O42

If a field (W) has a parameter, variable or resource associated with it, it acts as follows:

It assumes the value that its associate has when accessing the page.

- To continuously update the field value, use the (AUTOREFRESH) instruction as described later on.
- If its associate is a read-only type, the user will not be able to change the field value.
- It its associate is read/write type, the user may change the field value.
- When changing the value of the field (W), the value of its associate is also changed.
- On the other hand, when using the (AUTOREFRESH) instruction and the CNC or PLC changes the value of the associate, the value of the field is also changed.

;(AUTOREFRESH W2=FLWEX)

If (W2=FLWEX), it assigns the value of the X axis following error to the graphic element W2. This instruction updates that value periodically.

;(UNMODIFIED)

It indicates that the associated item must not pick up the editing focus.

It is programmed as a prefix of the instructions.

;(W1=GUP100)

Associates the value of a global parameter, variable or resource of the PLC with the (W1) data. The W1 item of the screen pick ups the editing focus.

;(AUTOREFRESH W6=FLWEX)

It refreshes (updates) the value of the graphic element W6 and it assigns the editing focus to it.

The resulting new instructions are:

;(UNMODIFIED W1=GUP170)

Associates the value of a global parameter, variable or resource of the PLC with the (W1) data. The W1 item of the screen pick ups the editing focus.

;(UNMODIFIED AUTOREFRESH W6=FLWEX)

It refreshes (updates) the value of the graphic element (item) W6, but it does not assign the editing focus to it.

;(FORMAT W8,LEDBARDEC)

It must be used with Ledbar type (W) data that have a decimal variable associated with them. (For example: X axis following error).

The values assigned, at the PLC, to the end and intermediate values of a LEDBAR element must be integer values and must be related to the variable associated at the CNC.

When the associated variable has a decimal format the following instruction must be used:

;(FORMAT W8,LEDBARDEC)

This instruction is used to convert coordinate values (decimal) to integer values by multiplying them by 10000

Examples:

To represent the % of axis feedrate, the FRO variable is used. The FRO values are integers (between 0 and 120) and, therefore, do not require LEDBARDEC.

;(AUTOREFRESH W9=FRO)

On the other hand, to represent the amount of following error on the X axis, the FLWEX variable is used. The values of FLWEX are not integers and, therefore, require LEDBARDEC (multiplying it by 10000) in order to make them integers.

;(FORMAT W11,LEDBARDEC)

;(AUTOREFRESH W11=FLWEX)

15.4 Arithmetic operations

Math operations are only available in version 2 of the configuration language. With math operations (arithmetic, trigonometric, logic, etc.) it is possible to assign to a "widget" element the result of several operations taking CNC variables, PLC resources, constants, etc. as operands.

A numeric value must be assigned through an internal CNC variable (GUP100 - GUP255, GUP1000 - GUP1255, GUP2000 - GUP2255). A constant (a number) cannot be assigned directly to a widget.

OPERATORS

An operator is a symbol that indicates the mathematical or logic operations to carry out. The CNC has arithmetic, relational, logic, binary, trigonometric operators and special operators.

Arithmetic operators:

+	add.	GUP100=3 + 4	GUP100=7
-	subtraction, also a negative value.	GUP101=5 - 2 GUP100= -(2 * 3)	GUP101=3 GUP103=-6
*	multiplication.	GUP104=2 * 3	GUP104=6
/	division.	GUP105=9 / 2	GUP105=4.5
MOD	Module or remainder of the division.	GUP106=7 MOD 4	GUP106=3
EXP	exponential.	GUP107=2 EXP 3	GUP107=8

Relational operators:

EQ	equal.	(GUP100 EQ GUP200)
NE	different.	(GUP101 NE 2.5)
GT	greater than.	(GUP102 EQ 0)
GE	greater than or equal to.	(GUP103 GE 1)
LT	Less than.	(GUP104 LT GUP100)
LE	Less than or equal to.	(GUP105 LE 66)

Logic and binary operators:

NOT, OR, AND, XOR: The act as logic operators between conditions and as binary operators between variables and constants.

```
IF ((FIRST) AND (GS1 EQ 1))
GUP1005 = (GUP100 AND (NOT(GUP200 OR GUP299)))
```

Trigonometric functions:

SIN	sine.	GUP101=SIN(30)	GUP101=0.5
COS	cosine.	GUP102=COS(30)	GUP102=0.8660
TAN	tangent.	GUP103=TAN(30)	GUP103=0.5773
ASIN	arcsine (degrees).	GUP104=ASIN(1)	GUP104=90º
ACOS	arccosine (degrees).	GUP105=ACOS(1)	GUP105=0º
ATAN	arctangent (degrees).	GUP106=ATAN(1)	GUP106=45º
ARG	ARG(x,y) arctangent y/x (degrees).	GUP107=ARG(-1 ,-2)	GUP107=243.4349º

There are two functions for calculating the arc tangent ATAN which returns the result between ±90° and ARG given between 0 and 360°.

Other functions:

ABS	absolute value.	GUP101=ABS(-8)	GUP101=8
LOG	decimal logarithm.	GUP102=LOG(100)	GUP102=2
SQRT	square root.	GUP103=SQRT(16)	GUP103=4
ROUND	rounding up an integer number.	GUP104=ROUND(5.83)	GUP104=6
FLOOR	It rounds down.	GUP105=FLOOR(5.423)	GUP105=5
CEIL	It rounds up.	GUP106=CEIL(5.423)	GUP106=6

Example:

Display the value of the X coordinate affected by the active zero offset. To do that, use the following control variables as an intermediate step: GUP100...GUP255, GUP1000...GUP1255 and GUP2000...GUP2255.

It is possible to write equations like:

```
;(GUP100 = (PPOSX - ORGX) * 25.4 / 2)
;(W1= GUP100)
```

To assign a value to a widget, that value must first be assigned to a GUP parameter of the CNC and, then, assign the value of this parameter to the widget. Otherwise, an error will be issued.

15.5 Conditional instructions.

Conditional instructions are only available in version 2 of the configuration language. These instructions may be used to have (IF...ELSE…) type structures that depending on particular conditions (PLC resources, etc.) it will be possible to do the following:

- Overlay one screen or another one or change screens.
- Assign different variables to the same "Widget" element.
- Temporarily cancel (disable) part of a screen.

Flow control instructions

This instruction analyzes the given condition that must be a relational expression. If the condition is true (result = 1), action 1 will be executed. Otherwise (result = 0), action 2 will be executed.

```
;(IF (condition))
    ;(<action1>)
;(ELSE)
    ;(<action2>)
;(ENDIF)
```

IF conditions cannot be nested; i.e. an IF instruction cannot be programmed inside another one. On the other hand, the ELSE part can be missing in the instruction; i.e., it is possible to program IF condition <action1>.

Example 1:

In this example, if P8 is other than 12.8, it executes the instruction WGDWIN 201. Otherwise, if P8 is equal to 12.8, it executes the instruction WGDWIN 202.

```
;(IF (GUP1000 NE 12.8))
    ;(WGDWIN 201)
;(ELSE)
    ;(WGDWIN 202)
;(ENDIF)
```

Example 2:

Example of an IF conditional instruction without the ELSE part.

```
;(IF (GUP100 EQ 12.8))
    ;(WGDWIN 201)
;(ENDIF)
```

Example 3:

The following program could be used to set a parameter to ·0· where the user has entered a value that is out of the permitted range. This may be particularly interesting in the MCO/TCO mode where the configuration file is only refreshed (updated) on start-up or after pressing the [ENTER] key.

```
;(IF (GUP100 GT 1000))
    ;(GUP100=0)
;(ENDIF)
    ;(W1=GUP100)
```

In this case, when entering the parameter value and pressing [ENTER], the configuration file is refreshed and if the parameter value is higher than ·1000·, the parameter takes the value of ·0·.

15.6 Example of a configuration file

```
;(PRGSCRIPT 1)
                                    Header
;;=============================================
;;      Screen (201) in JOG mode - Actual
;;=============================================
                                    Comment
;[JOG],PLCM1125
                                    To show the "JOG mode - Actual" screen when mark M1125=1
;(DEBUG)
                                    Starting at this line, program 999500 keeps a log of the errors originated
                                    when debugging the configuration file.
;(DISABLE 0)
                                    The OEM screen will replace the standard CNC screen.
;(WGDWIN 201)
                                    The OEM screen is 201

;;-------------------- X axis, coordinate, error and consumption
;(AUTOREFRESH W1=POSX)
                                    The graphic element W1 will always show the X axis position.
;(AUTOREFRESH W2=FLWEX)
                                    The graphic element W2 will always show the X axis following error.
;(FORMAT W3,LEDBARDEC)
;(AUTOREFRESH W3=ANAI1)
                                    The graphic element W3 (ledbar) will always show the X axis consumption
                                    (input ANAI1)

;;-------------------- Z axis, position, error and consumption
;(AUTOREFRESH W4=POSZ)
                                    The graphic element W4 will always show the Z axis position.
;(AUTOREFRESH W5=FLWEZ)
                                    The graphic element W5 will always show the Z axis following error.
;(FORMAT W6,LEDBARDEC)
;(AUTOREFRESH W6=ANAI2)
                                    The graphic element W6 (ledbar) will always show the Z axis consumption
                                    (input ANAI2)

;;-------------------- Feedrate, F, %
;(AUTOREFRESH W7=FEED)
                                    The graphic element W7 will always show the feedrate of the axes
;(AUTOREFRESH W8=FRO)
                                    The graphic element W8 will always show the selected % of feedrate override
                                    for the axes.

;;-------------------- Spindle, S, Smax, %
;(AUTOREFRESH W9=SPEED)
                                    The graphic element W9 will always show the spindle speed.
;(AUTOREFRESH W10=SLIMIT)
                                    The graphic element W10 will always show the maximum spindle speed
                                    allowed.

;(AUTOREFRESH W11=SSO)
```

The graphic element W11 will always show the selected % of spindle speed override.

;;---------- Tool and offset (T, D)

;(AUTOREFRESH W12=TOOL)

The graphic element W12 will always show the number of the selected tool.

;(AUTOREFRESH W13=TOD)

The graphic element W10 will always show the number of the selected tool offset.

;(END)

End of debug and end of the section corresponding to the screen

;;==========

;; Screen (202) in MC/TC/CO mode

;;==========

;[STDCONV],PLCM1125

We wish to customize the "Standard screen of the conversational mode" when mark M1125=1

;(DEBUG)

Starting at this line, program 999500 keeps a log of the errors originated when debugging the configuration file.

;(DISABLE 0)

The OEM screen will replace the standard CNC screen.

;(WGDWIN 202)

The OEM screen is 202

;;---------- Coordinates of the Z and X axes

;(AUTOREFRESH W1=POSZ)

The graphic element W1 will always show the Z axis position (coordinates)

;(AUTOREFRESH W2=POSX)

The graphic element W2 will always show the X axis position (coordinates)

;;---------- Machine cursors

;(AUTOREFRESH W3=POSZ)

The graphic element W3 (ledbar) will always show the Z axis position.

;(AUTOREFRESH W4=POSX)

The graphic element W4 (ledbar) will always show the X axis position.

;;---------- Axes feedrate (F)

;(AUTOREFRESH W5=FEED)

The graphic element W5 will always show the feedrate of the axes.

;(END)

End of debug, end of the portion of the configuration file corresponding to the screen and end of the configuration file.

15.7 Error log file (P999500)

Every time a customized screen is accessed, the CNC checks the section of that screen in the configuration file.

If it has errors, it displays the standard screen instead of the customized one.

If the error has been detected in a section defined after the DEBUG instruction, it generates, in program P999500 several lines indicating the detected error or errors.

The error log file (P999500) contains all the errors detected since the CNC was turned on. When the CNC is turned off, this error log file (P999500) is deleted.

Examples of detected errors:

Error due to a nonexistent variable. It must be FLWEX

```
;(AUTOREFRESH W2=FLWEXX)
; Syntax error ...
;       Unknown CNC variable name
;       Error on line: 12
;       Error on character: LF
```

Error caused for referring to a nonexistent graphic element (W33).

```
;(AUTOREFRESH W33=PLCR124)
; Warning...
;       Programmed Widget does not exist.
;       Warning in line: 15
```

CONFIGURABLE WORK MODE

16

The CNC offers a work mode that may be configured by the machine manufacturer. The basic screen provided by Fagor permits controlling the axes, the tool and the spindle.

In the configurable work mode, the manufacturer (OEM) can:

- Configure, partly or completely, the basic screen provided by Fagor.
- Create diagnosis screens.
- Create screens to consult and/or modify internal variables of the CNC, PLC or drive.
- Create screens for the operator to set zero offsets, etc.

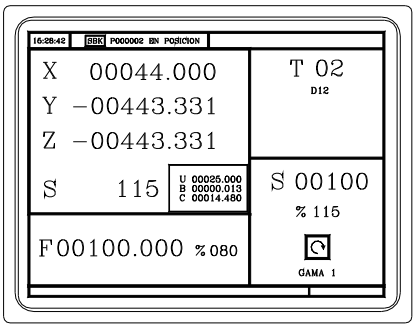
Since all the OEM screens have a subroutine associated, they can also create OEM cycles to:

- Consult inputs and outputs.
- Adjust the machine axes.
- Manage tool magazines.
- Manage external devices.
- etc.

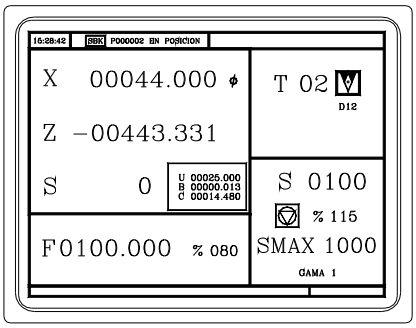
The OEM machining canned cycles may even be used to machine parts. A machining operation may be repeated as often as you wish, but it cannot be stored in memory.

With the keystroke sequence [SHIFT] [ESC] , it is possible to switch from M/T mode to configurable mode and vice versa.

Depending on the model, when accessing the configurable work mode, the CNC shows the following screen:



M model

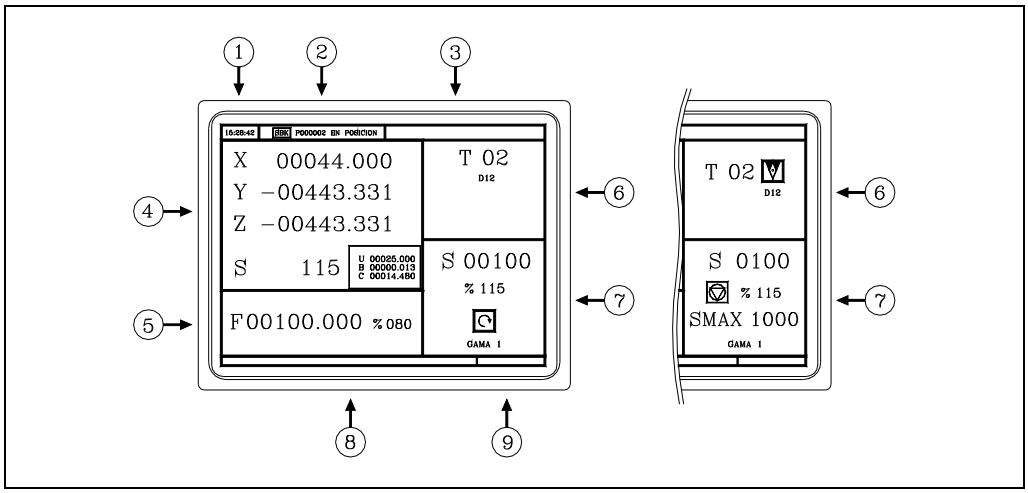


T model

The way to operate with either model is very similar. If any of the features described here is not common to both models, it will clearly indicate which model it corresponds to.

These screens may be customized entirely or by areas. See *"15.1 Configuration file"* on page 570.

The screen provided by Fagor contains the following information:



1. Clock.
2. This window may show the following data:

   SBK When "Single Block" execution mode is selected.

   DNC When the DNC mode is active.

   P... Number of the program currently selected.

   Message «In position» - «Execution» - «Interrupted» - «RESET».

   PLC messages.
3. This window shows the CNC messages.
4. The window provided by Fagor shows the position of the axes (those of the auxiliary axes highlighted) and the real spindle rpm (S).
5. The window provided by Fagor shows the axis feedrate (F) currently selected and the % of F being applied.
6. The window provided by Fagor shows the tool number (T) and the tool offset (D). If the tool number and offset number are the same, the CNC will not show the "D" value.

   The T model also shows the graphic representation for the location code (shape) associated with the tool.
7. The window provided by Fagor shows the associated spindle speed (S), the % being applied, the selected turning direction and the active spindle gear.

   The T model also shows the maximum rpm and the "CSS" value when working at Constant Surface Speed.
8. This window shows the help texts associated with the OEM screens. Refer to the WINDRAW55 manual.
9. Reserved.

16.1 Axis control

When accessing the customizable mode, the CNC assumes the work units "mm or inches", "radius or diameters", "mm/min. or mm/rev", etc. selected by machine parameter.

Coordinate preset

It must be done on one axis at a time and proceeding as follows:



o


o


Value


The CNC requests confirmation of the command.

Axis feedrate (F)

To set the axis feedrate, key in:



Value


Jog movement

Besides the continuous, incremental or handwheel jog, it is also possible to move the axes to a programmed position.

It is done on one axis at a time. With the feedrate «F» and % currently selected. To do that, press:



o


o


Value


16.2 Tool control

To select another tool, press:

 Value 

The CNC will manage the tool change and the T model updates its graphic representation.

Another offset may be assigned to the tool temporarily without modifying the one associated with it.

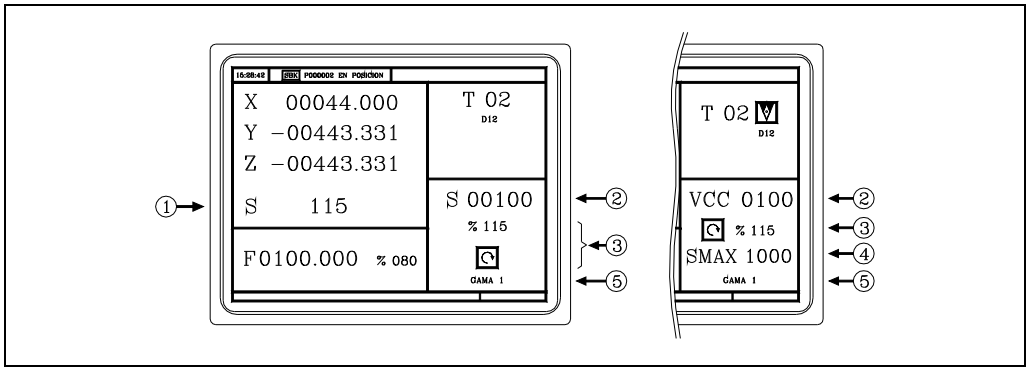
To access the "D" field, press:

  Tool offset number 

The CNC temporarily assumes the new offset for the current tool. The internal table is not modified, the tool's associated offset is still the one assigned to it when it was calibrated.

16.3 Spindle control

The CNC displays the following information:



1. Real spindle speed in rpm.
2. Theoretical spindle speed in rpm or m/min (ft/min) when working at Constant Surface Speed (CSS).

   To select another speed, press:

    Value 

   The CNC assumes this value and if the spindle is turning, it updates the real spindle speed (in rpm).
3. Spindle status and % of spindle speed being applied.
4. Maximum spindle rpm (T model).

   To select another speed, press:

     The CNC highlights the current value.

   Value  The CNC assumes this value and does not allow the spindle to exceed these rpm.
5. Spindle gear currently selected.

   This value cannot be changed when using an automatic gear change.

   To change gears, press

    y  until the new value is highlighted.

   Gear number  o 

Note: When the machine does not use spindle gears, this message makes no sense. That is why the CNC does not show this message when text 28 of program 999997 has not been defined.

16.4 MDI

To use the MDI option in this work mode, send the key code "$F01E (61470)" from the PLC to the CNC.

In the next example, there is an external push button connected to input I13 and the MDI mode is activated every time this button is pressed or when pressing a free key of the front panel.

( ) = MOV 0 R100 = MOV 1 R101 = MOV $F01E R102

Initializations.

DFU I13 OR DFU B?? R???

Every time one of the keys is pressed...

= CNCWR(R101, KEYSRC, M1)

... it indicates to the CNC that the keys come from the PLC.

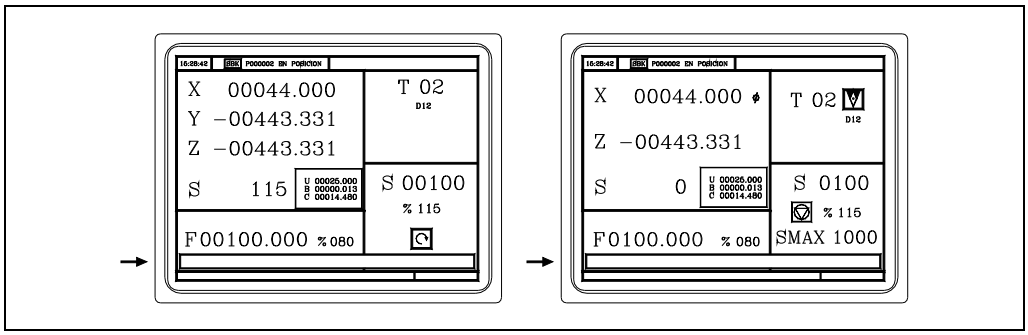
= CNCWR(R102, KEY, M1)

... a code is sent out to activate the MDI mode.

= CNCWR(R100, KEYSRC, M1)

... and it indicates to the CNC that the keys come from the CNC.

In MDI mode, the CNC shows a window at the bottom of the screen.



In this window, a block may be edited and later executed.

16.5 Screens, subroutines and cycles

User Fagor's WINDRAW55 (PC based) application to define the screens

Screens 201 through 255 must be used to customize the screens already stored at the CNC.

Screens 001 through 200 may be used to create new screens for diagnosis, variable consultation, setup, adjustments, device control, OEM canned cycles, etc.

Regardless of how they are used, screens 001 through 200 the have associated with them.

- The configuration file P999xxx (P999001 through P999200) that must be defined by the OEM.
- The subroutine that contains the executable 9xxx (9001 through 9200) that must be defined by the OEM.
- Program P999995 that contains the texts used by all the screens. Refer to the WINDRAW55 manual.
- Program P999994 that contains the help texts used by all the screens. Refer to the WINDRAW55 manual.

Thus, the configuration file P999004 and subroutine 9004 are associated with screen 4.

When pressing [HELP] , the CNC shows all the screens defined by the OEM with the WINDRAW55 application.

To delete any of them, position the cursor on it and press [CLEAR].

The CNC requests confirmation and the access code for customizing, if the OEM has previously defined it.

To access one of the screens 001 through 2000, position the cursor on it and press [ENTER].

Screens 201 through 255 cannot be accessed from this mode. Select the CNC screen associated with it.



In the WINDRAW55 application, the text for to the screen title should be defined with the same number as the screen.
This way, when pressing, the key [HELP], CNC will show the number and title (text of program P999995 with the same number) of the available screens.

16.6 Associated keys

Screens 001 through 200 are grouped in 20 groups. Keys F1 through F7 allow direct access to the first 7 groups; the rest of the grops are accessed by sending a key code from the PLC.

Each group has 10 different levels or screens. Once a group has been selected, send the code "$F01C (61468)" for the [LEVEL CYCLE] key from the PLC to see the next level.

Now, let us see how the screens are grouped and how to access them.

Key or code	Screens								
F1 key	1	21	41	61	81	101	121	141	161
F2 key	2	22	42	62	82	102	122	142	162
F3 key	3	23	43	63	83	103	123	143	163
F4 key	4	24	44	64	84	104	124	144	164
F5 key	5	25	45	65	85	105	125	145	165
F6 key	6	26	46	66	86	106	126	146	166
F7 key	7	27	47	67	87	107	127	147	167
$F108 (61704)	8	28	48	68	88	108	128	148	168
$F109 (61705)	9	29	49	69	89	109	129	149	169
$F10A (61706)	10	30	50	70	90	110	130	150	170
$F10B (61707)	11	31	51	71	91	111	131	151	171
$F10C (61708)	12	32	52	72	92	112	132	152	172
$F10D (61709)	13	33	53	73	93	113	133	153	173
$F10E (61710)	14	34	54	74	94	114	134	154	174
$F10F (61711)	15	35	55	75	95	115	135	155	175
$F110 (61712)	16	36	56	76	96	116	136	156	176
$F111 (61713)	17	37	57	77	97	117	137	157	177
$F112 (61714)	18	38	58	78	98	118	138	158	178
$F113 (61715)	19	39	59	79	99	119	139	159	179
$F114 (61716)	20	40	60	80	100	120	140	160	180

When accessing a group, the CNC will show the last screen (level) used in that group.

To exit the screen:

- Press the key again or send the group code.
  it shows the basic screen
- Press the key or send the code of another group.
  shows the screen of the new group.
- Press [ESC] twice.
  it shows the basic screen.

Example. It has screens 1, 21, 41, 2 and 22.

Press F1.

it shows screen 1.

It sends the level key code.

it shows screen 21.

It sends the level key code.

it shows screen 41.

Press F2.

it shows screen 2.

It sends the level key code.

it shows screen 22.

Press F2.

it exits the screen and shows the basic screen.

Press F1.

it shows screen 41 (last one used in the group).

It sends the level key code.

it shows screen 1 (there are only 1, 21, 41).

Press F2.

it shows screen 22 (last one used in the group).

It sends the level key code.

it shows screen 2 (there are only 2, 22).

Press [ESC] twice.

it shows the basic screen.

The next example shows an external push button connected to input I27 that selects and unselects the group of screens 13, 33, 53, etc.

( ) = MOV 0 R100 = MOV 1 R101 = MOV $F10D R102

Initializations.

DFU I27

Every time the external key is pressed...

= CNCWR(R101, KEYSRC, M1)

... it indicates to the CNC that the keys come from the PLC.

= CNCWR(R102, KEY, M1)

... it sends the key code for the group 13, 33, 53, etc.

= CNCWR(R100, KEYSRC, M1)

... and it indicates to the CNC that the keys come from the CNC.

16.7 OEM text in several languages.

By default, 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 error messages.
P999995	Texts and titles used by all the OEM screens.
P999994	Help texts of the OEM screens or cycles.

To have texts and messages in several languages, they must be grouped in a single program and set g.m.p. MSGFILE (P131) with that program number.

The MSGFILE program may be in user memory 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.

MSGFILE program structure

On each line, a text is defined preceded by ";" the "text number", a blank space and the "$" sign. Examples:

;116 $Axis feedrate (F)

;117 $Tool (T)

The texts must be grouped by subjects and languages.

The labels identify a group with a mnemonic in brackets and preceded by ";"

;[PLCMSG]	Texts for PLC messages (up to 256).
;[PLCERR]	Texts for PLC errors (up to 265).
;[CO999995]	Texts and titles used by all the OEM screens (up to 256).
;[CO999994]	Help texts of the OEM screens or cycles (up to 256).
;[OEMMSG]	Other texts used in OEM programs (up to 768).

The language number is indicated after the label and separated by a comma "," the same number used by g.m.p. LANGUAGE (P122):

(0) English	;[PLCMSG],0	(1) Spanish	;[PLCMSG],1
(2) French	;[PLCMSG],2	(3) Italian	;[PLCMSG],3
(4) German	;[PLCMSG],4	(5) Dutch	;[PLCMSG],5
(6) Portuguese	;[PLCMSG],6	(7) Czech	;[PLCMSG],7
(8) Polish	;[PLCMSG],8	(9) Mainland Chinese	;[PLCMSG],9
(10) Basque	;[PLCMSG],10	(11) Russian	;[PLCMSG],11
(12) Turkish	;[PLCMSG],12		

The text groups may be defined in the desired order, grouping them by subjects, languages, etc.

Programming example using MSGFILE (P131) = 12345

```
P12345          Texts for PLC messages in English
;[PLCMSG],0
;1 $Text 1
;2 $Text 2
----------
;[PLCERR],0     Texts for PLC error messages in English
;1 $Text 1
;2 $Text 2
----------
;[CO999994],0   Help texts of the OEM screens or cycles in English.
;1 $Text 1
;2 $Text 2
----------
;[CO999995],0   Texts and titles used by all the OEM screens in English.
;1 $Text 1
;2 $Text 2
----------
;[OEMMSG],0     Other texts used in OEM programs in English.
;1 $Text 1
;2 $Text 2
----------
```

The texts of the [OEMMSG] are directed to:

- Texts of the user channel used with the program control instructions MSG, ERR, IB, SK that depend on the selected work language.

  (MSG "HELLO") displays the text HELLO on the screen.

  (MSG 200) displays the text 200 of the [OEMMSG] group in the currently selected language.

  (MSG $C8) similar to the previous one. It is the text 200 in hexadecimal format.

  (MSG P100) shows the text of the [OEMMSG] group whose number is the same as the value of parameter 100.

- Texts associated with screens and symbols generated with program Draw55.

  Key in the text and press ENTER.

  Assign to it the number of one of the internal CNC texts.

  Assign to it the number of one of the OEM texts defined in the [OEMMSG] group. New option that is selected with the F2 softkey (OEM TEXT).

  Note: This type of texts can only be displayed properly at the CNC. The Draw55 application shows OEMtxtnn where "nn" means the number of the associated text; for example OEMtxt25.

  The OEM screens or cycles are created with the WINDRAW55 application.

The texts used by the application are created in the WINDRAW55.txt program and must be copied to the CNC as program P999995 or as part of the MSGFILE program within the [CO999995] group.

Use the Draw55 application to generate symbols or drawings used by those screens. If the texts of the symbols depend on the language, define them as part of the MSGFILE within the [OEMMSG] group.

Notes:

On power-up or after a Shift-Reset, the CNC runs the following checkl to find the work texts of each group or text type:

- It takes the group of the MSGFILE program if it has been defined in the selected language.
- If it is not defined, it takes the texts of the first group defined (another language).
- If there is none, it takes the texts of program PLCMSG, PLCERR, P999995 or P999994

It then looks for texts in those groups.

16.8 Associated programs

Programs P900000 through P999999 are reserved for the CNC itself; in other words, the user cannot use them as part-programs.

Some have a special meaning and must be used by the machine manufacturer (OEM).

P999001 P999200	Configuration files for OEM screens. P999001 corresponds to screen 1, P999002 to screen 2 and so on up to P999200 that corresponds to screen 200.
P999500	Program that logs (stores) the errors occurred when interpreting a configuration file.
P999994	Help texts of the OEM screens or cycles. Refer to the WINDRAW55 manual.
P999995	Texts and titles used by all the OEM screens. Refer to the WINDRAW55 manual.
P999999	Reserved for storing all the OEM subroutines It is empty.

The rest of the reserved programs are used internally by the CNC and cannot be erased.

16.9 Associated subroutines

Subroutines 0000 through 8999 are free to use and subroutines 9000 through 9999 are reserved for customizing the CNC.

It is recommended to define all the OEM subroutines in a program with a high number to prevent the user from modifying them. Program P999999 is free and may be used for this purpose.

When executing a cycle (screen), its associated subroutine is called upon indicating in local parameters A - Z (P0 - P25) the value used to define each field.

(PCALL 9001, A=11, B=22, C=33, D=44, E=0, F=1234, G=9999, H=1, I=1, J=12.34)

Parameter A (P0) indicates the value of the first editable field, B (P1) that of the second field and so on up to Z (P25) that indicates the value of the 26 field. The order is set when designing the screen with the WINDRAW55 application.

All the screens (cycles) have an associated subroutine, 9000 + "screen number" Subroutine 9001 corresponds to screen (cycle) 001, 9002 to 002 and so on up to 9200 that corresponds to screen (cycle) 200.

All these subroutines must be defined by the OEM and must contain all the necessary instructions to run the canned cycle.

Example:

```
( SUB 9005)          ; Definition of subroutine 9005.
                     ; Program blocks defined by the OEM.
(RET)                ; End of subroutine.
```

16.10 Configuration file

The configuration file is a CNC file written in high-level language (configuration language) that describes the operating characteristics of the various screen elements.

There is a configuration file for each screen. P999001 corresponds to screen 001, P999002 to 002 and so on up to P999200 that corresponds to screen 200.

The general characteristics of the configuration language are:

- All instructions are preceded by ";" and enclosed in parenthesis.
- The comments must be alone and preceded by ";;"
- The configuration file must begin with the line ";(PRGSCRIPT 1)"

  It indicates that it is a configuration file corresponding to the version being used (in this case "1").
- The configuration file should end with the line ";(END)"
- It is recommended to use the ";(DEBUG)" instruction for the CNC to check the configuration file. If an error occurs, it will log it in program 999500.

The configuration language has the following key words:

;(PRGSCRIPT 1)

Header of the configuration file and version used to edit it (in this case "1"). It must always be defined.

;(DEBUG)

It is optional. It indicates on which line program 999500 starts giving out information of the errors that have come up when debugging the configuration file.

The debugging of the configuration file begins on the first line ;(PRGSCRIPT 1) and ends on the ;(END) line.

;(DISABLE 1)

It indicates that the blue frame at the top right must not be displayed; it shows the current status of the machine (coordinates and machining conditions).

;(DISABLE 20)

It indicates that the "CYCLE START" icon must not be displayed when pressing [ESC].

To exit the screen, press [ESC] twice. The first time, it displays the "CYCLE START" icon at the top right of the screen offering the chance to either execute or simulate the associated program.

Some screens are merely informative and no associated program must be executed.

In these cases, it is recommended to program ;(DISABLE20) to exit the screen when press [ESC].

;(DISABLE 21)

It indicates that the spindle is not to be stopped at the end of the cycle.

By default, if ;(DISABLE21) is not programmed, the CNC adds the M5 instruction at the end of the associated program to stop the spindle at the end of the cycle.

;(HOTKEY W4,88)

It permits associating a key to the (W) data.

In this case, pressing the X key (ASCII value 88) selects the W4 data.

;(TEACHIN W5=POSX)

Permits assigning the value of a CNC variable to the data.

In this case, While the W5 data is selected, when pressing the Recall key, W5 will show the variable of POSX (X axis position). If Enter is then pressed, the cycle assumes this value.

;(FORMAT W7,INCH)

The value is shown in the work units (mm / inches) set by g.m.p. INCHES (P8).

;(PROFILE W12)

Permits associating a profile with the (W) field. The field must be an unsigned 3-digit integer.

Select the (W) field, enter the number of the profile to be edited, press Recall and it will access the profile editor.

The edited profile is stored as program P994xxx.

P994001 corresponds to profile 001, P994002 to 002 and so on up to P994999 that corresponds to profile 999.

;(P100=W13)

Permits assigning the value of the (W) data to a global parameter only when calling a subroutine.

When executing a cycle, the CNC calls the associated subroutine indicating with local parameters which values have been defined in each field. For example:

(PCALL 9001, A10, B12, C5, D8)

When using global parameters, the CNC uses another PCALL instruction to transfer the global parameters. For example:

(PCALL 9301, P100=22, P101=32, P102=48)

(PCALL 9001, A10, B12, C5, ... Y8, Z100)

The subroutine associated with the cycle is	9000 + cycle number
The auxiliary subroutine associated with the cycle is	9300 + cycle number

When using a (P100=W13) type instruction, the auxiliary subroutine must also be defined even if it only contains the SUB and RET instructions.

;(W1=GUP100)

Associates the value of a global parameter with the (W) data.

;(W2=PLCFRO)

Associates the value of a variable with the (W) data.

;(W3=PLCR127)

Associates the value of a PLC resource with the (W) data.

A register	;(W6=PLCR127)	
A mark	;(W6=PLCM1000,1)	first and how many
A group of inputs	;(W6=PLCI8,4)	first and how many
A group of outputs	;(W6=PLCO10,3)	first and how many

Associate only resources that are defined in the PLC program.

For marks, inputs and outputs, one must indicate how many of them, if none is indicated, 32 are assigned.

;(W6=PLCO11,4)	Assigns the value of O11, O12, O13, O14
;(W6=PLCO11)	Assigns the value of O11, O12 ... O41, O42.

If a field (W) has a parameter, variable or resource associated with it, it acts as follows:

- It assumes the value that its associate has when accessing the page.

  To continuously update the field value, use the (AUTOREFRESH) instruction as described later on.
- If its associate is a read-only type, the user will not be able to change the field value.
- It its associate is read/write type, the user may change the field value.

  When changing the value of the field (W), the value of its associate is also changed.

  On the other hand, when using the (AUTOREFRESH) instruction and the CNC or PLC changes the value of the associate, the value of the field is also changed.
- If an error occurs because the variable does not exist (PLC register) or the communication fails (drive variable), the field is momentarily disabled and it displays a gray window. The field is enabled again after 10 seconds.

;(AUTOREFRESH W6=FLWEX)

Refreshes (updates periodically) the value of graphic element W6 assigning it the X axis following error value.

;(SAVEINSUB)

It is programmed as a prefix of the instructions

;(W1=GUP100)

Associates the value of a global parameter, variable or resource of the PLC with the (W) data.

;(AUTOREFRESH W6=FLWEX)

Refreshes the value of graphic element W6.

The resulting new instructions are:

;(W1=GUP170)

;(SAVEINSUB W1=GUP170)

;(AUTOREFRESH W6=FLWEX)

;(SAVEINSUB AUTOREFRESH W6=FLWEX)

When recalling a cycle stored in memory [PPROG] , the CNC analyzes the type of instruction that each graphic element has associated with it and acts as follows:

- If it is of the ;(W1=GUP170) type, it assigns the current value of arithmetic parameter P170 to W1.
- If it is of the ;(SAVEINSUB W1=GUP170) , it restores the value that the arithmetic parameter had when the program was edited (when it was saved into memory) and it assigns that value to the graphic element W1 and to the arithmetic parameter P170.

Recommendations to use SAVEINSUB:

- Restrict its usage to when it is absolutely necessary because every time a stored cycle is saved or recalled, the value of the variable associated with the element changes.
- Use it only with global parameters and variables that may be read and written from the CNC See *"Summary of internal CNC variables."* on page 635.

;(UNMODIFIED)

It indicates that the associated item must not pick up the editing focus.

It is programmed as a prefix of the instructions.

;(W1=GUP100)

Associates the value of a global parameter, variable or resource of the PLC with the (W1) data. The W1 item of the screen pick ups the editing focus.

;(AUTOREFRESH W6=FLWEX)

It refreshes (updates) the value of the graphic element W6 and it assigns the editing focus to it.

The resulting new instructions are:

;(UNMODIFIED W1=GUP170)

Associates the value of a global parameter, variable or resource of the PLC with the (W1) data. The W1 item of the screen pick ups the editing focus.

;(UNMODIFIED AUTOREFRESH W6=FLWEX)

It refreshes (updates) the value of the graphic element (item) W6, but it does not assign the editing focus to it.

;(FORMAT W8,LEDBARDEC)

It must be used with Ledbar type (W) data that have a decimal variable associated with them. (For example: X axis following error).

The values assigned, at the PLC, to the end and intermediate values of a LEDBAR element must be integer values and must be related to the variable associated at the CNC.

When the associated variable has a decimal format the following instruction must be used:

;(FORMAT W8,LEDBARDEC)

This instruction is used to convert coordinate values (decimal) to integer values by multiplying them by 10000.

Examples:

To represent the % of axis feedrate, the FRO variable is used. The FRO values are integers (between 0 and 120) and, therefore, do not require LEDBARDEC.

;(AUTOREFRESH W9=FRO)

On the other hand, to represent the amount of following error on the X axis, the FLWEX variable is used. The values of FLWEX are not integers and, therefore, require LEDBARDEC (multiplying it by 10000) in order to make them integers.

;(FORMAT W11,LEDBARDEC)

;(AUTOREFRESH W11=FLWEX)

;(MODALCYCLE)

It indicates that cycle is modal. See programming manual.

The subroutine call is of the (MCALL 9001, A10, B12, C5,....) type

If after executing the cycle, several movements are carried out, the cycle will be executed again after each movement calling the (MCALL 9001, A10, B12, C5,...) subroutine again.

When using global parameters, the CNC only transfers the global parameters the first time.

First time:

(PCALL 9301, P100=22, P101=32, P102=48)

(MCALL 9001, A10, B12, C5, ... Y8, Z100)

The rest of the times:

(MCALL 9001, A10, B12, C5, ... Y8, Z100)

To cancel this mode, execute the (MDOFF) instruction.

;(END)

It indicates that it has finished debugging the configuration file.

It ignores the instructions programmed next.

16.11 Error log file (P999500)

There is a configuration file for each screen. P999001 corresponds to screen 001, P999002 to 002 and so on up to P999200 that corresponds to screen 200.

The CNC checks these programs when accessing each screen for the first time. When detecting an error, it displays a message window.

In all of them, if the error has been detected in a section defined after the DEBUG instruction, they generate, in program P999500, several lines indicating the detected error or errors.

The error log file (P999500) contains all the errors detected since the CNC was turned on. When the CNC is turned off, this error log file (P999500) is deleted.

Examples of detected errors:

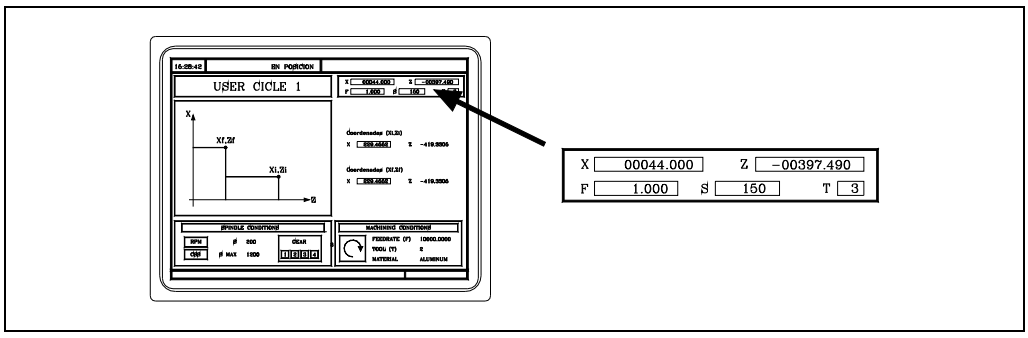
```
Error due to a nonexistent variable. It must be FLWEX.
    ;(AUTOREFRESH W2=FLWEXX)
    ; Syntax error ...
    ;     Unknown CNC variable name.
    ;     Error on line: 12.
    ;     Error on character: LF.
Error caused for referring to a nonexistent graphic element (W33).
    ;(AUTOREFRESH W33=PLCR124)
    ; Warning ...
    ;     Programmed Widget does not exist.
    ;     Warning in line: 15.
```



After modifying the configuration file, reset the CNC to debug it again when accessing its associated screen.

16.12 Cycle data entry

Once the cycle has been selected, the CNC shows the relevant screen. It may have a blue frame at the top right indicating the current status of the machine. Coordinates and machining conditions:



One of the data that define the cycle will be highlighted indicating that it is selected.

Use the keys to select another data



Select a numeric data

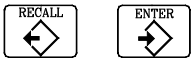
The numeric data is used for coordinates, feedrate, spindle speed, tool number, etc. (value in WINDRAW55).

X 229.4552

Key in the desired value and press



If the Teachin instruction is associated with it, the value of an internal variable (coordinate, tool number, etc.) may be assigned to this field. In those cases, press



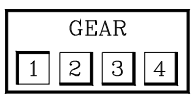
If this [Recall] key is not available, send the key code $F006 (61446) from the PLC to the CNC.

The following example shows how to use an external push button connected to input I25 so as to act like the [Recall] key.

```
( ) = MOV 0 R100 = MOV 1 R101 = MOV $F006 R102
    Initializations.
DFU I27
    Every time the external key is pressed...
= CNCWR(R101, KEYSRC, M1)
    ... it indicates to the CNC that the keys come from the PLC.
= CNCWR(R102, KEY, M1)
    ... the code for the Recall key is sent out.
= CNCWR(R100, KEYSRC, M1)
    ... and it indicates to the CNC that the keys come from the CNC.
```

Select an option among the ones available

It is used to select a button in a group of WINDRAW55 buttons.



Use the keys



Select one of the representations in an icon

It is used to select an icon in a multiple representation of WINDRAW55.



Press the key or send the key code $F01D (61469) from the PLC to the CNC until the desired icon or text appears.

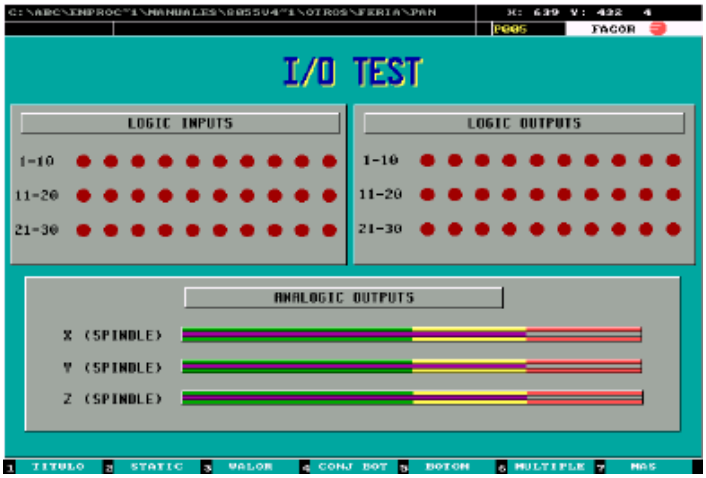
When pressing [ESC], the top of the screen shows a CYCLE START icon. It is then possible to:

- press [ESC] again to exit the cycle.
- Press the [Start] key. It calls the associated subroutine (9001) to execute the cycle.

16.13 Example. Consult inputs and outputs

It uses the 005 screen. The data shown by this screen have the following identifier (W).

Inputs I1 through I30	W1 through W30
Outputs O1 through O30	W31 through W60
X, Y, Z axis (spindle) output	W61, W62, W63



This screen has the P999005 configuration file associated with it which is defined as follows:

;(PRGSCRIPT 1)

Header and version.

The debugging of the configuration file begins.

;(DEBUG)

Starting at this line, program 999500 keeps a log of the errors originated when debugging the configuration file.

;(DISABLE 1)

The blue frame at the top right of the screen is not to be displayed.

;(AUTOREFRESH W1=PLCI1,1)

Refreshes (updates periodically) the value of graphic element W1 assigning the value of input I1 to it. Repeat this instruction for all the inputs.

;(AUTOREFRESH W31=PLCO1,1)

Refreshes (updates periodically) the value of graphic element W31 assigning the value of output O1 to it. Repeat this instruction for all the outputs.

;(FORMAT W61,LEDBARDEC)

;(AUTOREFRESH W61=ANAO1)

Refreshes (updates periodically) the value of the W61 ledbar assigning the value of analog output 1 (X axis analog voltage) to it. Repeat this instruction for all three axes.

;(END)

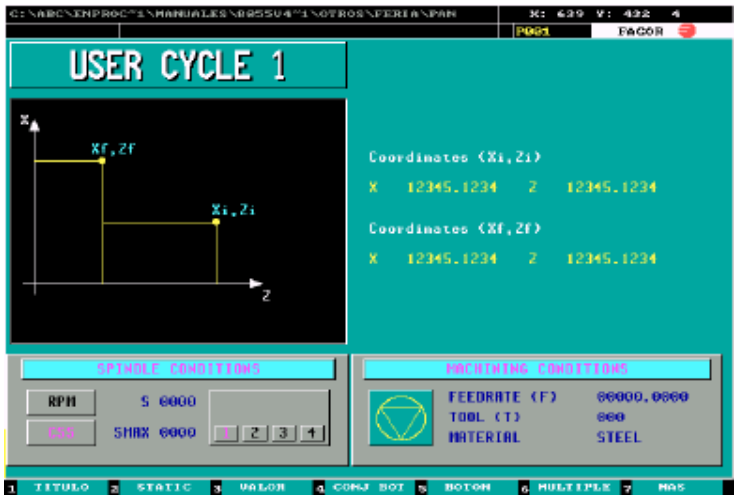
End of the debugging of the configuration file and end of program.

Since this screen does not belong to a cycle, its associated subroutine (9005) needs not be defined.

16.14 Example. Machining canned cycle

It uses the 001 screen. The data shown by this screen have the following identifier (W).

The data that the user may edit in this cycle have the W identifier, the number associated with each one indicates the order in which they are selected (W1, W2, etc.). W11, W12).



This screen has the P999001 configuration file associated with it. Some of its fields are defined as follows:

;(PRGSCRIPT 1)

;(HOTKEY W1,88)

;(TEACHIN W1=POSX)

The W1 field has the X hotkey (88) and the X axis coordinate associated with it. In other words:

Pressing the X key selects this field.

While this field is selected, if the Recall key is pressed, it will show the X axis position. If Enter is then pressed, the cycle assumes that value.

Repeat these instructions for the fields: W2, W3 and W4.

;(HOTKEY W6,83)

;(HOTKEY W7,83)

The W6 and W7 fields have the S (83) hotkey associated with them. In other words:

Pressing the S key selects the W6 field and pressing it again selects the W7 field.

Associate the F key with the W10 field and the T key with the W11 field.

;(END)

End of the debugging of the configuration file and end of program.

Pressing [ESC] displays a cycle-start icon at the top right of the screen and if the [CYCLE START] is pressed again, it calls the associated subroutine (9001) to execute the cycle.

The 9001 subroutine must be defined by the OEM and must contain all the necessary instructions to run the canned cycle.

16.

CONFIGURABLE WORK MODE

Example. Machining canned cycle

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

PLC PROGRAMMING EXAMPLE

17

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.

17.1 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

17.2 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

17.3 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 (3600000 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-COOLMA	The operator control the coolant. Manual mode.
I-COOLAU	The CNC controls the coolant. Automatic mode.
O-COOL	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:

SENDKEY	(Send Key) Calling parameter that must be activated whenever a key is to be sent.
KEYCODE	(Code of the key) Calling parameter that must contain the code corresponding to the key being simulated.
SENTOK	(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.

APPENDIX

A. Technical characteristics of the 8055iCNC 621
B. Central unit of the 8055 CNC 625
C. 11" LCD Monitor 629
D. Probe connection at the 8055i 631
E. Probe connection at the 8055 CNC 633
F. Summary of internal CNC variables. 635
G. Summary of PLC commands 643
H. Summary of PLC inputs and outputs 647
I. 2-digit BCD code output conversion table 653
J. Key code 655
K. Logic outputs of key status 665
L. Key inhibiting codes 675
M. Machine parameter setting chart 685
N. M functions setting chart 697
O. Leadscrew error compensation table 699
P. Cross compensation table 701
Q. Maintenance 703

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6X

TECHNICAL CHARACTERISTICS OF THE 8055iCNC

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

4 feedback inputs for the axes.
4 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 probes (TTL or 24 V DC).
Digital servo or CAN.

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.
56 optocoupled digital inputs
32 optocoupled digital outputs
Remote modules for digital I/O expansion.

32-bit processor
Math coprocessor
Graphics coprocessor.
1Mb CNC program memory.
3.5 ms block processing time , FL model.
0.9 ms block processing time, POWER model.
Sampling time configurable by the machine manufacturer; 2, 3, 4, 5 and 6 ms.

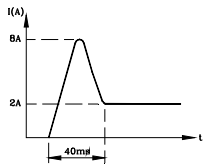
Approximate weight 7.5 Kg.
Maximum consumption of 60 W in normal operation.

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.

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: 135 kb.
Programming in mnemonics.
1 millisecond time unit.
512 inputs.
512 outputs.
3999 user marks.
499 32-bit registers.
256 32-bit counters.
512 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: Below +4 Vdc.
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.

Technical characteristics of the 8055iCNC

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

CENTRAL UNIT OF THE 8055 CNC

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

It has 8 feedback inputs up to 7 axes + spindle encoder or electronic handwheel.
It has 8 analog inputs for supervision and control of external devices.
It has 8 analog outputs of ±10V (one per axis + spindle).
Approximate weight 7 Kg. (that of 3 modules) and 10 kg. (that of 6 modules).
Maximum consumption of 80 W in normal operation.

Power supply

High performance switching power supply.
Universal power supply with any input between 84 Vac and 264 Vac.
Mains frequency of 50 - 60 Hz ±1% and ±2% during very short periods.
Power outages. Meets the "EN 61000-4-11" standard It can withstand microsurges of up to 10 milliseconds at 50 Hz starting at 0º and 180º (two polarities, positive and negative).
Harmonic distortion. Less than 10% of the total rms voltage between conductors under power (sum of the 2nd and 5th harmonics).

Packaging

Meets the "EN 60068-2-32" standard

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

Vibration

In duty cycle 10-50 Hz amplitude 0.2 mm.
While being shipped 10-50 Hz amplitude 1 mm, 50-300 Hz 5g acceleration.
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 : IP2X.
Operator panel: IP54
Accessible parts inside: IP1X.

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 more days, with the CNC off. It must be replaced.

Precaution, due to the risk of explosion or combustion.
Neither recharge the battery nor expose it to temperatures over 100 ºC (212 ºF). Do not short-circuit the terminals.

PLC

Memory: 135 kb.
Programming in mnemonics.
1 millisecond time unit.
512 inputs.
512 outputs.
3999 user marks.
499 32-bit registers.
256 32-bit counters.
512 32-bit timers.

–CPU– Module

32-bit processor
Math coprocessor
Graphics coprocessor.
1Mb CNC program memory.
3.5 ms block processing time , FL model.
0.9 ms block processing time, POWER model.
Sampling time configurable by the machine manufacturer; 2, 3, 4, 5 and 6 ms.
It has 1 RS232C communication line.

–Vpp axes– module

It has 8 feedback inputs for up to 7 axes + spindle encoder or electronic handwheel. Four of these inputs admit differential TTL and 1 Vpp sinusoidal signals; the other four admit differential TTL and non-differential TTL signals.
Feedrates between 0.0001 mm/min and 99999.9999 mm/min (0.00001 inches/min and 3937 inches/min).
Maximum travel: ±99999.9999 mm (±3937 inches).
Up to 2 digital probes (±5 V or ±10 V) may be connected .
It has 40 opto-coupled digital inputs.
It has 24 opto-coupled digital outputs.
It has 4 differential analog inputs of ±5 V or ±10 V (that may be selected by machine parameter) for monitoring external devices. Resolution 46.8 mV.
It has 8 analog outputs of ±10V (one per axis + spindle).

–Vpp SB axes– Module

It has 4 feedback inputs for up to 3 axes + spindle encoder or electronic handwheel. Two of these inputs admit differential TTL and 1 Vpp sinusoidal signals; the other two admit differential TTL and non-differential TTL signals.
Feedrates between 0.0001 mm/min and 99999.9999 mm/min (0.00001 inches/min and 3937 inches/min).
Maximum travel: ±99999.9999 mm (±3937 inches).
Up to 2 digital probes (±5 V or ±10 V) may be connected .
It has 40 opto-coupled digital inputs.
It has 24 opto-coupled digital outputs.
It has 4 differential analog inputs of ±5 V or ±10 V (that may be selected by machine parameter) for monitoring external devices. Resolution 46.8 mV.
It has 8 analog outputs of ±10V (one per axis + spindle).

–I/Os – (Inputs - Outputs)module

It has 64 opto-coupled digital inputs.
It has 32 opto-coupled digital outputs.

Digital inputs

Rated voltage +24 V DC.
Maximum rated voltage +30 Vdc.
Minimum rated voltage +18 Vdc.
High threshold (logic level "1") V_{IH}: From +18 V DC up.
Low threshold (logic level "0") V_{IL}: 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

Rated supply voltage +24 Vdc
Maximum rated voltage +30 Vdc.
Minimum rated voltage +18 Vdc.
Output voltage V_{out} = Supply voltage (Vdc) - 3 V
Maximum output current 100 mA
Protection by means of galvanic isolation by optocouplers.
Protection through an external 3 A fuse against reverse connection up to -30 Vdc and against over-voltage of the external power supply greater than 33 Vdc.

Analog inputs

At the –Vpp axes– and –Vpp SB axes– modules, voltage within the ±5 V or ±10 V range (may be selected by machine parameter).
Resolution 46.8 mV.
Shielded cable should be used.

Analog outputs

Command voltage within ±10 V.
Minimum impedance of the connected connector 10 kΩ.
Shielded cable should be used.

5 V probe input.

Typical value: 0.25 mA. ≅ Vin = 5 V.
High threshold (logic level "1") V_{IH}: From +2.4 V DC up.
Low threshold (logic level "0") V_{IL}: Under +0.9 V DC.
Rated voltage Vimax = +15 V DC.

24 V probe input.

Typical value: 0.30 mA. ≅ Vin = 24 V.
High threshold (logic level "1") V_{IH}: From +12.5 V DC up.
Low threshold (logic level "0") V_{IL}: Under +4 V DC.
Rated voltage Vimax = +35 V DC.

B.

Central unit of the 8055 CNC

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

11" LCD MONITOR

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.

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.

Power supply

Universal AC Power supply 84-264 Vac
Mains frequency: 50 - 60 Hz ±1.
Consumption: 20 W in normal operation and 3W in low consumption.

Connectors

Monitor supply: Bipolar connection base + ground connection, according to IEC-320 and EEC-22 standards.
Video signals: 25 pin SUB-D connector (male).
Keyboard connection: 25 pin SUB-D connector (female).

Packaging

Meets the "EN 60068-2-32" standard

Ambient conditions

Relative humidity: 20% ÷ 80%.
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.

Electromagnetic compatibility and Safety

Refer to the section on safety conditions in the introduction of this manual.

Degree of protection

Front panel: IP54
Rear panel: IP2X.
Accessible parts inside: IP1X.

C.

11" LCD Monitor

FAGOR

CNC 8055
CNC 8055i

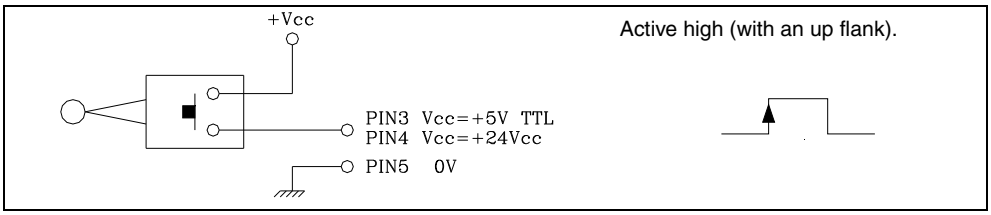
SOFT: V01.6x

PROBE CONNECTION AT THE 8055i

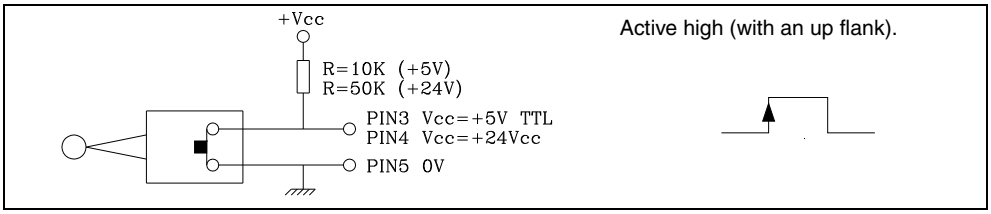
The CNC has two probe inputs (of 5Vdc and 24Vdc) at connector X3.

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.

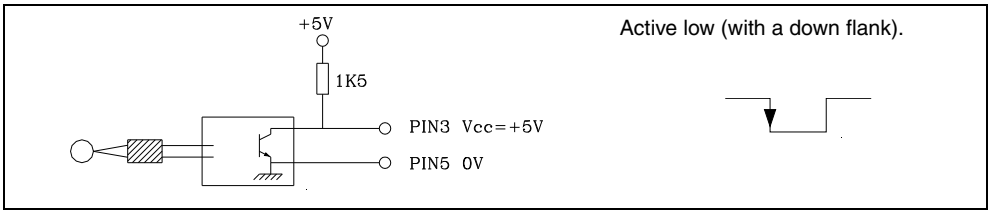
Probe with "normally open contact" output,



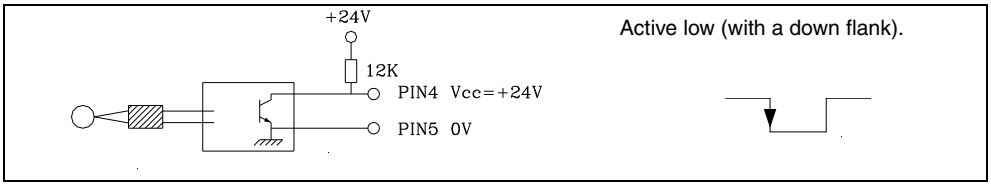
Probe with "normally closed contact" output.



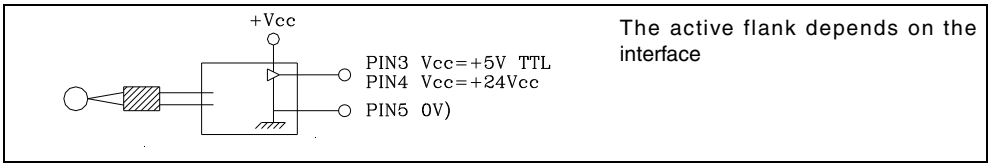
Interface with output in open collector Connection to +5 V.



Interface with output in open collector Connection to +24 V.



Interface with output in PUSH-PULL



D.

Probe connection at the 8055i

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

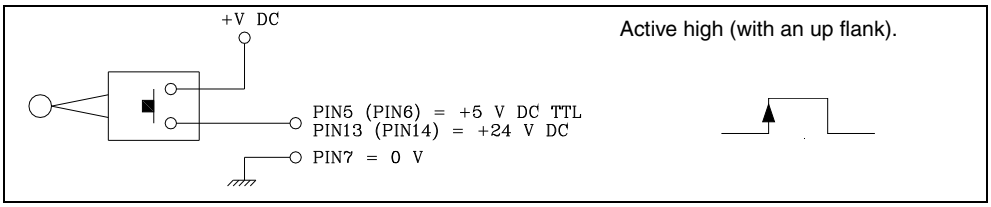
PROBE CONNECTION AT THE 8055 CNC

-Vpp axes and –Vpp SB Axes– module

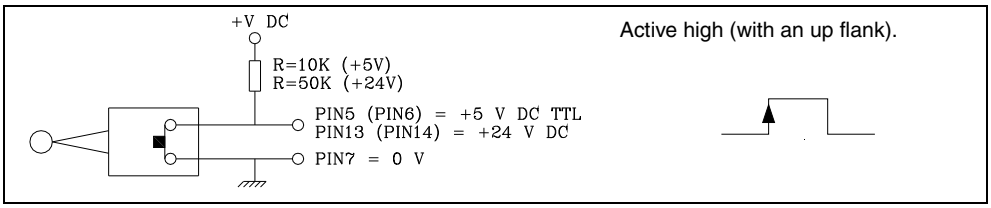
The CNC has two probe inputs (of 5Vdc and 24Vdc) at connector X7 of the –Vpp Axes– module and the -Vpp SB- module.

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.

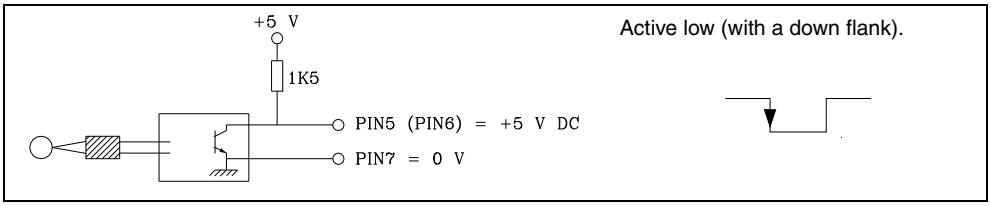
Probe with "normally open contact" output,



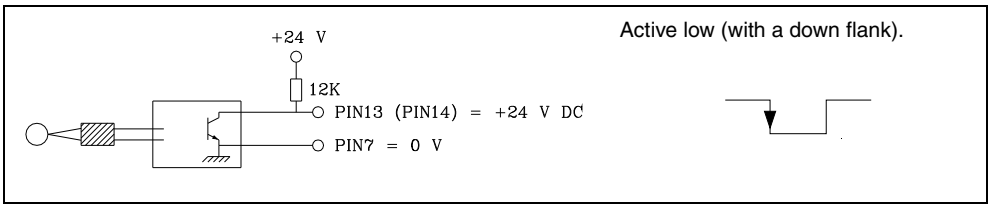
Probe with "normally closed contact" output.



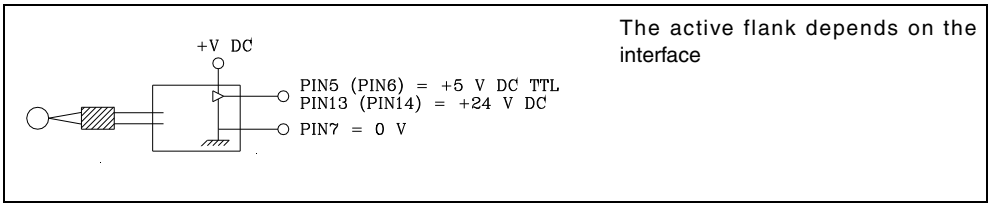
Interface with output in open collector Connection to +5 V



Interface with output in open collector Connection to +24 V



Interface with output in PUSH-PULL



E.

Probe connection at the 8055 CNC

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

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 13.1)
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	-	Cutter width of indicated tool.
CUTAn	R/W	R/W	-	Cutting angle of indicated tool.

Variables associated with zero offsets.

Variable	CNC	PLC	DNC	(section 13.2)
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 function G49

Variables associated with the definition of function G49.

Variable	CNC	PLC	DNC	(section 13.3)
ORGROX	R	R	R	X coordinate of the new part zero with respect to home.
ORGROY	R	R	R	Y coordinate of the new part zero with respect to home.
ORGROZ	R	R	R	Z coordinate of the new part zero with respect to home.
ORGROA	R	R	R	Value assigned to parameter A.
ORGROB	R	R	R	Value assigned to parameter B.
ORGROC	R	R	R	Value assigned to parameter C.
ORGROI	R	R	R	Value assigned to parameter I.
ORGROJ	R	R	R	Value assigned to parameter J.
ORGROK	R	R	R	Value assigned to parameter K.
ORGROQ	R	R	R	Value assigned to parameter Q.
ORGROR	R	R	R	Value assigned to parameter R.
ORGROS	R	R	R	Value assigned to parameter S.
GTRATY	R	R	R	Type of G49 programmed.

Variables updated by the CNC once function G49 is executed.

TOOROF	R/W	R/W	R/W	Position to be occupied by the spindle's main rotary axis.
TOOROS	R/W	R/W	R/W	Position to be occupied by the spindle's 2nd rotary axis.

Variables associated with machine parameters.

Variable	CNC	PLC	DNC	(section 13.4)
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.
MPSSn	R	R	-	Value assigned to machine parameter (n) of the second spindle.
MPASn	R	R	-	Value assigned to machine parameter (n) of the auxiliary spindle.
MPLCn	R	R	-	Value assigned to machine parameter (n) of the PLC.

Variables associated with work zones.

Variable	CNC	PLC	DNC	(section 13.5)
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 13.6)
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 G94.

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 G94.

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 13.7*)
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.
DEFLEX	R	R	R	Probe deflection along X axis.
DEFLEY	R	R	R	Probe deflection along Y axis.
DEFLEZ	R	R	R	Probe deflection along Z 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.
DRPO(X-C)	R	R	R	Position indicated by the Sercos drive of the selected axis.
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 13.8*)
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 13.9*)
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.
SASINS	R	R	R	"A" signal of the CNC sinusoidal feedback for the second spindle.
SBSINS	R	R	R	"B" signal of the CNC sinusoidal feedback for the second spindle.

Variables associated with the main spindle.

Variable	CNC	PLC	DNC	(*section 13.10*)
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 -3600000 and 3600000) and from the CNC in degrees (between -360 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).
DRPOS	R	R	R	Position indicated by the Sercos drive.
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.
SYNCER	R	R	R	Error of second spindle (synchronized) following the main spindle.

Variables associated with the second spindle.

Variable	CNC	PLC	DNC	(*section 13.11*)
SSREAL	R	R	R	Real spindle speed
SFTEOS	R	R	R	Theoretical spindle speed.

Variables associated with spindle speed.

SSPEED	R	R	R	Active spindle speed at the CNC.
SDNCS	R	R	R/W	Spindle speed selected via DNC.
SPLCS	R	R/W	R	Spindle speed selected via PLC.
SPRGS	R	R	R	Spindle speed selected by program.

Variables associated with constant cutting speed (lathe model).

SCSS	R	R	R	Constant surface speed active at the CNC.
SDNCCS	R	R	R/W	Constant surface speed selected via DNC.
SPLCCS	R	R/W	R	Constant surface speed selected via PLC.
SPRGCS	R	R	R	Constant surface speed selected by program.

Variables associated with the spindle override.

SSSO	R	R	R	Spindle Speed Override (%) active at the CNC.
SPRGSO	R/W	R	R	Override (%) selected by program.
SDNCSO	R	R	R/W	Override (%) selected via DNC.
SPLCSO	R	R/W	R	Override (%) selected via PLC.
SCNCSO	R	R	R	Spindle Speed Override (%) selected from front panel.

Speed limit related variables.

SSLIMI	R	R	R	Spindle speed limit active at the CNC.
SDNCSL	R	R	R/W	Spindle speed limit selected via DNC.
SPLCSL	R	R/W	R	Spindle speed limit selected via PLC.
SPRGSL	R	R	R	Spindle speed limit selected by program.

Position related variables.

SPOSS	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).
SRPOSS	R	R	R	Real Spindle position. Reading from the PLC in ten-thousandths of a degree (between -3600000 and 3600000) and from the CNC in degrees (between -360 and 360).
STPOSS	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).
SRTPOS	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).
SDRPOS	R	R	R	Position indicated by the Sercos drive.
SPRGSP	R	R	R	Position programmed in M19 via program for the second spindle.

Variables related to the following error.

SFLWES	R	R	R	Spindle following error.

Variables associated with the live tool.

Variable	CNC	PLC	DNC	(*section 13.12*)
ASPROG	R	R	-	Speed programmed in M45 S (within the associated subroutine).
LIVRPM	R	R	-	Speed of the live tool in TC mode.

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 13.13*)
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).

Sercos variables.

Variable	CNC	PLC	DNC	(*section 13.14*)
SETGE(X-C)	W	W	-	Gear ratio and parameter set of the (X-C) axis drive
SETGES	W	W	-	Gear ratio and parameter set of the main spindle
SSETGS	W	W	-	Gear ratio and parameter set of the second spindle
SVAR(X-C) id	R/W	-	-	Sercos variable sercos for the (X-C) axis "id"
SVARS id	R/W	-	-	Sercos variable sercos for the main spindle "id"
SSVARS id	R/W	-	-	Sercos variable sercos for the second spindle "id"
TSVAR(X-C) id	R	-	-	Third attribute of the sercos variable for the (X-C) axis "id"
TSVARS id	R	-	-	Third attribute of the sercos variable for the main spindle "id"
TSSVAR id	R	-	-	Third attribute of the sercos variable for the second spindle "id"

Software & hardware configuration variables.

Variable	CNC	PLC	DNC	(*section 13.15*)
HARCON	R	R	R	It indicates, with bits, the CNC's hardware configuration.
HARCOA	R	R	R	It indicates, with bits, the CNC's hardware configuration.
IDHARH	R	-	R	Hardware identifier (8 least significant bits).
IDHARL	R	R	R	Hardware identifier (4 most significant bits).
SOFCON	R	R	R	Software version of the CNC (bits 15-0) and HD (bits 31-16)
HDMEGA	R	R	R	Hard disk size (in megabytes).
KEYIDE	R	R	R	Keyboard code, according to the auto-identification system.
MODEL	R	R	R	It identifies the CNC model, mill or lather.

Variables associated with telediagnosis.

Variable	CNC	PLC	DNC	(*section 13.16*)
HARSWA	R	R	R	Hardware configuration.
HARSWB	R	R	R	Hardware configuration.
HARTST	R	R	R	Hardware test.
MEMTST	R	R	R	Memory test.
NODE	R	R	R	Node number in the Sercos ring.
VCHECK	R	R	R	Software version checksum.
IONODE	R	R	R	Position of the "ADDRESS" switch of the I/O CAN bus.
IOSLOC	R	R	R	Number of local I/O available.
IOSREM	R	R	R	Number of remote I/O available.

Operating-mode related variables.

Variable	CNC	PLC	DNC	(*section 13.17*)
OPMODE	R	R	R	Operating mode.
OPMODA	R	R	R	Operating mode when working in the main channel.
OPMODB	R	R	R	Type of simulation.
OPMODC	R	R	R	Axes selected by handwheel.

Other variables.

Variable	CNC	PLC	DNC	(*section 13.18*)
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 through 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.
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.
AXICOM	R	R	R	Pairs of axes switched with function G28.
TANGAN	R	R	R	Angular position with respect to the path (G45).
TPIOUT(X-C)	R	R	R	Output of the PI of the tandem master axis in rpm.
DNCSTA	-	R	-	DNC transmission status.
TIMEG	R	R	R	Remaining time to finish the dwell block (in hundredths of a second)..

Variable	CNC	PLC	DNC	(*section 13.18*)
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 drilling, mill or lathe type withdrawal iscompleted.
RIP	R	R	R	Linear theoretical feedrate resulting from the next loop (in mm/min).
TEMPIn	R	R	R	It returns the temperature in tenths of a degree detected by the PT100.
TIPPRB	R	R	R	PROBE cycle being executed.
PANEDI	R	R	R	WINDRAW55 application. Number of screen being executed.
DATEDI	R	R	R	WINDRAW55 application. Number of element (item) being executed.
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.
FBDIF(X-C)	R	R	R	For monitoring the difference between the coordinates (position values) of the first and second feedback on the oscilloscope.
CYCLEV	R	R	R	It indicates on the conversational model, the tab number being displayed at a particular moment.
CYCEDI	R	R	R	It indicates on the conversational model, the cycle or screen number being displayed at a particular moment.
DISBLO	R	R	R	It indicates the value of the total programmed distance in blocks with look-ahead.
MIXPO(X..C)	R	R	R	Variable that indicates the axis position with the combined feedback.
FLWAC(X..C)	R	R	R	Variable that indicates the real error considering the second feedback.
RADIOC	R	-	-	It indicates the programmed radius value when executing a G15 R.

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.

SUMMARY OF PLC COMMANDS

PLC resources

Inputs:	I 1/512
Outputs:	O 1/512
User marks:	M 1/2000 and 2049/3999
Arithmetic flag marks:	M 2003
Clock marks:	M 2009/2024
Fixed status marks:	M 2046/2047
Marks associated with messages:	M 4000/4254
Marks associated with errors:	M 4500/4627
Screen marks:	M 4700/4955
CNC communication marks:	M 5000/5957
Timers:	T 1/512
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 1/2000	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	1/512	Timers
C	1/256	Counters
B	0/31 R 1/499	Register bit

Flank detection instructions

DFU (Up flank detection)	I 1/512
DFD (Down flank detection)	O 1/512
	M 1/5957

Comparing instructions

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	

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.

Conditional binary action 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	I 1/512
= RES	O 1/512
= CPL	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.
MACHMOVE	M5012	With coordinate transformation, moves coincide with machine axes.
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.
TOOLMOVE	M5021	With coordinate transformation, moves coincide with tool axes.
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.
TOOLINSP	M5050	Tool inspection available in MC, MCO, TC, TCO modes
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.
CAXSEROK	M5055	Drive ready to work as "C" axis. When it is shared with the spindle. Lathe.
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 to the next hole after drilling or mill type threading withdrawal.
RETRACYC	M5065	It indicates tha the Z axis has stopped before starting the withdrawal.
SETTMEM	M5066	It can activate an error during a tool change.
RESTMEM	M5067	It cancels the error state of the CNC.

Axis logic inputs.

	Axis 1 Axis 2	Axis 3 Axis 4	Axis 5 Axis 6	Axis 7	
LIMIT+	M5100 M5150	M5200 M5250	M5300 M5350	M5400	Travel limit overrun. Stops the axes and the spindle. Displays the error.
LIMIT-*	M5101 M5151	M5201 M5251	M5301 M5351	M5401	Travel limit overrun. Stops the axes and the spindle. Displays the error.
DECEL*	M5102 M5152	M5202 M5252	M5302 M5352	M5402	Home switch pressed.
INHIBIT*	M5103 M5153	M5203 M5253	M5303 M5353	M5403	Inhibits axis movement.
MIRROR*	M5104 M5154	M5204 M5254	M5304 M5354	M5404	Applies mirror image.
SWITCH*	M5105 M5155	M5205 M5255	M5305 M5355	M5405	Swaps commands (axes with 1 drive)
DRO*	M5106 M5156	M5206 M5256	M5306 M5356	M5406	DRO axis. (DRO*=1 and SERVOON*=0).
SERVO*ON	M5107 M5157	M5207 M5257	M5307 M5357	M5407	Servo signal. (=1) closes the position loop
AXIS+*	M5108 M5158	M5208 M5258	M5308 M5358	M5408	Moves the axis in JOG mode. Similar to JOG keys.
AXIS-*	M5109 M5159	M5209 M5259	M5309 M5359	M5409	Moves the axis in JOG mode. Similar to JOG keys.
SPENA*	M5110 M5160	M5210 M5260	M5310 M5360	M5410	With Sercos. Speed enable signal of the drive.
DRENA*	M5111 M5161	M5211 M5261	M5311 M5361	M5411	With Sercos. Drive enable signal of the drive.
SYNCHRO*	M5112 M5162	M5212 M5262	M5312 M5362	M5412	Couples the axis with the one indicated by SYNCHRO*.
ELIMINA*	M5113 M5163	M5213 M5263	M5313 M5363	M5413	It does not display the axis and cancels the feedback alarms.
SMOTOF*	M5114 M5164	M5214 M5264	M5314 M5364	M5414	Cancels the SMOTIME filter, a.m.p. SMOTIME (P58).
LIM*OFF	M5115 M5165	M5215 M5265	M5315 M5365	M5415	It ignores the software limits.
MANINT*	M5116 M5166	M5216 M5266	M5316 M5366	M5416	Activate the additive handwheel in each axis.

Spindle logic inputs.

Main		Second		
LIMIT+S	M5450	LIMIT+S2	M5475	Travel limit overrun. Stops the axes and the spindle. Displays the error.
LIMIT -S	M5451	LIMIT -S2	M5476	Travel limit overrun. Stops the axes and the spindle. Displays the error.
DECELS	M5452	DECELS2	M5477	Home switch pressed.
SPDLEINH	M5453	SPDLEIN2	M5478	Outputs a zero command for the spindle.
SPDLEREV	M5454	SPDLERE2	M5479	Reverses the spindle turning direction.
SMOTOFS	M5455	SMOTOFS2	M5480	Cancels the SMOTIME filter, s.m.p. SMOTIME (P46)
SERVOSON	M5457	SERVOSO2	M5482	Servo signal. (=1) to move the spindle in closed loop (M19).
GEAR1	M5458	GEAR12	M5483	Spindle gear 1 selected.
GEAR2	M5459	GEAR22	M5484	Spindle gear 2 selected.
GEAR3	M5460	GEAR32	M5485	Spindle gear 3 selected.
GEAR4	M5461	GEAR42	M5486	Spindle gear 4 selected.
SPENAS	M5462	SPENAS2	M5487	With Sercos. Speed enable signal of the drive.
DRENAS	M5463	DRENAS2	M5488	With Sercos. Drive enable signal of the drive.
PLCFM19	M5464	PLCFM192	M5489	Rapid synchronization feedrate, in M19.
M19FEED	R505	M19FEED2	R507	Rapid synchronization feedrate, in M19.
PLCCNTL	M5465	PLCCNTL2	M5490	Spindle controlled directly by the PLC.
SANALOG	R504	SANALOG2	R506	Spindle velocity command. Only for spindle controlled by PLC.
ELIMIS	M5456	ELIMIS2	M5481	The CNC does not display the spindle although it keeps controlling it.

Logic inputs of the auxiliary spindle

SPENAAS	M5449	With Sercos. Speed enable signal of the drive.
DRENAAS	M5448	With Sercos. Drive enable signal of the drive.
PLCCNTAS	M5056	The auxiliary spindle is controlled directly by the PLC.
SANALOAS	R509	Velocity command for the auxiliary spindle. Only for spindle controlled by PLC.
ELIMIAS	M5062	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.
CUSTOM	M5512	Selected work mode (=0) M or T; (=1) MC, MCO, TC or TCO
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.
S2MAIN	M5536	Indicates which spindle is controlled by the CNC.
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).
DM45	M5548	Auxiliary spindle or live tool selected (M45).
TANGACT	M5558	G45 active
SYNCPOSI	M5559	Spindles synchronized in position.
SYNSPEED	M5560	Spindles synchronized in speed.
SYNCHRON	M5561	G77S function selected (spindle synchronization).
SERPLCAC	M5562	Requested change of parameter set and gear ratio in progress.
RETRACT	M5567	It indicates that the drilling or mill type threading operation or the lathe type threading withdrawal has stopped.
TMINEM	M5569	It indicates tha the CNC has detected an error during tool change.

Axis logic outputs.

	Axis 1 **Axis 2**	**Axis 3** **Axis 4**	**Axis 5** **Axis 6**	**Axis 7**	
ENABLE*	M5600 M5650	M5700 M5750	M5800 M5850	M5900	Enables axis movement.
DIR*	M5601 M5651	M5701 M5751	M5801 M5851	M5901	Indicate axis moving direction.
REFPOIN*	M5602 M5652	M5702 M5752	M5802 M5852	M5902	Home search done.
DRSTAF*	M5603 M5653	M5703 M5753	M5803 M5853	M5903	With Sercos. They indicate servo drive status.
DRSTAS*	M5604 M5654	M5704 M5754	M5804 M5854	M5904	With Sercos. They indicate servo drive status.
ANT*	M5606 M5656	M5706 M5756	M5806 M5856	M5906	If distance < MINMOVE (P54), ANT*=1
INPOS*	M5607 M5657	M5707 M5757	M5807 M5857	M5907	Axis in position.

Spindle logic outputs.

Main		**Second**		
ENABLES	M5950	ENABLES2	M5975	Enables spindle movement.
DIRS	M5951	DIRS2	M5976	Spindle turning direction
REFPOINS	M5952	REFPOIS2	M5977	The spindle has been already referenced (homed).
DRSTAFS	M5953	DRSTAFS2	M5978	With Sercos. They indicate servo drive status.
DRSTASS	M5954	DRSTASS2	M5979	With Sercos. They indicate servo drive status.
CAXIS	M5955	CAXIS2	M5980	"C" axis active.
REVOK	M5956	REVOK2	M5981	Spindle rpm correspond to programmed speed.
INPOSS	M5957	INPOSS2	M5982	Spindle in position.

Logic outputs of the auxiliary spindle

DRSTAFAS	M5557	With Sercos. They indicate servo drive status.
DRSTASAS	M5556	

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	

2-DIGIT BCD CODE OUTPUT CONVERSION TABLE

S Programmed	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

S Programmed	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

S Programmed	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

I.

2-digit BCD code output conversion table

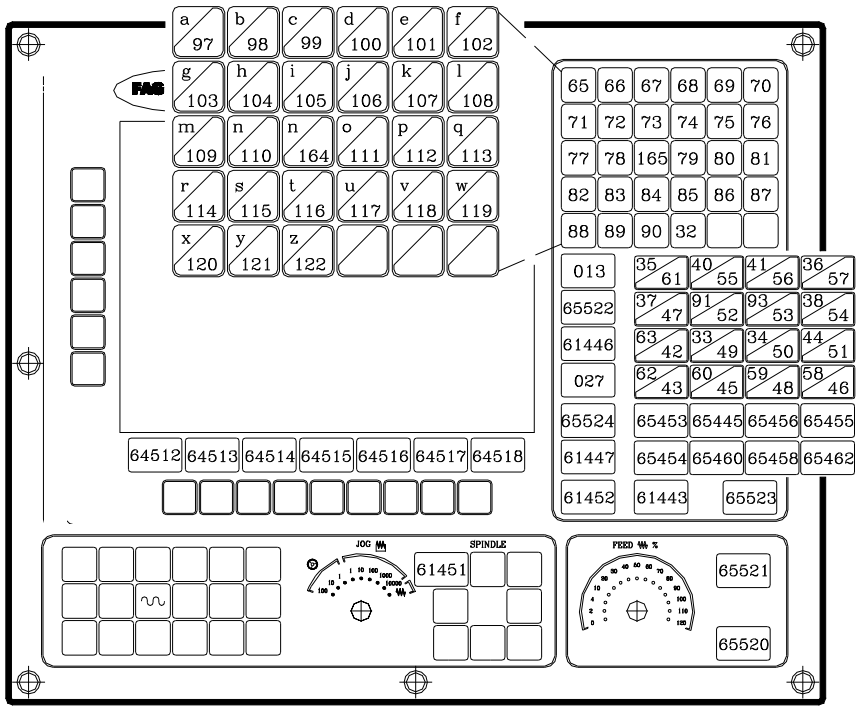
FAGOR

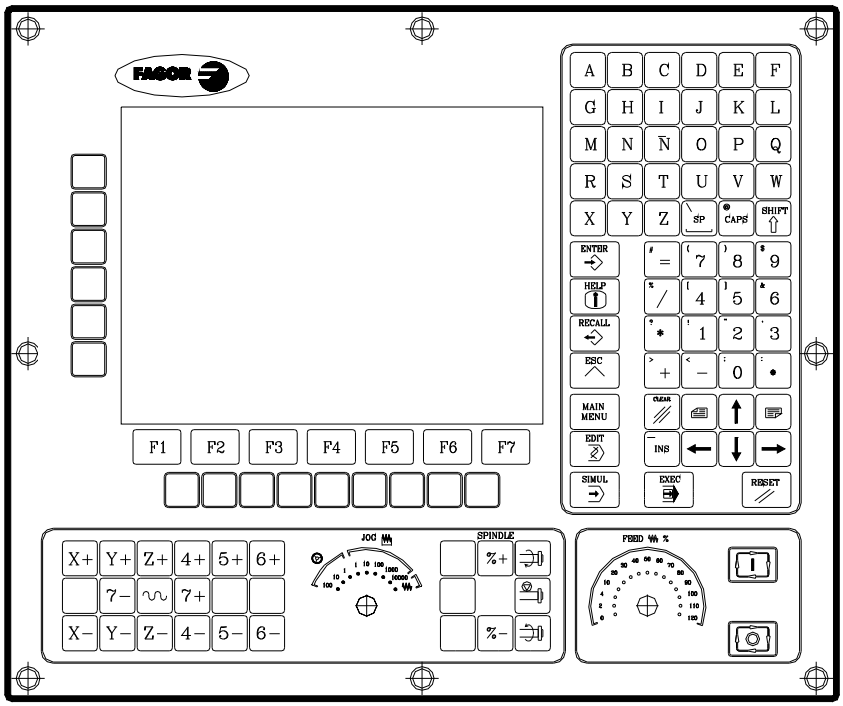
CNC 8055
CNC 8055i

SOFT: V01.6x

KEY CODE

Alphanumeric keyboard and monitor (M-T models)





FAGOR

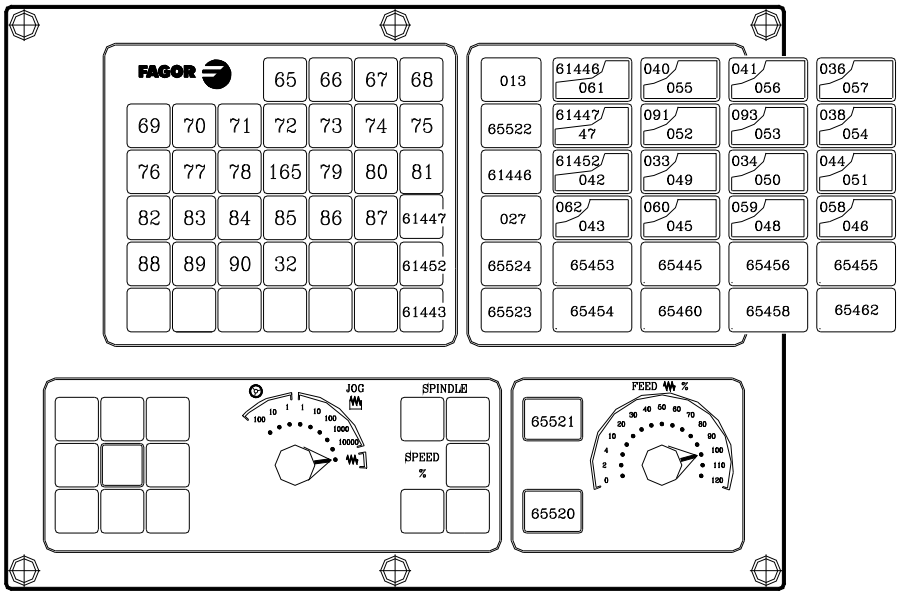
CNC 8055
CNC 8055i

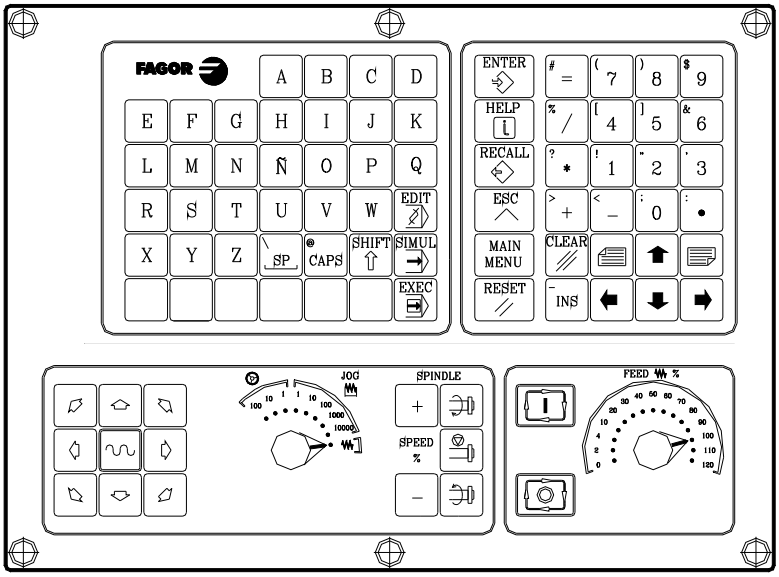
SOFT: V01.6X

J.

Key code

Alphanumeric operator panel (M-T models)



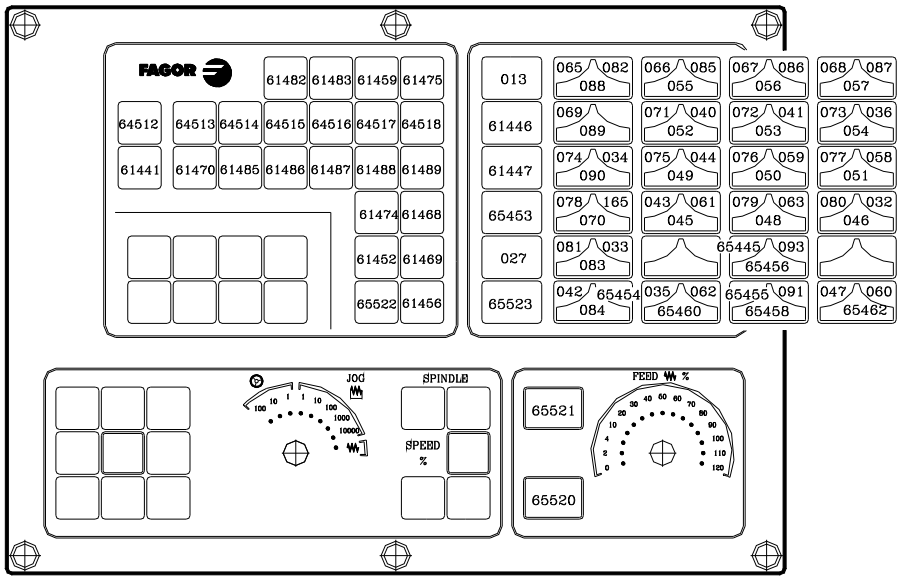


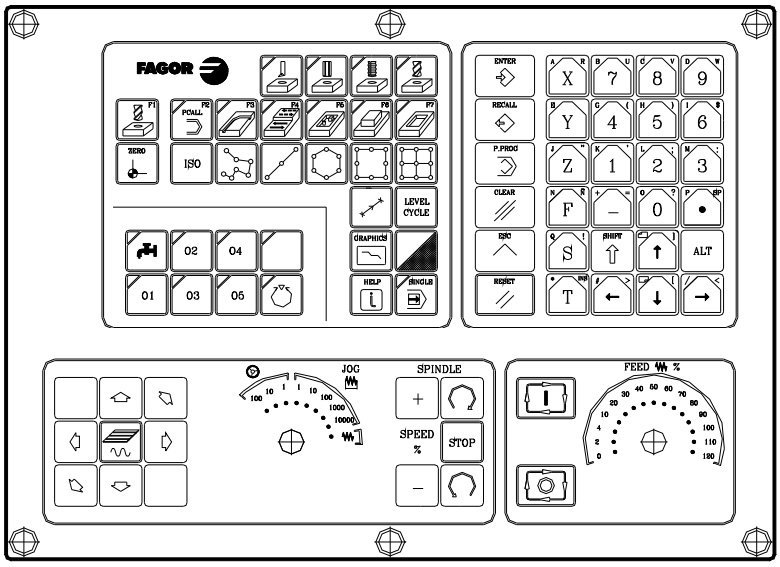
FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

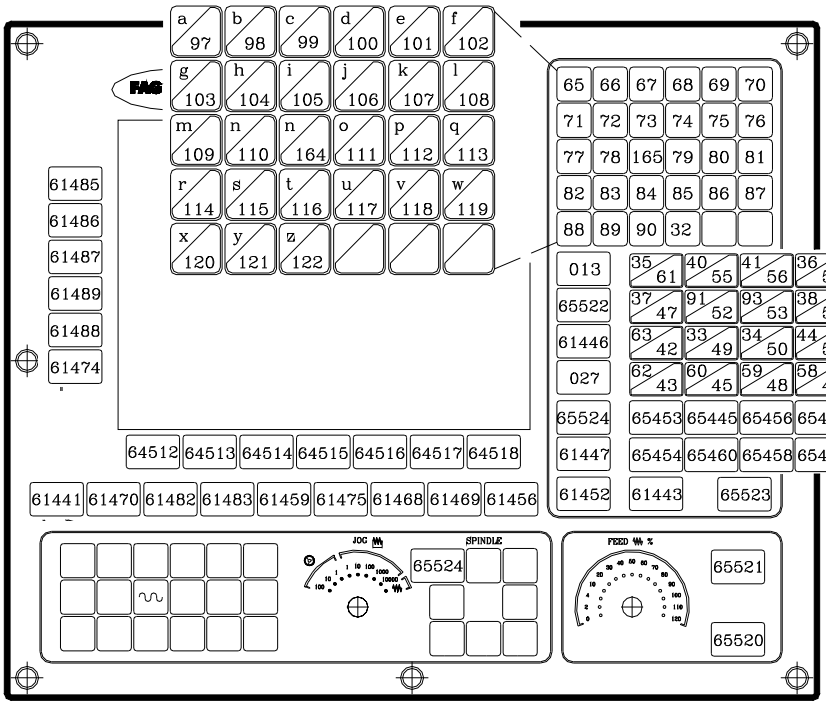
MC operator panel

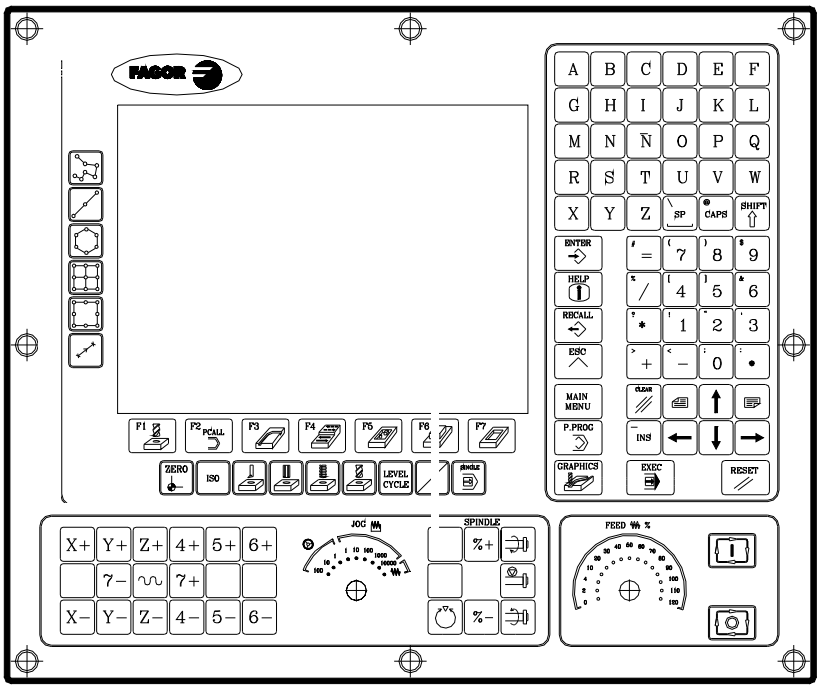




J.

Key code



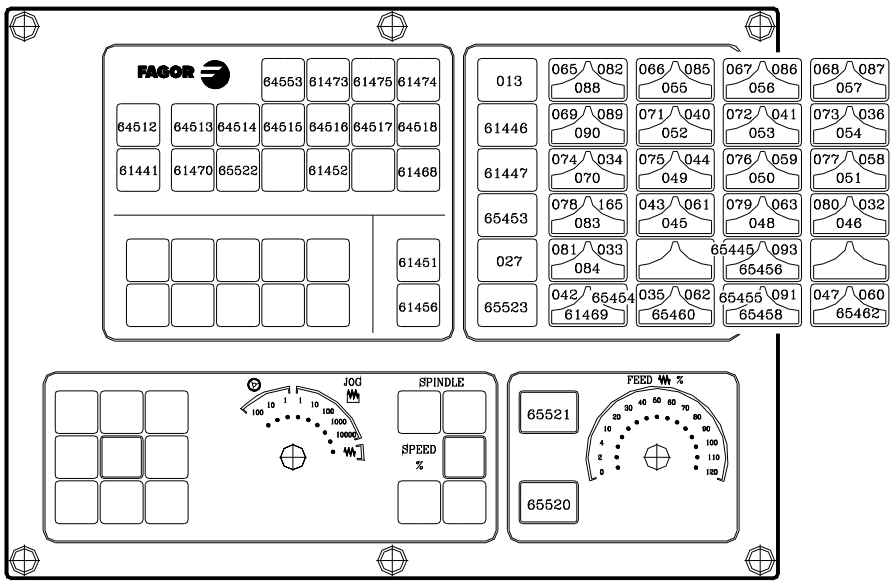


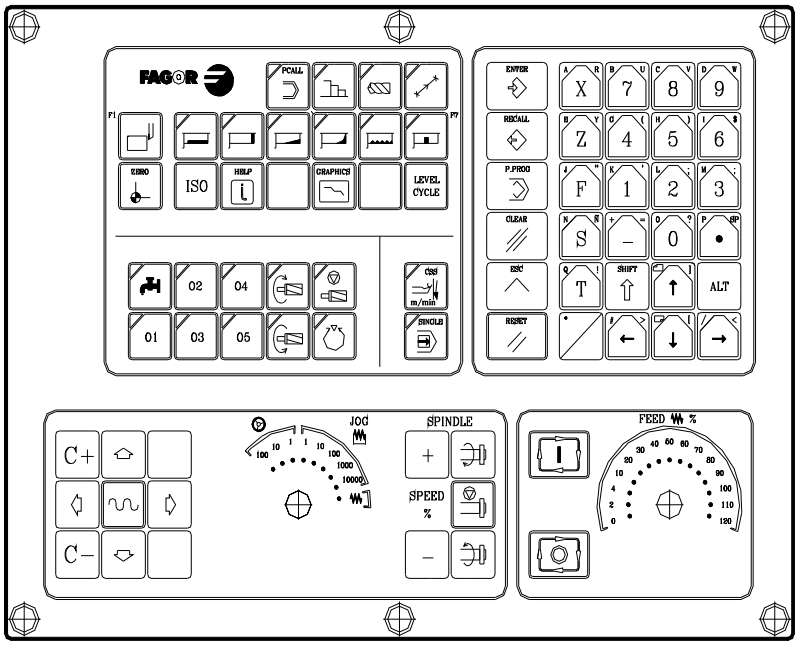
FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

TC operator panel





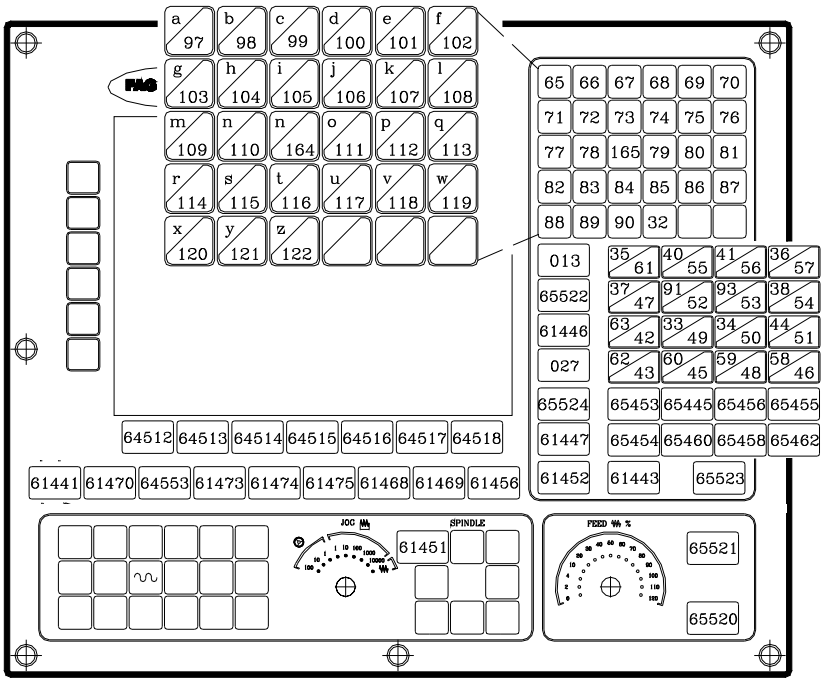
FAGOR

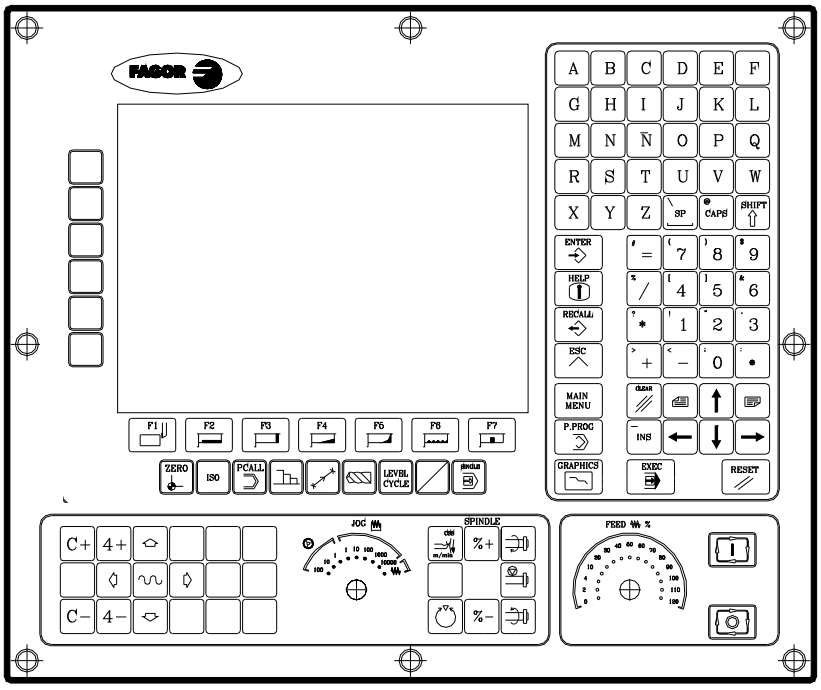
CNC 8055
CNC 8055i

SOFT: V01.6X

J.

Key code



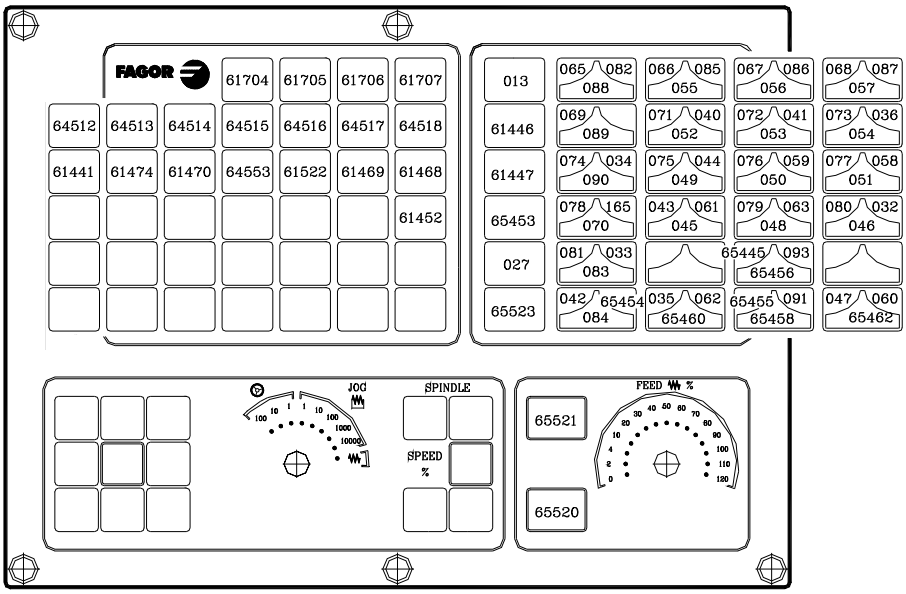


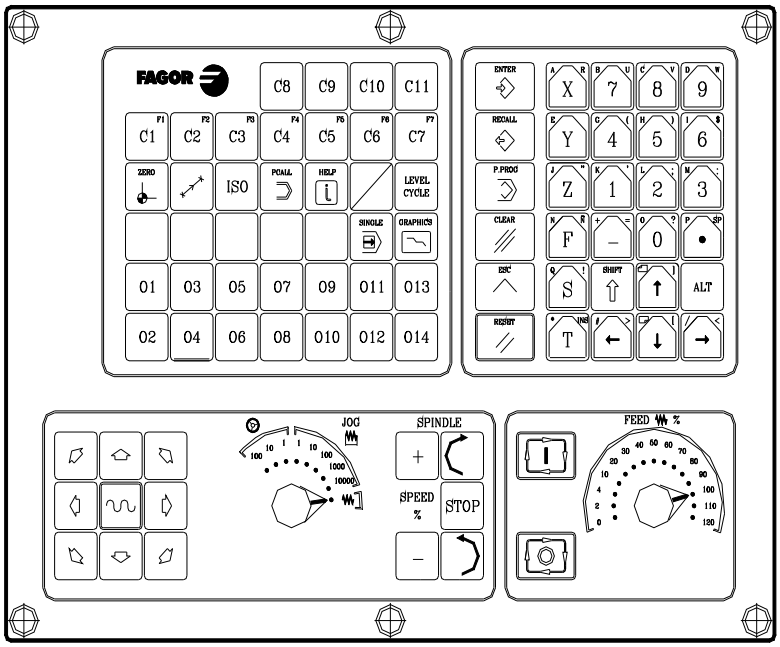
FAGOR

CNC 8055
CNC 8055i

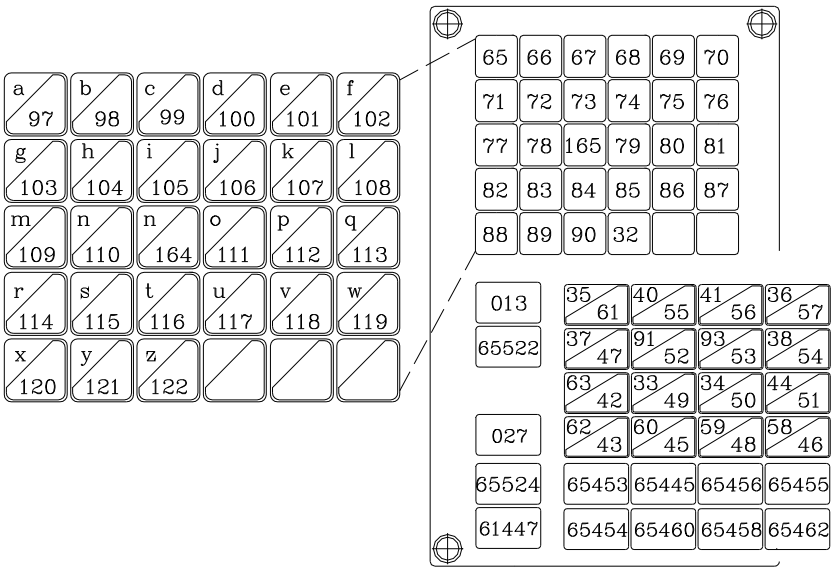
SOFT: V01.6x

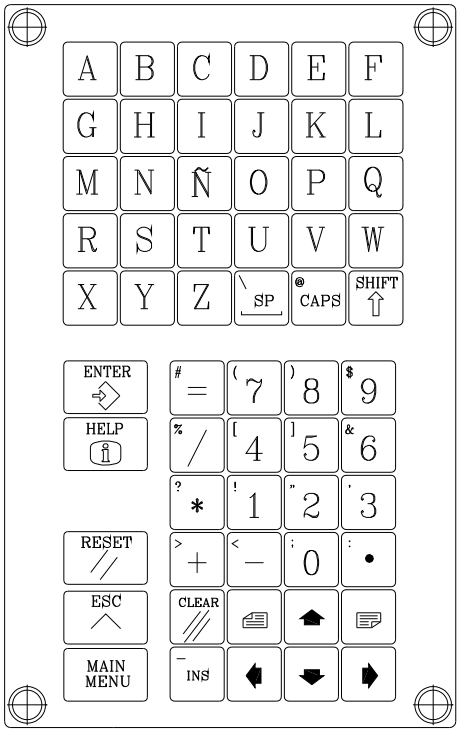
MCO/TCO operator panel





Alphanumeric keyboard



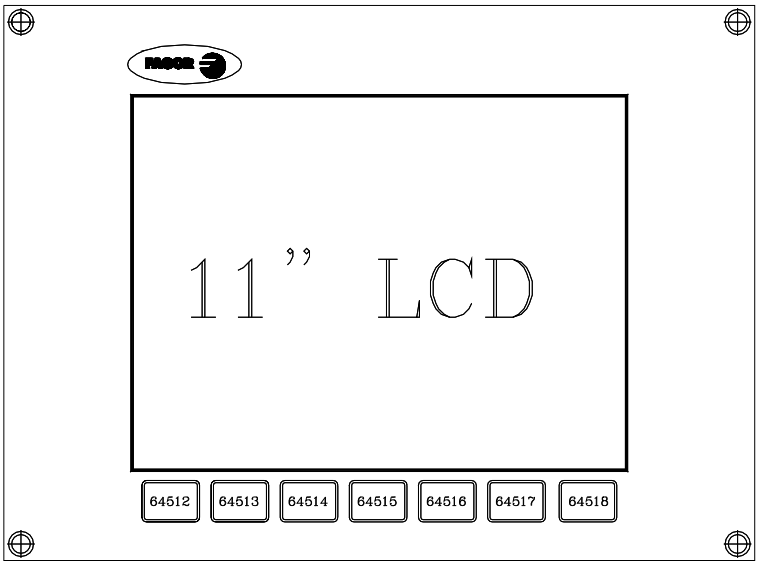


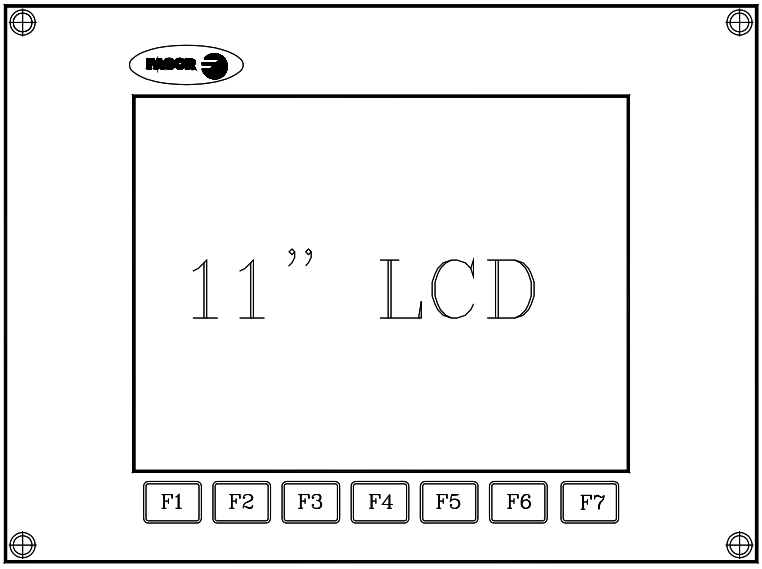
FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

11" LCD Monitor





J.

Key code

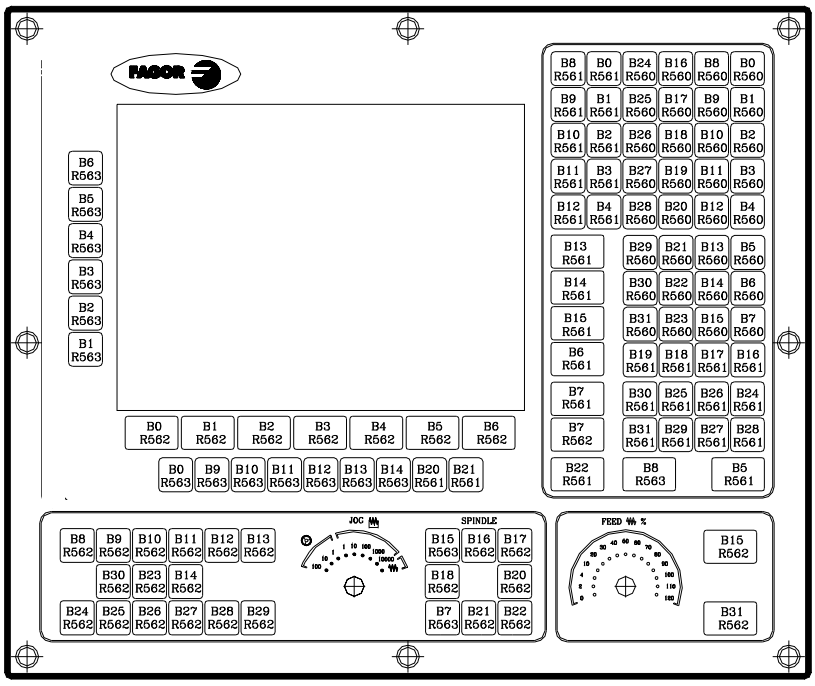
FAGOR

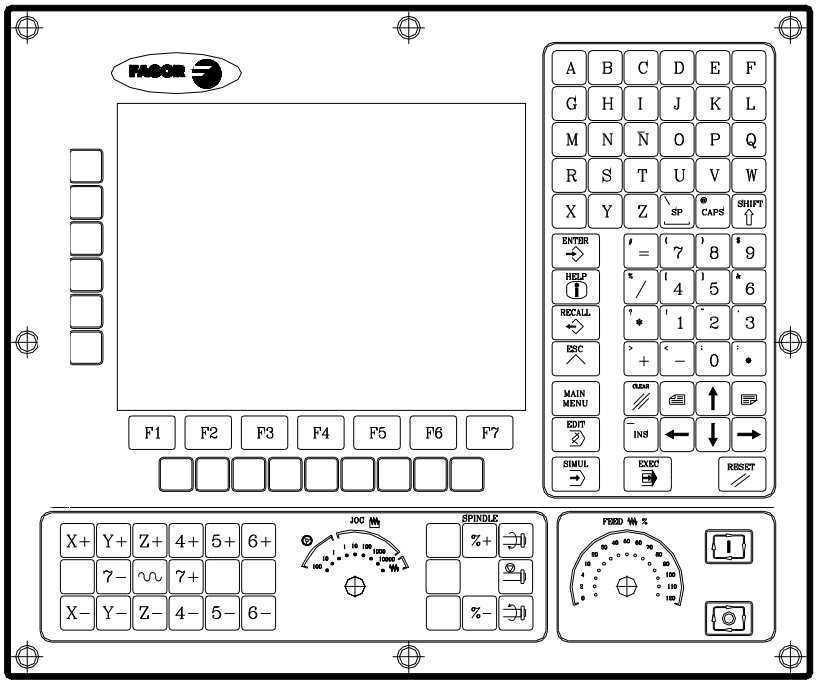
CNC 8055
CNC 8055i

SOFT: V01.6x

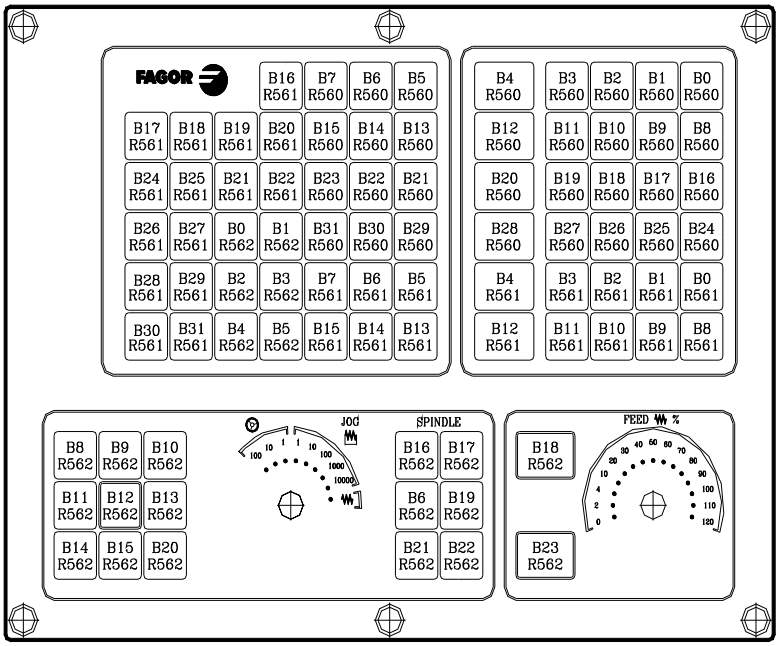
LOGIC OUTPUTS OF KEY STATUS

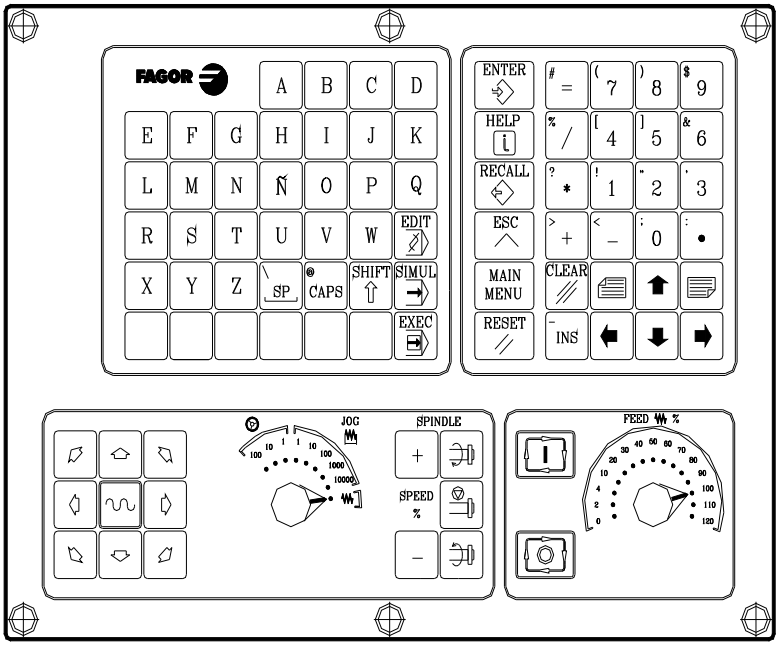
Alphanumeric keyboard and monitor (M-T models)



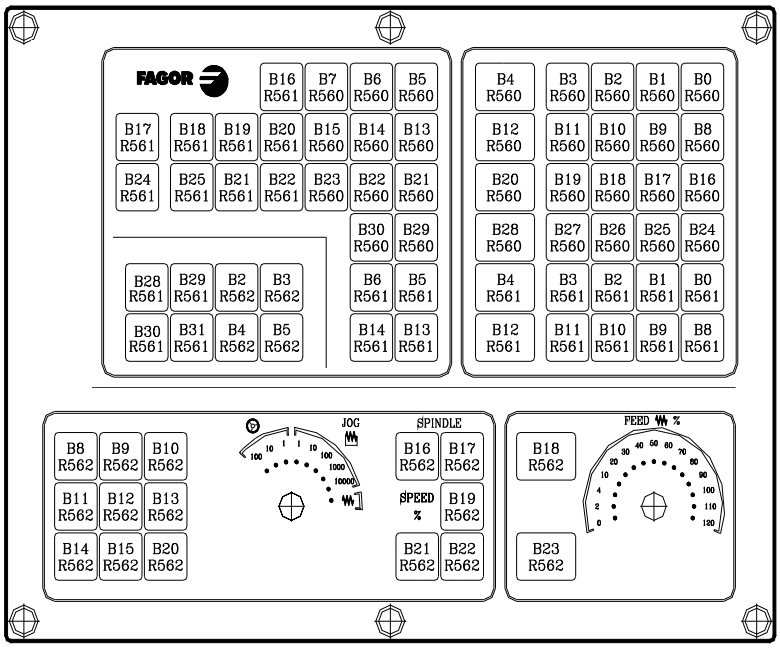


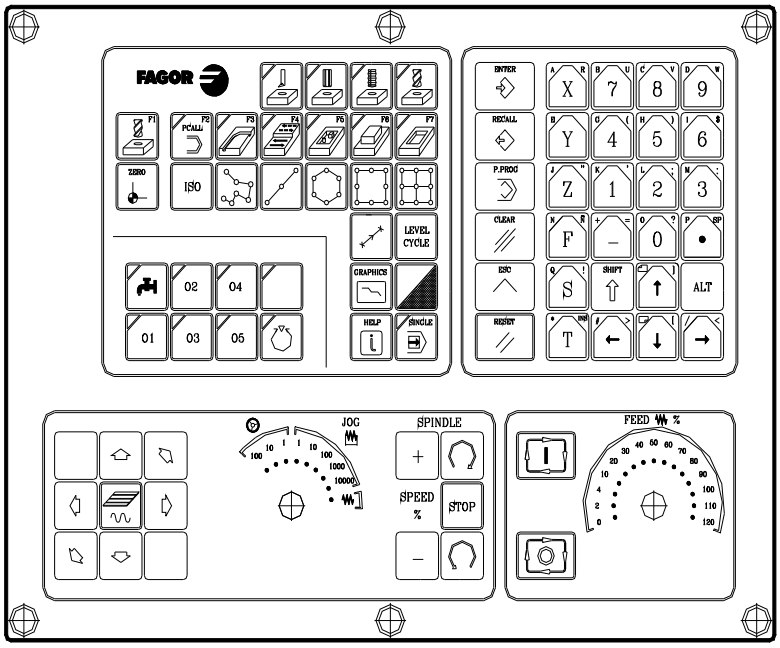
Alphanumeric operator panel (M-T models)

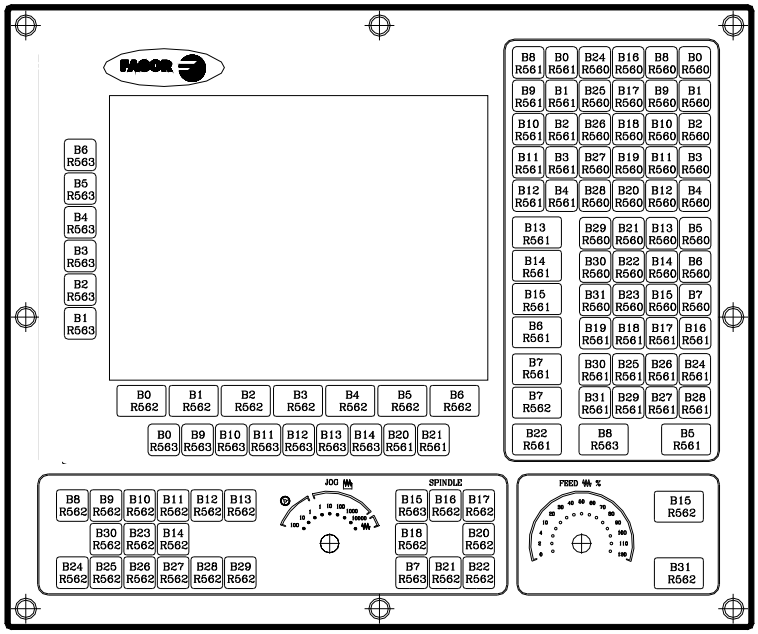


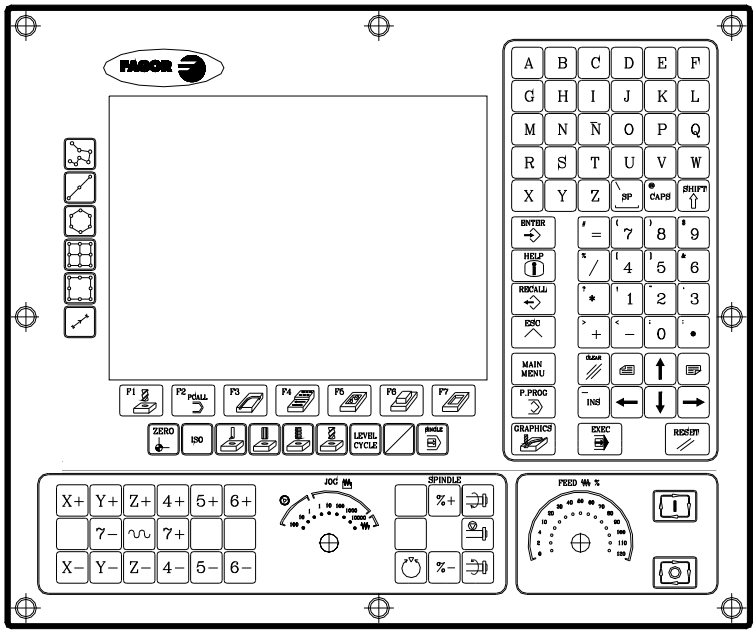


MC operator panel







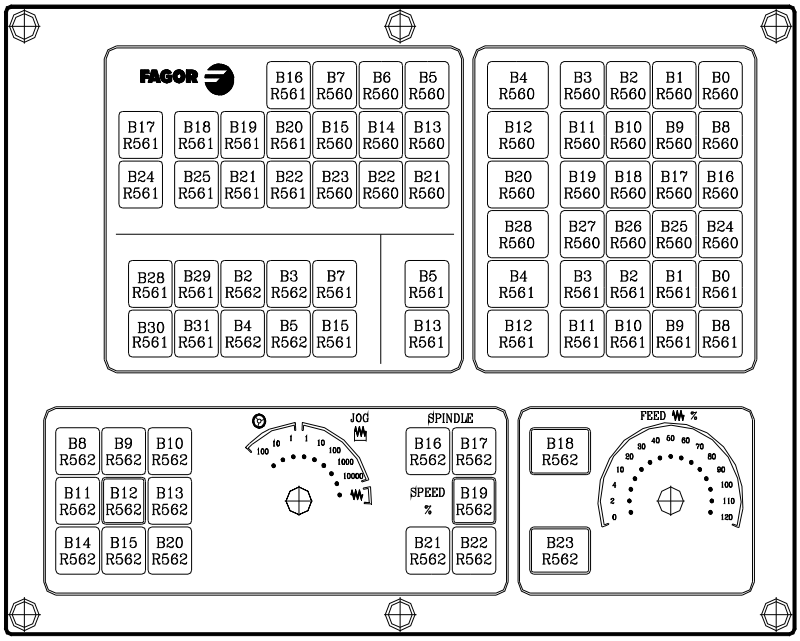


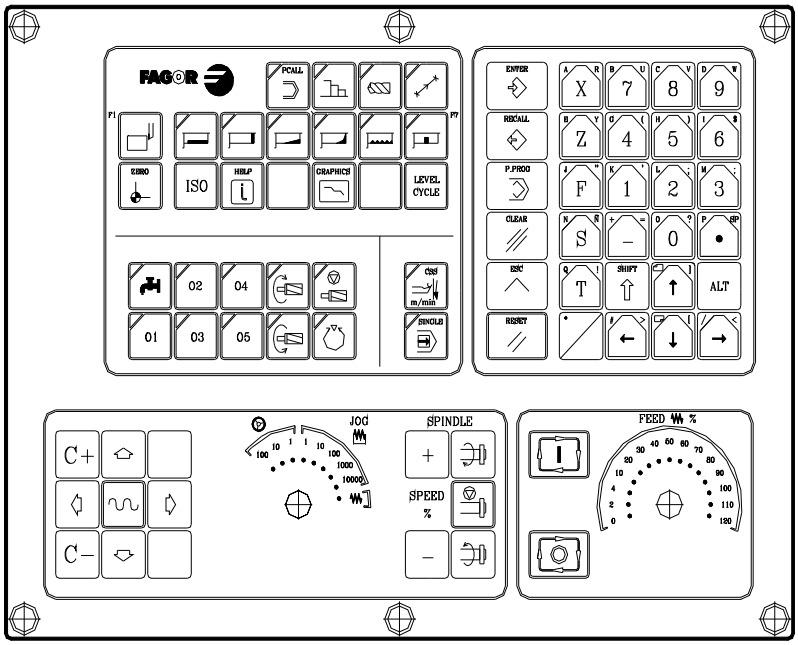
FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

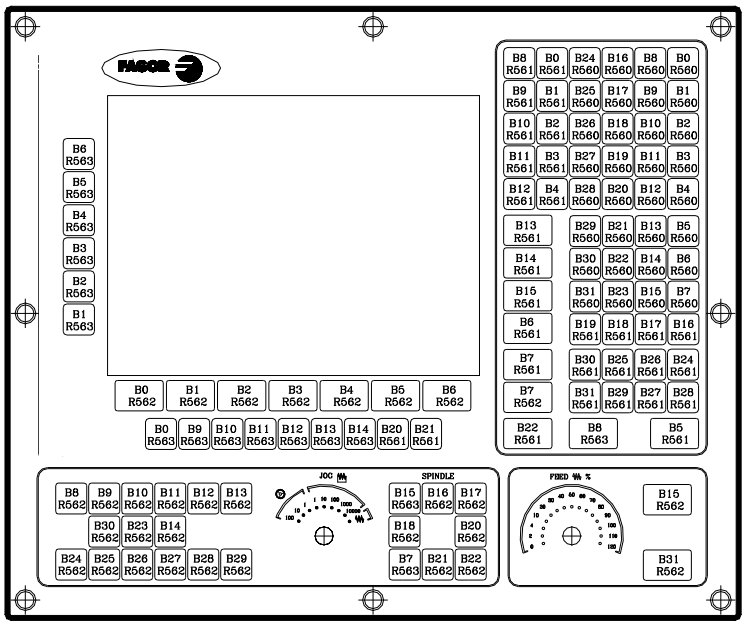
TC operator panel

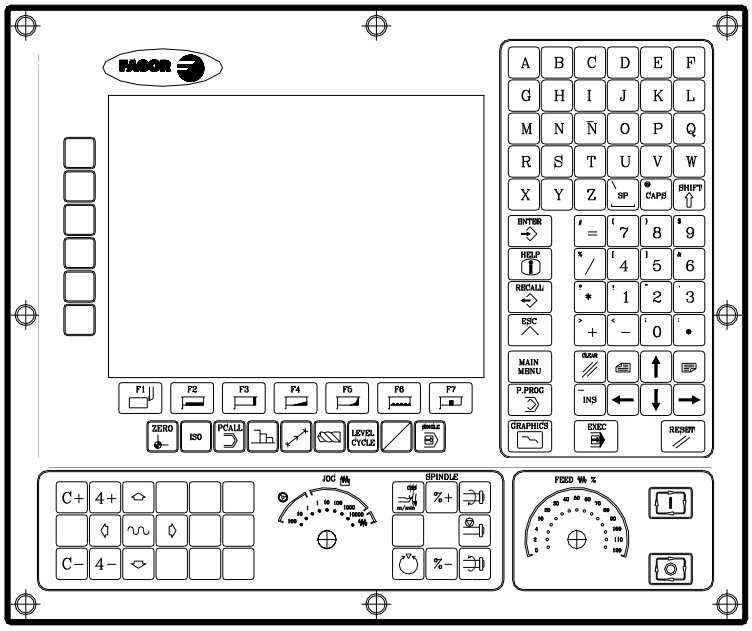




K.

Logic outputs of key status



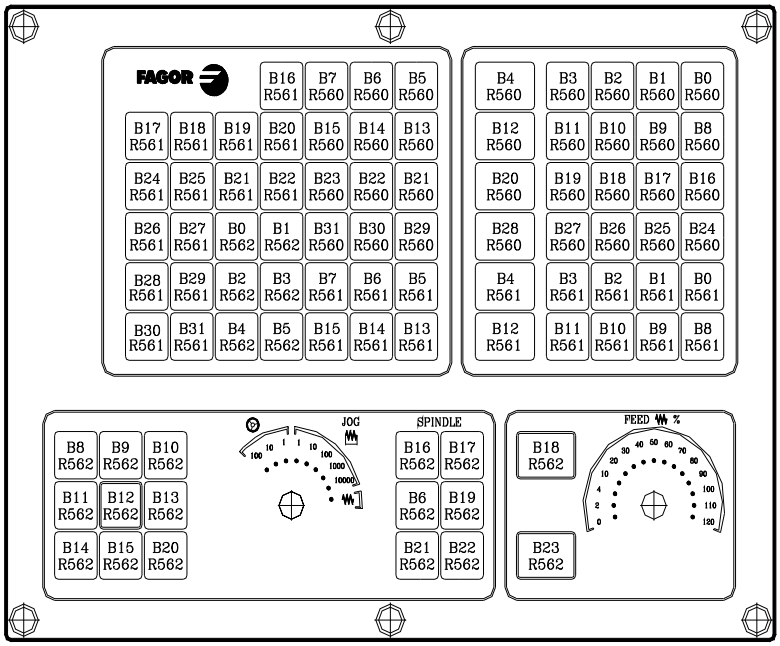


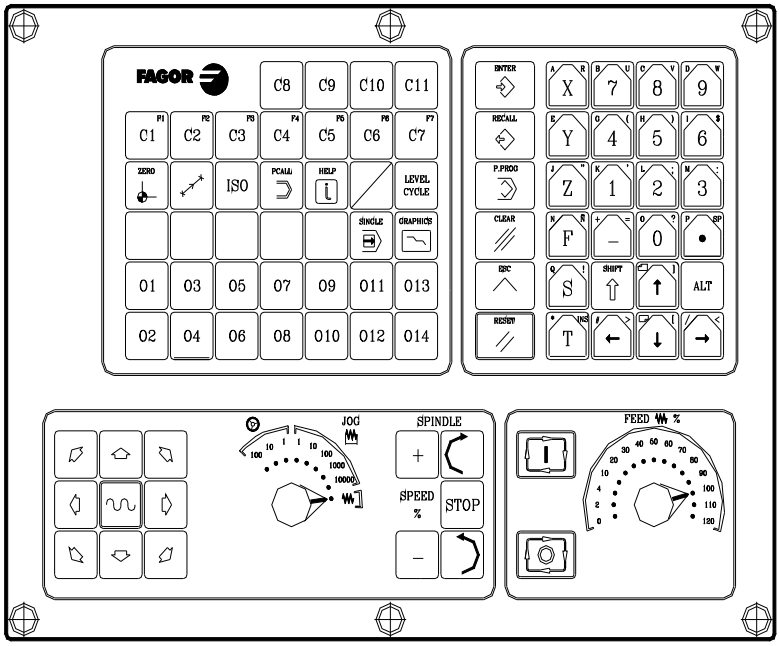
FAGOR

CNC 8055
CNC 8055i

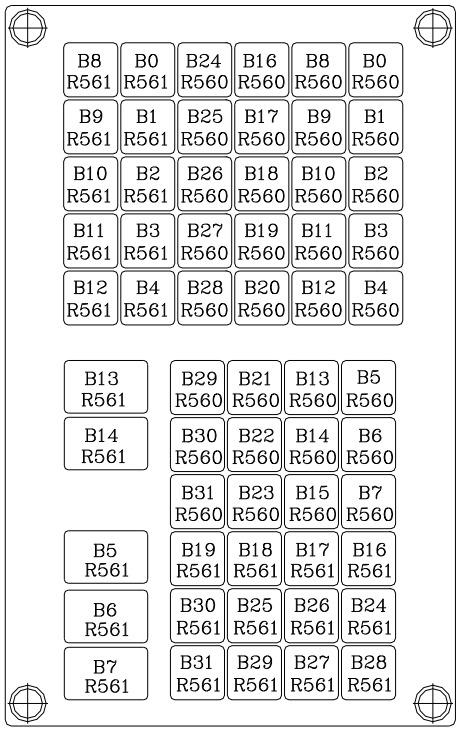
SOFT: V01.6X

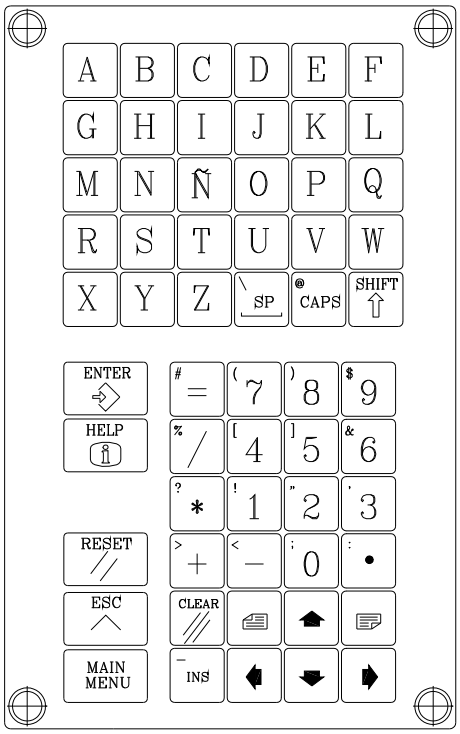
MCO/TCO operator panel



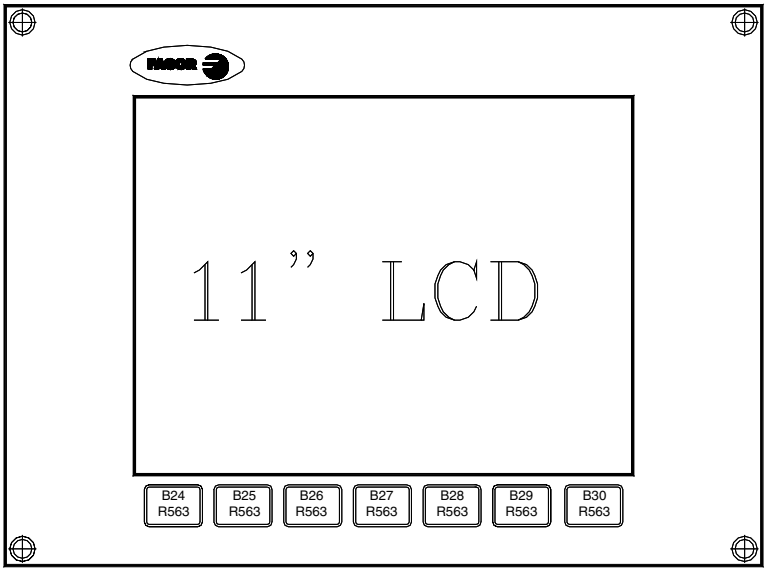


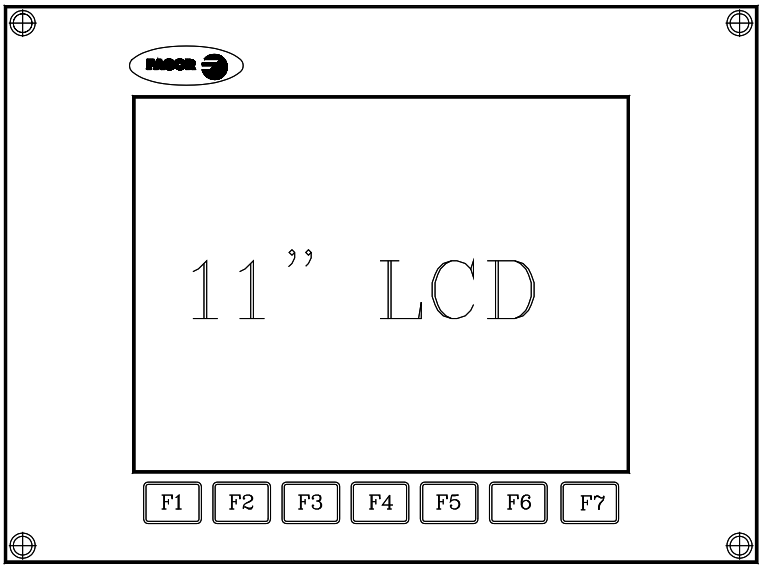
Alphanumeric keyboard





11" LCD Monitor





K.

Logic outputs of key status

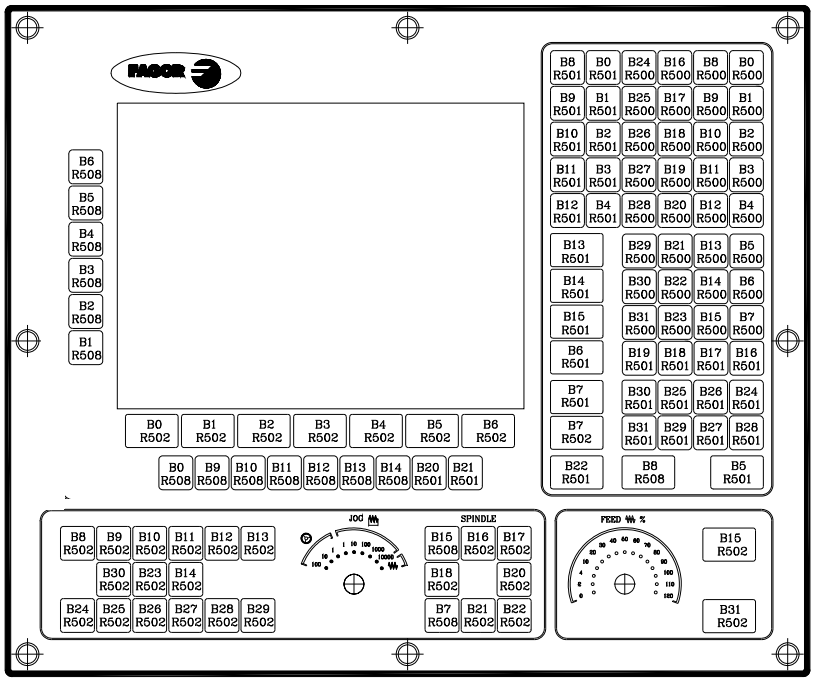
FAGOR

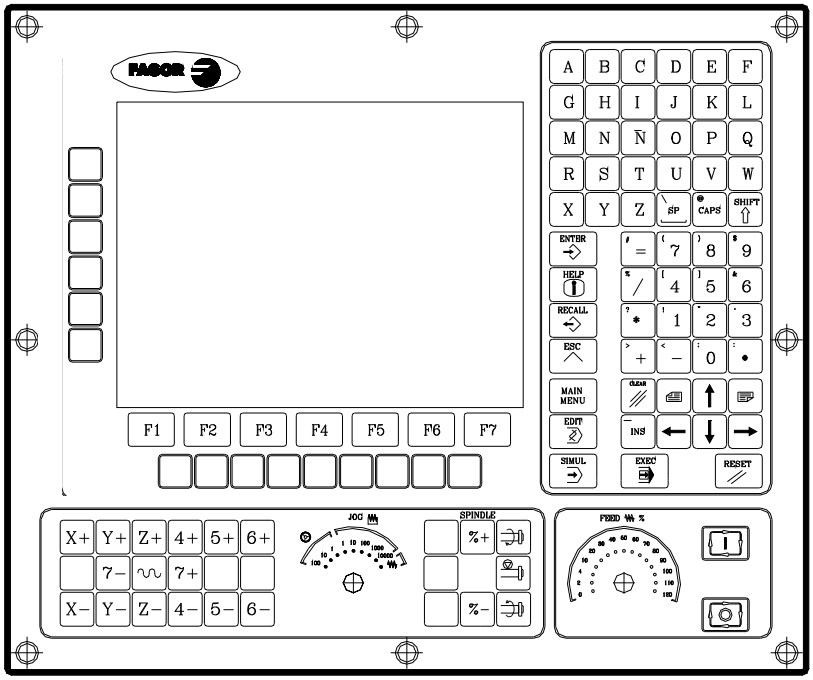
CNC 8055
CNC 8055i

SOFT: V01.6x

KEY INHIBITING CODES

Alphanumeric keyboard and monitor (M-T models)



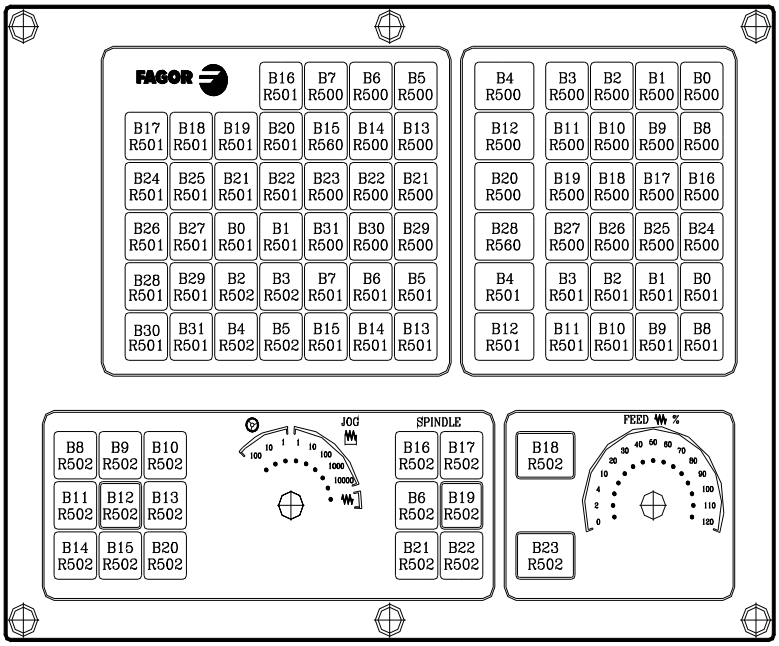


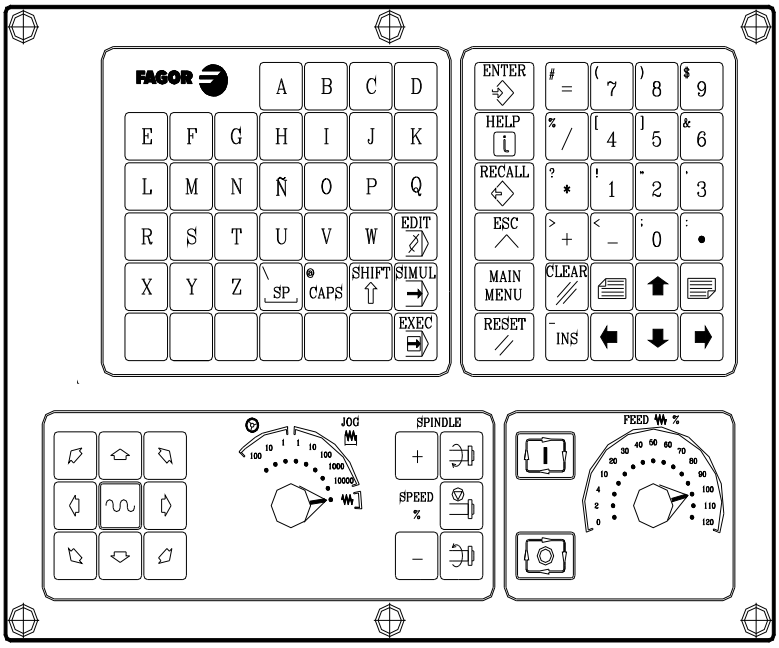
FAGOR

CNC 8055
CNC 8055i

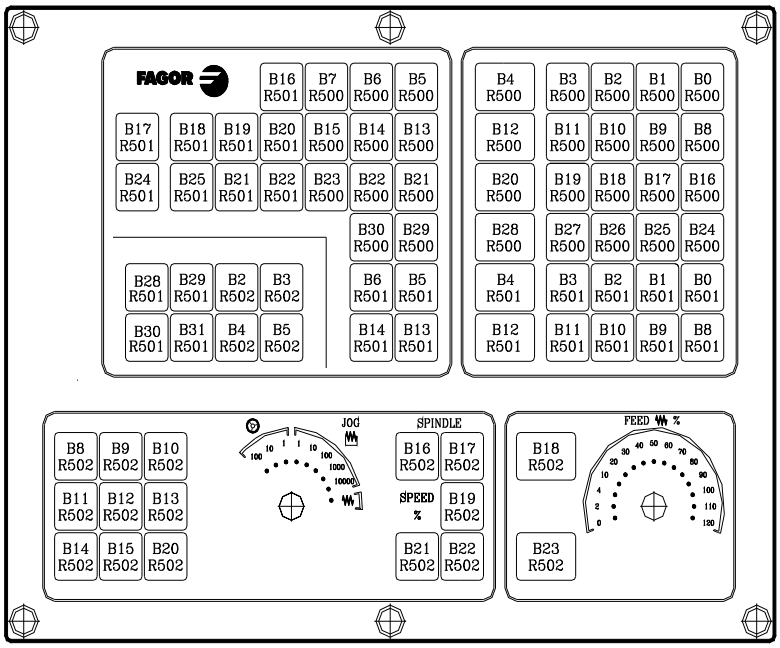
SOFT: V01.6X

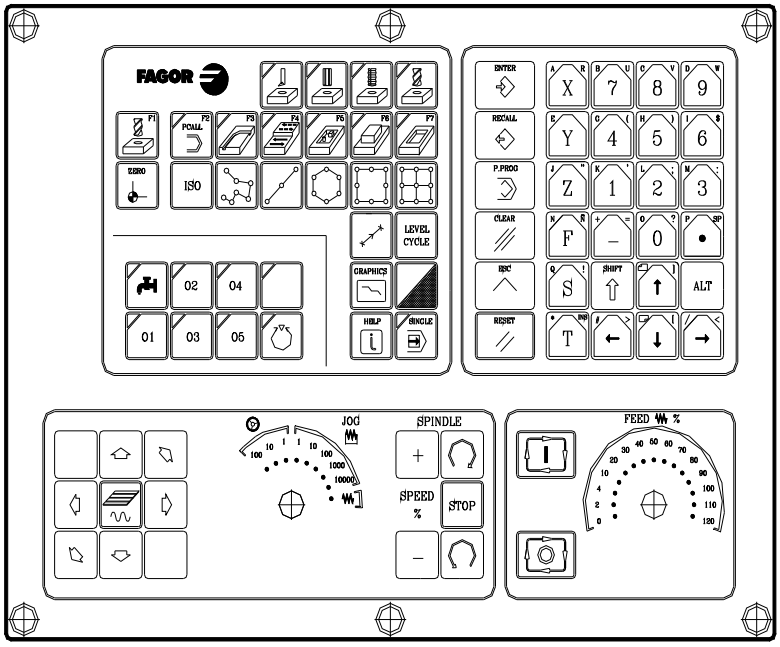
Alphanumeric operator panel (M-T models)

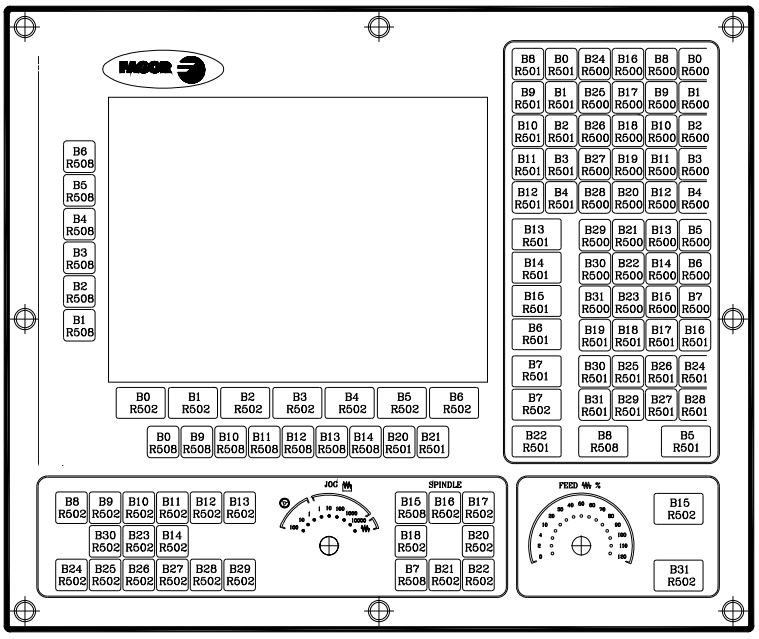


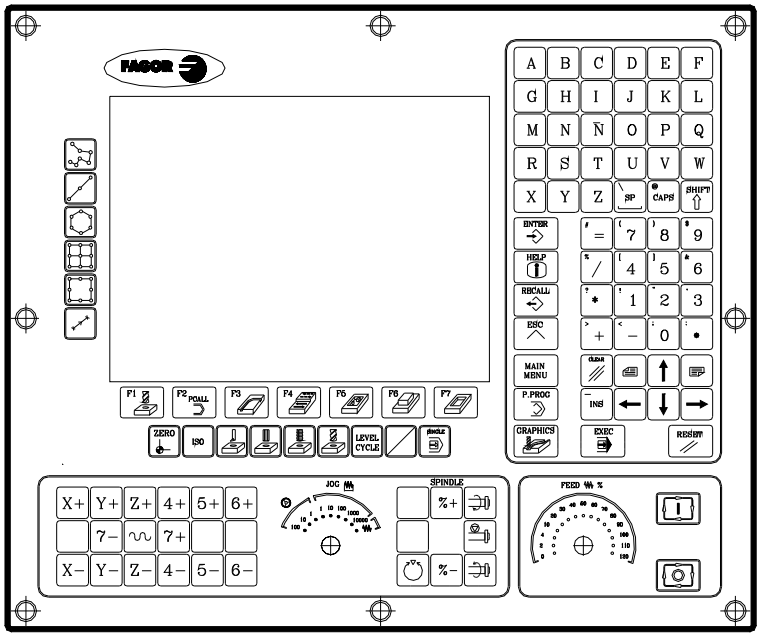


MC operator panel







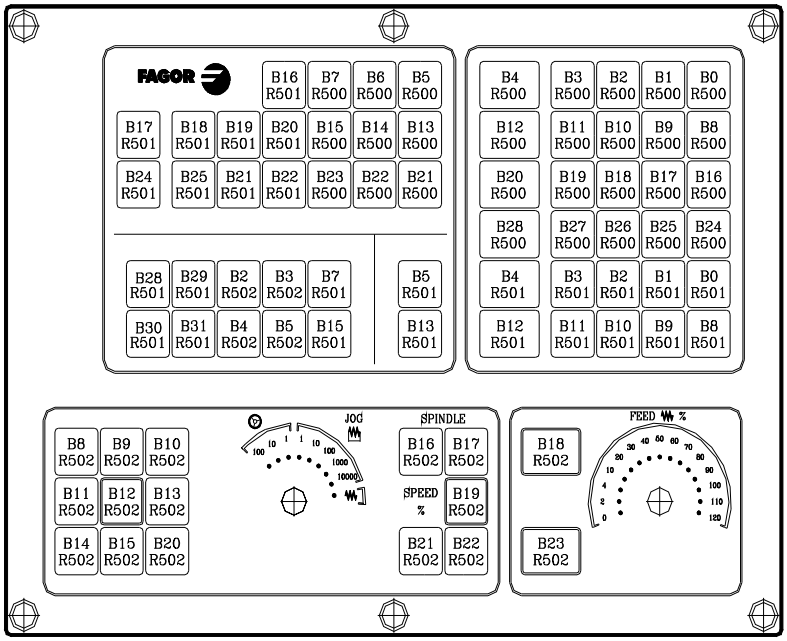


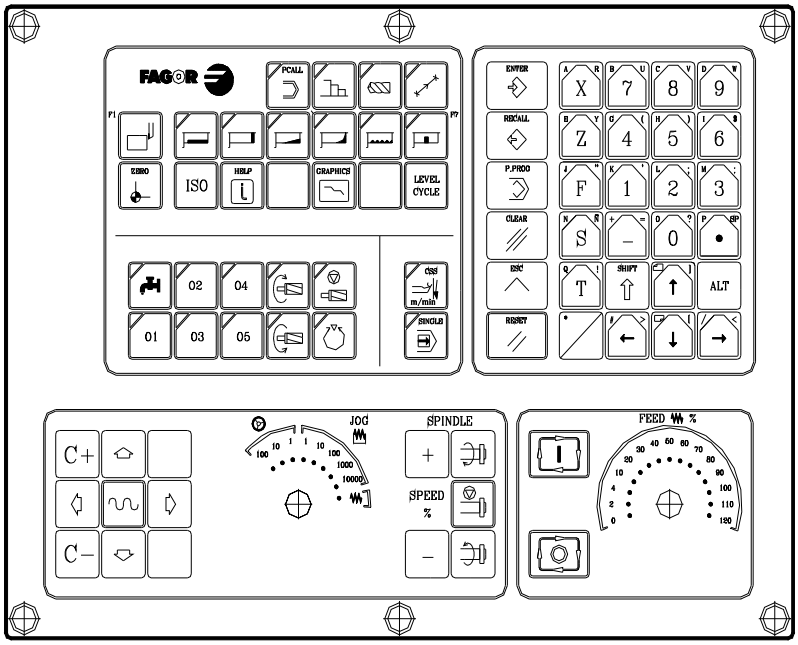
FAGOR

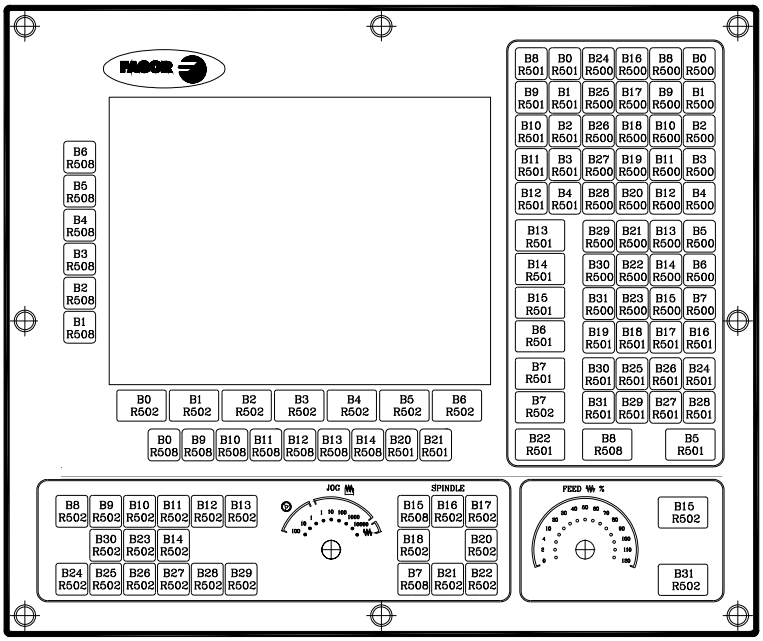
CNC 8055
CNC 8055i

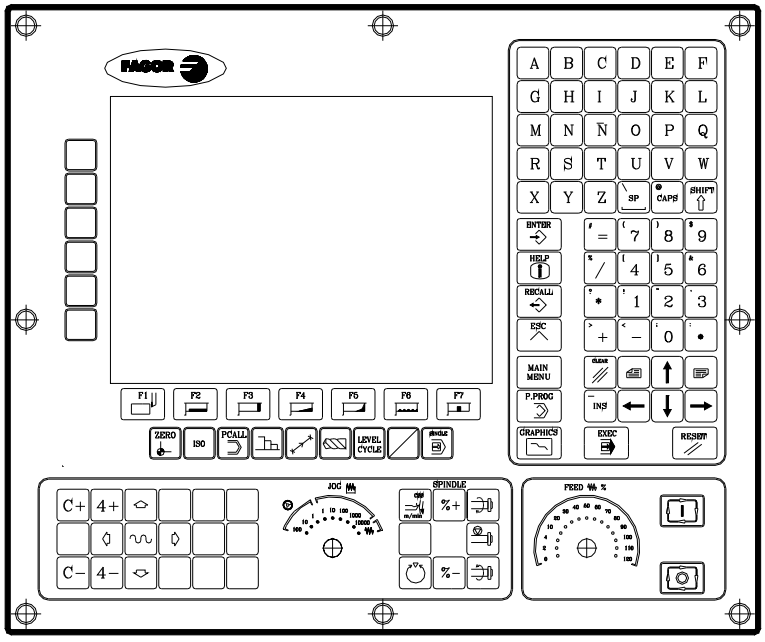
SOFT: V01.6X

TC operator panel







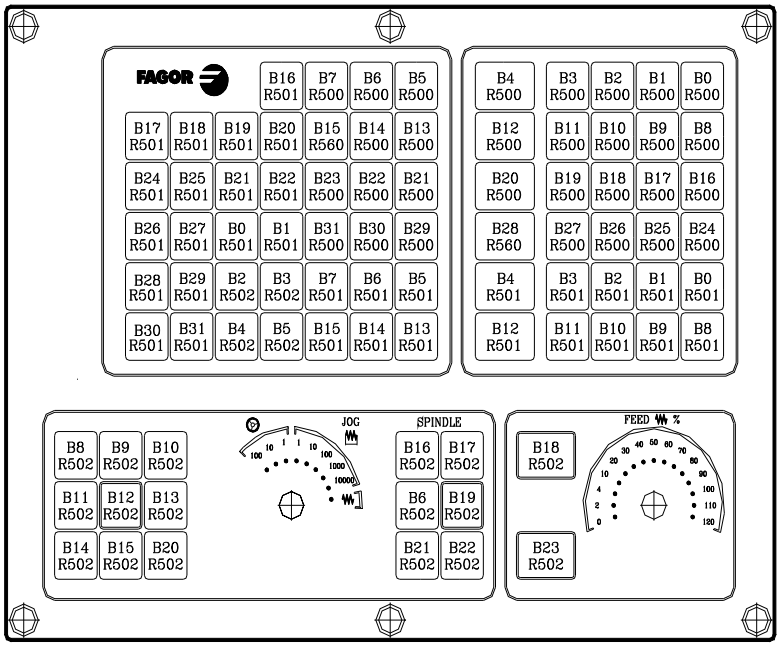


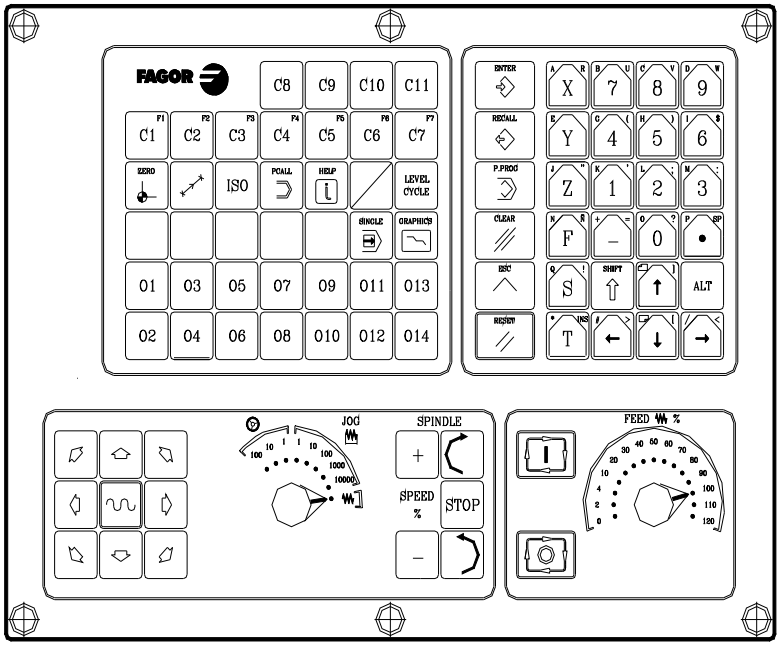
FAGOR

CNC 8055
CNC 8055i

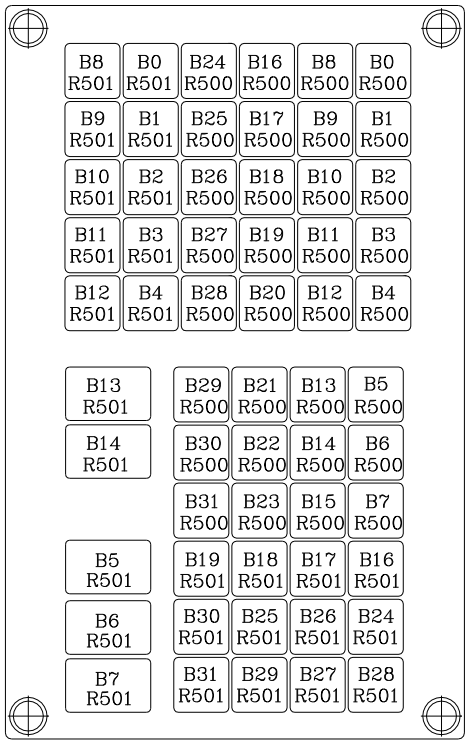
SOFT: V01.6X

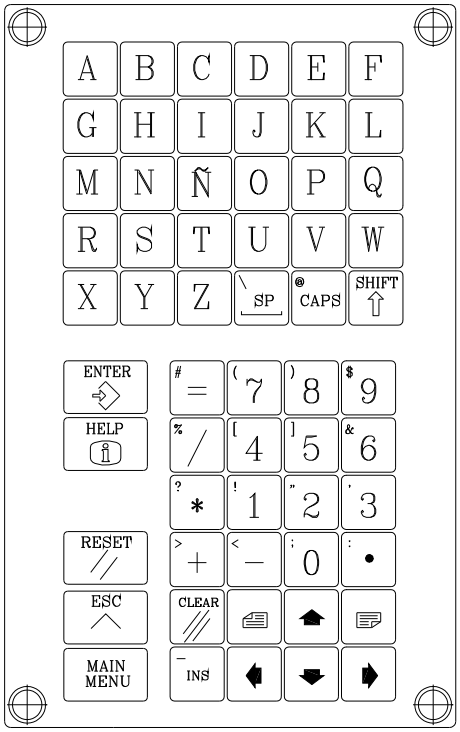
MCO/TCO operator panel





Alphanumeric keyboard



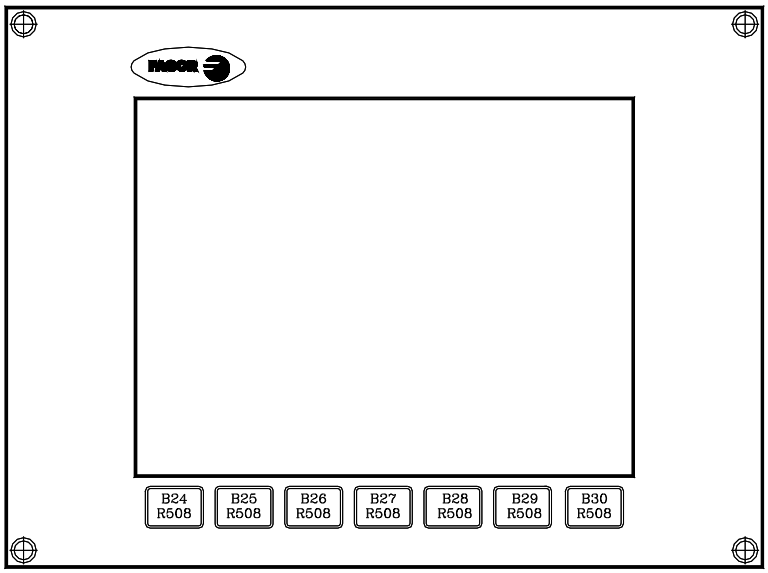


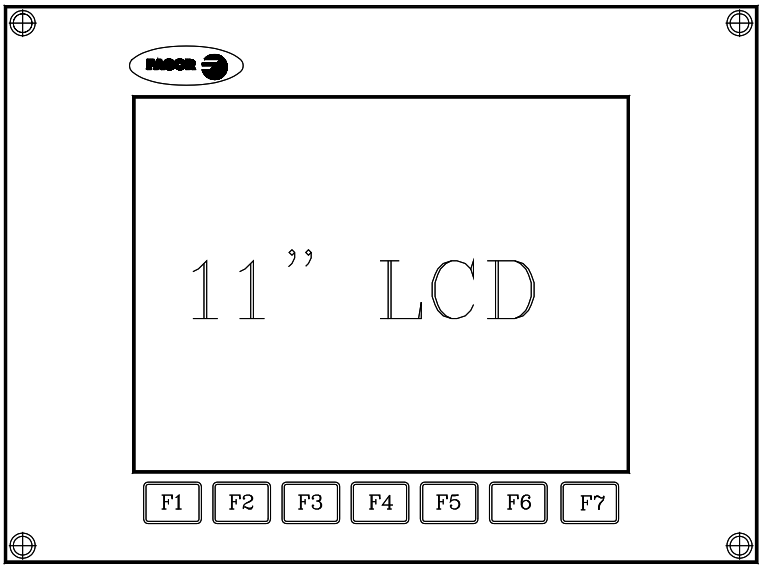
FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

11" LCD Monitor





L.

Key inhibiting codes

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

MACHINE PARAMETER SETTING CHART

General machine parameters

P0 P1		P50 P51		P100 P101		P150 P151	
P2 P3		P52 P53		P102 P103		P152 P153	
P4 P5		P54 P55		P104 P105		P154 P155	
P6 P7		P56 P57		P106 P107		P156 P157	
P8 P9		P58 P59		P108 P109		P158 P159	
P10 P11		P60 P61		P110 P111		P160 P161	
P12 P13		P62 P63		P112 P113		P162 P163	
P14 P15		P64 P65		P114 P115		P164 P165	
P16 P17		P66 P67		P116 P117		P166 P167	
P18 P19		P68 P69		P118 P119		P168 P169	
P20 P21		P70 P71		P120 P121		P170 P171	
P22 P23		P72 P73		P122 P123		P172 P173	
P24 P25		P74 P75		P124 P125		P174 P175	
P26 P27		P76 P77		P126 P127		P176 P177	
P28 P29		P78 P79		P128 P129		P178 P179	
P30 P31		P80 P81		P130 P131		P180 P181	
P32 P33		P82 P83		P132 P133		P182 P183	
P34 P35		P84 P85		P134 P135		P184 P185	
P36 P37		P86 P87		P136 P137		P186 P187	
P38 P39		P88 P89		P138 P139		P188 P189	
P40 P41		P90 P91		P140 P141		P190 P191	
P42 P43		P92 P93		P142 P143		P192 P193	
P44 P45		P94 P95		P144 P145		P194 P195	
P46 P47		P96 P97		P146 P147		P196 P197	
P48 P49		P98 P99		P148 P149		P198 P199	

_____ *axis machine parameters*

P0 P1		P50 P51		P100 P101		P150 P151	
P2 P3		P52 P53		P102 P103		P152 P153	
P4 P5		P54 P55		P104 P105		P154 P155	
P6 P7		P56 P57		P106 P107		P156 P157	
P8 P9		P58 P59		P108 P109		P158 P159	
P10 P11		P60 P61		P110 P111		P160 P161	
P12 P13		P62 P63		P112 P113		P162 P163	
P14 P15		P64 P65		P114 P115		P164 P165	
P16 P17		P66 P67		P116 P117		P166 P167	
P18 P19		P68 P69		P118 P119		P168 P169	
P20 P21		P70 P71		P120 P121		P170 P171	
P22 P23		P72 P73		P122 P123		P172 P173	
P24 P25		P74 P75		P124 P125		P174 P175	
P26 P27		P76 P77		P126 P127		P176 P177	
P28 P29		P78 P79		P128 P129		P178 P179	
P30 P31		P80 P81		P130 P131		P180 P181	
P32 P33		P82 P83		P132 P133		P182 P183	
P34 P35		P84 P85		P134 P135		P184 P185	
P36 P37		P86 P87		P136 P137		P186 P187	
P38 P39		P88 P89		P138 P139		P188 P189	
P40 P41		P90 P91		P140 P141		P190 P191	
P42 P43		P92 P93		P142 P143		P192 P193	
P44 P45		P94 P95		P144 P145		P194 P195	
P46 P47		P96 P97		P146 P147		P196 P197	
P48 P49		P98 P99		P148 P149		P198 P199	

_____ *axis machine parameters*

P0 P1		P50 P51		P100 P101		P150 P151	
P2 P3		P52 P53		P102 P103		P152 P153	
P4 P5		P54 P55		P104 P105		P154 P155	
P6 P7		P56 P57		P106 P107		P156 P157	
P8 P9		P58 P59		P108 P109		P158 P159	
P10 P11		P60 P61		P110 P111		P160 P161	
P12 P13		P62 P63		P112 P113		P162 P163	
P14 P15		P64 P65		P114 P115		P164 P165	
P16 P17		P66 P67		P116 P117		P166 P167	
P18 P19		P68 P69		P118 P119		P168 P169	
P20 P21		P70 P71		P120 P121		P170 P171	
P22 P23		P72 P73		P122 P123		P172 P173	
P24 P25		P74 P75		P124 P125		P174 P175	
P26 P27		P76 P77		P126 P127		P176 P177	
P28 P29		P78 P79		P128 P129		P178 P179	
P30 P31		P80 P81		P130 P131		P180 P181	
P32 P33		P82 P83		P132 P133		P182 P183	
P34 P35		P84 P85		P134 P135		P184 P185	
P36 P37		P86 P87		P136 P137		P186 P187	
P38 P39		P88 P89		P138 P139		P188 P189	
P40 P41		P90 P91		P140 P141		P190 P191	
P42 P43		P92 P93		P142 P143		P192 P193	
P44 P45		P94 P95		P144 P145		P194 P195	
P46 P47		P96 P97		P146 P147		P196 P197	
P48 P49		P98 P99		P148 P149		P198 P199	

_____ *axis machine parameters*

P0 P1		P50 P51		P100 P101		P150 P151	
P2 P3		P52 P53		P102 P103		P152 P153	
P4 P5		P54 P55		P104 P105		P154 P155	
P6 P7		P56 P57		P106 P107		P156 P157	
P8 P9		P58 P59		P108 P109		P158 P159	
P10 P11		P60 P61		P110 P111		P160 P161	
P12 P13		P62 P63		P112 P113		P162 P163	
P14 P15		P64 P65		P114 P115		P164 P165	
P16 P17		P66 P67		P116 P117		P166 P167	
P18 P19		P68 P69		P118 P119		P168 P169	
P20 P21		P70 P71		P120 P121		P170 P171	
P22 P23		P72 P73		P122 P123		P172 P173	
P24 P25		P74 P75		P124 P125		P174 P175	
P26 P27		P76 P77		P126 P127		P176 P177	
P28 P29		P78 P79		P128 P129		P178 P179	
P30 P31		P80 P81		P130 P131		P180 P181	
P32 P33		P82 P83		P132 P133		P182 P183	
P34 P35		P84 P85		P134 P135		P184 P185	
P36 P37		P86 P87		P136 P137		P186 P187	
P38 P39		P88 P89		P138 P139		P188 P189	
P40 P41		P90 P91		P140 P141		P190 P191	
P42 P43		P92 P93		P142 P143		P192 P193	
P44 P45		P94 P95		P144 P145		P194 P195	
P46 P47		P96 P97		P146 P147		P196 P197	
P48 P49		P98 P99		P148 P149		P198 P199	

_____ *axis machine parameters*

P0 P1		P50 P51		P100 P101		P150 P151	
P2 P3		P52 P53		P102 P103		P152 P153	
P4 P5		P54 P55		P104 P105		P154 P155	
P6 P7		P56 P57		P106 P107		P156 P157	
P8 P9		P58 P59		P108 P109		P158 P159	
P10 P11		P60 P61		P110 P111		P160 P161	
P12 P13		P62 P63		P112 P113		P162 P163	
P14 P15		P64 P65		P114 P115		P164 P165	
P16 P17		P66 P67		P116 P117		P166 P167	
P18 P19		P68 P69		P118 P119		P168 P169	
P20 P21		P70 P71		P120 P121		P170 P171	
P22 P23		P72 P73		P122 P123		P172 P173	
P24 P25		P74 P75		P124 P125		P174 P175	
P26 P27		P76 P77		P126 P127		P176 P177	
P28 P29		P78 P79		P128 P129		P178 P179	
P30 P31		P80 P81		P130 P131		P180 P181	
P32 P33		P82 P83		P132 P133		P182 P183	
P34 P35		P84 P85		P134 P135		P184 P185	
P36 P37		P86 P87		P136 P137		P186 P187	
P38 P39		P88 P89		P138 P139		P188 P189	
P40 P41		P90 P91		P140 P141		P190 P191	
P42 P43		P92 P93		P142 P143		P192 P193	
P44 P45		P94 P95		P144 P145		P194 P195	
P46 P47		P96 P97		P146 P147		P196 P197	
P48 P49		P98 P99		P148 P149		P198 P199	

_____ *axis machine parameters*

P0 P1		P50 P51		P100 P101		P150 P151	
P2 P3		P52 P53		P102 P103		P152 P153	
P4 P5		P54 P55		P104 P105		P154 P155	
P6 P7		P56 P57		P106 P107		P156 P157	
P8 P9		P58 P59		P108 P109		P158 P159	
P10 P11		P60 P61		P110 P111		P160 P161	
P12 P13		P62 P63		P112 P113		P162 P163	
P14 P15		P64 P65		P114 P115		P164 P165	
P16 P17		P66 P67		P116 P117		P166 P167	
P18 P19		P68 P69		P118 P119		P168 P169	
P20 P21		P70 P71		P120 P121		P170 P171	
P22 P23		P72 P73		P122 P123		P172 P173	
P24 P25		P74 P75		P124 P125		P174 P175	
P26 P27		P76 P77		P126 P127		P176 P177	
P28 P29		P78 P79		P128 P129		P178 P179	
P30 P31		P80 P81		P130 P131		P180 P181	
P32 P33		P82 P83		P132 P133		P182 P183	
P34 P35		P84 P85		P134 P135		P184 P185	
P36 P37		P86 P87		P136 P137		P186 P187	
P38 P39		P88 P89		P138 P139		P188 P189	
P40 P41		P90 P91		P140 P141		P190 P191	
P42 P43		P92 P93		P142 P143		P192 P193	
P44 P45		P94 P95		P144 P145		P194 P195	
P46 P47		P96 P97		P146 P147		P196 P197	
P48 P49		P98 P99		P148 P149		P198 P199	

_____ *axis machine parameters*

P0 P1		P50 P51		P100 P101		P150 P151	
P2 P3		P52 P53		P102 P103		P152 P153	
P4 P5		P54 P55		P104 P105		P154 P155	
P6 P7		P56 P57		P106 P107		P156 P157	
P8 P9		P58 P59		P108 P109		P158 P159	
P10 P11		P60 P61		P110 P111		P160 P161	
P12 P13		P62 P63		P112 P113		P162 P163	
P14 P15		P64 P65		P114 P115		P164 P165	
P16 P17		P66 P67		P116 P117		P166 P167	
P18 P19		P68 P69		P118 P119		P168 P169	
P20 P21		P70 P71		P120 P121		P170 P171	
P22 P23		P72 P73		P122 P123		P172 P173	
P24 P25		P74 P75		P124 P125		P174 P175	
P26 P27		P76 P77		P126 P127		P176 P177	
P28 P29		P78 P79		P128 P129		P178 P179	
P30 P31		P80 P81		P130 P131		P180 P181	
P32 P33		P82 P83		P132 P133		P182 P183	
P34 P35		P84 P85		P134 P135		P184 P185	
P36 P37		P86 P87		P136 P137		P186 P187	
P38 P39		P88 P89		P138 P139		P188 P189	
P40 P41		P90 P91		P140 P141		P190 P191	
P42 P43		P92 P93		P142 P143		P192 P193	
P44 P45		P94 P95		P144 P145		P194 P195	
P46 P47		P96 P97		P146 P147		P196 P197	
P48 P49		P98 P99		P148 P149		P198 P199	

_____ *axis machine parameters*

P0 P1		P50 P51		P100 P101		P150 P151	
P2 P3		P52 P53		P102 P103		P152 P153	
P4 P5		P54 P55		P104 P105		P154 P155	
P6 P7		P56 P57		P106 P107		P156 P157	
P8 P9		P58 P59		P108 P109		P158 P159	
P10 P11		P60 P61		P110 P111		P160 P161	
P12 P13		P62 P63		P112 P113		P162 P163	
P14 P15		P64 P65		P114 P115		P164 P165	
P16 P17		P66 P67		P116 P117		P166 P167	
P18 P19		P68 P69		P118 P119		P168 P169	
P20 P21		P70 P71		P120 P121		P170 P171	
P22 P23		P72 P73		P122 P123		P172 P173	
P24 P25		P74 P75		P124 P125		P174 P175	
P26 P27		P76 P77		P126 P127		P176 P177	
P28 P29		P78 P79		P128 P129		P178 P179	
P30 P31		P80 P81		P130 P131		P180 P181	
P32 P33		P82 P83		P132 P133		P182 P183	
P34 P35		P84 P85		P134 P135		P184 P185	
P36 P37		P86 P87		P136 P137		P186 P187	
P38 P39		P88 P89		P138 P139		P188 P189	
P40 P41		P90 P91		P140 P141		P190 P191	
P42 P43		P92 P93		P142 P143		P192 P193	
P44 P45		P94 P95		P144 P145		P194 P195	
P46 P47		P96 P97		P146 P147		P196 P197	
P48 P49		P98 P99		P148 P149		P198 P199	

Machine parameters for the main spindle

P0 P1		P50 P51		P100 P101		P150 P151	
P2 P3		P52 P53		P102 P103		P152 P153	
P4 P5		P54 P55		P104 P105		P154 P155	
P6 P7		P56 P57		P106 P107		P156 P157	
P8 P9		P58 P59		P108 P109		P158 P159	
P10 P11		P60 P61		P110 P111		P160 P161	
P12 P13		P62 P63		P112 P113		P162 P163	
P14 P15		P64 P65		P114 P115		P164 P165	
P16 P17		P66 P67		P116 P117		P166 P167	
P18 P19		P68 P69		P118 P119		P168 P169	
P20 P21		P70 P71		P120 P121		P170 P171	
P22 P23		P72 P73		P122 P123		P172 P173	
P24 P25		P74 P75		P124 P125		P174 P175	
P26 P27		P76 P77		P126 P127		P176 P177	
P28 P29		P78 P79		P128 P129		P178 P179	
P30 P31		P80 P81		P130 P131		P180 P181	
P32 P33		P82 P83		P132 P133		P182 P183	
P34 P35		P84 P85		P134 P135		P184 P185	
P36 P37		P86 P87		P136 P137		P186 P187	
P38 P39		P88 P89		P138 P139		P188 P189	
P40 P41		P90 P91		P140 P141		P190 P191	
P42 P43		P92 P93		P142 P143		P192 P193	
P44 P45		P94 P95		P144 P145		P194 P195	
P46 P47		P96 P97		P146 P147		P196 P197	
P48 P49		P98 P99		P148 P149		P198 P199	

Machine parameters for the second spindle

P0 P1		P50 P51		P100 P101		P150 P151	
P2 P3		P52 P53		P102 P103		P152 P153	
P4 P5		P54 P55		P104 P105		P154 P155	
P6 P7		P56 P57		P106 P107		P156 P157	
P8 P9		P58 P59		P108 P109		P158 P159	
P10 P11		P60 P61		P110 P111		P160 P161	
P12 P13		P62 P63		P112 P113		P162 P163	
P14 P15		P64 P65		P114 P115		P164 P165	
P16 P17		P66 P67		P116 P117		P166 P167	
P18 P19		P68 P69		P118 P119		P168 P169	
P20 P21		P70 P71		P120 P121		P170 P171	
P22 P23		P72 P73		P122 P123		P172 P173	
P24 P25		P74 P75		P124 P125		P174 P175	
P26 P27		P76 P77		P126 P127		P176 P177	
P28 P29		P78 P79		P128 P129		P178 P179	
P30 P31		P80 P81		P130 P131		P180 P181	
P32 P33		P82 P83		P132 P133		P182 P183	
P34 P35		P84 P85		P134 P135		P184 P185	
P36 P37		P86 P87		P136 P137		P186 P187	
P38 P39		P88 P89		P138 P139		P188 P189	
P40 P41		P90 P91		P140 P141		P190 P191	
P42 P43		P92 P93		P142 P143		P192 P193	
P44 P45		P94 P95		P144 P145		P194 P195	
P46 P47		P96 P97		P146 P147		P196 P197	
P48 P49		P98 P99		P148 P149		P198 P199	

Machine parameters for auxiliary spindle

P0 P1		P50 P51		P100 P101		P150 P151	
P2 P3		P52 P53		P102 P103		P152 P153	
P4 P5		P54 P55		P104 P105		P154 P155	
P6 P7		P56 P57		P106 P107		P156 P157	
P8 P9		P58 P59		P108 P109		P158 P159	
P10 P11		P60 P61		P110 P111		P160 P161	
P12 P13		P62 P63		P112 P113		P162 P163	
P14 P15		P64 P65		P114 P115		P164 P165	
P16 P17		P66 P67		P116 P117		P166 P167	
P18 P19		P68 P69		P118 P119		P168 P169	
P20 P21		P70 P71		P120 P121		P170 P171	
P22 P23		P72 P73		P122 P123		P172 P173	
P24 P25		P74 P75		P124 P125		P174 P175	
P26 P27		P76 P77		P126 P127		P176 P177	
P28 P29		P78 P79		P128 P129		P178 P179	
P30 P31		P80 P81		P130 P131		P180 P181	
P32 P33		P82 P83		P132 P133		P182 P183	
P34 P35		P84 P85		P134 P135		P184 P185	
P36 P37		P86 P87		P136 P137		P186 P187	
P38 P39		P88 P89		P138 P139		P188 P189	
P40 P41		P90 P91		P140 P141		P190 P191	
P42 P43		P92 P93		P142 P143		P192 P193	
P44 P45		P94 P95		P144 P145		P194 P195	
P46 P47		P96 P97		P146 P147		P196 P197	
P48 P49		P98 P99		P148 P149		P198 P199	

Machine parameters for serial line

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	

Ethernet machine parameters

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 M	S S								
M M	S S								
M M	S S								
M M	S S								
M M	S S								
M M	S S								
M M	S S								
M M	S S								
M M	S S								

M function	Associated subroutine	Setting bits							
		7	6	5	4	3	2	1	0
M M	S S								
M M	S S								
M M	S S								
M M	S S								
M M	S S								
M M	S S								
M M	S S								
M M	S S								
M M	S S								

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6X

N.

M functions setting chart

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

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

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

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

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

Moving axis _____		MOVAXIS (P55)
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 (P56)
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

Moving axis _____		MOVAXIS (P58)
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 (P59)
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

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 short-circuit the internal circuitry, especially under high humidity conditions.

To clean the operator panel and the monitor, a smooth cloth should be used which has been dipped into de-ionized water and /or non abrasive dish-washer soap (liquid, never powder) or 75º alcohol.

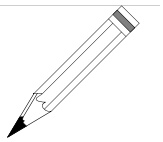
Do not use highly compressed air to clean the unit because it could generate electrostatic discharges.

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.



FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x

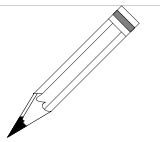


Q.

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6X



Q.

FAGOR

CNC 8055
CNC 8055i

SOFT: V01.6x