



(Soft M: V11.1x) (Soft T: V12.1x) Ref. 0612-a



REF. 0612-A



All rights reserved. No part of this documentation may be copied, transcribed, stored in a data backup system or translated into any language without Fagor Automation's explicit consent.

The information described in this manual may be modified for technical reasons. FAGOR AUTOMATION S. COOP. Reserves the right to modify the contents of this manual without having to communicate such modifications.

 $\label{lem:microsoft} \mbox{Microsoft and Windows are registered trademarks of Microsoft Corporation USA.} \mbox{The other commercial trademarks belong to their respective owners.}$ 

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

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	1
		ation of conformity	
		n history (M)	
	Versio	n history (T)	XI
	Safety	conditions	XVII
	Warra	nty terms	. XXI
	Materi	al returning terms	XXIII
	Additio	onal remarks	XXV
	Fagor	documentation	XXVII
CHAPTER 1	8040	CNC CONFIGURATION	
	1.1	CNC structure	1
	1.1.1	Connectors	
	1.2	Operator panel	
	1.2.1	Alphanumeric keyboard (optional)	
CHAPTER 2	HEAT	DISSIPATION	
	2.1	Heat dissipation by natural convection	30
	2.2	Heat dissipation by forced convection with inside fan	
	2.3	Heat dissipation by air flow to the outside using a fan	
CHAPTER 3	REMO	TE MODULES (BUS CAN WITH CANOPEN PROTOCOL)	
	3.1	Installation of the modules	35
	3.2	Power supply	
	3.3	Digital inputs and digital outputs (single module).	
	3.4	Digital inputs and digital outputs (double module)	
	3.5	Electrical characteristics of the inputs and outputs	
	3.6	Numbering of the digital inputs and outputs	
CHAPTER 4	MACH	IINE AND POWER CONNECTION	
	4.1	Digital inputs and outputs	52
	4.2	Digital inputs and outputs.	
	4.3	Setup	
	4.4	Connection of the emergency input and output	
CHAPTER 5	MACI	HINE PARAMETERS	
	5.1	Parameters that may be modified from the oscilloscope, OEM program or OEM	
		subroutine	
	5.2	General machine parameters	
	5.3	Axis parameters	
	5.4	Spindle parameters	
	5.4.1	Machine parameters for the main and 2nd spindles	
	5.4.2	Machine parameters for auxiliary spindle	
	5.5	Drive parameters	
	5.5.1	Friction compensation	
	5.6	Serial line parameters	
	5.7	Ethernet parameters	
	5.8	PLC Parameters	
	5.9	Tables	
	5.9.1	Miscellaneous (M) function table	
	5.9.2	Leadscrew error compensation table	
	5.9.3	Cross compensation parameter table	1/1



**CNC 8040** 

#### **CHAPTER 6**

#### CONCEPTS

6.1	Axes and coordinate systems	173
6.1.1	Rotary axes	
6.1.2	Gantry axes	
6.1.3	Slaved axes and synchronized axes	
6.1.4 6.1.5	Incline axis	
6.1.5	Jog	
6.2.1	Relationship between the axes and the JOG keys	
6.2.2	Path-jog mode	
6.3	Movement with an electronic handwheel	
6.3.1	Standard handwheel	193
6.3.2	Path handwheel	
6.3.3	Feed handwheel mode	
6.3.4	"Additive handwheel" mode	
6.4 6.4.1	feedback system	
6.4.2	Counting speed limitation	
6.5	Axis adjustment	
6.5.1	Drive setting	
6.5.2	Gain setting	
6.5.3	Proportional gain setting	208
6.5.4	Feed-forward gain setting	
6.5.5	Derivative (AC-forward) gain setting	
6.5.6	Leadscrew backlash compensation	
6.5.7 6.5.8	Leadscrew error compensation	
6.6	Circle geometry test	
6.6.1	Home search	
6.6.2	Setting on systems without distance-coded feedback	
6.6.3	Setting on systems with distance-coded feedback	
6.6.4	Axis travel limits (software limits)	
6.7	Unidirectional approach	223
6.8	Auxiliary M, S, T function transfer	
6.8.1	Transferring M, S, T using the AUXEND signal	
6.8.2	Transferring the auxiliary (miscellaneous) M functions without the AUXEND signal.	
6.9	Main and second spindle	
6.9.1 6.9.2	Spindle types	
6.9.3	Spindle gear change	
6.9.4	Spindle in closed loop	
6.10	Auxiliary spindle controlled by PLC	
6.11	Treatment of emergency signals	
6.12	Digital servo (CAN or Sercos)	
6.12.1	Communications channel	
6.12.2	Drive's absolute feedback	
6.13	Axes (2) controlled by a single drive	
6.14	Additive coupling between axes	
6.15 6.16	Machine safety related functions	
6.16.1	Maximum machining spindle speed	
6.16.2	Cycle start disabled when hardware errors occur.	
6.17	Configuring a CNC as two and a half axes	
6.17.1	Machine parameter setting	
6.17.2	PLC program	264
6.18	Tool change via PLC	266
INTRO	DUCTION TO THE PLC	
	21.0.2	
7.1	PLC Resources	
7.2	PLC program execution	
7.3 7.4	Cycle time	
7.4 7.4.1	Modular structure of the program	
7.4.1	Main module (PRG)	
7.4.3	Periodic execution module (PE t)	
7.4.4	Priority of execution of the PLC modules	
PLC R	ESOURCES	
8.1	Inputs	
8.2 8.3	Outputs	
8.4	Registers	
	- U 2	_55



**CHAPTER 7** 

**CHAPTER 8** 

**CNC 8040** 

	8.5 Timers	
		284
		286
	·	out
	· ·	290
		292
	8.6.1 Operating mode of a counter	295
HAPTER 9	PLC PROGRAMMING	
	9.1 Module structure	299
		303
	S .	305
	· · · · · · · · · · · · · · · · · · ·	306
	· ·	308
		310
		312
	•	314
HAPTER 10	CNC-PLC COMMUNICATION	
		318
		321
		D signal322
		) M functions without the AUXEND signal 323
		s324
		326
	10.5 Access to the PLC from a PC, via DNC.	327
HAPTER 11	LOGIC CNC INPUTS AND OUTPUTS	
	11.1 General logic inputs	330
	3 - 1	
	<u> </u>	344
		350
		351
	, , ,	352
	• .	354
	<u> </u>	361
	• ,	364
	11.11 Logic outputs of key status	The state of the s
HAPTER 12	ACCESS TO THE INTERNAL CNC VARIABLES	s
		371
	•	meters
		378
		ndwheels
	•	dle
		oindle
	<del>-</del>	pal parameters392
		393
	•	bles
		396
	. •	398 401
HAPTER 13	AXES CONTROLLED FROM THE PLC	
		408
		e PLC
		CNC414
	13.2 Action CNCEX1	416



**CNC 8040** 

#### **CHAPTER 14 CUSTOMIZABLE SCREENS** 14 1 14.2 14.3 14.4 14.5 **CHAPTER 15 CONFIGURABLE WORK MODE** Axis control 433 15.1 15.2 15.3 15.4 15.5 15.6 15.7 15.8 15.9 15.10 15.11 15.12 15.13 15.14 PLC PROGRAMMING EXAMPLE **CHAPTER 16** 16.1 16.2 16.3 Main module 459 **APPENDIX** Α В С Summary of internal CNC variables. ...... 477 D Ε F G Н I J Machine parameter setting chart ...... 533 Κ L M

N

Maintenance 551



**CNC 8040** 

# **ABOUT THE PRODUCT**

## Basic characteristics.

Block processing time	12 ms
RAM memory	256 Kb expandable to 1Mb
Memkey Card memory	512 Kb expandable to 2 Mb

## Hardware options.

	Analog	Digital
Hard disk / compact flash	Option	Option
Ethernet	Option	Option
RS-232 serial line.	Standard	Standard
16 digital inputs and 8 outputs (I1 to I16 and O1 to O8)	Standard	Standard
Another 40 digital inputs and 24 outputs (I65 to I104 and O33 to O56)	Option	Option
Probe inputs	Standard	Standard
Spindle (feedback input and analog output)	Standard	Standard
Electronic handwheels	Standard	Standard
4 axes (feedback and analog voltage)	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
1M RAM - 2M Flash	Option	Option



Before start-up, check that machine carrying this CNC meets the 89/392/CEE ruling.



# Software options.

		Model					
	GP	М	МС	мсо	т	тс	тсо
Number of axes with standard software	4	4	4	4	2	2	2
Number of axes with optional software					4	4	4
Electronic threading		Stand	Stand	Stand	Stand	Stand	Stand
Tool magazine management:		Stand	Stand	Stand	Stand	Stand	Stand
Machining canned cycles		Stand	Stand		Stand	Stand	
Multiple machining		Stand	Stand				
Solid graphics		Stand	Stand	Stand	Stand	Stand	Stand
Rigid tapping	Stand						
Tool live monitoring		Opt	Opt	Opt	Opt	Opt	Opt
Probing canned cycles		Opt	Opt	Opt	Opt	Opt	Opt
DNC	Stand						
COCOM version	Opt						
Profile editor	Stand						
Tool radius compensation	Stand						
Tangential control	Opt						
Retracing	Opt						
Setup assistance	Stand						
Irregular pockets with islands		Stand	Stand	Stand			
Digitizing		Opt	Opt				
Telediagnosis	Opt						



# **DECLARATION OF CONFORMITY**

#### The manufacturer:

Fagor Automation, S. Coop.

Barrio de San Andrés s/n, C.P. 20500, Mondragón -Guipúzcoa- (Spain).

#### We declare:

We declare under our exclusive responsibility the conformity of the product:

# Numerical Control Fagor 8040 CNC

Referred to by this declaration with following directives:

#### Safety regulations.

EN 60204-1 Machine safety. Electrical equipment of the machines.

### Regulation on electromagnetic compatibility.

EN 61000-4-3	Generic regulation on emissions in industrial environments.
EN 55011	Radiated. Class A, Group 1.
EN 61000-6-2	Generic regulation on immunity in industrial environments.
EN 61000-4-2	Electrostatic Discharges.
EN 61000-4-3	Radiofrequency Radiated Electromagnetic Fields.
EN 61000-4-4	Bursts and fast transients.
EN 61000-4-6	Conducted disturbance induced by radio frequency fields.
EN 61000-4-8	Magnetic fields to Mains frequency
EN 61000-4-11	Voltage fluctuations and Outages.
ENV 50204	Fields generated by digital radio-telephones

As instructed by the European Community Directives: 73/23/CEE modified by 93/68/EEC on Low Voltage and 89/336/CEE modified by 92/31/EEC and 93/68/EEC on Electromagnetic Compatibility and their updates.

In Mondragón, July 15th, 2005.

Fcio.: Julen Busturia

# **VERSION HISTORY (M)**

## (mill model)

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

The version history uses the following abbreviations:

INST Installation manual
PRG Programming manual
OPT Operation manual

OPT-MC Operation manual for the MC option.

OPT-CO Manual of the CO manual

Software V05.3x April 2002

List of features	Manual
New expansion board models for the 8055i.	INST / PRG
Bus CANOpen to control remote digital I/Os at the 8055i	INST / OPT
New PLC instructions. IREMRD and OREMWR.	INST
Leadscrew error compensation on rotary axes between 0-360 degrees.	INST
PLC statistic deletion in a single softkey.	OPT
Show only the XY plane in top-view graphics	OPT
Absolute reference mark management via Sercos (a.m.p. REFVALUE).	INST

Software V07.0x December 2002

List of features	Manual
New languages; Basque and Russian.	INST
Additional command pulse. Exponential backlash peak.	INST
Additional command pulse. Eliminate inside backlash peaks when changing quadrants.	INST
Improved non-random tool magazine management.	INST
Safety limit for the axis feedrate.	INST
Safety limit for the spindle speed.	INST
Execute the additional compensation block at the beginning of the next block.	INST
Jerk control in look-ahead.	INST
New graphics option. Mill graphics with changed line graphics.	INST
Path-jog mode.	INST / OPT
Update the variables of the machine parameters from the OEM program or subroutine.	INST / PRG
HARCON variable returns the type of LCD and turbo board.	INST / PRG
Variables to consult the actual (real) and theoretical feedrate of each axis.	INST / PRG
Variable to consult the coordinates shown on the screen for each axis.	INST / PRG
Variable to consult the position indicated by the Sercos drive of each axis.	INST / PRG
Variable to consult the coordinate programmed in a block of a program.	INST / PRG
Variable to consult the position indicated by the Sercos drive of the first and second spindles.	INST / PRG
Variable to consult the theoretical turning speed of the first and second spindles.	INST / PRG
Variable to consult the maximum spindle speed.	INST / PRG
Feedback related variables.	INST / PRG
Variable to consult a single PLC mark.	INST / PRG
Variable to consult the PROBE cycle being executed.	INST / PRG
Variable to know the number of the screen, created with WGDRAW, that is being consulted.	INST / PRG
Variable to know the number of the item (element), created with WGDRAW, that is being	INST / PRG
consulted.	
Machine safety. Hardware errors disable the [CYCLE START] key.	INST
Machine safety. Maximum machining feedrate.	INST



List of features	Manual
Mandatory axis homing after feedback alarm when using direct feedback.	INST
The "SERCOS816" board is now recognized.	11101
Write-protect the user arithmetic parameters (P1000-P1255) and OEM arithmetic parameters	INST
(P2000-P2255).	INST
New command of the configuration language. UNMODIFIED command.	INST
Synchronize a PLC axis with a CNC axis.	INST
Axes (2) controlled by a drive. The direction of the command LOOPCHG is defined on both axes.	INST
Tool change via PLC.	INST
New user arithmetic parameters (P1000 - P1255).	PRG
New OEM arithmetic parameters (P2000 - P2255).	PRG
Improved PROBE 1 cycle. Calibrate and/or measure tool radius wear and/or tool length wear.	PRG
RPT instruction. Execute blocks of a program located in RAM memory.	PRG
Look-ahead. It analyzes up to 75 blocks in advance.	PRG
DEM (manufacturer's) subroutines. Range SUB10000 - SUB20000.	PRG
	OPT
Oscilloscope function.	OPT
Load the version without using an external microprocessor.	OPT
The screen data (feedrates, coordinates, etc.) do not show their zeros to the left.	OPT
Blackbox for registering errors.	
Telediagnosis through the RS232 serial line.	OPT OPT
Telediagnosis via WinDNC.	_
Correct tool wear from tool inspection mode.	OPT
Improvements to the profile editor.	OPT
Save and load the parameters of the Sercos or CAN drive through the serial line.	INST
MC option. Restore the last F, S and Smax values on power-up.	INST
MC option. Possibility of hiding unused operations or cycles.	INST
MC option. There are auxiliary M functions in the cycles.	INST
MC option. Improvements for executing a part-program.	OPT-MC
MC option. It shows warning messages on a green stripe.	OPT-MC
MC option. Limitations to tool calibration when there is a program in execution or in tool inspection	OPT-MC
mode.	
MC option. Tool measuring and calibrating cycle.	OPT-MC
MC option. Icon to select between available options.	OPT-MC
MC option. Threading cycle. It is possible to indicate the type of thread defining the pitch and	OPT-MC
the speed (S) or the feedrate (F) and the speed (S).	
MC option. Milling cycle. When defining the points of the profile, when leaving a data blank, the	OPT-MC
cycle assumes that it is the repetition of the previous one.	
MC option. Milling cycle. It is possible to define the incremental points or coordinates.	OPT-MC
MC option. Multi-point multiple positioning cycle. When defining the points of the profile, when	OPT-MC
leaving a data blank, the cycle assumes that it is the repetition of the previous one.	
MC option. Multi-point multiple positioning cycle. It is possible to define the incremental points	OPT-MC
or coordinates.	

Software V07.1x July 2003

List of features	Manual
New 24-character validation codes.	
Pin ·9· of connector X1 (RS232 serial line) no longer supplies 5 V.	INST
Smooth stop of probing movement.	INST
Machining in round corner mode when changing the tool offset.	INST
Bi-directional leadscrew compensation.	INST
The management of the distance-coded reference mark via Sercos may be carried out through	
the drive's second feedback. The drive version must be V4.10 or V5.10 ( or greater).	
Machine parameters modifiable from an OEM program using variables	INST / OEM
General machine parameters modifiable from the oscilloscope: TLOOK.	OPT
Axis machine parameters that may be modified from the oscilloscope: MAXFEED, JOGFEED.	OPT
Improvements to look ahead. Machining feedrate variations are now smoother thanks to filtered acc/dec.	



List of features	Manual
MEM CARD slot as floppy drive (CARD-A).	INST / OPT
Tandem axes	INST
Stop block preparation when executing a new "T".	INST
Execute the stop signal when done with the "T" change.	INST
Compact-flash type hard disk and Ethernet.	INST
Incline axis.	INST / PRG
Select the behavior of the feedrate for F0.	INST
On Gantry axes, cross compensation is also applied to the slave axis.	INST
Variable to select the active probe input.	INST / PRG
Variable to know the address of the I/O CAN.	INST / PRG
Variables to read the number of local and remote digital I/O available.	INST / PRG
The HARCON variable recognizes Ethernet and compact flash.	INST / PRG
While compiling the PLC program, the outputs are initialized to zero.	
New marks to park the spindles.	INST
Name the logic inputs and outputs with the axis name	INST
Parameter RAPIDEN assumes the value of ·2·. Rapid key controlled by PLC.	INST
Ending the execution of a block using a PLC mark (BLOABOR, BLOABORP).	INST
Additive coupling between axes.	INST
The EXEC and OPEN instructions may be used with Ethernet.	PRG
G2/G3. The center coordinates may be left out if their value is zero.	PRG
General parameters that may be modified from the oscilloscope or from an OEM program:	INST/PRG/OPT
CODISET.	
Axis parameters that may be modified from the oscilloscope or from an OEM program: MAXFLWE1, MAXFLWE2.	INST/PRG/OPT
Connection to a remote hard disk.	INST / OPT
Connection to a PC through WinDNC.	INST / OPT
Access the CNC's hard disk from a PC via FTP.	INST / OPT
Telediagnosis. Normal phone call.	OPT
Telediagnosis. Disable the CNC screen and keyboard from the PC.	OPT
Functions M41 through M44 admit subroutines when the gear change is automatic.	PRG
MC option. Configuration as two and half axes.	INST
MC option. Accessing cycles and programs from the auxiliary screen.	INST
MC option. ISO programming assistance.	OPT-MC
MC option. Zero offset table management.	OPT-MC
MC option. After an execution or simulation error, it indicates the erroneous cycle.	OPT-MC
MC option. In execution or simulation, it indicates the cycle number.	OPT-MC
MC option. The CNC highlights the axis being moved in jog or with handwheels.	OPT-MC
MC option. Copying a profile.	OPT-MC
MC option. Select a program by indicating its number.	OPT-MC
MC option. Selection of the starting point in rectangular pockets and bosses.	OPT-MC
MC option. Pockets and bosses may be assigned to multiple positioning cycles.	OPT-MC
MC option. Configuration of two and half axes.	OPT-MC
CO option. Copying a profile.	OPT-CO

# Software V09.1x December 2004

List of features	Manual
Calculation of central unit heat dissipation .	INST
Frequency filters for axes and spindles.	INST
When activating tool radius compensation in the first motion block even if there is no movement	INST
of the plane axes.	
CAN servo system.	INST
8055i model. New board "Axes2".	INST
Sercos transmission speed at 8 MHz and 16 MHz.	INST
Retracing function. If RETRACAC=2, the retrace function does not stop at the M functions.	INST
Retracing function. The RETRACAC parameter is initialized with [SHIFT][RESET].	
Retracing function. The number of blocks being retraced has been increased to 75.	INST
New variables for APOS(X-C) and ATPOS(X-C) to consult part coordinates.	INST
New variable DNCSTA to consult DNC communication status.	INST
New variable TIMEG to consult the status of the timer programmed with G4.	INST
Manual intervention with additive handwheel.	INST / OPT
A CNC emergency disables the SPENA signals and the Sercos drive brakes using the emergency	INST
ramps.	
Maintain G46 when the home search does not involve any axis of the angular transformation.	INST / PRG



List of features	Manual
COMPMODE (P175). New tool radius compensation methods.	INST / PRG
Automatic keyboard type identification.	INST
Variable to indicate whether the handwheel selector button has been pressed or not.	INST
Protect the access from the network to the hard disk using a password.	INST
The HARCON variable recognizes the new axis board "Axes2".	INST / PRG
Variable to analog the value of the analog inputs.	INST / PRG
New MEXEC instruction. Execute a modal part-program.	PRG
Look-ahead. Functions G74, G75 and G76 are possible.	PRG
Up to 319 G functions now available.	PRG
Machine parameters that may be modified from the oscilloscope or from an OEM program: REFVALUE, REFDIREC, FLIMIT, SLIMIT.	INST/PRG/OPT
Accessing the variables of the auxiliary spindle drive from the oscilloscope.	OPT
The simulations without axis movement ignore the G4.	OPT
Sharing the CNC's hard disk with password.	INST / OPT
Telediagnosis. Advanced phone call.	OPT
Telediagnosis through the Internet.	OPT
Telediagnosis. Disconnect the CNC from the Ethernet during telediagnosis.	OPT
Maintain the feedrate selected in simulation.	OPT

Software V9.13 April 2005

List of features	Manual
New table to define the Sercos power with the Sercos816 board.	INST
600 µs delay at the Sercos bus for transmissions at 8 MHz and 16 MHz.	INST
Hirth axis pitch may be set in degrees via parameters.	INST
Rollover positioning axis. Movement in G53 via the shortest way.	INST

Software V9.14 May 2005

List of features	Manual
New table to define the Sercos power with the Sercos816 board.	INST

Software V11.01 Ref. 0508

List of features	Manual
Analog I/O expansion and PT100	INST
CAN bus speed with remote digital I/O modules	INST
The CNC supports Memkey Card + Compact Flash or KeyCF	OPT
File explorer to show the contents of the storage devices	INST / OPT
Loading the version from the Memkey card o from the hard disk	OPT
New way to search home that may be selected through g.m.p. I0TYPE=3	INST
Improved block search. Switching from simulation to execution	INST / OPT
New repositioning mode that is activated by setting g.m.p. REPOSTY=1	INST/PRG/OPT
Square-sine ramps on open-loop spindle	INST
Numbering of the local inputs/outputs of the expansion modules using plc machine parameters	INST
Default value of axis and spindle machine parameter ACFGAIN = YES	INST
Setting axis parameters FFGAIN and FFGAIN2 with two decimals	INST
Up to 400 (DEF) symbols now available at the PLC	INST
New HTOR variable that indicates the tool radius being used by the CNC	INST
Longitudinal axis definition with G16	INST / PRG
Part centering with a probe	OPT-MC
Rectangular part centering cycle (PROBE 10)	PRG
Circular part centering cycle (PROBE 11)	PRG
Generating an ISO-coded program	OPT-MC



# Software V11.11 Ref. 0602

List of features	Manual
New G145. Temporary cancellation of tangential control	PRG
Handwheel feedback taken to a free feedback connector	INST
New variables: RIP, GGSE, GGSF, GGSG, GGSH, GGSI, GGSJ, GGSK, GGSL, GGSM, PRGSP,	INST
SPRGSP and PRBMOD	
Automatic keyboard type identification	INST / PRG
Improved part centering with a probe (PROBE 10 - PROBE 11)	PRG
G04 K0. Block preparation interruption and coordinate update	PRG
Possibility to view all the active PLC messages	OPT-MC
Improved part centering with a probe	OPT-MC

Software V11.13 Ref. 0606

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

Software V11.14 Ref. 0608

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





# **VERSION HISTORY (T)**

## (lathe model)

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

The version history uses the following abbreviations:

INST Installation manual
PRG Programming manual
OPT Operation manual

OPT-TC Operation manual for the TC option.

OPT-CO Manual of the CO manual

Software V06.3x April 2002

List of features	Manual
New expansion board models for the 8055i.	INST / PRG
Bus CANOpen to control remote digital I/Os at the 8055i	INST / OPT
New PLC instructions. IREMRD and OREMWR.	INST
Leadscrew error compensation on rotary axes between 0-360 degrees.	INST
PLC statistic deletion in a single softkey.	OPT
Absolute reference mark management via Sercos (a.m.p. REFVALUE).	INST

Software V08.0x December 2002

List of features	Manual
New languages; Basque and Russian.	INST
Additional command pulse. Exponential backlash peak.	INST
Additional command pulse. Eliminate inside backlash peaks when changing quadrants.	INST
Improved non-random tool magazine management.	INST
Safety limit for the axis feedrate.	INST
Safety limit for the spindle speed.	INST
Execute the additional compensation block at the beginning of the next block.	INST
Path-jog mode.	INST / OPT
Update the variables of the machine parameters from the OEM program or subroutine.	INST / PRG
HARCON variable returns the type of LCD and turbo board.	INST / PRG
Variables to consult the actual (real) and theoretical feedrate of each axis.	INST / PRG
Variable to consult the coordinates shown on the screen for each axis.	INST / PRG
Variable to consult the position indicated by the Sercos drive of each axis.	INST / PRG
Variable to consult the coordinate programmed in a block of a program.	INST / PRG
Variable to consult the position indicated by the Sercos drive of the first and second spindles.	INST / PRG
Variable to consult the theoretical turning speed of the first and second spindles.	INST / PRG
Variable to consult the maximum spindle speed.	INST / PRG
Feedback related variables.	INST / PRG
Variable to consult a single PLC mark.	INST / PRG
Variable to consult the PROBE cycle being executed.	INST / PRG
Variable to know the number of the screen, created with WGDRAW, that is being consulted.	INST / PRG
Variable to know the number of the item (element), created with WGDRAW, that is being	INST / PRG
consulted.	
Machine safety. Hardware errors disable the [CYCLE START] key.	INST
Machine safety. Maximum machining feedrate.	INST
Mandatory axis homing after feedback alarm when using direct feedback.	INST
The "SERCOS816" board is now recognized.	



List of features	Manual
Write-protect the user arithmetic parameters (P1000-P1255) and OEM arithmetic parameters	INST
(P2000-P2255).	
New command of the configuration language. UNMODIFIED command.	INST
Synchronize a PLC axis with a CNC axis.	INST
Axes (2) controlled by a drive. The direction of the command LOOPCHG is defined on both axes.	INST
Tool change via PLC.	INST
New user arithmetic parameters (P1000 - P1255).	PRG
New OEM arithmetic parameters (P2000 - P2255).	PRG
RPT instruction. Execute blocks of a program located in RAM memory.	PRG
Look-ahead. It analyzes up to 75 blocks in advance.	PRG
OEM (manufacturer's) subroutines. Range SUB10000 - SUB20000.	PRG
Oscilloscope function.	OPT
Load the version without using an external microprocessor.	OPT
The screen data (feedrates, coordinates, etc.) do not show their zeros to the left.	
Blackbox for registering errors.	OPT
Telediagnosis through the RS232 serial line.	OPT
Telediagnosis via WinDNC.	OPT
Correct tool wear from tool inspection mode.	OPT
Improvements to the profile editor.	OPT
G60, G61 and G63. Minimum drilling peck.	PRG
G86 and G87. "V" parameter for multi-entry threads.	PRG
Save and load the parameters of the Sercos or CAN drive through the serial line.	INST
TC option. Restore the last F, S and Smax values on power-up.	INST
TC option. Possibility of hiding unused operations or cycles.	INST
TC option. Improvements for executing a part-program.	OPT-TC
TC option. It shows warning messages on a green stripe.	OPT-TC
TC option. Limitations to tool calibration when there is a program in execution or in tool inspection	OPT-TC
mode.	
TC option. Turning cycle. It is now possible to define the finishing stock in X and Z.	OPT-TC
TC option. Turning cycle. New level to define the type of corner.	OPT-TC
TC option. Facing cycle. It is now possible to define the finishing stock in X and Z.	OPT-TC
TC option. Facing cycle. New level to define the type of corner.	OPT-TC
TC option. Taper turning cycle. New level to define the Z distance and the taper angle.	OPT-TC
TC option. Threading cycle. It is possible to define the type of standardized thread (except face	OPT-TC
threading).	
TC option. Threading cycle. It is possible to repeat the last pass.	OPT-TC
TC option. Threading cycle. New level for multi-entry threads.	OPT-TC
TC option. Grooving cycle. New level for cut-off operations.	OPT-TC
TC option. Profiling cycle. After roughing, it does not withdraw to the safety distance.	OPT-TC
TC option. Profiling cycle. The first point of the profile is assumed as the starting point.	OPT-TC
TC option. Profiling cycle. Finishing stock in XC and ZC profile cycles	OPT-TC
TC option. Profiling cycle. Finishing tool for the XC and ZC cycles.	OPT-TC

Software V08.1x July 2003

List of features	Manual
New 24-character validation codes.	
Pin ·9· of connector X1 (RS232 serial line) no longer supplies 5 V.	INST
Smooth stop of probing movement.	INST
Machining in round corner mode when changing the tool offset.	INST
Bi-directional leadscrew compensation.	INST
The management of the distance-coded reference mark via Sercos may be carried out through	
the drive's second feedback. The drive version must be V4.10 or V5.10 ( or greater).	
Machine parameters modifiable from an OEM program using variables	INST / OEM
General machine parameters modifiable from the oscilloscope: TLOOK.	OPT
Axis machine parameters that may be modified from the oscilloscope: MAXFEED, JOGFEED.	OPT
Improvements to look ahead. Machining feedrate variations are now smoother thanks to filtered	
acc/dec.	
G86 and G87. "J" parameter. Thread exit through the last point.	PRG
TC option. Threading cycle. Thread exit through the last point.	OPT-TC



List of features	Manual
MEM CARD slot as floppy drive (CARD-A).	INST / OPT
Tandem axes.	INST
Stop block preparation when executing a new "T".	INST
Execute the stop signal when done with the "T" change.	INST
Compact-flash type hard disk and Ethernet.	INST
Incline axis.	INST / PRG
Select the behavior of the feedrate for F0.	INST
On Gantry axes, cross compensation is also applied to the slave axis.	INST
Variable to select the active probe input.	INST / PRG
Variable to select the programming mode, radius or diameter.	INST / PRG
Variable to know the address of the I/O CAN.	INST / PRG
Variables to read the number of local and remote digital I/O available.	INST / PRG
The HARCON variable recognizes Ethernet and compact flash.	INST / PRG
While compiling the PLC program, the outputs are initialized to zero.	
New marks to park the spindles.	INST
Name the logic inputs and outputs with the axis name	INST
Parameter RAPIDEN assumes the value of ·2 ·. Rapid key controlled by PLC.	INST
Ending the execution of a block using a PLC mark (BLOABOR, BLOABORP).	INST
Additive coupling between axes.	INST
The EXEC and OPEN instructions may be used with Ethernet.	PRG
G2/G3. The center coordinates may be left out if their value is zero.	PRG
General parameters that may be modified from the oscilloscope or from an OEM program: CODISET.	INST/PRG/OPT
Axis parameters that may be modified from the oscilloscope or from an OEM program: MAXFLWE1, MAXFLWE2.	INST/PRG/OPT
Connection to a remote hard disk.	INST / OPT
Connection to a PC through WinDNC.	INST / OPT
Access the CNC's hard disk from a PC via FTP.	INST / OPT
Telediagnosis. Normal phone call.	OPT
Telediagnosis. Disable the CNC screen and keyboard from the PC.	OPT
Functions M41 through M44 admit subroutines when the gear change is automatic.	PRG
TC option. Accessing cycles and programs from the auxiliary screen.	INST
TC option. ISO programming assistance.	OPT-TC
TC option. Zero offset table management.	OPT-TC
TC option. Point-to-point profiling cycle. When defining the points of the profile, when leaving a	OPT-TC
data blank, the cycle assumes that it is the repetition of the previous one.	
TC option. Tool calibration cycle.	OPT-TC
TC option. After an execution or simulation error, it indicates the erroneous cycle.	OPT-TC
TC option. In execution or simulation, it indicates the cycle number.	OPT-TC
TC option. The CNC highlights the axis being moved in jog or with handwheels.	OPT-TC
TC option. Copying a profile.	OPT-TC
TC option. Select a program by indicating its number.	OPT-TC
CO option. Copying a profile.	OPT-CO

Software V10.1x December 2004

List of features	Manual
Calculation of central unit heat dissipation .	INST
Frequency filters for axes and spindles.	INST
When activating tool radius compensation in the first motion block even if there is no movement	INST
of the plane axes.	
CAN servo system.	INST
8055i model. New board "Axes2".	INST
Sercos transmission speed at 8 MHz and 16 MHz.	INST
Retracing function. If RETRACAC=2, the retrace function does not stop at the M functions.	INST
Retracing function. The RETRACAC parameter is initialized with [SHIFT][RESET].	
Retracing function. The number of blocks being retraced has been increased to 75.	INST
New variables for APOS(X-C) and ATPOS(X-C) to consult part coordinates.	INST
New variable DNCSTA to consult DNC communication status.	INST
New variable TIMEG to consult the status of the timer programmed with G4.	INST
Manual intervention with additive handwheel.	INST / OPT
A CNC emergency disables the SPENA signals and the Sercos drive brakes using the emergency	INST
ramps.	
Maintain G46 when the home search does not involve any axis of the angular transformation.	INST / PRG



List of features	Manual
COMPMODE (P175). New tool radius compensation methods.	INST / PRG
Automatic keyboard type identification.	INST
Variable to indicate whether the handwheel selector button has been pressed or not.	INST
Protect the access from the network to the hard disk using a password.	INST
The HARCON variable recognizes the new axis board "Axes2".	INST / PRG
Variable to analog the value of the analog inputs.	INST / PRG
New MEXEC instruction. Execute a modal part-program.	PRG
Look-ahead. Functions G74, G75 and G76 are possible.	PRG
Up to 319 G functions now available.	PRG
Machine parameters that may be modified from the oscilloscope or from an OEM program:	INST/PRG/OPT
REFVALUE, REFDIREC, FLIMIT, SLIMIT.	
Accessing the variables of the auxiliary spindle drive from the oscilloscope.	OPT
The simulations without axis movement ignore the G4.	OPT
Sharing the CNC's hard disk with password.	INST / OPT
Telediagnosis. Advanced phone call.	OPT
Telediagnosis through the Internet.	OPT
Telediagnosis. Disconnect the CNC from the Ethernet during telediagnosis.	OPT
Maintain the feedrate selected in simulation.	OPT
G151-G152. Programming in diameter or radius.	PRG
TC option. Displaying the transformed coordinates with the C axis active	OPT-TC

Software V10.13 April 2005

List of features	Manual
New table to define the Sercos power with the Sercos816 board.	INST
600 µs delay at the Sercos bus for transmissions at 8 MHz and 16 MHz.	INST
Hirth axis pitch may be set in degrees via parameters.	INST
Rollover positioning axis. Movement in G53 via the shortest way.	INST

Software V10.14x May 2005

List of features	Manual
New table to define the Sercos power with the Sercos816 board.	INST

Software V12.01 Ref. 0508

List of features	Manual
Analog I/O expansion and PT100	INST
CAN bus speed with remote digital I/O modules	INST
The CNC supports Memkey Card + Compact Flash or KeyCF	OPT
File explorer to show the contents of the storage devices	INST / OPT
Loading the version from the Memkey card o from the hard disk	OPT
New way to search home that may be selected through g.m.p. I0TYPE=3	INST
Improved block search. Switching from simulation to execution	INST / OPT
New repositioning mode that is activated by setting g.m.p. REPOSTY=1	INST/PRG/OPT
Square-sine ramps on open-loop spindle	INST
Numbering of the local inputs/outputs of the expansion modules using plc machine parameters	INST
Default value of axis and spindle machine parameter ACFGAIN = YES	INST
Setting axis parameters FFGAIN and FFGAIN2 with two decimals	INST
Up to 400 (DEF) symbols now available at the PLC	INST
New HTOR variable that indicates the tool radius being used by the CNC	INST
Spindle override in the whole threading cycle at 100%	PRG
Generating an ISO-coded program	OPT-TC



# Software V12.11 Ref. 0602

List of features	Manual
New G145. Temporary cancellation of tangential control	PRG
Handwheel feedback taken to a free feedback connector	INST
New variables: RIP, GGSE, GGSF, GGSG, GGSH, GGSI, GGSJ, GGSK, GGSL, GGSM, PRGSP,	INST
SPRGSP and PRBMOD	
Automatic keyboard type identification	INST / PRG
G04 K0. Block preparation interruption and coordinate update	PRG
Possibility to view all the active PLC messages	OPT-TC
In the cycles, when selecting CSS, the screen shows "CSS" instead of "S"	OPT-TC
Icon for coolant ON/OFF in positioning cycles 1 and 2	OPT-TC

Software V12.13 Ref. 0606

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

Software V12.14 Ref. 0608

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





# **SAFETY CONDITIONS**

Read the following safety measures in order to prevent damage to personnel, to this product and to those products connected to it.

This unit must only be repaired by personnel authorized by Fagor Automation.

Fagor Automation shall not be held responsible for any physical or material damage derived from the violation of 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 $^{\rm o}C$ (113° F).
Do not work in explosive environments
In order to avoid risks, damage, do no work in explosive environments.

## **Precautions against product damage**

■ Working environment
This unit is ready to be used in industrial 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 to install the CNC away from coolants, chemical products, possible blows etc. which 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 / TC 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 decoupled all those elements capable of generating interference (relay coils, contactors, motors, etc.)

- DC relay coils. Diode type 1N4000.
- AC relay coils. RC connected as close to the coils as possible with approximate values of R=220  $\Omega$  / 1 W and C=0,2  $\mu F$  / 600 V.
- AC motors. RC connected between phases, with values of R=300  $\Omega/6$  W and C=0,47  $\mu F$  / 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  $^{\circ}$ C and +40  $^{\circ}$ C (41 $^{\circ}$ F and 104 $^{\circ}$ F) The storage temperature must be between -25  $^{\circ}$ C and +70  $^{\circ}$ C. (-13  $^{\circ}$ F and 158  $^{\circ}$ F)

☐ Central unit enclosure (8055i CNC)

Guarantee the required gaps between the central unit and each wall of the enclosure. Use a DC fan to improve enclosure ventilation.

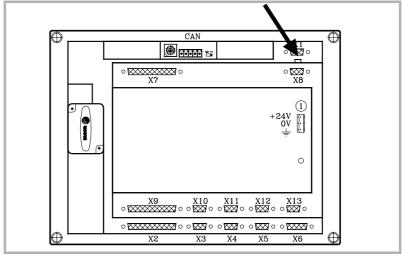
■ Power switch

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



#### □ Central Unit

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



## ■ Inputs-Outputs

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

## **Precautions during repair**



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

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

# Safety symbols

☐ Symbols which may appear on the manual.



Symbol for danger or prohibition.

It indicates actions or operations that may cause damage to people or to units.



Warning or caution 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 suggestions.





# **WARRANTY TERMS**

All products manufactured or marketed by Fagor Automation has a warranty period of 12 months from the day they are shipped out of our warehouses.

The mentioned warranty covers repair material and labor costs, at Fagor facilities, incurred in the repair of the products.

Within the warranty period, Fagor will repair or replace the products verified as being defective.

Fagor is committed to repairing or replacing its products from the time when the first such product was launched up to 8 years after such product has disappeared from the product catalog.

It is entirely up to Fagor to determine whether a repair is to be considered under warranty.

### Excluding clauses

The repair will take place at our facilities. Therefore, all shipping expenses as well as travelling expenses incurred by technical personnel are NOT under warranty even when the unit is under warranty.

This warranty will be applied so long as the equipment has been installed according to the instructions, it has not been mistreated or damaged by accident or negligence and has been manipulated by personnel authorized by Fagor.

If once the service call or repair has been completed, the cause of the failure is not to be blamed the FAGOR product, the customer must cover all generated expenses according to current fees.

No other implicit or explicit warranty is covered and Fagor Automation shall not be held responsible, under any circumstances, of the damage which could be originated.

#### Service agreements

Service and Maintenance Contracts are available for the customer within the warranty period as well as outside of it.





# **MATERIAL RETURNING TERMS**

When returning the remote modules or the central unit, pack it in its original package and with its original packaging material. If not available, pack it as follows:

- 1. Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 kg (375 lb).
- 2. Attach the unit label indicating the owner of the unit, his/her address, the name of the contact person, the type of unit and the 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, above all protect the screen
- 6. Pad the unit inside the cardboard box with polyurethane foam on all sides.
- 7. Seal the cardboard box with packing tape or industrial staples.





# **ADDITIONAL REMARKS**

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

In case of a malfunction or failure, disconnect it and call the technical service. Do not open this 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.

#### **WGDRAW Manual**

Directed to people who use the WGDRAW to create screens.

It is supplied in a floppy disk with the application.





# **8040 CNC CONFIGURATION**

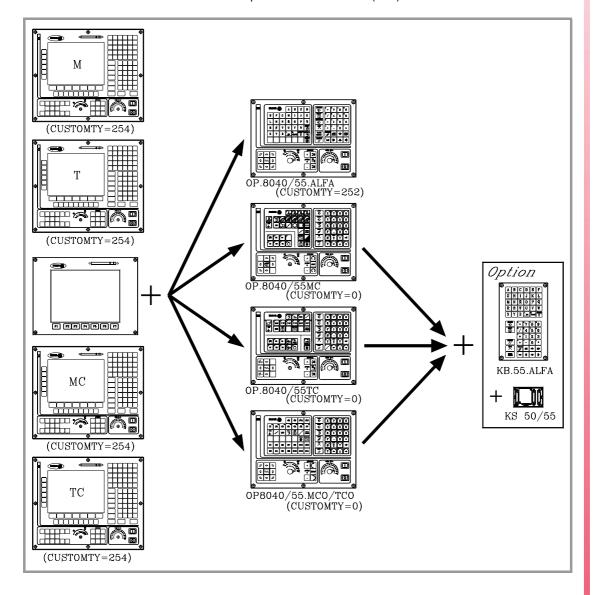
1

The CNC can control machine movements and devices.

### 1.1 CNC structure

The central unit (cpu) is on the back 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 machine parameter CUSTOMY (P92).



FAGOR

**CNC 8040** 

**CNC** structure

### Operator panels with integrated monitor

CNC 8040-M-COL-K

CNC 8040-T-COL-K

CNC 8040-T-MON-K

CNC 8040-T-MON-K

CNC 8040-MC-MON-K

CNC 8040-TC-MON-K

CNC 8040-TC-MON-K

All these keyboards have an auto-identification system. See "Keyboard auto-identification" on page 2.

### Independent monitors

CNC 8040-M-COL CNC 8040-M-MON CNC 8040-T-COL CNC 8040-T-MON

### Independent operator panels

M model Operator panel: OP.8040/55.ALFA
T model Operator panel: OP.8040/55.ALFA
MC model Operator panel: OP.8040/55.MC
TC model Operator panel: OP.8040/55.TC

MCO/TCO model Operator panel: OP.8040/55.MCO/TCO

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 (KB55.ALFA). This keyboard is connected to the operator panel through the KS50/55 adapter.

### 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.



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

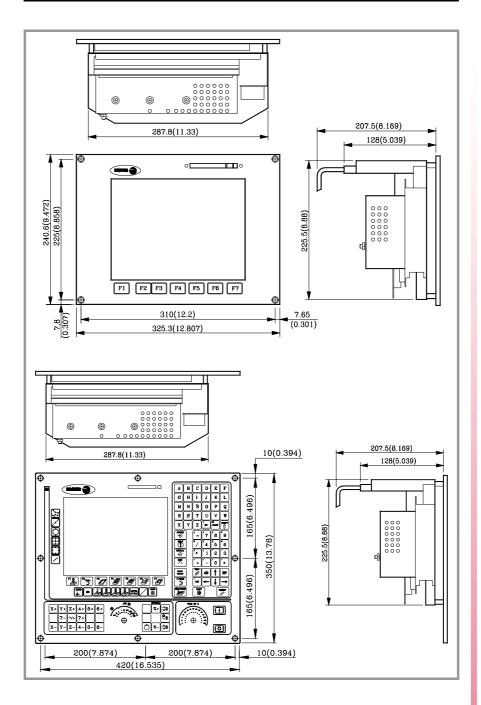
If an auto-identifying keyboard is connected to a CNC that has an older software version, the keyboard will beep. In this case, disable the auto-identification hardware of the keyboard by setting the identification switch to zero.

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 has been set for a lathe and the CNC shows the error message.



**CNC 8040** 

### **Dimensions**



CNC structure 8040 CNC CONFIGURATION

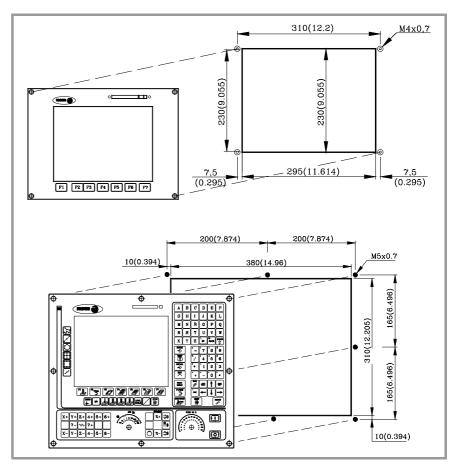


**CNC 8040** 

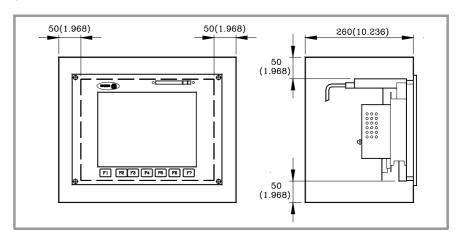


**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x)



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 5° C and +50° C (41° F and 122° 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 Operation manual, chapter on Diagnosis, section on Hardware configuration.

### 1.1.1 **Connectors**

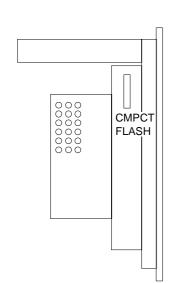


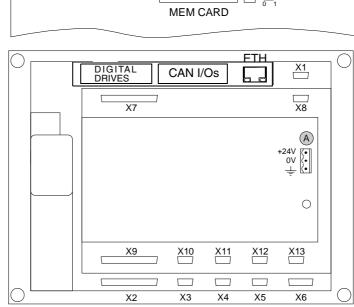
From versions, there is a new axes board that includes the recognizance of 24V at the inputs and outputs. This board will appear in Diagnosis > Configuration > Hardware with the name of "Axes 2".

The board is compatible with previous software versions, it will not recognize the 24V at the inputs and outputs.

They are located in the rear.







(A)	Power supply.				
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 analog voltage connection.				
X9	For digital I/O 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.				
CAN I/O	Connector for CAN connection with the remote I/O's.				
DIGITAL DRIVES	Connector for the digital servo system (CAN or Sercos).				
ETH	Ethernet connector to configure the CNC within a local network.				
CMPCT FLASH	Slot for the local hard disk (compact flash).				
MEM CARD	IEM CARD Slot for the CNC configuration card (memkey card).				

CNC structure 8040 CNC CONFIGURATION



**CNC 8040** 



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

Do not handle the connectors with the unit connected to mains. Before 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.

### Signal adapters

The following signal adapters are available.

SA-TTL-TTLD Adapter for "Non-differential TTL" to "differential TTL" signals

SA-FS-P Adapter for Fagor sinusoidal signals to Vpp signals

### **Power supply**

3-prong male Phoenix connector, 7.65 mm pitch.

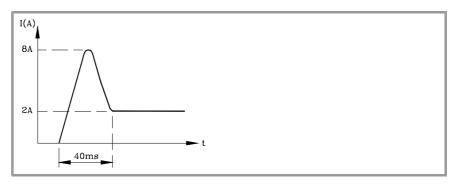
	Pin	Signal and function		
+24V	1	+24 V	Power supply.	
	2	0 V	Power supply.	
	3	Chassis	Shield.	
<u></u>				

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:





**CNC 8040** 

### **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
0 1	3	TxD
6 0 0 2	4	DTR
7 0 0 3 4 9 0 0 5	5	GND ISO
	6	DSR
5	7	RTS
7	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<sup>2</sup> section.

### **Transmission speed**

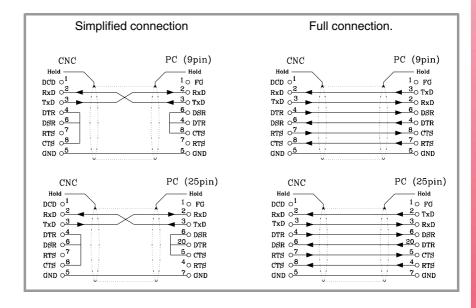
The CNC can operate at up to 115,200 Baud.

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

### **Ground connection**

It is suggested to reference all control and data signals to the same ground cable (pin 7 GND) thus, avoiding reference points at different voltages especially in long cables.

### Recommended RS232C interface connection





**CNC 8040** 

All shields must only be connected to ground at the CNC end through pin 37 of the connector leaving the other end free. The wires of the shielded cables cannot be unshielded for more than 75mm (about 3 inches).

 $\triangle$ 

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 a	nd function
37	1 2 3 4 5	24 V O1 O3 O5 O7	External power supply. / Emergency output.
	6 7 8 9 10	   I1	/ Emergency input.
	11 12 13 14 15	13 15 17 19 111	
	16 17 18 19	I13 I15 0 V 0 V	External power supply. External power supply.
24	20 21 22 23 24	24 V O2 O4 O6 O8	External power supply.
,	25 26 27 28 29	   12	
	30 31 32 33 34	14 16 18 110 112	
	35 36 37	I14 I16 Chassis	Shield.

8040 CNC CONFIGURATION

CNC structure



**CNC 8040** 

### Connectors X3 For probe connection

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

Pin		Signal an	nd function
	1	Chassis	Shield.
9 0 5	2	+5 V	Probe 1. +5 V output for the probe.
8 0 0 4	3	PRB1_5	5 V TTL input.
$  \gamma   \circ {}^{0}  _{2}^{3}$	4	PRB1_24	Probe 1. 24 Vdc input.
$6 \left[ \begin{array}{c c} & 0 & 2 \\ \hline \end{array} \right] $	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 Vdc 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 specifications of the CNC" on page 471.

The appendix of this manual describes the recommended connection circuits. See *"Probe connection"* on page 475.

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

### Connector X4 For analog spindle connection

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

Pin		Signal a	nd function
	1	Α	
	2	/A	
	3	В	Feedback signals.
	4	/B	
10	5	10	
150.05	6	/10	
14 000 4	7	AL	
13 0 0 3 12 0 0 2	8	/AL	
11000	9	+5 V	+5 V output for feedback.
6	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 (double-ended) and non-differential (single-ended) TTL feedback.

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



**CNC 8040** 

### **Connector X5**

### For electronic handwheel connection

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

Pin		Signal and function	
	1	A1	
	2	/A1	Feedback signals of first handwheel.
	3	B1	
	4	/B1	
10	5	A2	
15 00 5	6	/A2	Feedback signals of second handwheel.
14 0 0 4	7	B2	
12 000 2	8	/B2	
11 000 1	9	+5 V	Supply output.
6	10	+5 V	Supply output.
	11	GND	Supply output.
	12	GND	Supply output.
	13	100P	Push button of Fagor 100P handwheel.
	14		
	15	Chassis	Shield

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

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

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

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

For further information on the characteristics of the feedback inputs (square and sinusoidal signals), see the appendix at the end of this manual. See "*Technical specifications of the CNC*" on page 471.

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

### **Connector X6**

### **For Operator Panel connection**

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

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

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

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



**CNC 8040** 

### 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.

All shields must only be connected to ground at the CNC end through pin 37 of the connector leaving the other end free. The wires of the shielded cables cannot be unshielded for more than 75mm (about 3 inches).

Pin		Signal a	nd function
	1	24 V	External power supply.
	2	O33	
	3	O35	
	4	O37	
	5	O39	
	6	041	
	7	O43	
	8	O45	
	9	O47	
37 0 0 19	10	O49	
36 0 0 18	11	O51	
34 0 16	12	O53	
1 33 0 0 15	13	O55	
32 0 0 14	14	197	
$\frac{31}{30}$ 0 12	15	199	
29 0 0 11	16	l101	
20 0 A A	17	I103	
27 0 0 8	18	0 V	External power supply.
25 0 0 7	19	0 V	External power supply.
24 0 0 5	20	24 V	External power supply.
00 2 9 4	21	O34	
21 6 0 3	22	O36	
20 0 2	23	O38	
	24	O40	
	25	O42	
	26	O44	
	27	O46	
	28	O48	
	29	O50	
	30	O52	
	31	O54	
	32	O56	
	33	198	
	34	I100	
	35	l102	
	36	I104	
	37	Chassis	Shield.

1.

8040 CNC CONFIGURATION
CNC structure



**CNC 8040** 

Pin		Signal a	nd function		
		1	Chassis	Shield.	
			2	Cons 1	Velocity command output for the first axis.
9	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.	
	1	6	GND		
	-	7	GND	Analog voltage reference signals.	
	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).

1.

8040 CNC CONFIGURATION
CNC structure



**CNC 8040** 

### Digital inputs (I65 to I96)

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 to pins 18 and 19 (for 0V) of the connector.

All shields must only be connected to ground at the CNC end through pin 37 of the connector leaving the other end free. The wires of the shielded cables cannot be unshielded for more than 75mm (about 3 inches).

Pin		Signal a	nd function
	1		External power supply.
	2	165	
	3	167	
	4	169	
	5	l71	
	6	173	
	7	175	
	8	177	
	9	179	
37 0 0 19	10	l81	
36 0 18 36 0 17	11	183	
35 0 16	12	I85	
33 0 0 10	13	187	
32 0 0 13	14	189	
30 0 12	15	I91	
29 0 0 11	16	193	
27 0 0 9	17	195	
	18	0 V	External power supply.
25 0 0 7	19	0 V	External power supply.
24 0 0 5	20		
00 - 0 4	21	166	
21 0 0 0	22	168	
20 0 2	23	170	
,	24	172	
	25	174	
	26	176	
	27	178	
	28	180	
	29	182	
	30	l84	
	31	186	
	32	188	
	33	190	
	34	192	
	35	194	
	36	196	
	37	Chassis	Shield.

1.

8040 CNC CONFIGURATION

CNC structure



**CNC 8040** 

### Connectors X10, X11, X12 and 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 a	nd function
	1	Α	
	2	/A	
	3	В	
	4	/B	Feedback signals.
10	5	10	
15 <b>0 0</b> 5 14 <b>0 0</b> 4	6	/10	
	7	AL	
13 0 0 3	8	/AL	
110001	9	+5 V	Voltage supply for the feedback system.
6	10	+5 V	
Ŭ	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

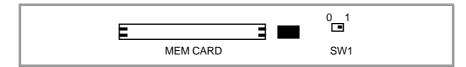
It detects overcurrents or short-circuits at the feedback devices issuing the relevant error message.

"Supply voltage error on axis feedback \*".



**CNC 8040** 

### Slot "MEM CARD" For the memkey card (CNC configuration card)



This slot admits "linear memory card" type cards.

Slot MEM CARD is used for the "memkey card" and for upgrading software versions. Selector SW1 must be in the ·0· position and it must only be handled when updating the software.

The memkey card 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 "Memkey Card" may be used to store data on machine customizing (user screens, PLC program backup and/or machine parameters, etc.) as well as user partprograms.



The memkey card must be inserted into the CNC so part-programs can be executed. With the memory card that contains the software version, the CNC CANNOT execute anything.



If the CNC is turned on with the "Memkey card" in and the SW1 switch set to ·1·, the CNC does not come on, but its data is NOT affected.

Reinstall the CNC software when replacing the -Hard Disc- module. The CNC software and the -Hard Disc- module must be compatible.

### Slot MEM CARD" as floppy disk drive

The slot of the memkey card can also be used as a floppy disk drive. To do this, use the softkey menu to identify the slot as CARDA. As a storing device, use either Fagor memory cards or standard off-the-shelf cards "INTEL series 100".

In order to be able to use a memory card, it must be formatted in Fagor mode.

- This operation may be carried out at the CNC from the mode: "Diagnosis / Tests / Format CARDA". If a memkey card has been inserted, it does not show the formatting option.
- At the PC, the memory card may be formatted using WinDNC.

The software configuration screen shows the card's hardware ID. If a floppy disk type memory card is inserted, the ID will be FFFFFFE.



When powering up the CNC with a memory card in, it considers it invalid and it issues an error indicating minimum configuration. On the other hand, that memory card will not be recognized as such until it is extracted from the CNC and inserted back.

### Using a memory card as floppy disk

Only part-programs and PLC message and error programs may be stored in the memory card.

The memory card may be inserted and extracted from the CNC while the CNC is on. The memkey card cannot be extracted during execution while being accessed. For example, if it is executing from CARDA, it is displaying screens from CARDA, etc.

**CNC** structure 8040 CNC CONFIGURATION



**CNC 8040** 

**CNC** structure

### Slot "CMPCT FLASH"

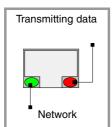
### Slot for the local hard disk (compact flash).

It has a compact flash type hard disk to store user programs. The hard disk cannot be accessed from the outside.

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

### **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.

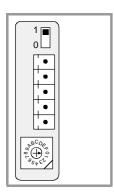


The NFS protocol is used to communicate with the remote hard disk. This protocol must be available at the PC that is used as server.



**CNC 8040** 

### 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 inputs and outputs, Depending on the elements used, each remote module may have the following configurations:

- 24 digital inputs and 16 digital outputs.
- 48 digital inputs and 32 digital outputs.
- 72 digital inputs and 48 digital 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 element integrated into the CAN 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 module 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  $\cdot 1$  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

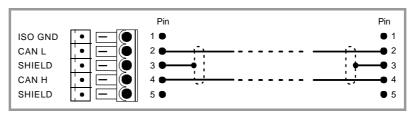
The "Line\_Term" switch identifies which elements are occupying the ends of the CAN bus; i.e. the first and last physical elements of the connection.

The central unit is always one end of the line. The other end will be the last group of remote modules.

The switch of the elements at the ends must be in the "1" position and the rest of the elements in the "0" position.

### **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.



**CNC 8040** 

**CNC** structure

### 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  $(1 \times 2 \times 0.22 \text{ mm}^2)$ .

Flexibility: Superflexible. Minimum static bending radius of 50 mm and

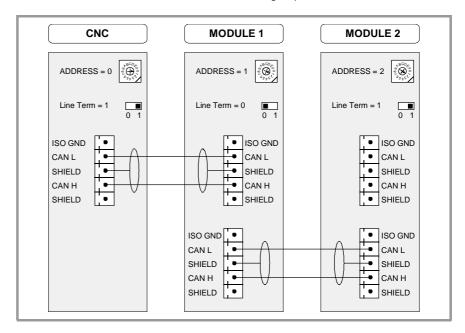
a dynamic radius of 95 mm.

Cover: PUR.

Impedance: Cat.5 (100  $\Omega$  - 120  $\Omega$ ).

### Interconnection of modules

They are connected in series using any of the two connectors. The figure shows the CAN connection between the central unit and 2 groups of remote modules.





**CNC 8040** 

### **DIGITAL DRIVES**

### Digital servo (CAN or Sercos)

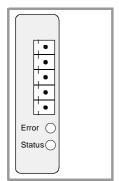
Two types of digital servo system may be used to communicate with Fagor servo drives:

- 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

CAN servo is not available when the CNC has the CPU-Turbo board.



### Module identification at the bus

Each element integrated into the CAN 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.

Although the switch has 16 positions, only positions 1 through 8 are valid. The CNC does not have a switch and always assumes position 0 inside the bus. The rest occupy consecutive positions (recommended) starting from  $\cdot 1 \cdot$ .

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 elements are occupying the ends of the CAN bus; i.e. the first and last physical elements of the connection.

The central unit is always one end of the line. The other end will be the last group of remote modules.

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 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  $(1 \times 2 \times 0.22 \text{ mm}^2)$ .

Flexibility: Superflexible. Minimum static bending radius of 50 mm and

a dynamic radius of 95 mm.

Cover: PUR

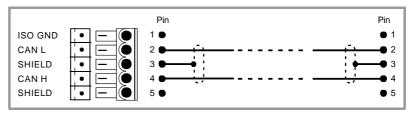
Impedance: Cat.5 (100  $\Omega$  - 120  $\Omega$ )



**CNC 8040** 

### **CAN** connector pinout

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

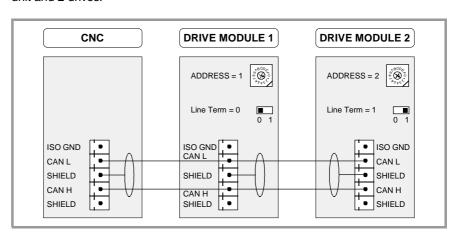


Signal	Description
ISO GND	Ground / 0 V.
CAN L	Bus signal (LOW).
SHIELD	CAN shield.
CAN H	Bus signal (HIGH).
SHIELD	CAN shield.

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.

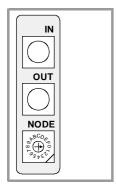




**CNC 8040** 

### Sercos digital servo system

The -Sercos- module must be used when the CNC has the CPU- Turbo board.



### Module identification at the bus

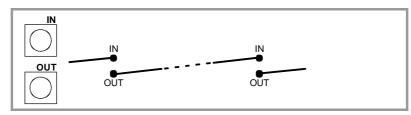
Each element integrated into the Sercos bus is identified with the 16-position rotary switch (0-15) "Address" (also known as Node\_Select). This rotary switch selects the address (node) occupied by each element integrated in the bus.

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

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



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

### **Cable characteristics**

Fiber optic cable of Ø3,6 mm or Ø6 mm and very flexible.

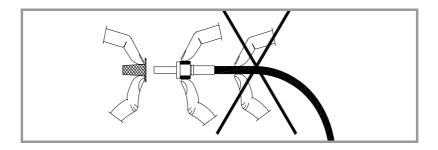
- For the Ø3,6 mm cable, a minimum static bending radius of 30 mm and a dynamic radius of 60 mm.
- For the Ø6 mm cable, a minimum static bending radius of 50 mm and a dynamic radius of 70 mm.

PUR type covering.

### Cable handling

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

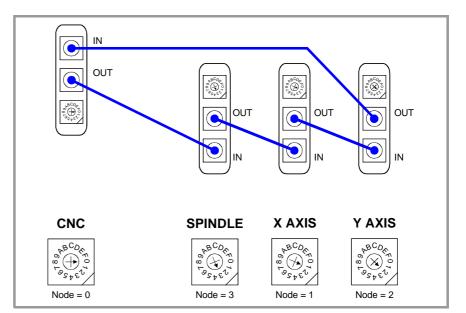
To remove the terminal protecting cap or to connect or disconnect the cable, hold the cable by its terminal; never pull the cable by holding its plastic part because it could render it useless.





**CNC 8040** 

It is connected line 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.





8040 CNC CONFIGURATION
CNC structure

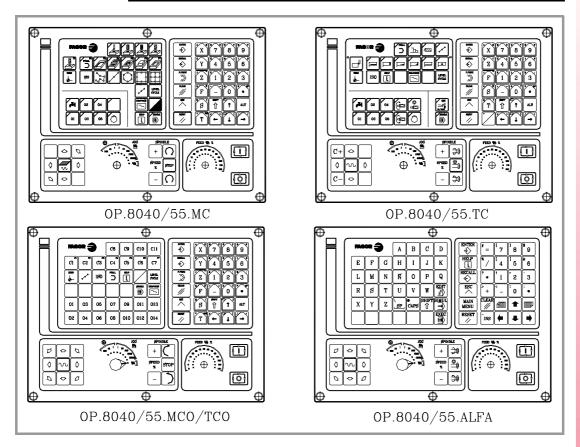


**CNC 8040** 

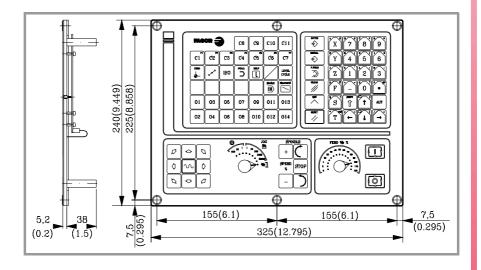
### 1.2 **Operator panel**

The operator panel depends on the CNC model.

M model	Operator panel: OP.8040/55.ALFA
T model	Operator panel: OP.8040/55.ALFA
MC model	Operator panel: OP.8040/55.MC
TC model	Operator panel: OP.8040/55.TC
MCO/TCO model	Operator panel: OP.8040/55.MCO/TCO



### Dimensions:

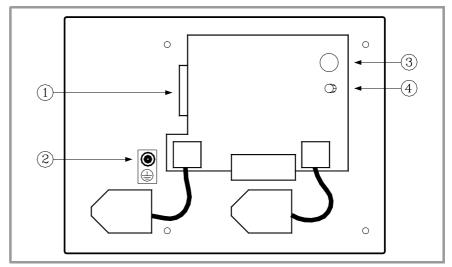


Operator panel 8040 CNC CONFIGURATION

**FAGOR** 

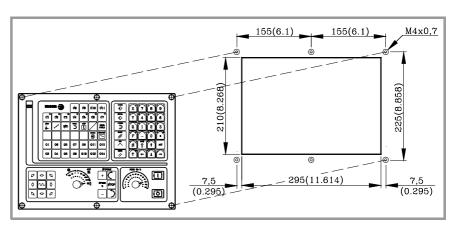
**CNC 8040** 

### 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:

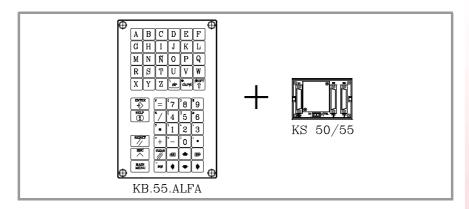




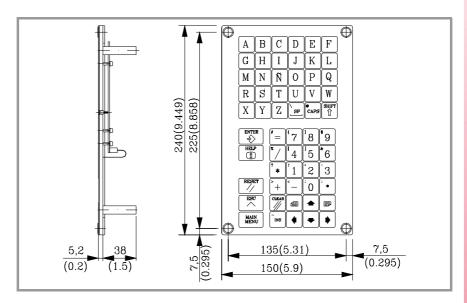
**CNC 8040** 

### 1.2.1 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 (KB55.ALFA). This keyboard is connected to the operator panel through the KS50/55 adapter.



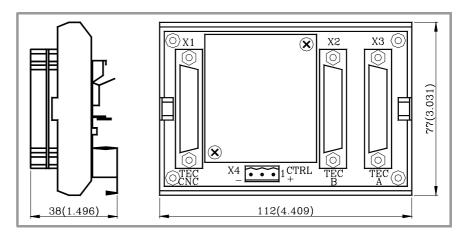
### Dimensions:





**CNC 8040** 

### 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.

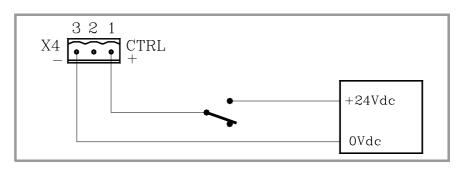
Pin	Value	Meaning
1	0 V	The CNC attends to the operator panel.
	24 V	The CNC attends to the alphanumeric keyboard.
2		Not being used.
3	GND	External power supply.

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





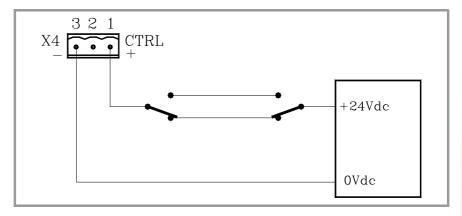
CNC CONFIGURATION
Operator panel



**CNC 8040** 

8040 CNC CONFIGURATION Operator panel

### With two switchers



### With a PLC

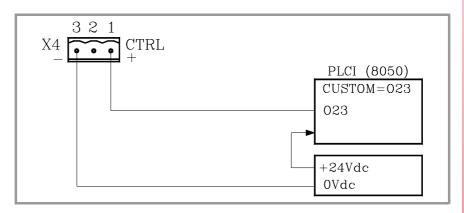
CNC general logic output CUSTOM (M5512) indicates to the PLC the currently selected work mode.

CUSTOM (M5512) = 0M 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.





**CNC 8040** 



**CNC 8040** 

### **HEAT DISSIPATION**

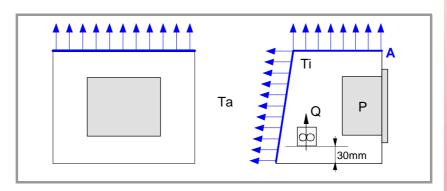
2

The temperature of the enclosure for the central unit must not exceed 45  $^{\circ}$ C (113  $^{\circ}$ F) while the unit is running. 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 whose walls are 2 mm thick and is 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 (°C) Ambient temperature outside the enclosure.
- Ti (°C) Temperature inside the enclosure.
- ∆t (°C) Temperature difference (Ti-Ta).
- Q (m<sup>3</sup>/h) Air flow provided by the fan, if there is one.

### **Dissipating surface**

Only the top and rear of the enclosure are considered as convection heat dissipating surfaces. The rest of the surfaces will not be considered in the total surface.

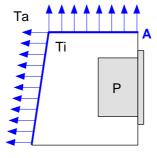
### Power dissipated by the CNC

The maximum power dissipated by the CNC is 60 W, power supply not included.



**CNC 8040** 

## 2.1 Heat dissipation by natural convection



Surface without paint.

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

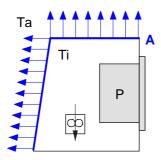
Surface with smooth metallic enamel.

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

**CNC 8040** 

### 2.2 Heat dissipation by forced convection with inside fan

Fan whose air flow is  $Q = 13.6 \text{ m}^3/\text{h}$  facing down.



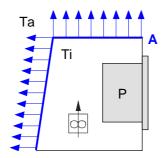
Surface without paint.

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

Surface with smooth metallic enamel.

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

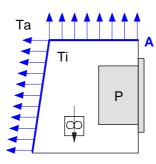
Fan whose air flow is  $Q = 13.6 \text{ m}^3/\text{h}$  facing up.



Surface without paint.

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

Fan whose air flow is  $Q = 30 \text{ m}^3/\text{h}$  facing down.



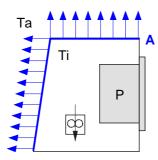
Surface without paint.

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

Surface with smooth metallic enamel.

$$A = \frac{P}{9.1 \cdot \Delta T}$$

Fan whose air flow is  $Q = 102 \text{ m}^3/\text{h}$  facing down.



Surface without paint.

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

Surface with smooth metallic enamel.

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



**CNC 8040** 

HEAT DISSIPATION

Heat dissipation by air flow to the outside using a fan

Surface without paint.

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

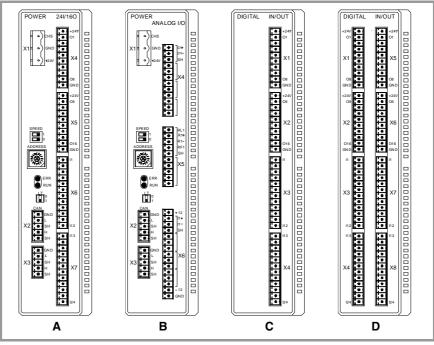
It must be borne in mind that this air flow through the unit allows extracting hot air out, but it may let dust in. A filter should be placed in order to maintain the permitted ambient conditions.

**CNC 8040** 

# REMOTE MODULES (BUS CAN WITH CANOPEN PROTOCOL)

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.

### 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.



**CNC 8040** 

### Probe connection.

The digital inputs let you handle the signal of the two probes (5 Vdc or 24 Vdc). Use machine parameters to define which digital input is associated with each probe and their type of pulse.



**CNC 8040** 

# REMOTE MODULES (BUS CAN WITH CANOPEN PROTOCOL)

### 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

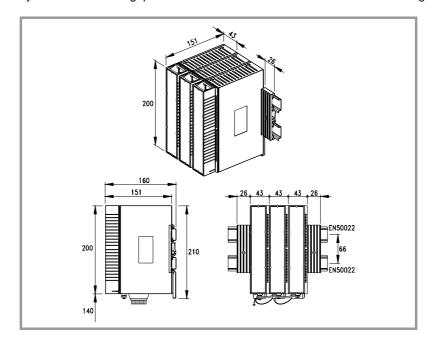
### **Dimensions of the modules**

maintaining the right gap between the profiles.

Installation of the modules

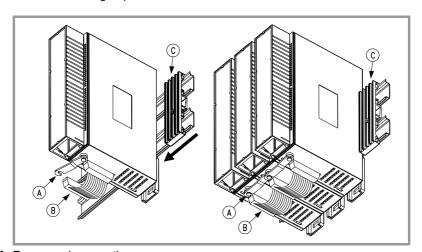
3.1

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 ends.

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

**FAGOR** 

**CNC 8040** 

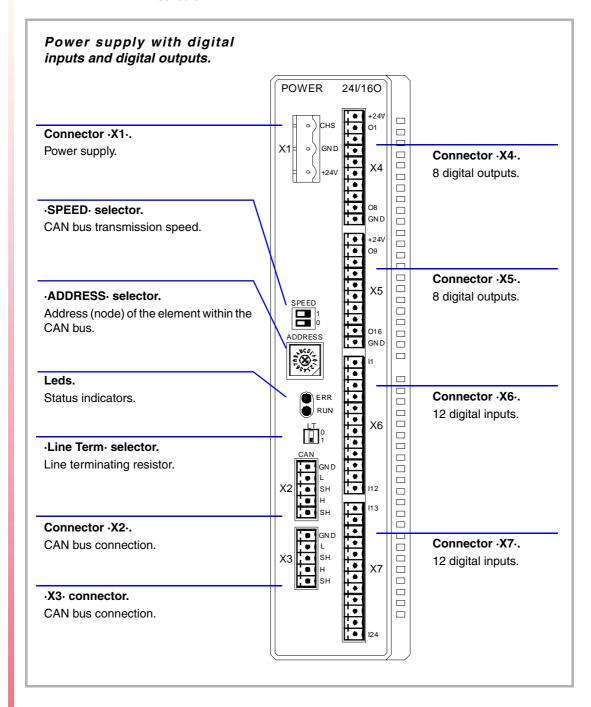
Power supply

### 3.2 Power supply

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

There are two power supply models

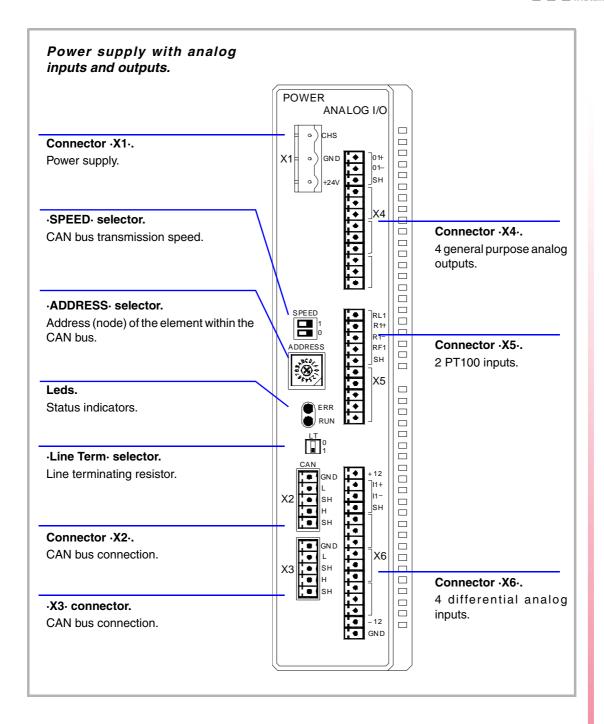
- Power supply with 24 digital inputs and 16 digital outputs.
- Power supply with 4 analog inputs, 4 analog outputs and 2 inputs for temperature sensors.





**CNC 8040** 





#### **Description of the connectors (power supply)**

#### Connector ·X1·. Power supply.

3-pin male Phoenix connector (7,62 mm pitch).

	Pin	Signal and function	
о)снѕ	1	Chassis	Shield.
□ ○ GND	2	GND	Power supply.
+24V	3	+24 V	Power supply.

3.

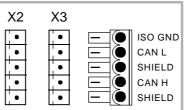
N PROTOCOL)

Power supply

FAGOR =

**CNC 8040** 

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



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.

#### Led ·ERR·. Status indicator LED.

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

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.	

#### Led ·RUN·. Status indicator LED.

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

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.



Power supply



**CNC 8040** 

Power supply

#### ·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 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.

#### 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.

X4		X5	
. •	+24V		+24V
	01	•	O9
	O2	•	O10
	О3	•	011
	O4	•	012
	O5	•	013
•	O6	•	014
•	07	•	O15
. ●	O8		016
	GND	•	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).

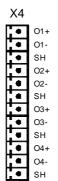
X6		X7	
. •	l1	. •	l13
•	12		114
•	13	+	115
	14	•	I16
•	15	•	117
•	16	•	l18
	17	•	119
•	18	•	120
. •	19	٠	121
•	I10	•	122
•	l1 1	•	123
•	l12	•	124

Signal	Function
l1 - l12	Digital inputs.
l13 - l24	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
01+ 01-	Analog output.
O2+ O2-	Analog output.
O3+ O3-	Analog output.
O4+ O4-	Analog output.
SH	Grid connection.



**FAGOR** 

**CNC 8040** 

Power supply

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

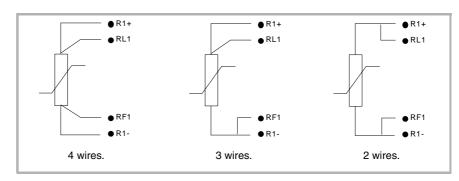
#### Connector ·X5·. Inputs for the temperature sensor PT100 (2 inputs).

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

X5	
. •	RL1
	R1+
. •	R1-
. •	RF1
	SH
•	RL2
•	R2+
•	R2-
<b>,</b> •	RF2
	SH

Signal	Function
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).

X6	
	+12
	l1+
	l1-
	SH
	12+
	12-
	SH
	l3+
	l3-
	SH
	14+
. •	14-
	SH
	-12
	GND
	•

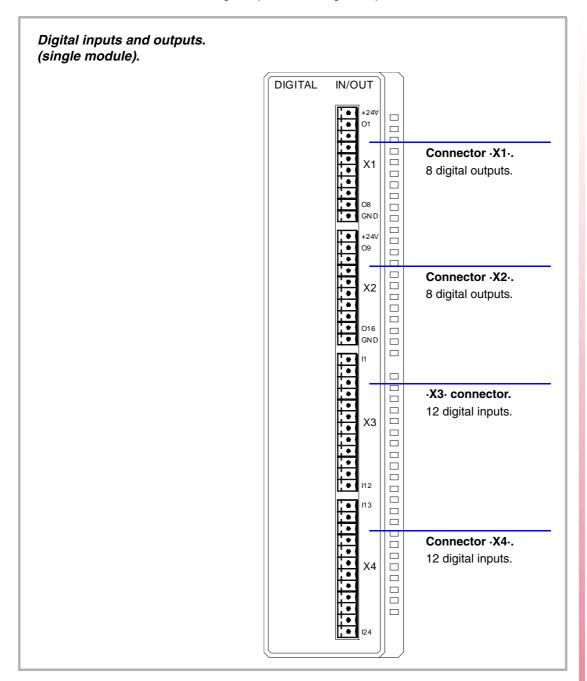
Signal	Function
l1+ l1-	Analog input
12+ 12-	Analog input
13+ 13-	Analog input
14+ 14-	Analog input
SH	Grid connection.
+12	Reference signals.
-12	
GND	

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



**CNC 8040** 

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



3.

REMOTE MODULES (BUS CAN WITH CANOPEN PROTOCOL)

Digital inputs and digital outputs (single module).



**CNC 8040** 

#### 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.

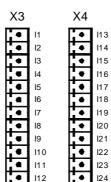
GND

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).

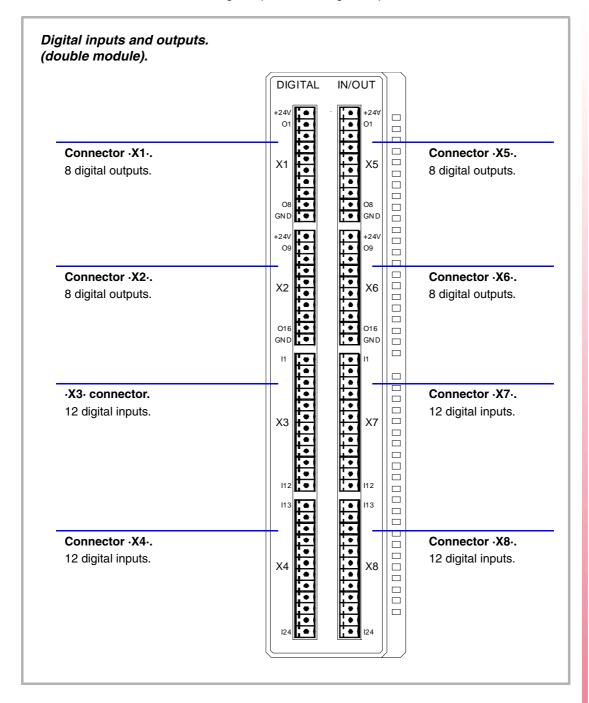


Signal	Function
l1 - l12	Digital inputs.
l13 - l24	Digital inputs.



**CNC 8040** 

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



3.

REMOTE MODULES (BUS CAN WITH CANOPEN PROTOCOL)

Digital inputs and digital outputs (double module).



**CNC 8040** 

#### Description of the connectors (digital inputs and outputs)

Connector  $\cdot$ X1· &  $\cdot$ X2· &  $\cdot$ X5· &  $\cdot$ 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.

X1-X5		X2-X6	
	+24V	. •	+24V
•	01	. •	O9
	O2	•	O10
	О3	•	011
	04	•	012
•	O5		O13
•	O6	. •	014
•	07	. •	O15
	08		O16
•	GND	•	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).

- pa.oox				
X3-X7			X4-	X8
. •	11	Г	•	113
	12		•	114
•	13		•	115
	14		٠	I16
•	15		٠	117
4	16		•	l18
	17		•	119
•	18		•	120
	19		٠	121
•	I10		•	122
•	l111		٠	123
•	l12		•	124

Signal	Function
l1 - l12	Digital inputs.
l13 - l24	Digital inputs.



**CNC 8040** 

REMOTE MODULES (BUS CAN WITH CANOPEN PROTOCOL)

#### Electrical characteristics of the inputs and outputs. 3.5

#### 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 DC and +30 V DC).
High threshold "1"	Over +18 V DC.
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 DC 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 ΚΩ.
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:

Voltage range	±10 V
Resolution	16 bits
Minimum impedance of the connected device	10 ΚΩ.
Maximum cable length (unshielded)	75 mm.

Electrical characteristics of the inputs and outputs.



**CNC 8040** 

#### 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 $^{\circ}$ C ( -328 $^{\circ}$ F) and +850 $^{\circ}$ C (1562 $^{\circ}$ F).	
Resolution	0.1 ºC	
Typical consumption of each input	2 mA.	
Maximum cable length (unshielded)	75 mm.	



**CNC 8040** 

#### Numbering of the digital inputs and outputs 3.6

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 "5.8 PLC Parameters" on page 160.





**CNC 8040** 



**CNC 8040** 

# MACHINE AND POWER CONNECTION

4



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 to install the CNC away from coolants, chemical products, possible blows etc. which could damage it.

#### Mains connection of the central unit

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

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

Use an independent external power supply with the following specifications:

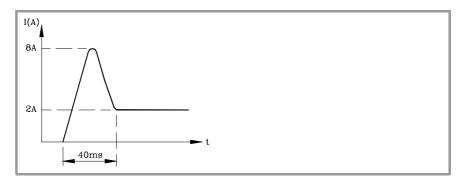
Nominal voltage:	20 V minimum	30 V maximum
Ripple:	4 V	
Nominal current:	2 A	
Current peak on power-up:	8 A	

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



**CNC 8040** 

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



#### **Machine connection**

The machine tool must have decoupled all those elements capable of generating interference (relay coils, contactors, motors, etc.)

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

 $C 0.2 \mu F / 600 V$ 

· AC motors.

RC connected between phases with values:

R 300  $\Omega$  / 6 W

 $C 0.47 \mu F / 600 V$ 

#### **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).

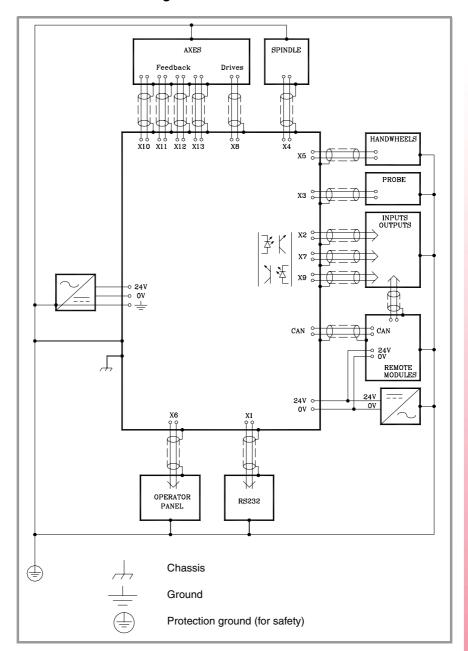
When a second point is necessary, it is recommended to join both points with a cable whose section is no smaller than 8 mm<sup>2</sup>.

Verify that the impedance between the central point of each connector housing and the main ground point is less than 1  $\Omega$ 



**CNC 8040** 

#### Ground connection diagram







**CNC 8040** 

#### 4.1 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 Vdc.

Maximum voltage value +30 V.

Minimum voltage value +18 V.

Output voltage 2 V less than the supply voltage.

Maximum output current 100 mA

All outputs are protected by means of:

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

#### **Digital inputs**

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

The electrical characteristics of these inputs are:

Nominal voltage value +24 Vdc

Maximum voltage value +30 Vdc

Minimum voltage value +18 Vdc

High threshold voltage (logic level 1) from +18 V up.

Low threshold voltage (logic level 0) Under +5 V

Typical consumption for each input 5 mA.

Maximum consumption for 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.



**CNC 8040** 

MACHINE AND POWER CONNECTION

### 4.2 Digital inputs and outputs.

#### **Analog outputs**

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

Analog voltage 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 8040 CNC configuration".



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



CNC 8040

#### 4.3 Setup

#### General considerations

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.



**CNC 8040** 

#### 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 "5 Machine parameters".

These values, shown in the parameter tables, may be modified manually from the CNC's keyboard or from a peripheral (cassette reader, floppy disk reader, computer, etc.) via the two serial communication ports RS 232C and RS 422.

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

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

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

#### Setting of the machine parameters for the axes

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

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

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

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

- · Adjust the axes one by one.
- Connect the power output of the drive corresponding to the axis being adjusted.
- Selecting the Jog mode at the CNC, jog the axis to be adjusted.
  - In case of runaway, the CNC will display the relevant following error and the machine parameter labelled LOOPCHG (corresponding to the sign of the analog output of the CNC) will have to be changed.
- If the axis does not run away; but the direction of the move is not the desired one, parameters labelled AXISCHG (P13) (axis feedback counting direction) and LOOPCHG (P26) (sign of the analog 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.



**FAGOR** 

**CNC 8040** 

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 lo 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.



**CNC 8040** 

#### 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 analog 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 analog input.

#### Maximum feedrate adjustment

It is recommended to adjust the drives so the maximum feedrate is obtained with an analog signal 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:

 $(20 \text{ V} / 1000 \text{ rpm}) \times 3000 \text{ rpm.} = 60 \text{ V}$ 



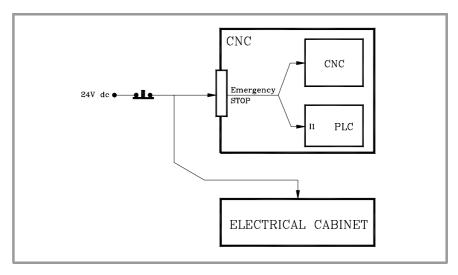


**CNC 8040** 

#### 4.4 Connection of the emergency input and output

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

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



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:

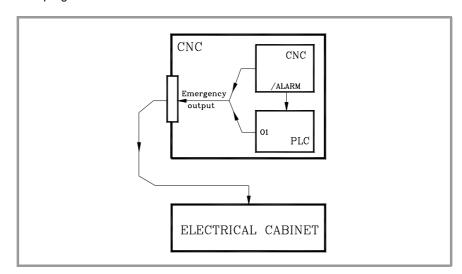
- E-stop has been pressed.
- The travel limit of any axis has be exceeded.
- There is a malfunction on a drive or it is locked without analog signal.



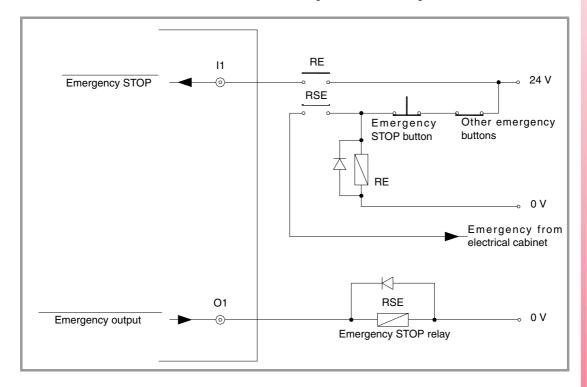
**CNC 8040** 

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

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



The recommended connection diagram is the following:





FAGOR

**CNC 8040** 



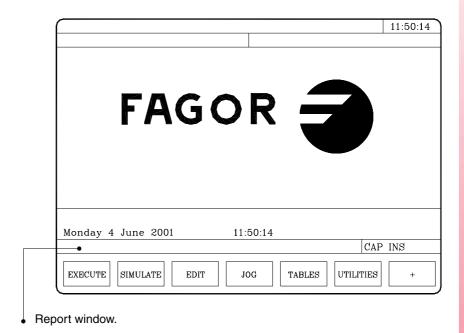
**CNC 8040** 

## **MACHINE PARAMETERS**



It is recommended to save the CNC machine parameters into the memkey card (CARD A) 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 operation 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 memkey card (CARD A).



8040 CNC

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 ports 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.
1	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  $[\mbox{\bf 1}]$  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.



8040 CNC

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 memkey card (CARD

A), a peripheral device or a PC.

SAVE Save the tables into the memkey card" (CARD A), a peripheral

device or a PC.

MM/INCHES See the parameter values in the desired units. Only those

parameters affected by this conversion will be altered. It will not change the g.m.p. INCHES (P8) that indicates machine units.

#### Machine parameter setting

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

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

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

There are some parameters to indicate whether the machine has cross compensation or not. 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 memkey card (Card A).



8040 CNC

## 5.1 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 "12.3 Variables associated with machine parameters" on page 376.

General machine parameters:

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

#### Machine parameters of an axis:

Parameter	Number	Update	
BACKLASH	P14	Immediate	
ACCTIME	P18	Beginning of the next block	
INPOSW	P19	Immediate	
MAXFLWE1	P21	Immediate	
MAXFLWE2	P22	Immediate	
PROGAIN	P23	Immediate	
DERGAIN	P24	Immediate	
FFGAIN	P25	Immediate	
MINANOUT	P27	Immediate	
SERVOFF	P28	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	

#### 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

**5**.

Parameters that may be modified from the oscilloscope, OEM program or OEM subroutine



8040 CNC

Parameter	Number	Update
PROGAIN	P23	Immediate
DERGAIN	P24	Immediate
FFGAIN	P25	Immediate
MINANOUT	P27	Immediate
SERVOFF	P28	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

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



8040 CNC

#### **General machine parameters** 5.2

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 analog voltage output and the default values associated with each parameter.

Parameter		Feedback	Analog voltage	Default value	
		(connector)		- M -	- 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	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)

**MACHINE PARAMETERS** General machine parameters



8040 CNC

#### 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 "6.3 Movement with an electronic handwheel" on page 192.

#### 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).

#### **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

Default value: 0

#### **IPLANE (P11)**

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

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

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

#### ILCOMP (P12)

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

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

Default value: 0



8040 CNC

#### 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:

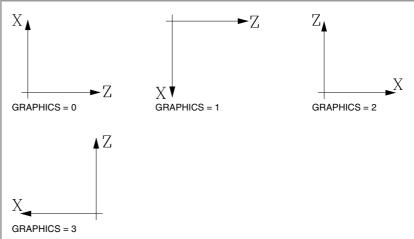
Value	Meaning
0	Real position values.
1	Theoretical position values.

Default value: 0

#### **GRAPHICS (P16)**

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

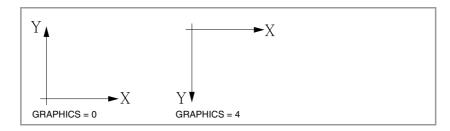
Value	Meaning
0	Mill graphics.
1	Mill graphics with added W axis.
2	Boring Mill graphics.
3	Boring Mill graphics with added W axis.
4	Mill graphics (changed line graphics).

Default value: 0



8040 CNC

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



#### **RAPIDOVR (P17)**

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

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

Default value: NO

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

The feedrate % can always be changed in JOG movements.

#### **MAXFOVR (P18)**

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

Possible values	
Integer numbers between 0 and 255.	
	Default value: 120

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

#### **CIRINLIM (P19)**

Indicates the maximum angular feedrate value for circular interpolations.

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

#### Possible values

Integer numbers between 0 and 65535.

Default value: 0 (it is not limited)

#### Example

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

The theoretical angular speed is:

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

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

Feedrate to be applied =  $1500 \times 0.5 = 750 \text{ mm/min}$ .



8040 CNC

#### **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 an	d 255.
	Default values 100 (for the M model)

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.



8040 CNC

# MACHINE PARAMETERS General machine parameters

#### **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 whether the machine is a machining center or not.

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.	
	D - f   k   00

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 "6.8 Auxiliary M, S, T function transfer" on page 224.

#### 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)



8040 CNC

#### **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)



8040 CNC

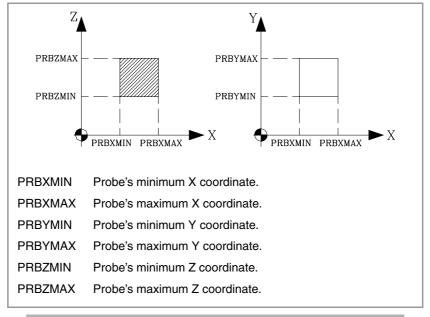
# MACHINE PARAMETERS General machine parameters

# PRBPULSE (P39) Indicates whether the probe functions of the CNC react to the up-flank (leading edge) or down-flank (trailing edge) of the probe signal. This probe is connected to the connector X7 of the AXES module.

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.



### 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)



8040 CNC

### **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. (Internally between 100 and 299).

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)



8040 CNC

# General machine parameters

**MACHINE PARAMETERS** 

### 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	en 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)



8040 CNC

### 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)	Not being used.
TRPROG (P63)	Not being used.
TRDERG (P64)	Not being used.
MAXDEFLE (P65)	Not being used.
MINDEFLE (P66)	Not being used.
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.



8040 CNC

### 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: 120

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

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

### PERCAX (P70)

Not being used.

### TAFTERS (P71)

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

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

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

Default value: NO

### **LOOPTIME (P72)**

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

Value	Meaning
0	4 ms period (standard).
16	Period in milliseconds.



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

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

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

### **IPOTIME (P73)**

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

For instance, a 2 msec sampling and interpolation time results in block processing time of 4.5 msec. for a 3-axis linear interpolation with no tool compensation.

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



8040 CNC

### **COMPTYPE (P74)**

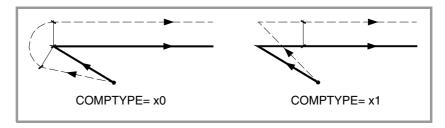
It determines how tool radius compensation is applied. This parameter has three digits.

### (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

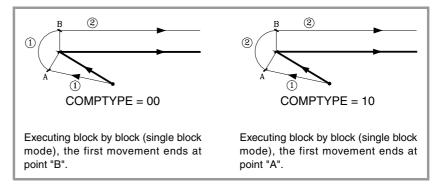


### (tens) Additional compensation block.

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

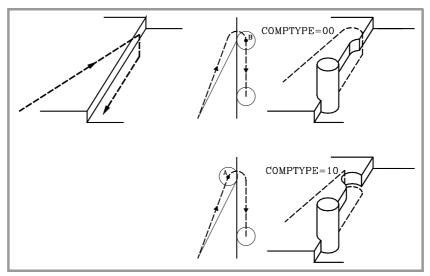


When the beginning or the end of the compensation takes place in a different plane (there is an intermediate vertical movement) and with angle greater than  $270^{\circ}$ , one should be analyze the CNC's behavior as shown next:

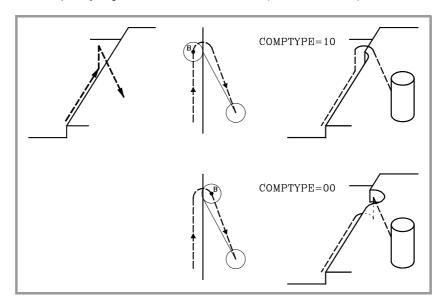
At the beginning of the compensation, the tool should be positioned before
penetrating into the part. The additional block must be executed in the upper plane
and, consequently, together with the first block (COMPTYPE=00).



8040 CNC



 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.

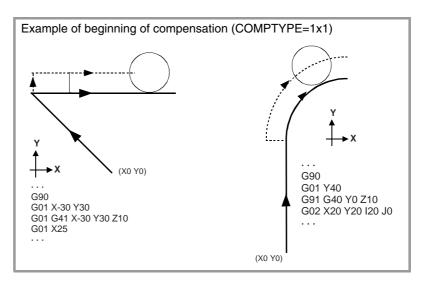
Default value: 000

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.



8040 CNC



### 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



8040 CNC

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	0.001 mm	0.010 mm	0.100 mm	
4.4"	0.0001"	0.0010"	0.0100"	
4.4 mm	0.0001 mm	0.0010 mm	0.0100 mm	
3.5"	0.00001"	0.00010"	0.00100"	
6.2 mm	0.01 mm	0.10 mm	1.00 mm	
5.3"	0.001"	0.010"	0.100"	



8040 CNC

**FAGOR** 

### MPGNPUL (P82)

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

### Possible values

Integer numbers between 0 and 65535.

Default value: 0 (means 25)

## Example

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

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

With 5.3mm type display format, set MPGRES=1

With 4.4mm type display format, set MPGRES=2

With 6.2mm type display format, set MPGRES=0

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



8040 CNC

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).

**5**.

MACHINE PARAMETERS
General machine parameters



8040 CNC

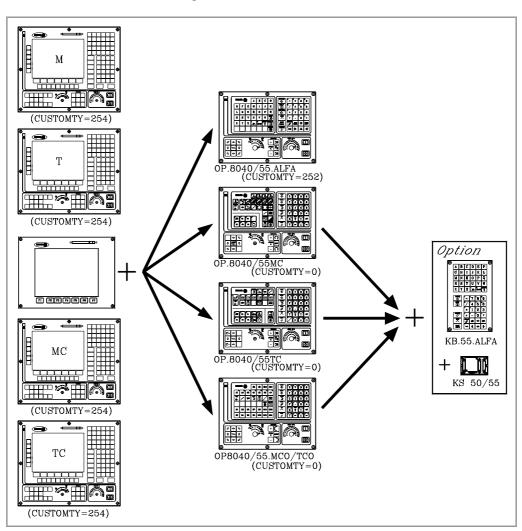
i

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

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

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 has been set for a lathe and the CNC shows the error message.

### Possible configurations.



**5**.

MACHINE PARAMETERS
General machine parameters



8040 CNC

### Keyboards having auto-identification.

CNC 8040-M-COL-K

CNC 8040-MC-COL-K

CNC 8040-TC-COL-K

CNC 8040-TC-COL-K

CNC 8040-TC-MON-K

CNC 8040-TC-MON-K

OP.8040/55.ALFA OP.8040/55.MC

OP.8040/55.TC OP.8040/55.MCO/TCO

XFORM (P93) Not being used.

XFORM1 (P94) Not being used.

XFORM2 (P95) Not being used.

Not being used.

XDATA0 (P96)

U (P90)

XDATA1 (P97) XDATA2 (P98)

**XDATA3 (P99)** 

XDATA4 (P100)

XDATA5 (P101)

XDATA6 (P102) XDATA7 (P103)

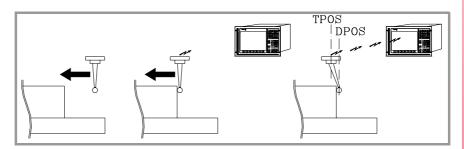
**XDATA8 (P104)** 

**XDATA9 (P105)** 

PRODEL (P106)

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

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.

**5**.

MACHINE PARAMETERS
General machine parameters



8040 CNC

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

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

This parameter has 16 bits counted from right to left. Each bit has a function or work mode associated with it. By default, all the bits will be assigned the value of .0. Assigning the value of ·1· activates the corresponding function.

Bit	Meaning	Bit	Meaning
0		8	G51
1		9	G50
2		10	
3		11	
4	JOG	12	
5	Rigid tapping	13	G33
6	G95	14	G01
7	G75 / G76	15	G00

Default value in all the bits: 0

Every time each of these functions or work modes is activated, the CNC checks the value assigned to the corresponding bit and acts as follows:

bit = 0It applies the first of the sets "ACCTIME, PROGAIN, etc."

It applies the second of the sets "ACCTIME2, PROGAIN2, etc" bit = 1

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, 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.



8040 CNC

### DIPLCOF (P110)

**HANDWIN** 

**HANDWHE1** 

HANDWHE4 (P115)

(P111)

(P112) HANDWHE2

(P113) HANDWHE3

(P114)

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.

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.  ${\sf HANDWIN}$  (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:

l17	I33	1225	I241	button signal from the handwheel with selector button (only the first one)
l18	I34	1226	1242	A signal from the first handwheel.
119	I35	1227	1243	B signal from the first handwheel.
120	I36	1228	1244	A signal from the second handwheel.
l21	I37	1229	1245	B signal from the second handwheel.
122	I38	1230	1246	A signal from the third handwheel.
123	I39	1231	1247	B signal from the third handwheel.
124	I40	1232	1248	A signal from the fourth handwheel.
125	l41	1233	1249	B signal from the fourth handwheel.



MACHINE PARAMETERS
General machine parameters

FAGOR =

8040 CNC

### 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 inches/min 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



8040 CNC

# **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

The speeds of 8 MHz and 16 MHz will only be available when using the Sercos816 board. Previous boards only admitted 2 MHz and 4 MHz; when defining a higher value, the speed is limited to 4 MHz.



When using the Sercos816 board and versions older than V9.11 (mill) or V10.11 (lathe), the transmission speed will be limited to 4 MHz.

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 total length of the cable being used. Set it with the same value used by the drives.

### Possible values (Sercos board).

Value	Meaning
2	For cables of up to 7 meters (23 ft).
4	For cables between 7 and 15 meters (23 ft and 49 ft).
6	For cables longer than 15 meters (23 ft).

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
1, 2, 3, 4	For cables of up to 15 meters (23 ft).
5, 6	For cables between 15 and 30 meters (23 ft and 49 ft).
7	For cables between 30 and 45 meters (23 ft and 49 ft).
8	For cables longer than 45 meters (23 ft).

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	6	Portuguese
1	Spanish	7	Czech
2	French	8	Polish
3	Italian	9	Mainland Chinese
4	German	10	Basque
5	Dutch	11	Russian

Default value: 0



8040 CNC

**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) Not being used.

**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 "AUXTYPE (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 whethe the CNC displays the real or theoretical RPM (affected by the %) of the main spindle.

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

Default value: 0

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



8040 CNC

# General machine parameters

**MACHINE PARAMETERS** 

### 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  $\cdot 0 \cdot$ . Assigning the value of  $\cdot 1 \cdot$  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 whether the rapid key affects the execution and simulation or not. The behavior of this key is managed using the EXRAPID mark.

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

Default value: 0

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

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

### MSGFILE (P131)

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

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

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

PLCMSG	Texts for PLC messages
PLCERR	Texts for PLC errors
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 memkey card. If it is in both places, it takes the one in user memory.



8040 CNC

### 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.

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.

ı	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



8040 CNC

**AXIS9 (P136)** 

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).

General machine parameters **MACHINE PARAMETERS** 



8040 CNC

**ACTBAKAN (P145)** It is related to a.m.p. BAKANOUT (P29) and BAKTIME (P30), additional analog 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  $\cdot 0 \cdot$ . Assigning the value of  $\cdot 1 \cdot$  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).



8040 CNC

MACHINE PARAMETERS

### CODISET (P147)

This parameter is associated with the MC 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  $\cdot 0 \cdot$ . Assigning the value of  $\cdot 1 \cdot$  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 - 15	Not being used.

Default value in all the bits: 0

### 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.

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.

Each parameter has 16 bits counted from right to left. By default, all the bits will be assigned the value of  $\cdot 0 \cdot$  (option available). Assigning the value of  $\cdot 1 \cdot$  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.



8040 CNC

## Meaning of the parameters in the MC work mode.

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

Parameter	Multiple positioning			
COCYZPOS	at several points (Bit 0)	In line (Bit 3)	in arc (Bit 6)	in polar arc (Bit 7)
	in grid pattern (Bit 11)	in parallelogram (Bit 12)	(= 11 3)	(=1117)

## 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					

**MACHINE PARAMETERS** General machine parameters



8040 CNC

### JERKACT (P160) Jerk ON.

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  $\cdot 0 \cdot$ . Assigning the value of  $\cdot 1 \cdot$  activates the corresponding function.

Bit	Meaning
0	Apply jerk control in look-ahead.
1 - 15	Not being used.

Default value in all the bits: 0

### 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.

### **TLOOK (P161)** Real block processing time for look-ahead.

Possible values	
Integers between 0 and 65535 ms.	
	Default value: 0

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

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

### MAINTASF (P162) 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  $\cdot 1 \cdot$  (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) Not being used.



8040 CNC

### 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.
1	Several tools per 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 defined in diameter at the lathe model.

### 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".

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.

### 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.



8040 CNC

**MACHINE PARAMETERS** 

### 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.



8040 CNC

FEEDTYPE (P170) Behavior of the feedrate when programming F0.

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.

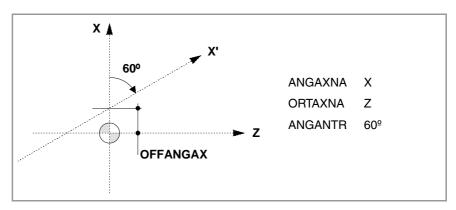
If set to 0, it is not possible to program F0 or execute motion blocks with F0 active.

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



8040 CNC

# General machine parameters

**MACHINE PARAMETERS** 

### ORTAXNA (P172) Axis perpendicular to the Cartesian axis associated with the incline 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 0°, 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 $300^\circ$ , 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 $300^{\circ}$ , it calculates the intersection. In the rest of the cases such as COMPMODE = 0.

Default value: 0

### COMPMODE = 0.

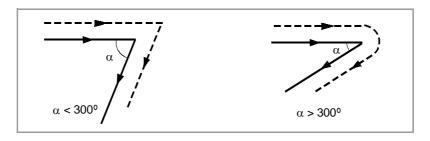
The compensation method depends on the angle between paths.

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

### COMPMODE = 2.

The compensation method depends on the angle between paths.

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





8040 CNC

### **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  $\cdot 0 \cdot$ . Assigning the value of  $\cdot 1 \cdot$  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.

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

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

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

### Bit 13. Manual intervention enabled with look-ahead.

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

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

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

### Bit 15. Manual intervention with additive handwheel is available.

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

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



8040 CNC

### 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 400 µs and this parameter may be used to set a delay of 600 µs.

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  $400 \mu s$ ) 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.



The 0 value (400  $\mu$ s) is compatible with all versions except V9.11, V10.11, V9.12 and V10.12. When updating the software from one of these versions, the value of SERCDEL1 must be set to 600.

SERCDEL2 (P179) Not being used.

### **EXPLORER (P180)** It sets how to access the explorer.

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

Default value: 0

### **REPOSTY (P181)** It may be used to select the repositioning mode:

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

Default value: 1

### **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.



8040 CNC

## For the block search: Disabling in execution:

DISSIMU	L =	xxxx	XXXX	0/1 x x x	XXXX	
bit 7 = 1	EXE	C G SEA	RCH			
bit 6 = 1	EXE	C GMST	SEARCH			

For simulation: Disabling in simulation:

DISSIMUL	.= 0/1 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



8040 CNC

# 5.3 Axis parameters

### **AXISTYPE (P0)** It sets the type of axis and whether it is governed by the CNC or by the PLC.

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

Default value: 0



By default, rotary axes are Rollover and are displayed between  $0^{\circ}$  and  $359.9999^{\circ}$ . 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 10°, and its target position is 350°, the axis will go through, 10°, 9°, ... 352, 351, 350.

See "6.1 Axes and coordinate systems" on page 173.

# **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 X 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".



8040 CNC

### 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 X 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



8040 CNC

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 = 36°.

### 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.



8040 CNC

### **NPULSES (P8)**

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

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  $\cdot 0 \cdot$ , the CNC will assume the equivalent values of the drive.

### **DIFFBACK (P9)**

Indicates whether the spindle encoder uses differential signals (double ended) or not.

Value	Meaning
NO	It does NOT use differential signals.
YES	It uses differential signals.

Default value: YES

### SINMAGNI (P10)

Indicates the multiplying factor (x1, x4, x20, etc.) that the CNC must apply only to sinusoidal feedback signal.

For square feedback signals, this parameter must be set to 0 and the CNC will always apply a multiplying factor of x4.

Possible values	
Integer numbers between 0 and 255.	
	Default value: 0

The axis feedback resolution must be set using a.m.p. PITCH (P7), NPULSES (P8) and SINMAGNI (P10) as shown in the following table:

	PITCH (P7)	NPULSES (P8)	SINMAGNI (P10)
Square signal encoder	Leadscrew pitch	Nr of pulses	0
Sinusoidal signal encoder	Leadscrew pitch	Nr of pulses	multiplying factor
Square signal linear encoder	linear encoder pitch	0	0
Sinusoidal signal linear encoder	linear encoder pitch	0	multiplying factor

### FBACKAL (P11)

This parameter is to be used only when the feedback signals are sinusoidal or differential (double ended).

Indicates whether the feedback alarm for this axis will be ON or OFF.

Value	Meaning
OFF	No feedback alarm desired, it is canceled.
ON	Feedback alarm is being used.

Default value: ON



8040 CNC

### **FBALTIME (P12)**

It indicates the maximum time that the axis may stay without properly responding to the CNC's command.

Depending on the command for an axis, the CNC calculates the number of feedback pulses that it must receive for each sample period.

The axis will be considered that it is working fine when the number of pulses received is between 50% and 200% of the calculated number.

If at any time, the number of feedback pulses received is out of this range, the CNC will keep checking that axis until it detects that the number of pulses received has come back to normal. But if more time elapses than the one indicated in this parameter without the axis coming back to normal, the CNC will issue the relevant error message.

### Possible values

Integers between 0 and 65535 ms.

Default value: 0 (it is not checked)

### **AXISCHG (P13)**

Indicates the counting direction. If correct, leave it as is, but to change it, select YES if it was set to NO and viceversa. When changing this parameter, also change a.m.p. LOOPCHG (P26).

### Possible values

NO / YES.

Default value: NO

BACKLASH (P14) Indicates the amount of backlash. Enter 0 when using linear encoders.

### Possible values

Within ±99999.9999 degrees or millimeters.

Within ±3937.00787 inches.

Default value: 0

LSCRWCOM (P15) Indicates whether the CNC should apply leadscrew error compensation or not.

Value	Meaning
OFF	Leadscrew compensation not being used.
ON	Leadscrew compensation being used.

Default value: OFF

### **NPOINTS (P16)**

Indicates the number of leadscrew error compensation points available in the table. The values in this table will be applied if a.m.p. "LSCRWCOM" (P15) is ON.

### Possible values

Integer numbers between 0 and 255.

Default value: 30

### DWELL (P17)

Indicates the dwell from the moment the "ENABLE" signal is activated until the analog voltage 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)



8040 CNC

### 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 =  $(9.5/20000) \times 1000 = 475 \text{ mV}$ 

Therefore "PROGAIN" = 475



8040 CNC

Indicates the value of the derivative gain. Its value represents the analog voltage (in millivolts) corresponding to a change in following error of 1mm (0.03937 inches) in 10 milliseconds.

This analog voltage will be added to the one calculated for the proportional gain.

**Analog voltage** 

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

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

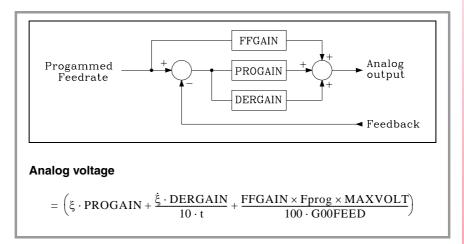
### Possible values

Integer numbers between 0 and 65535.

Default value: 0 (derivative gain not applied)

FFGAIN (P25)

Indicates the % of the analog voltage 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.



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

### Possible values

Between 0 and 100.99 rpm (numbers with 2 decimals).

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

Usually, a value between 40% and 80% is assigned depending mainly on the type of machine and its characteristics.



When having a CNC with a version V11.01 or later where FFGAIN or FFGAIN2 have two decimals, when passing them to an older version, those parameters lose their decimals.

LOOPCHG (P26)

Indicates the sign of the analog output. 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

**5**.

MACHINE PARAMETERS
Axis parameters



8040 CNC

MINANOUT (P27) Indicates the minimum analog output for this 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 voltage of 10V.

Default value: 0

MINANOUT	Minimum analog output
1	0.3 mV.
3277	1 V.
32767	10 V.

**SERVOFF (P28)** Indicates the analog offset value for the spindle 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 voltage of 10V.

Default value: 0 (not applied)

SERVOFF	Analog voltage
-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 voltage of 10V.

Default value: 0 (not applied)

BAKANOUT	Additional analog voltage
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 analog voltage will be applied for a period of time indicated in the a.m.p. BAKTIME (P30).



8040 CNC

### **BAKTIME (P30)**

It indicates the duration of the additional velocity command pulse to make up for backlash in movement reversals.

Possi	ble	val	ues	

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 inches/min 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.



8040 CNC

### **MAXVOLT (P37)**

Indicates the maximum analog voltage 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 inches/min 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 inches/min 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 inches/min and 7874.01574 inches/min.

Default value: 1000 mm/min.

# FAGOR =

8040 CNC

### 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.

(SOFT M: V11.1x) (SOFT T: V12.1x) **MAXCOUPE (P45)** Indicates the maximum difference allowed between the following errors of the axes electronically coupled (by program, PLC or as Gantry axes).

### Possible values

Between 0.0001 and 99999.9999 degrees or millimeters.

Between 0.00001 and 3937.00787 inches.

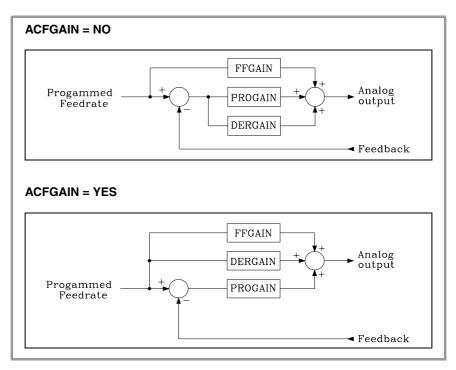
Default value: 1 mm.

### **ACFGAIN (P46)**

Indicates whether or not the value assigned to a.m.p. DERGAIN (P24) is applied onto the variations of the programmed feedrate (AC-forward).

Value	Meaning
NO	It is applied on variations of following error (derivative gain).
YES	It is applied on the variations of the programmed feedrate that are due to acceleration/deceleration (AC-forward).

Default value: YES



### **REFSHIFT (P47)**

This parameter is used when once the machine has been all set up, it is necessary to reinstall the feedback system and the new machine reference point (home) no longer coincides physically with the previous one.

It indicates the difference existing between the two reference points, the previous one and the current one.

### Possible values

Within ±99999.9999 degrees or millimeters.

Within ±3937.00787 inches.

Default value: 0

If this parameter has a value other than 0, once the home search has been carried out (the reference mark of the feedback device has been detected), the CNC moves the distance indicated by a.m.p. REFSHIFT (P47). This way, the machine reference point will always be the same.

This movement is carried out at the feedrate indicated by a.m.p. REFEED2 (P35).

### STOPTIME (P48) STOPMOVE (P49)

These parameters are used in conjunction with a.m.p. "STOPAOUT (P50)" with function G52 (move to hardstop).

**MACHINE PARAMETERS** Axis parameters



8040 CNC

Axis parameters

### 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 analog voltage 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 voltage of 10V.

Default value: 0

STOPAOUT	Minimum analog output
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: 0.01 mm.

It should be assigned a value 10 times the value of "INPOSW"

### 8040 CNC

**FAGOR** 

**IOTYPE (P52)** 

Axis machine parameter I0TYPE has two digits:

**MACHINE PARAMETERS** 

Value	Meaning
x0	normal I0.
x1	"A" type distance-coded I0.
x2	Type B distance-coded reference mark (only linear encoder COVS).
x3	Normal I0 (search with retraction)

When using linear encoders with distance-coded reference marks (I0), set a.m.p. I0CODI1 (P68) and I0CODI2 (P69).

### Tenths:

It defines whether it stops smoothly or not when detecting the reference mark of the axes or not.

Value	Meaning
0x	Normal stop on I0.
1x	Smooth stop on I0.

When setting the smooth stop, parameters "DERGAIN" and "FFGAIN" should be set to zero.

### ABSOFF (P53)

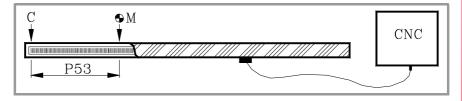
The CNC takes this parameter into account when a.m.p. I0TYPE (P52) has been set with a value other than 0.

Linear encoders having a distance-coded reference mark indicate the machine position with respect to the "zero" of the linear encoder.

# Possible values Within ± 99999.9999 millimeters. Within ±3937.00787 inches. Default value: 0

respect to the machi

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).





8040 CNC

### 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.	
-		D = f = 4 =   0

Default value: 0

It is recommended (not necessary) that the Can addresses of the various axes and spindles be consecutive and start from number  $\cdot 1 \cdot$  (the address of the CNC is always  $\cdot 0 \cdot$ ). For example, with 3 Sercos axes and 1 Sercos spindle, the values of this parameter must be 1, 2, 3 and 4.



From version V9.11 on, this parameter is called DRIBUSID and is valid for both Sercos and CAN servo systems. In previous versions, this parameter was called SERCOSID.

### **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	
EXTMULT (P57)	Period of the graduation on the glass (mechanical period or Period of the feedback signal (electrical period)
	Default value: 0

### Example:

E.g. Fagor linear encoder "FOT" has a graduation pitch of 100  $\mu$ m and a feedback signal period of 20  $\mu$ m.

EXTMULT = 100 / 20 = 5



8040 CNC

### Values to be assigned for Fagor encoders with distance-coded IO.

Linear	encod	ers				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)
НО	SO	90000 pulses	1000	1001	5
НО	SO	180000 pulses	1000	1001	10
HOP	SOP	18000 pulses	1000	1001	1

**5**.

MACHINE PARAMETERS
Axis parameters



8040 CNC

### **SMOTIME (P58)**

Sometimes the axis does not respond as desired on particular movements (handwheel movements, etc.).

In these cases, the axis response may be smoothed by applying a filter to speed variations. This filter is set by parameter SMOTIME that indicates the duration of the filter in milliseconds, value given by g.m.p. LOOPTIME (P72).

### Possible values

Integers between 0 and 64 times the value assigned to g.m.p. LOOPTIME (P72).

If LOOPTIME = 0 (4 ms), the maximum value that could be assigned to SMOTIME will be  $64 \times 4 = 256$  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.

Firs	t set	Secon	d 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).



From version V9.11 on, this parameter is called DRIBUSLE and is valid for both Sercos and CAN servo systems. In previous versions, this parameter was called SERCOSLE.



8040 CNC

# MACHINE PARAMETERS Axis parameters

### 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 to the machine reference point (or marker pulse).

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: YES

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 "6.13 Axes (2) controlled by a single drive" on page 250.

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 "6.14 Additive coupling between axes" on page 253.

Value	Meaning	Value	Meaning
0	None.	6	With the W axis.
1	With the X axis.	7	With the A axis.
2	With the X axis.	8	With the B axis.
			D ( 1: 1 0

Default value: 0

8040 CNC

Value	Meaning	Value	Meaning
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

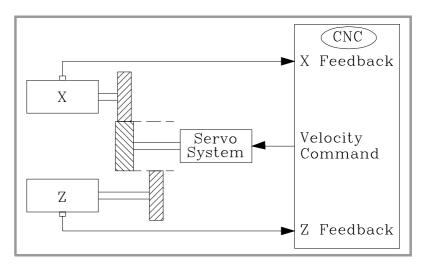
Parameter SWINBACK (P66) must also be set when having two axes controlled by a servo system or wishing to do an additive coupling,

### 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.
10	For an additive coupling.

Default value: 0

See "6.13 Axes (2) controlled by a single drive" on page 250.

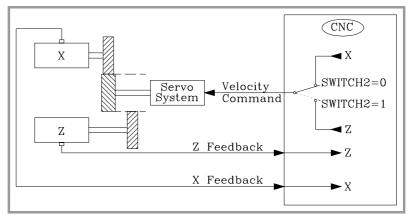
The following examples show the various possibilities of two axes controlled by a single servo system. In all of them, the toggling of the analog voltage must be done from the PLC using the SWTCHZ mark.



8040 CNC

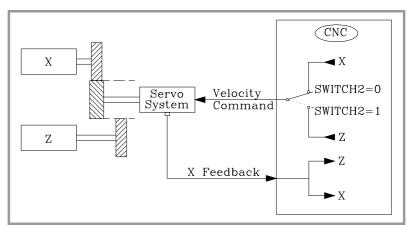
A. Each axis has its own feedback device.

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



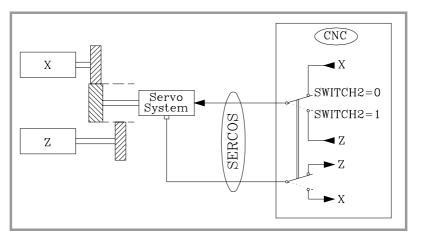
**B.** The two axes share the same feedback device. It must be connected to the feedback connector of the main axis.

X axis (main) SWINBACK of the X axis = 0. Z axis (secondary) SWINBACK of the Z axis = 0.



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

X axis (main) SWINBACK of the X axis = 0. Z axis (secondary) SWINBACK of the 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.



8040 CNC

### 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<sup>3</sup>.

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 IOCODD1 (P68) = 1000. Default value: For IOCODD2 (P69) = 1001.

### **Example with Fagor linear encoder**

Gap between two fixed distance-coded  $\,$  20 000  $\mu m$ 

I<sub>0</sub>'s

Gap between two variable distance-coded  $\,$  20 020 $\mu m$ 

l₀'s

Signal period 20  $\mu m$ 

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 IO.

Linear	encode	ers				I0CODI1 (P68)	10CODI2 (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



8040 CNC

Rotary	y encoder		I0CODI1 (P68)	I0CODI2 (P69)	EXTMULT (P57)
НО	SO	90000 pulses	1000	1001	5
НО	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.

### Possible values

From 0 to 4.

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

When applying a filter, it must be set with an order of ·3·. Before setting it to another value, consult with Fagor Automation's technical service.



If the filter has been designed wrong, it will not be applied.

The filters are not applied while moving with an electronic handwheel or a mechanical handwheel or while tracing.

It is recommended not to activate these filters on machines carrying out movements against a hard stop.

### **TYPE (P71)**

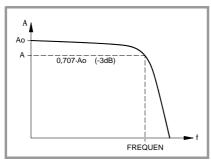
Filter type. Two types of filters may be used, namely "low passing" or "antiresonance". To obtain a good machining quality, all the axes and the spindle interpolating with each other should be defined with the same type of filter and with the same frequency.

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

Default value: 0

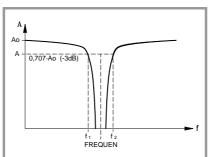
When defining anti-resonance filters, parameters NORBWIDTH and SHARE must also be set.

### "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.



8040 CNC

FREQUEN (P72)

The meaning of this parameter depends on the type of filter being applied.

For the "low passing" filter, it indicates the cutoff frequency or frequency at which its amplitude drops 3 dB or 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)** 

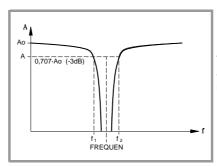
Standardized bandwidth.

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)

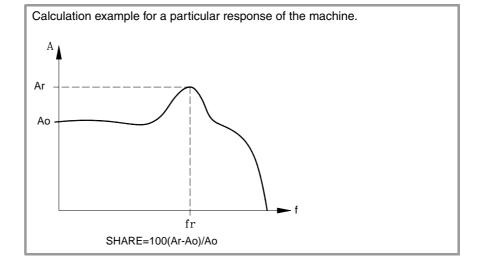
Signal percentage that passes through the filter. This value must be equivalent to the percentage overshooting of the resonance because it has to make up for it.

This parameter is only taken into account for the "anti-resonance (notch)" filter type.

## Possible values

between 0 and 100

Default value: 100



FAGOR

8040 CNC

### FLIMIT (P75)

Maximum safety limit for the axis feedrate. This limit is activated from the PLC and is applied to all the work modes, including the PLC channel.

### Possible values

Between 0 and 99999.9999 degrees/min or mm/min.

Between 0 inches/min and 3937.00787 inches/min.

Default value: 0

This limit is activated for all the axes using the mark FLIMITAC (M5058). When the limit is canceled, the CNC recovers the programmed feedrate.

This limit permits clearing the axis feedrate temporarily via PLC, e.g. when opening the doors, etc.

### **TANSLAID (P76)**

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%.

Possi	hla	VO	11100

Between 0 and 100% (both included).

Default value: 50



8040 CNC

### 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 = 
$$k_p \cdot T_{error}$$

### 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%.

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

### **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 = 
$$k_i \cdot \sum T_{error}$$

### Possible values

Integers between 0 and 65535 ms.

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



8040 CNC

### 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: 10000 mm/min.

This limit does not affect the rapid jog which will still have the value of parameter G00FEED.

### PITCHB (P86)

Leadscrew pitch.

This parameter must only be set when using CAN servo system. When not using CAN servo system, the leadscrew pitch is defined with parameter PITCH (P7).

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



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

### HPITCH (P89)

On Hirth axes, it indicates its pitch in degrees. When set to  $\cdot 0 \cdot$ , it assumes a pitch value of  $1^{\circ}$ .

### 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 10° (and under 10°). If the pitch value is greater than 10°, it will move a single step.

MACHINE PARAMETERS
Axis parameters



8040 CNC

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



8040 CNC

## 5.4 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.



8040 CNC

### 5.4.1 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.
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 analog for M03 and M04.

Value	Meaning
+ sign	Positive analog.
- sign	Negative analog.

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) signal with the indicated sign.



8040 CNC

### **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) = 120.

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	
rossible values	
Integer numbers between 0 and 25	.5

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 spindle encoder uses differential signals (double ended) or not.

Value	Meaning
NO	It does NOT use differential signals.
YES	It uses differential signals.
	D (        VEO

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

8040 CNC

### **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 analog voltage 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.



8040 CNC

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 voltage 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 =  $1000^{\circ}/\text{min}$  (2.778 rev/rpm).

Command from the drive: 9.5V for 500 rpm

Analog output corresponding to S = 1000 °/min. (2.778 rpm)

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 voltage (in millivolts) corresponding to a change in following error of 1mm (0.03937 inches) in 10 milliseconds.

This analog voltage will be added to the one calculated for the proportional gain.

### **Analog voltage**

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

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)

### FFGAIN (P25)

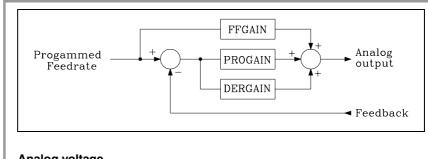
The CNC takes this parameter into account when operating in closed loop (M19).

Indicates the % of the analog voltage 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.

MACHINE PARAMETERS
Spindle parameters



8040 CNC



### **Analog voltage**

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

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)

Usually, a value between 40% and 80% is assigned depending mainly on the type of machine and its characteristics.

### LOOPCHG (P26)

Indicates the sign of the analog output. 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



Two axes controlled by a single drive.

In versions older than V7.01 and V8.01, the sign of the velocity command of the associated axis corresponded to that of the main axis. From this version on, you must define the sign of the command in both axes.

When updating the software version, the secondary axis may run away. Before installing the new version, make sure that parameter LOOPCHG of the associated axis has the same value as that of the main axis.

### **MINANOUT (P27)**

Indicates the minimum value for the spindle analog output.

### Possible values

It is given in D/A converter units and it admits integer values between 0 and 32767 which corresponds to an analog voltage of 10V.

Default value: 0

MINANOUT	Minimum analog output
1	0.3 mV.
3277	1 V.
32767	10 V.



8040 CNC

### SERVOFF (P28)

Indicates the analog offset value for the spindle drive.

### Possible values

It is given in D/A converter units and it admits integer values between 0 and  $\pm 32767$  which corresponds to an analog voltage of 10V.

Default value: 0 (not applied)

SERVOFF	Analog voltage
-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 to 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.	
		D ( ); 1

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.

**5**.

MACHINE PARAMETERS
Spindle parameters



8040 CNC

### 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)

Indicates the analog voltage corresponding to the maximum speed of gears 1, 2, 3 and 4.

MAXVOLT 3 (P39) MAXVOLT 4 (P40)	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 voltage 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



8040 CNC

### 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  $\cdot 1 \cdot$  (the address of the CNC is always  $\cdot 0 \cdot$ ). For example, with 3 Sercos axes and 1 Sercos spindle, the values of this parameter must be 1, 2, 3 and 4.



From version V9.11 on, this parameter is called DRIBUSID and is valid for both Sercos and CAN servo systems. In previous versions, this parameter was called SERCOSID.

### **OPLACETI (P45)**

When working in open loop (M3, M4) spindle speed 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.

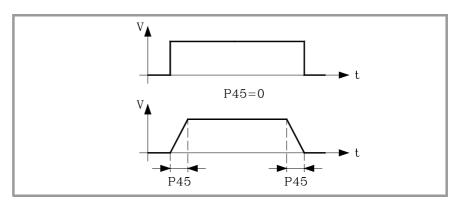
Possible values	
Integers between 0 and 65535 ms.	
	Default value: 0 (in steps).

5.

MACHINE PARAMETERS
Spindle parameters



8040 CNC



### **SMOTIME (P46)**

Sometimes the axis does not respond as desired on particular movements (handwheel movements, etc.).

In these cases, the response of the spindle may be smoothed by applying a filter to the speed changes.

This filter is set by parameter SMOTIME that indicates the duration of the filter in milliseconds, value given by g.m.p. LOOPTIME (P72).

### Possible values

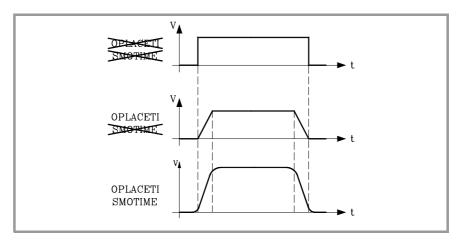
Integers between 0 and 64 times the value assigned to g.m.p. LOOPTIME (P72).

If LOOPTIME = 0 (4 ms), the maximum value that could be assigned to SMOTIME will be  $64 \times 4 = 256$  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.



FAGOR

8040 CNC

(SOFT M: V11.1x) (SOFT T: V12.1x) 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		Secon	d set
ACCTIME PROGAIN DERGAIN	(P18) (P23) (P24)	ACCTIME2 PROGAIN2 DERGAIN2	(P48)
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).

**MACHINE PARAMETERS** 

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).



From version V9.11 on, this parameter is called DRIBUSLE and is valid for both Sercos and CAN servo systems. In previous versions, this parameter was called SERCOSLE.

### When using a communication interface via Sercos bus.

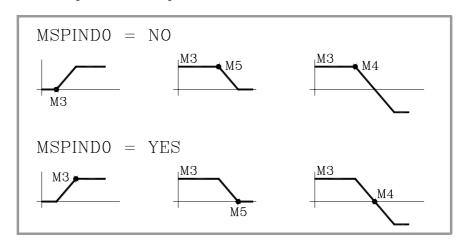
DRIBUSLE = 0	The CNC controls the position loop. The axis feedback is done via connector. The command to the drive is sent out via Sercos.
DRIBUSLE = 1	The CNC controls the position loop.  The axis feedback is done via Sercos. First feedback (motor feedback).  The command to the drive is sent out via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

### MSPIND0 (P52)

Indicates when functions M3, M4, M5 are to be sent out. While the spindle is accelerating and decelerating.





8040 CNC

### 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	Third set	
ACCTIME	(P18)	ACCTIME2	` '	ACCTIME3	(P55)	
PROGAIN	(P23)	PROGAIN2		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.

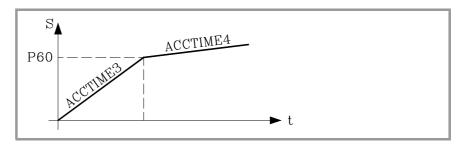
Default value: For ACCTIME3 (P55) = 4000 ms .
For PROGAIN3 (P56) = 50 mV/degree.
For DERGAIN3 (P57) = 0.
For FFGAIN3 (P58) = 100.

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.





8040 CNC

MACHINE PARAMETERS

### Possible values

Integers between 0 and 65535 ms.

Parameter ACCTIME4 is set like ACCTIME3.

Default value: 8000.

### SECACESP (P60)

Parameter SECACESP (P60) indicates at what speed the acceleration change takes place. If P60=0, it always applies ACCTIME3.

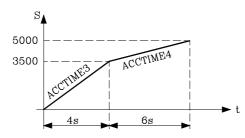
### Possible values

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

 $ACCTIME3(P55) = 6000 \times 4 / 3500 = 6857 \text{ ms.}$ 

 $ACCTIME4(P59) = 6000 \times 6 / 1500 = 24000 \text{ ms.}$ 

### 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.



8040 CNC

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

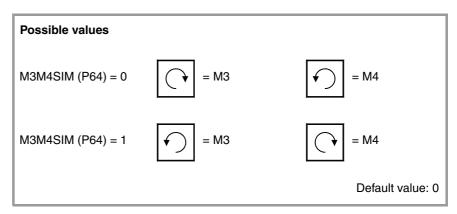
Integers between 0 and 65535 rpm.

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.



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 0.001° 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



8040 CNC

MACHINE PARAMETERS

### 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.

### Possible values

From 0 to 4.

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

When applying a filter, it must be set with an order of  $\cdot 3 \cdot$ . Before setting it to another value, consult with Fagor Automation's technical service.



If the filter has been designed wrong, it will not be applied.

The filters are not applied while moving with an electronic handwheel or a mechanical handwheel or while tracing.

It is recommended not to activate these filters on machines carrying out movements against a hard stop.

### **TYPE (P68)**

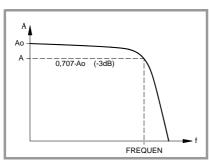
Filter type. Two types of filters may be used, namely "low passing" or "antiresonance". To obtain a good machining quality, all the axes and the spindle interpolating with each other should be defined with the same type of filter and with the same frequency. For the spindle, the filters are only applied 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.

Default value: 0

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

### "Low passing" filter.

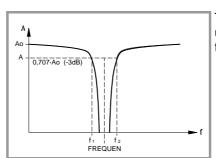


The "low passing" filter is used to limit the jerk by making the movements smoother although it has the drawback that it rounds the corners slightly.



8040 CNC

### Anti-resonance filter (notch filter).



The "anti-resonance" (notch) filter must be used when the machine has a resonance frequency to be eliminated.

# FREQUEN (P69)

The meaning of this parameter depends on the type of filter being applied.

For the "low passing" filter, it indicates the cutoff frequency or frequency at which its amplitude drops 3 dB or 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)**

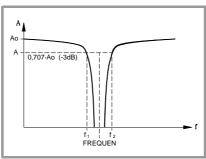
Standardized bandwidth.

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.

$$NORBWID = \frac{FREQUEN}{(f_2 - f_1)}$$



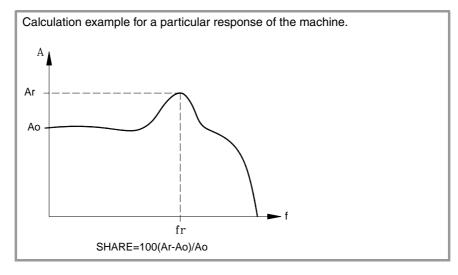
8040 CNC

### 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.





INPREV1 (P72) OUTPREV1 (P73) INPREV2 (P74) OUTPREV2 (P75) INPREV3 (P76) OUTPREV3 (P77) INPREV4 (P78) OUTPREV4 (P79) When using a CAN servo system (only with DRIBUSLE = 0), these parameters set the gear ratios in each range.

Parameters INPREV1 trough INPUTRE4 indicate the input speed for each gear.

Parameters OUTPREV1 through OUTPREV4 indicate the output speed for each gear.



When using a CAN servo system, if parameter NPULSES and parameters INPREV and OUTPREV of all the gears are set with a ·0· value, the CNC will assume the equivalent ones of the drive.

### JERKLIM (P80)

Square-sine (bell shape) ramp acceleration. This type of ramp is used to gain in smoothness. This spindle parameter becomes effective with RESET in machine parameters.

Value	Meaning
JERKLIM = 0	Linear acceleration ramp

A JERKLIM value other than zero activates the square-sine ramp.

It comes in degrees/ $s^3$ , in other words, a parameter value of 20 means a jerk of 20000 degrees/ $s^3$ .

This parameter only affects the spindle acceleration in open loop (M3, M4, M5).

The parameter value so maximum acceleration (resulting from OPLACETI) is reached in half the acceleration time up to MAXGEAR1 is calculated as follows:

JERKLIM = 6000 · MAXGEAR1 / OPLACETI <sup>2</sup>

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.

**5**.

MACHINE PARAMETERS
Spindle parameters



Default value: 0

8040 CNC

# 5.4.2 Machine parameters for auxiliary spindle

MAXSPEED (P0) Indicates the maximum speed of the auxiliary spindle.

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 analog voltage 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 voltage of 10V.

Default value: 0

MINANOUT	Minimum analog output
1	0.3 mV.
3277	1 V.
32767	10 V.

### SERVOFF (P3)

Indicates the analog offset value for the spindle drive.

### Possible values

It is given in D/A converter units and it admits integer values between 0 and  $\pm 32767$  which corresponds to an analog voltage of 10V.

Default value: 0 (not applied)

SERVOFF	Analog voltage
-32767	-10 V.
-3277	-1 V.
1	0.3 mV.
3277	1 V.
32767	10 V.



8040 CNC

MAXVOLT (P4)

Indicates the analog voltage 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)

**MACHINE PARAMETERS** 

### **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  $\cdot 1 \cdot$  (the address of the CNC is always  $\cdot 0 \cdot$ ). For example, with 3 Sercos axes and 1 Sercos spindle, the values of this

parameter must be 1, 2, 3 and 4.



From version V9.11 on, this parameter is called DRIBUSID and is valid for both Sercos and CAN servo systems. In previous versions, this parameter was called SERCOSID.



8040 CNC

# 5.5 Drive parameters

This option is available when the CNC uses digital servo drive system; i.e. the drives are connected to the CNC via 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.

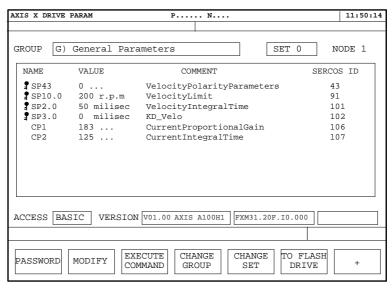
#### CAN interface

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 memkey card" (Card A) and the sofkeys 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 memkey card (Card A).

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].



Drive parameters



8040 CNC

• 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.



8040 CNC

### Save

It makes a copy of the parameters of the drive's RAM memory into the CNC's CARD A 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 CARD A 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  $[\P]$   $[\P]$  keys.

### **Options**

It shows a screen where one can select to display either all the parameters and variables or just the ones than can be modified.

Press the [Modify Option] softkey to change it and [ENTER] to validate it. This option is common to all the axes.



8040 CNC

**MACHINE PARAMETERS** 

# 5.5.1 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
```

FAGOR

8040 CNC

# 5.6 Serial line parameters

### **BAUDRATE (P0)** Indicates the communication speed, in baud, between the CNC and the peripherals.

It is given in baud and it is selected with the following code:

Value	Meaning	Value	Meaning
0	110 baud.	7	9.600 baud.
1	150 baud.	8	19.200 baud.
2	300 baud.	9	38.400 baud.
3	600 baud.	10	57.600 baud.
4	1.200 baud.	11	115.200 baud.
5	2.400 baud.	12	Reserved.
6	4.800 baud.		

Default value: 7 (9600 baud)

# **NBITSCHR (P1)** Indicates the number of data bits per transmitted character.

Value	Meaning
0	Uses the 7 least significant bits of an 8-bit character. It is used when transmitting ASCII characters (standard).
1	Uses all 8 bits of the transmitting character. Used when transmitting special characters whose codes are greater than 127.

Default value: 1

# **PARITY (P2)** Indicates the type of parity check used.

Value	Meaning
0	No parity.
1	Odd parity.
2	Even parity.

Default value: 0

### STOPBITS (P3) Indicates the number of stop bits at the end of each transmitted word.

Value	Meaning
0	1 STOP bit.
1	2 STOP bits.

Default value: 0

### **PROTOCOL (P4)** Indicates the type of communications protocol to be used.

Value	Meaning
0	Communications protocol for general device.
1	DNC communications protocol.
2	Communications protocol for Fagor floppy disk unit.

Default value: 1 (DNC)

### **PWONDNC (P5)** Indicates whether the DNC feature will be active on power-up or not.

Value	Meaning
NO	Not active on power-up.
YES	Active on power-up

Default value: NO



8040 CNC

# **5**.

# MACHINE PARAMETERS Serial line parameters

### **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



8040 CNC

**MACHINE PARAMETERS** Ethernet parameters

#### 5.7 **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 or CARDA nor read variables, tables,

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)** 

Not being used.

CNHDDIR1 (P6)

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).

**FAGOR** 

8040 CNC

EXTNAME2 (P8)

Not being used.

SERUNI2 (P21)

# 5

MACHINE PARAMETERS
Ethernet parameters

### **DNCEACT (P22)** DNC number to be used by Ethernet.

All three communication devices (serial lines 1 and 2, and Ethernet) cannot be active. In order to use Ethernet, one of the serial lines must be disabled.

Value	Meaning
0	There is no active DNC associated with Ethernet.
1	DNC 1 associated with Ethernet. The RS-422 serial line is disabled.
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.

# FAGOR

8040 CNC

# CNC connection in an Ethernet network. Connection using Windows<sup>®</sup> 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 lexplorer browser, it is called "Favorites". This name may vary depending on the Web browser being used.



8040 CNC

# Ethernet parameters

**MACHINE PARAMETERS** 

# CNC connection in an Ethernet network. Connection using Windows<sup>®</sup> 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

 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.
- 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.



8040 CNC

#### **PLC Parameters** 5.8

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

Parameters "USER0" through "USER23" do not mean anything to the CNC. USER0 (P2)

**USER23 (P25)** 

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.

### With CPU turbo:

Value	Meaning
0	1 ms every 8 samplings.
1	1 ms every 4 samplings.
2	1 ms every 2 samplings.
3	1 ms every sampling.
4	2 ms every sampling. With LOOPTIME = 3, 4, 5 or 6
5	3 ms every sampling. With LOOPTIME = 4, 5 or 6
6	4 ms every sampling. With LOOPTIME 5 or 6.
7	5 ms every sampling. With LOOPTIME = 6

Default value: 0



8040 CNC

**MACHINE PARAMETERS** 

### Without CPU turbo:

Value	Meaning
0	1 ms every 8 samplings.
1	1 ms every 4 samplings.
2	1 ms every 2 samplings.
3	1 ms every sampling. With LOOPTIME = 4, 5 or 6
4	2 ms every sampling. With LOOPTIME = 4, 5 or 6
5	3 ms every sampling. With LOOPTIME 5 or 6.
6	4 ms every sampling. With LOOPTIME = 6
7	4 ms every sampling. With LOOPTIME = 6

Default value: 0

The sampling period is determined by the g.m.p. LOOPTIME (P72). Hence, for a sampling period of 4 msec. and a CPUTIME=0, the system CPU dedicates 1 millisecond every 8 samplings (thus, 32 milliseconds) to the PLC.

The Status window of the PLC statistics screen indicates the time the system CPU dedicates to the PLC. Refer to the operation 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)

They are used in the data exchange via Sercos between the CNC and the drives.

SRR739 (P67)

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

 $P32 \Rightarrow R704$ 

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.



8040 CNC

### SWR800 (P68)

They are used in the data exchange via Sercos between the CNC and the drives.

### ...

SWR819 (P87)

They indicate what type of information is put in registers R800 through R819 and which drive will be assigned that value.

 $P68 \Rightarrow R800$   $P69 \Rightarrow R801$ 

P71 ⇒ R803

P72 ⇒ R804

 $\begin{array}{l} \text{P70} \Rightarrow \text{R802} \\ \text{etc.} \end{array}$ 

**5**.

PLC Parameters

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)

They are used to set the remote modules.

IOCANID2 (P91) IOCANID3 (P92) IOCANID4 (P93)

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



8040 CNC

NUICAN1 (P102) NUOCAN1 (P103) NUICAN2 (P104) NUOCAN2 (P105) NUICAN3 (P106) UOCAN3 (P107) NUICAN4 (P108) NUOCAN4 (P109) They are used to set the remote modules.

NUICAN\* indicates the number of the first input and NUOCAN\* that of the first output of the group.

The central unit reserves local inputs I1 through I64 and local outputs O1 through O32. When having an expansion board that includes I/Os, it also reserves local inputs I65 through I128 and local output O33 through O64.

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).

Possible NUICAN values without the expansion board that has I/O:

```
65, 73, 81, 89, 97, 105, 113, 121, 129, 137, 145, 153....
```

Possible NUICAN values with the expansion board that has I/O:

```
129, 137, 145, 153, 161, 169, 177, 185, 193, 201, 209....
```

Possible NUOCAN values without the expansion board that has I/O:

```
33, 41, 49, 57, 65, 73, 81, 89, 97, 105, 113, 121, 129....
```

Possible NUOCAN values without the expansion board that has I/O:

```
65, 73, 81, 89, 97, 105, 113, 121, 129, 137, 145, 153....
```

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	IOCANID1=2
ICAN1=48	ICAN1=24
OCAN1=32	OCAN1=16
NUICAN1=0	NUICAN1=0
NUOCAN1=0	NUOCAN1=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.

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	IOCANID1=2
ICAN1=48	ICAN1=24
OCAN1=32	OCAN1=16
NUICAN1=129	NUICAN1=0
NUOCAN1=65	NUOCAN1=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.

FAGOR 🚄

8040 CNC

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 IOCANID1=2 ICAN1=48 ICAN1=24 OCAN1=32 OCAN1=16 NUICAN1=129 NUICAN1=201 NUOCAN1=65 NUOCAN1=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

IANALOG1 (P110) IANALOG2 (P113) They are used to set the remote modules.

IANALOG3 (P116) IANALOG4 (P119)

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.

OANALOG2(P114) OANALOG3(P117)

OANALOG1 (P111) 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 OANALOG4 (P120) 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 0000 0011 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_	$_{-}1 = x x x x   x x x x x x x x 0/1 0/1 x 0/1 0/1$	
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
	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.

8040 CNC

**FAGOR** 

**NUIANA1 (P122) NUIANA2 (P124)**  They are used to set the remote modules.

**NUIANA3 (P126) NUIANA4 (P128)** 

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)

Not being used.

### NUILO1 (P131) NUOLO1 (P132)

These PLC machine parameters may be used to redefine the numbering of the inputs/outputs of the local expansion module without having to modify the PLC program.

Value	Meaning
NUILO1	Numbering of the first input of the expansion module with inputs/outputs (I/O).
NUOLO1	Numbering of the first output of the expansion module with inputs/outputs (I/O).

The CPU of the  $8040\,\text{CNC}$  may have digital 16l/80 and optionally a single local digital expansion of 40l/240.

The inputs/outputs that do not belong to the expansion module are numbered starting from I1 and O1 and cannot be set by parameters.

**IMPORTANT:** The numbering of both the first local input and the first local output of the expansion module must be a multiple of 8 plus 1 (1+8n).



If incoherent parameter settings are detected on power-up, an error message will be issued indicating it.

Inside the expansion module, the numbering of the rest of inputs/outputs will be sequential from the first one on.

The numbering for the inputs/outputs of the expansion module will be different depending on the values entered in parameters NUILO1 and NUOLO1.

To properly number the inputs/outputs of the expansion module follow the indications of the example.

### **Example:**

Let's assume an 8040 CNC with 16i/8O and a local expansion module with 40l/24O. How should all the inputs/outputs of the expansion module be numbered?

- ▶ The inputs/outputs that do not belong to the expansion module (16l/8O) cannot be set by parameters; therefore, the first input is always numbered as I1 and the first output as O1.
- ▶ The inputs/outputs of the expansion module with the PLC parameters set to zero:

Parameters of the expansion module
NUILO1 = 0
NUOLO1 = 0

will be:

I/O of the expa	nsion module	
l17 - l56	09 - 032	

The inputs of the expansion module are numbered sequentially after the last input that does not belong to the expansion module (I16+1 = I17). Follow the same procedure for the outputs.



8040 CNC

PLC Parameters

► The inputs/outputs of the expansion module with the PLC parameters set to a value other than zero and multiple of (1+8n) where "n" is a natural number:

## Parameters of the expansion module

NUILO1 = 65

NUOLO1 = 33

will be:

### I/O of the expansion module

l65 - l104 O33 - O56

The inputs of the expansion module are numbered sequentially after the value assigned to parameter NUILO1 (I65) chosen at will with the restriction (8n+1). Follow the same procedure for the outputs.

The total limit of inputs (local + remote) is 512.

The total limit of outputs (local + remote) is 512.



The values of the PLC machine parameters mentioned earlier should be multiple of 16 for best managing the inputs and outputs in time.

NUILO2 (P133) NUOLO2 (P134) NUILO3 (P135) NUOLO3 (P136) NUILO4 (P137) NUOLO4 (P138) Not being used.



8040 CNC

#### **Tables** 5.9

#### 5.9.1 Miscellaneous (M) function table

The number of M functions in this table is determined by the g.m.p. NMISCFUN (P29), being possible to define up to 255 M functions.

It must borne in mind that functions: M00, M01, M02, M03, M04, M05, M06, M8, M9, M19, M30, M41, M42, M43, M44 and M45, besides what is indicated in this table, have specific meanings when programming the CNC.

M FUNCTION TABLE	P N	11:50:14	
Miscellaneous Function	 Subroutine	Customizing Bits	
miscenaneous runction	Bubroutille	Customizing bits	
M????	\$0000	00000000	
M???? \$0000 00000000		00000000	
_	_		
CAP INS			
EDIT MODIFY FIND ERASE LOAD SAVE			

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.

**MACHINE PARAMETERS** 



8040 CNC

**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.

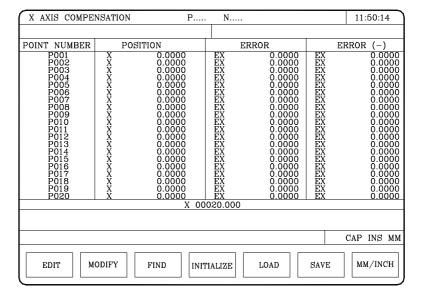


8040 CNC

# 5.9.2 Leadscrew error compensation table

The CNC provides a table for each one of the axes requiring leadscrew compensation. The CNC will provide a table for each one of the axes having leadscrew compensation. This type of compensation is selected by setting a.m.p. LSCRWCOM (P15).

The number of elements of the table must be set by a.m.p. NPOINTS (P16), being possible to define up to 255 points per axis. Different compensation values may be defined at each point for each moving direction.



Each parameter of the table represents a point of the profile to compensate. The following information is defined at each point:

• The position occupied by the point in the profile (position to compensate). This position is defined by its coordinate referred to machine reference zero.

### Possible values

Within ±99999.9999 mm or ±3937.00787 inches.

• The error of the leadscrew at that point, when moving in the positive direction.

### Possible values

Within ±99999.9999 mm or ±3937.00787 inches.

• The error of the leadscrew at that point, when moving in the negative direction.

### Possible values

Within ±99999.9999 mm or ±3937.00787 inches.

For each axis position, define the amount of error to be compensated in both directions. If the amount of error in the negative direction is zero in all points, it assumes that the amount of error defined for the positive direction is valid for both directions.

### Leadscrew error compensation on rotary axes

On rotary axes, although the display is limited between 0 and  $360^\circ$ , the internal count is accumulative. When using leadscrew error compensation, set positions  $0^\circ$  and  $360^\circ$ , 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.

**5**.

Tables

**MACHINE PARAMETERS** 



8040 CNC

8040 CNC

(SOFT M: V11.1x) (SOFT T: V12.1x)

### **Considerations and limitations**

When defining the profile points in the table, the following requirements must be met:

- The axis points must be in sequential order starting from the most negative (least positive) point to be compensated.
- For those points outside the compensation zone, the CNC will apply the compensation value corresponding to the table point closest to them.
- The amount of error of the machine reference point may have any value.
- The error difference between two consecutive points must not be greater than the distance between them (maximum slope= 100%).



Bidirectional compensation of the leadscrew error is available from versions V7.11 (mill) and V8.11 (lathe) on.

When updated from a version that does not have bidirectional compensation, it keeps the error values in the positive direction and it sets a zero error in the negative direction for all the points.

When changing to a version that does not have bidirectional compensation, it keeps the error values in the positive direction, but it loses the error values in the negative direction. Also, the amount of error for the machine reference point must be zero.

MACHINE PARAMETERS

# 5.9.3 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.

CROSSED COMP.TABLE	P N	11:50:14	
ERROR POINT	POSITION	ERROR	
P 001	X 0.0000	EV 0.0000	
P 002	X 0.0000	EV 0.0000	
P 003	X 0.0000	EV 0.0000	
P 004	X 0.0000	EV 0.0000	
P 005	X 0.0000	EV 0.0000	
P 006	X 0.0000	EV 0.0000	
P 007	X 0.0000	EV 0.0000	
P 008	X 0.0000	EV 0.0000	
P 009	X 0.0000	EV 0.0000	
P 010	X 0.0000	EV 0.0000	
P 011	X 0.0000	EV 0.0000	
P 012	X 0.0000	EV 0.0000	
P 013	X 0.0000	EV 0.0000	
l <b>_</b>			
		CAP INS	
EDIT MODIFY	FIND INITIALIZE LO	AD SAVE MM /	
		INCHES INCHES	

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.



8040 CNC



8040 CNC

# **CONCEPTS**



It is recommended to save the machine parameters as well as the PLC program and files into the memkey card" (CARDA) or in a peripheral or PC to avoid losing them.

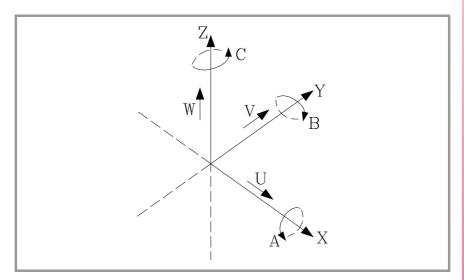
# 6.1 Axes and coordinate systems

Given that the objective 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 the 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.

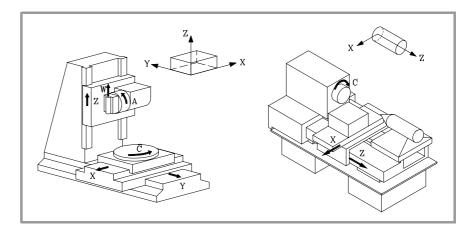
U, V, W auxiliary axes parallel to X, Y, Z respectively.

A, B, C Rotary axes on each axis X, Y, Z.



**CNC 8040** 

In the figure (below) 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 machine parameters of axes which appear in the Installation and Start-up Manual.

There is no limitation to the programming of the axes, and interpolations can be made simultaneously with up to 7 axes.

# Example of milling.

The machine has three regular linear axes: X, Y and Z, one normal 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 or	utput.
--	--------

Likewise, a.m.p DFORMAT (P1) and s.m.p. DOFORMAT (P1) must be properly set to indicate their display formats.



**CNC 8040** 

# Example of lathe.

The machine has two regular linear axes: X and Z, 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.
AXIS3 (P1) = 3	Z axis	associated with feedback X2 and output O2.
AXIS5 (P2) = 10	Spindle (S)	associated with feedback X3 and output O3.
AXIS4 (P3) = 13	Auxiliary spindle	associated with feedback X5 (1-6) and output O5.
AXIS5 (P4) = 0		
AXIS6 (P5) = 0		
AXIS7 (P6) = 0		
AXIS8 (P7) = 0		

The CNC activates a machine parameter table for each axis (X, Z), 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

s.m.p. SPDLTYPE (P0) must be set as follows:

Spindle	SPDLTYPE $(P0) = 0$	±10V spindle analog output.

Likewise, a.m.p DFORMAT (P0) and s.m.p. DOFORMAT (P0) must be properly set to indicate their display formats.

CONCEPTS

**FAGOR** 

**CNC 8040** 

# 6.1.1 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 axis AXISTYPE (P0) = 3
Rotary Hirth axis AXISTYPE (P0) = 4

By default, their position is always displayed between 0 and 360° (Rollover axis). If these limits are not to be set, modify a.m.p. ROLLOVER (P55).

ROLLOVER = YES Rotary axis display between 0 and 360°

ROLLOVER = NO No display limits.

Although the display is limited between 0 and  $360^{\circ}$ , 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 "5.3 Axis parameters" on page 105.

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 "6.5.7 Leadscrew error compensation" on page 212.

### **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 360°, the axis will turn more than once before positioning at the desired point.

Normal rotary axis		
AXISTYPE=2	_	
LIMIT+ = 8000	ROLLOVER=YES	It counts between 0° and 360°. G90 The sign indicates the turning direction. G91 The sign indicates the turning direction.
LIMIT- =-8000	ROLLOVER=NO	It counts between 7999.9999° and -7999.9999°. G90 and G91 as linear axis.
LIMIT+ = 0 LIMIT- =0	ROLLOVER=YES	It counts between 0° and 360°. G90 The sign indicates the turning direction. G91 The sign indicates the turning direction.
	ROLLOVER=NO	There are 2 loops, one between 0° and 360° and the other between 0° and -360° 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 10° and 350°. With G90 and G91 like when LIMIT+=8000. An error message is issued if the target position is beyond the limits.



**CNC 8040** 

CONCEPTS

# Positioning-only 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 360°, the axis will turn more than once before positioning at the desired point.

Positioning-only axis		
AXISTYPE=3		
LIMIT+ = 8000 LIMIT- =-8000	ROLLOVER=YES	It counts between 0° and 360°. G90 does not admit negative values. Always via the shortest path. G91 The sign indicates the turning direction.
	ROLLOVER=NO	It counts between 7999.9999° and -7999.9999°. G90 and G91 as linear axis.
LIMIT+ = 0 LIMIT- =0	ROLLOVER=YES	It counts between 0° and 360°. G90 does not admit negative values. Always via the shortest path. G91 The sign indicates the turning direction.
	ROLLOVER=NO	There are 2 loops, one between 0° and 360° and the other between 0° and -360° 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 10° and 350°. With G90 and G91 like when LIMIT+=8000. An error message is issued if the target position is beyond the limits.

# **Rotary Hirth axis**

It is a positioning-only axis which cannot take decimal coordinates. All positioning movements must be in whole degrees.



**CNC 8040** 



**CNC 8040** 

Rotary Hirth axis (whole degrees)		
AXISTYPE=3		
LIMIT+ = 8000 LIMIT- =-8000	ROLLOVER=YES	It counts between 0° and 360°. G90 does not admit negative values. G91 The sign indicates the turning direction.
	ROLLOVER=NO	It counts between 7999.9999° and -7999.9999°. G90 and G91 as linear axis.
LIMIT+ = 0 LIMIT- =0	ROLLOVER=YES	It counts between 0° and 360°.  G90 does not admit negative values. Always via the shortest path.  G91 The sign indicates the turning direction.
	ROLLOVER=NO	There are 2 loops, one between 0° and 360° and the other between 0° and -360° 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 10° and 350°. With G90 and G91 like when LIMIT+=8000. An error message is issued if the target position is beyond the limits.

CONCEPTS

# 6.1.2 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.

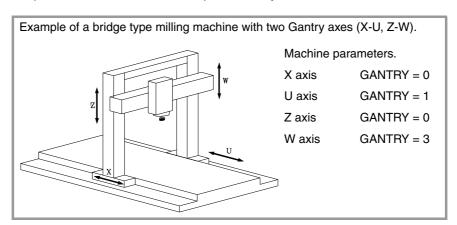
Only the movements of one of those axes must be programmed and it is called the main axis. The other axis is referred to as "slave axis".

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.





**CNC 8040** 

# 6.1.3 Slaved axes and synchronized axes

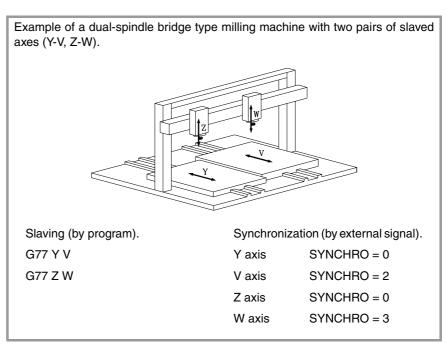
Coupled or synchronized axes are two or more axes which are normally independent, but, sometimes need to be moved at the same time and in synchronism (temporarily slaved, versus permanently as by machine parameter). For example on multi-spindle milling machines.

### 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.



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.

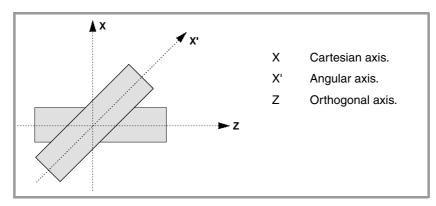




**CNC 8040** 

With the angular transformation of an incline axis, it is possible to make movements along an axis that is not perpendicular to anoher.

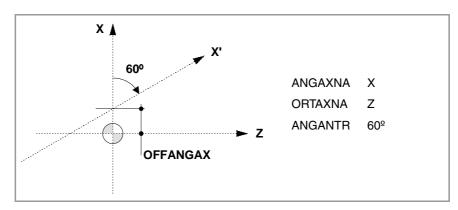
On certain machines, the axes are configured in a Cartesian way, they are not perpendicular to each other. A typical case is the X axis of a lathe that due to sturdiness reasons is not perpendicular to the Z axis.



Programming in the Cartesian system (Z-X) requires activating an angular transformation of an incline 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.



# 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 0°, there is no need to do an angular transformation.



CONCEPTS
Axes and coordinate systems



**CNC 8040** 

## **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 incline 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 making up the configuration of the incline axis.



**CNC 8040** 

CONCEPTS

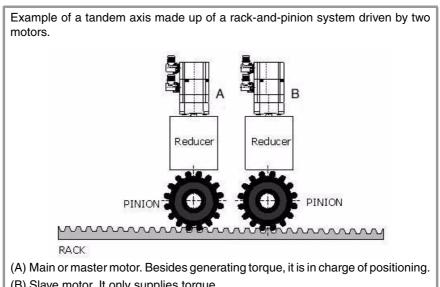
#### 6.1.5 **Tandem axes**

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.

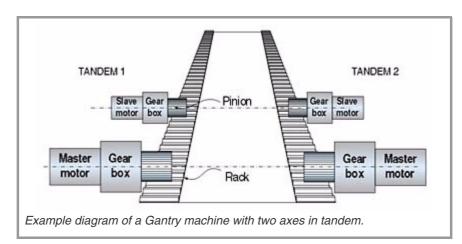
The following aspects may be pointed out in this type of configuration:

- . 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.



(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.



**FAGOF** 

**CNC 8040** 

Axes and coordinate systems

## 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.

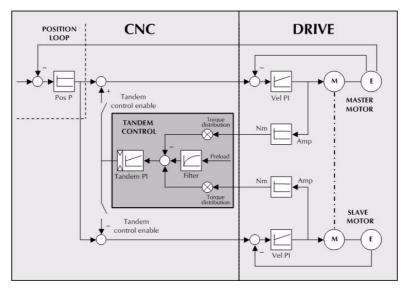




**CNC 8040** 

## **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.

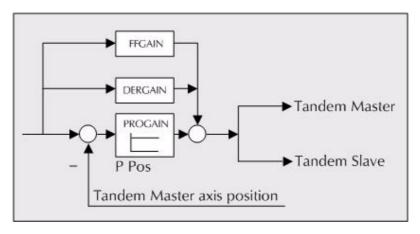


The block diagram has an area for the drive and another one for the CNC that comprises the position loop and the control of the tandem.

### 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.



CONCEPTS Axes and coordinate systems



**CNC 8040** 

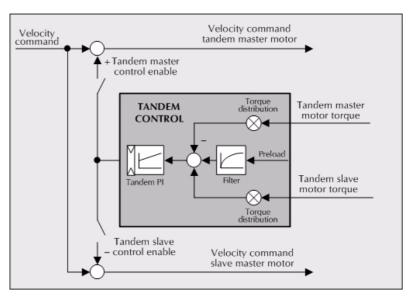
6.

CONCEPTS
Axes and coordinate systems

FAGOR

**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x) 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.

CONCEPTS

## 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.

**FAGO** 

**CNC 8040** 

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.

**6.** 

CONCEPTS
Axes and coordinate systems

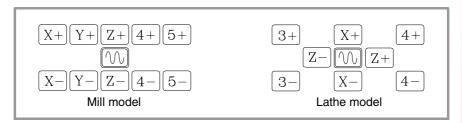


**CNC 8040** 

# **6.2** Jog

# 6.2.1 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.



The  $X,\,Y$  and Z axes always use their own nomenclature 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-]



CONCEPTS



**CNC 8040** 

## 6.2.2 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 the continuous or incremental jog positions.

The CNC assumes as "Path jog" the keys associated with the X axis.

## Feature setting

This feature must be managed from the PLC.

To activate or cancle the "Path jog" work mode, act upon the logic CNC input "MASTRHND" M5054.

M5054 = 0 "Path JOG" function off.

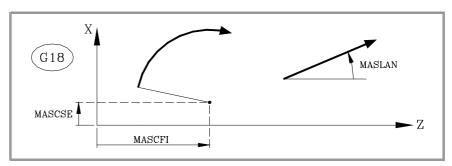
M5054 = 1 "Path JOG" function on.

To indicate the type of movement, use CNC logic input "HNLINARC" M5053.

M5053 = 0 Linear path.

M5053 = 1 Arc path.

For a linear path, indicate the path angle in the MASLAN variable (value in degrees between the linear path and the first axis of the plane). For an arc, indicate the arc center coordinates in the MASCFI, MASCSE variables (for the first and second axes of the main plane).



Variables MASLAN, MASCFI and MASCSE may be read and written from the CNC, DNC and PLC.

### Operation of the "path jog" feature

The "path jog" mode is only available with the X axis keys. When pressing one of the keys associated with the X axis, the CNC behaves as follows:

Switch position	Path jog	Type of movement	
Continuous jog	OFF	Only the axis and in the indicated direction	
	ON	Both axes in the indicated direction and along the indicated path	
Incremental jog	OFF	Only the axis, the selected distance and in the indicated direction	
	ON	Both axes, the selected distance and in the indicated direction, but along the indicated path	
Handwheel		It ignores the keys.	

The rest of the jog keys always work in the same way, whether "path jog" is on or off. The rest of the keys move only the axis and in the indicated direction.





**CNC 8040** 

## 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.



ONCEPTS



**CNC 8040** 

#### Movement with an electronic handwheel 6.3

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.



Movement with an electronic handwheel



**CNC 8040** 

CONCEPTS

## 6.3.1 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.



Movement with a



**CNC 8040** 

## 6.3.2 Path handwheel

With this feature, it is possible to jog two axes at the same time along a linear path (chamfer) or circular path (rounding) with a single handwheel.

The CNC assumes as the path handwheel the general handwheel or, when this one is missing, the one associated with the X axis (Mill) or Z axis (lathe).

### Feature setting

This feature must be managed from the PLC.

To activate or cancle the "Path jog" work mode, act upon the logic CNC input "MASTRHND" M5054.

M5054 = 0 "Path JOG" function off.

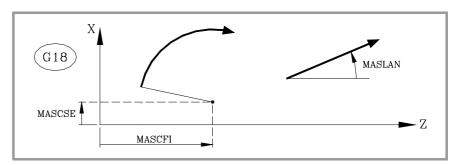
M5054 = 1 "Path JOG" function on.

To indicate the type of movement, use CNC logic input "HNLINARC" M5053.

M5053 = 0 Linear path.

M5053 = 1 Arc path.

For a linear path, indicate the path angle in the MASLAN variable (value in degrees between the linear path and the first axis of the plane). For an arc, indicate the arc center coordinates in the MASCFI, MASCSE variables (for the first and second axes of the main plane).



Variables MASLAN, MASCFI and MASCSE may be read and written from the CNC, DNC and PLC.

The next example uses the [O2] key to activate and deactivate the "path handwheel" mode and the [O3] key to indicate the type of movement.

DFU B29 R561 = CPL M5054

Activate or cancel the "path handwheel" mode.

DFU B31 R561 = CPL M5053

Select the type of movement; straight section or arc section.

#### Simultaneous handwheels

When selecting the path handwheel mode, the CNC behaves as follows:

- If there is a general handwheel, it will be the one working in path handwheel mode.
   The individual handwheels, if any, will remain associated with the corresponding axes.
- If there is no general handwheel, one of the individual handwheel starts working in path handwheel mode. The one associated with the X axis if Mill model or the one associated with the Z if lathe model.



**CNC 8040** 

### 6.3.3 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 from 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



FAGOR

**CNC 8040** 

L101

END



CONCEPTS

Movement with an electronic handwheel



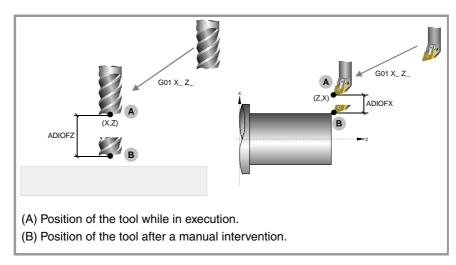
**CNC 8040** 

CONCEPTS

## 6.3.4 "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.



The intervention with an additive handwheel is only possible in execution mode, even while the program is interrupted. However, it is not allowed in the tool inspection mode.

The additive handwheel may be enabled for a coordinate transformation G46 (inclined axis) where the handwheel movements are applied to the machining operation even if they are not shown on the graphics screen.

The offset caused by the additive handwheel stays active after disabling the handwheel and it is reset to zero after a home search. The offset stays active or is reset after an M02 or M30 and after an emergency or a reset depending on the setting of g.m.p. ADIMPG (P176).

#### Considerations

- The movement with the additive handwheel on the master axis is also applied to the slave axis when using axes that are gantry, slaved or synchronized by PLC.
- When testing the software limits during block preparation, it checks the theoretical coordinate ignoring the additional movement of the additive handwheel.
- The mirror image by PLC is not applied to the additive handwheel movement.

FAGOR =

**CNC 8040** 

## 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.



**CNC 8040** 

# 6.4 feedback system

The various feedback inputs available at the CNC admit sinewave and squarewave differential signals from feedback systems. The following axis machine parameters indicate the type of feedback system and the resolution utilized for each axis.

· When using linear feedback devices.

PITCH (P7) Leadscrew pitch or that of the linear encoder being

used.

NPULSES (P8) = 0

DIFFBACK (P9) Indicates whether the feedback device uses

differential signals (double ended) or not.

SINMAGNI (P10) Feedback multiplying factor applied by the CNC.

FBACKAL (P11) Feedback alarm (only with differential signals).

· When using rotary encoders.

PITCH (P7) Number of degrees per encoder turn.

NPULSES (P8) Number of pulses (lines) per encoder turn.

DIFFBACK (P9) Indicates whether the feedback device uses

differential signals (double ended) or not.

SINMAGNI (P10) Feedback multiplying factor applied by the CNC.

FBACKAL (P11) Feedback alarm (only with differential signals).

Next, the feedback counting speed (frequency) limitation is described as well as how to set these machine parameters for the axes.



**CONCEPTS** eedback system



**CNC 8040** 

eedback system

## 6.4.1 Counting speed limitation

## Sinewave signals

The maximum counting speed (frequency) for sinewave feedback is 250 KHz.

The maximum feedrate for each axis will depend upon the selected resolution and the signal pitch (distance per pulse) in use while with 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  $\mu m$ . Therefore, with a counting resolution of 1  $\mu m$ , the maximum feedrate will be:

20  $\mu$ m/pulse x 250,000 pulses/sec = 300 m/min.

When using Fagor linear encoder, the maximum feedrate is limited by their own characteristics to 60 m/min.

## Example 2:

Using an indexer with a sinewave Fagor encoder of 3600 lines per turn. Therefore, for a feedback resolution of 1 µm, the maximum axis feedrate will be:

(360 degrees/turn / 3600 pulses/turn) x 250,000 pulses/s. = 25,000 degrees/s. = 1,500,000 degrees /min

Since Fagor sine-wave encoders admit a frequency of up to 200 KHz, the maximum feedrate will be:

(360 degrees/turn / 3600 pulses/turn) x 200,000 pulses/s. =

= 20,000 degrees/sec.= 1,200,000 degrees/min.

## Squarewave signals

The maximum frequency (speed) for squarewave differential feedback is 425 KHz. with a separation of 450 ns between A and B flanks. Which is equivalent to  $90^{\circ} \pm 20^{\circ}$ .

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 encoder, 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).



**CNC 8040** 

## 6.4.2 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  $\mu$ m or 100  $\mu$ m).

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 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, just enter "0". If a gear box is used on the axis, the whole assembly must be taken into account when setting 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 Leadscrew pitch	Nr of pulses	0
Sinusoidal signal encoder		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  $2\mu m$  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 which provides the following number of pulses (lines) per turn.

Nr of pulses = ballscrew pitch / (multiplying factor x Resolution)

Nr pulses =  $5000 \mu m / (4 \times 2 \mu m) = 625 \text{ pulses/turn}$ 

## Therefore:

INCHES = 0 PITCH=5.0000 NPULSES = 625 SINMAGNI=0

Although the CNC accepts a maximum squarewave frequency of 400 KHz, when using Fagor squarewave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

Max. feedrate = (200.000 pulses/sec. / 625 pulses/turn) x 0.2 inch/turn

Max. feedrate = 1600 mm/s = 96 m/min.



**CONCEPTS** feedback system



**CNC 8040** 



**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x)

#### Example 2:

## Resolution in "mm" with sinusoidal signal encoder

We would like to obtain a 2µm resolution by using a 250-line sinewave 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)

 $SINMAGNI = 5000 \mu m / (250 x 2 \mu m) = 10$ 

Therefore:

INCHES = 0 PITCH=5.0000 NPULSES = 250 SINMAGNI=10

Although the CNC accepts a maximum squarewave frequency of 250 KHz, when using Fagor sine-wave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

Max. feedrate = (200.000 pulses/sec. / 250 pulses/turn) x 0.2 inch/turn

Max. feedrate = 4.000 mm/s = 240 m/min.

### Example 3:

### Resolution in "mm" with squarewave linear encoder

Since the CNC applies a x4 multiplying factor to squarewave signals, we must select a linear encoder whose grading pitch is 4 times the desired resolution.

FAGOR linear encoders use a grading pitch of either 20  $\mu$ m or 100  $\mu$ m. Therefore, the resolution that can be obtained with them are: 5  $\mu$ m (20/4) or 25  $\mu$ m (100/4).

Therefore:

INCHES = 0 PITCH=0.0200 NPULSES = 0 SINMAGNI=0 PITCH=0.1000

The CNC's maximum squarewave feedback input frequency is 400 KHz which means that the maximum feedrate obtainable with a 20 µm pitch linear encoder is:

Max. Feed =  $20 \mu m/pulse \times 400,000 pulses/sec.$ 

Max. feedrate = 8000 mm/s = 480 m/min.

When using Fagor linear encoder, 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 sinewave linear encoder with a 20  $\mu m$  pitch and we would like to obtain 1  $\mu m$  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 =  $20 \mu m / 1 \mu m = 20$ 

Therefore:

INCHES = 0 PITCH=0.0200 NPULSES = 0 SINMAGNI=20

The CNC's maximum sinewave feedback input frequency is 250 KHz which means that the maximum feedrate for this axis will be:

Max. Feed =  $20 \mu m/pulse \times 250,000 pulses/sec.$ 

Max. feedrate = 5.000 mm/s = 300 m/min.

When using Fagor linear encoder, 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 which provides the following number of pulses (lines) per turn.

Nr of pulses = ballscrew pitch / (multiplying factor x Resolution)

Nr pulses =  $0.25 / (4 \times 0.0001) = 625 \text{ pulses/turn}$ 

#### Therefore:

INCHES = 1 PITCH=0.25000 NPULSES = 625 SINMAGNI=0

Although the CNC accepts a maximum squarewave frequency of 400 KHz, when using Fagor squarewave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

Max. feedrate = (200.000 pulses/sec. / 625 pulses/turn) x 0.255 inch/turn

Max. feedrate = 80 inch/sec. = 4800 inch/min.

### Example 6:

#### Resolution in "inches" with sinewave encoder

We would like to obtain a0.0001 inch resolution by using a 250-line sinewave encoder mounted on a leadscrew with a 5 turns/inch pitch.

We must calculate the multiplying factor "SINMAGNI" to be applied by the CNC to the pulses provided by the encoder in order to obtain the desired resolution.

SINMAGNI = ballscrew pitch / (Nr pulses x Resolution)

SINMAGNI = 0.2 inch/turn / (250 x 0.0001) = 8

### Therefore:

INCHES = 1 PITCH=0.20000 NPULSES = 250 SINMAGNI=8

Although the CNC accepts a maximum squarewave frequency of 250 KHz, when using Fagor sine-wave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

Max. feedrate = (200.000 pulses/sec. / 250 pulses/turn) x 0.2 inch/turn

Max. feedrate = 160 inch/sec. = 9.600 inch/min.

#### Example 7:

### Resolution in "degrees" with squarewave encoder

We would like to obtain a  $0.0005^{\circ}$  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 which provides the following number of pulses (lines) per turn.

Nr of pulses =  $\frac{9}{\text{turn}}$  / (multiplying factor x gear ratio x Resolution)

Nr of pulses =  $360 / (4 \times 10 \times 0.0005) = 18,000 \text{ pulses/turn}$ 

#### Therefore:

INCHES = 0 PITCH=36.0000 NPULSES = 18000 SINMAGNI=0



CONCEPTS feedback system



**CNC 8040** 

Although the CNC accepts a maximum squarewave frequency of 400 KHz, when using Fagor squarewave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

Max. feedrate = (200,000 pulses/sec) / (18,000 pulses/turn)

Max. feedrate =11.111 turns/sec = 666.666 rpm

## Example 8:

## Resolution in "degrees" with sinewave encoder

We would like to obtain a 0.001° resolution by using a 3600 pulse/rev sinusoidal encoder.

We must calculate the multiplying factor "SINMAGNI" to be applied by the CNC to the pulses provided by the encoder in order to obtain the desired resolution.

SINMAGNI = degrees per turn / (Nr. of pulses x Resolution)

SINMAGNI = 360 / (3600 x 0.001) = 100

Therefore:

INCHES = 0 PITCH=360.0000 NPULSES = 3600 SINMAGNI=100

Although the CNC accepts a maximum squarewave frequency of 250 KHz, when using Fagor sine-wave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

Max. feedrate = (200,000 pulses/sec) / (3,600 pulses/turn)

Max. feedrate =55.5556 turns/sec = 3333.33 rpm



**CNC 8040** 

# 6.5 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).



**CONCEPTS**Axis adjustment



**CNC 8040** 

Axis adjustment

## 6.5.1 Drive setting

## Offset (drift) adjustment

This adjustment is made on one axis at a time:

- Select the JOG mode at the CNC and press the softkey sequence: [Display]
   [Following error]. The CNC shows the current following Error (axis lag) of the axes.
- Adjust the offset by turning the offset potentiometer at the drive (NOT AT THE CNC) until a "0" following error is obtained.

## Maximum feedrate adjustment

The drives should be adjusted so they provide maximum axis feedrate when receiving an analog voltage (velocity command) of 9.5 V.

Set each a.m.p. MAXVOLT (P37) = 9500 so the CNC outputs a maximum analog voltage of 9.5 V.

The maximum axis feedrate, a.m.p. MAXFEED (P42), depends on the motor rpm as well as on the gear reduction 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 which 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 voltage provided by the CNC to the drive and adjust the feed potentiometer at the drive (NOT AT THE CNC) until reaching  $9.5~\rm V$ .

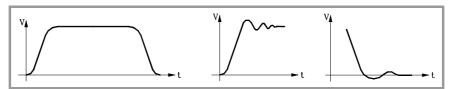


**CNC 8040** 

## 6.5.2 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 brake down:



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 analog output corresponding to a feedrate resulting in  $1^{\varrho}$  of following error.

It is defined with a.m.p. PROGAIN (P23).

## Feed-forward gain

It sets the percentage of analog output dependent of 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 analog output applied depending on the fluctuations of following error.

The "AC-forward gain" sets the percentage of analog output 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.



**CONCEPTS**Axis adjustment



**CNC 8040** 

Axis adjustment

# 6.5.3 Proportional gain setting

In a "pure" proportional positional loop, the analog output 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.

Analog output = 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 analog output 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 15m/min, but we would like to limit its maximum programmable machining feedrate (F) to 3 m/min with a gain of 1 mm lag 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 an analog voltage of 9.5 V.

a.m.p. MAXFEED (P42) must be set to 3,000 (3 m/min).

Analog output 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, the great majority of machines show an excellent behavior with a unitary gain (gain of 1, as shown in the previous examples).



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.



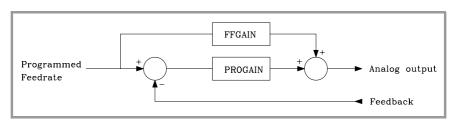
**CNC 8040** 

## 6.5.4 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 set the percentage of analog output 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.



For example, if a.m.p. FFGAIN (P25) has been set to "80", the axis analog voltage 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 voltage at the drive.
- 3. Set parameter MAXVOLT (P37) to a value 10 times the measured value. For example, If the measured 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%



**CONCEPTS**Axis adjustment



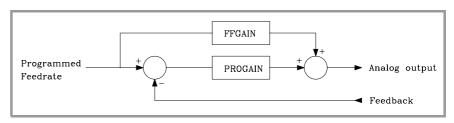
**CNC 8040** 

# 6.5.5 Derivative (AC-forward) gain setting

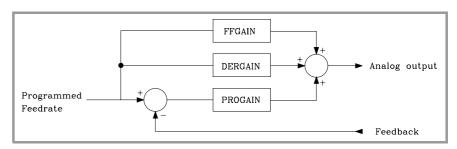
With the derivative gain, it is possible to reduce the following error during the acc./ dec. stages.

Its value is given by a.m.p. DERGAIN (P24).

When this additional analog voltage 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.



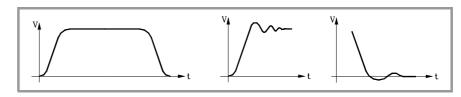
Best results are usually obtained when using it as AC-forward Gain, "ACFGAIN" (P46) = YES together with feed-forward gain.

This gain is only to be used when operating with acceleration / deceleration.

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 is no oscillations on following error, In other words, that it is not unstable.
- Check, with an oscilloscope, the tacho voltage or the analog voltage at the drive (velocity command), verify that it is stable (left graph) and that there are neither instabilities when starting up (center graph) nor when braking down (right graph).



6.

**CONCEPTS**Axis adjustment

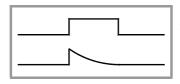


**CNC 8040** 

## 6.5.6 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 analog pulse may also be needed to recover the possible backlash when reversing the axis movement. The additional velocity command (analog voltage) 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 analog voltage and a.m.p. BACKTIME (P30) indicates the duration of this additional analog pulse and general machine parameter ACTBAKAN (P145) indicates the type of backlash peak applied.



**CONCEPTS**Axis adjustment



**CNC 8040** 

## 6.5.7 Leadscrew error compensation

The CNC provides a table for each one of the axes requiring leadscrew compensation. It is possible to define different compensation values for each moving direction. This type of compensation is activated by setting a.m.p. LSCRWCOM (P15)=ON for the desired axis.

The CNC enables one leadscrew error compensation table for each axis. The number of elements of the table is determined by the a.m.p. NPOINTS (P16), being possible to define up to 255 points per axis.

X AXIS COMPE	NSATION	P	N			11:50:14
POINT NUMBER	POSITION			RROR		RROR (-)
P001 P002 P003 P004 P006 P006 P007 P019 P011 P012 P013 P015 P016 P016 P016 P016 P018 P019	X	0000 0000 0000 0000 0000 0000 0000 0000 0000	EX EX EX EX EX EX EX EX EX EX EX EX EX E	0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000	EX E	0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000 0.0000
						CAP INS MM
EDIT	ODIFY FINE	INIT	TALIZE	LOAD	SAVE	MM/INCH

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  $360^{\circ}$ , the internal count is accumulative. When using leadscrew error compensation, set positions  $0^{\circ}$  and  $360^{\circ}$ , 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.



**CONCEPTS**Axis adjustment



**CNC 8040** 

#### **Considerations and limitations**

When defining the profile points in the table, the following requirements must be met:

- The axis points must be in sequential order starting from the most negative (least positive) point to be compensated.
- For those points outside the compensation zone, the CNC will apply the compensation value corresponding to the table point closest to them.
- The amount of error of the machine reference point may have any value.
- The error difference between two consecutive points must not be greater than the distance between them (maximum slope= 100%).



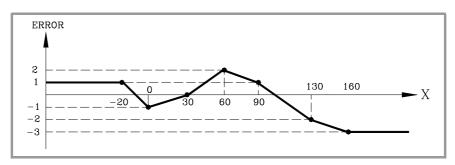
Bidirectional compensation of the leadscrew error is available from versions V7.11 (mill) and V8.11 (lathe) on.

When updated from a version that does not have bidirectional compensation, it keeps the error values in the positive direction and it sets a zero error in the negative direction for all the points.

When changing to a version that does not have bidirectional compensation, it keeps the error values in the positive direction, but it loses the error values in the negative direction. Also, the amount of error for the machine reference point must be zero.

## Setting example:

The X axis ballscrew must be compensated in the positive direction for between X-20 and X160 according to the leadscrew error graph below:



Set a.m.p. LSCRWCOM (P15) = ON and NPOINTS (P16) = 7

Considering that the Machine Reference Point (physical location of the marker pulse) is located 30 mm from HOME (machine reference zero), at X30. The leadscrew error compensation parameters must be set as follows:

Point	Pos	ition	Posi	Positive error		ative error
P001	Х	-20,000	EX	0,001	EX	0
P002	Х	0,000	EX	-0,001	EX	0
P003	Х	30,000	EX	0,000	EX	0
P004	Х	60,000	EX	0,002	EX	0
P005	Х	90,000	EX	0,001	EX	0
P006	Х	130,000	EX	-0,002	EX	0
P007	Χ	160,000	EX	-0,003	EX	0



**CONCEPTS**Axis adjustment



**CNC 8040** 

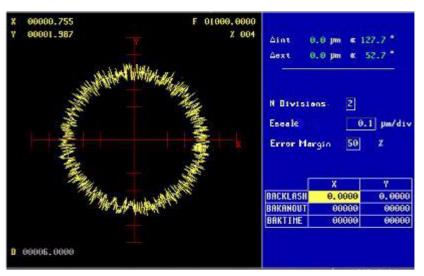
# 6.5.8 Circle geometry test

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 used 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 assigns the new values to the relevant machine parameters; therefore, it is recommended to jot the initial values down.

One 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.

6.

CONCEPTS
Axis adjustment



**CNC 8040** 

#### **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.

 $\Delta$  internal  $\,$   $\,$  Maximum negative value of the error over the theoretical value, in

microns or ten-thousandths of an inch and its angular position.

 $\Delta$  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.



**CONCEPTS**Axis adjustment



CNC 8040

Reference systems

# 6.6 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 system of coordinates of the machine.

#### Part zero

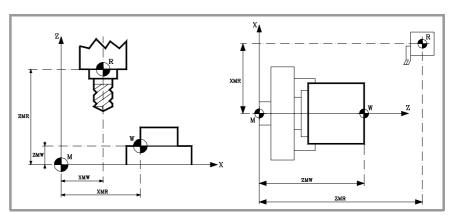
Part origin point. This is the point of origin which is set for programming the measurements of the part. It can be freely selected by the programmer, and its zero machine reference can be set by the zero offset.

### 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 coded marker pulse, lo), this point is only used when leadscrew error compensation must be applied onto the axis.

When the feedback is a regular incremental system (without coded marker pulse, 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



**CNC 8040** 

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 zero offset will be maintained and the CNC will display the position value with respect to the zero offset (or part zero) active before the home search.

- On axes with no distance-coded feedback system.
  - ☐ The CNC will move all selected axes which have a home switch and in the direction indicated by a.m.p. REFDIREC (P33) for each axis.

This movement will be carried out at the feedrate established by a.m.p. REFEED1 (P34) for each axis 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 is found.

□ If machine parameter I0TYPE (P52) =3, the home search procedure is the following:

The CNC will move all selected axes which have a home switch and in the direction indicated by a.m.p. REFDIREC (P33) for each axis.

This movement will be carried out at the feedrate established by a.m.p. REFEED1 (P34) for each axis 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) for each axis until the marker pulse is found.

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. This movement is carried out at the feedrate indicated by a.m.p. REFEED2 (P35).



CONCEPTS
Reference systems



**CNC 8040** 

# FAGOR =

**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x)

#### **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 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 slaved axis is found and then, it will look for the marker pulse from the main axis.

☐ If machine parameter IOTYPE=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 contact 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 slaved 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 slaved 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 slaved 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.

## **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 lo 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:

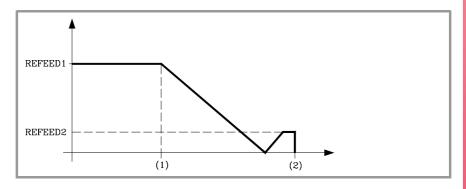
- After allocating this new value, press SHIFT + RESET or turn the CNC off and back on in order for the CNC to assume this new value.
- The axis must be homed again in order for it to assume its right reference values.

#### Considerations

If at the time when the home search is requested, the axis is sitting on the home switch, the axis will back up (in the direction opposite to the one indicated by "REFDIREC (P33)") until it is off the switch and then, it will go on to searching 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.





CONCEPTS
Reference systems



**CNC 8040** 



**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x) 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 lo every 50mm (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 (lo) used by the CNC.

## Offset adjustment

The offset of the linear encoder must be adjusted on one axis at a time, preferably, following this procedure:

**1.** Set the following a.m.p:

REFDIREC (P33) Homing direction.

"REFEED2" (P35) Homing feedrate.

2. Verify that the value allocated to a.m.p. REFPULSE (P32) (type of marker pulse of the feedback system) is correct.

To do this, set a.m.p. DECINPUT (P31) = NO and a.m.p. I0TYPE (P52) = 0 Then perform a home search.

If assumed immediately, change a.m.p. REFPULSE (P32) and check again.

- 3. Set a.m.p. I0TYPE (P52) = 1 and ABSOFF (P53) = 0.
- **4.** Once in JOG mode and after positioning the axis in the proper area, home the axis. The new position value displayed by the CNC is the distance from the current point to the origin of the linear encoder.
- 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. IOTYPE (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.



CONCEPTS
Reference systems



**CNC 8040** 

## 6.6.4 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.

6.

CONCEPTS
Reference systems



**CNC 8040** 

The FAGOR 8055 CNC provides a number of machine parameters to help improve the repetitiveness when positioning the axes in rapid (G00) by always approaching the end point in the same direction.

#### "UNIDIR" (P39)

Indicates the direction of unidirectional approach.

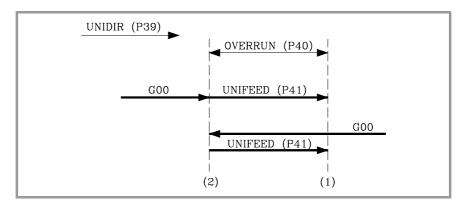
#### **OVERRUN**

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).



CONCEPTS
Unidirectional approach



**CNC 8040** 

## 6.8 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 "10.1 Auxiliary M, S, T functions" on page 318. See "11.7 General logic outputs" on page 354. See "5.9 Tables" on page 167.

#### 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).

 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

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.



**CNC 8040** 

CONCEPTS

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).

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 q.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.



FAGOR 🕣

**CNC 8040** 

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.

**6.** 

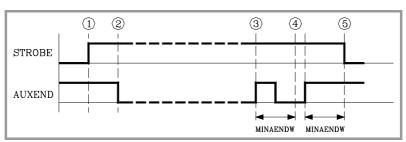
CONCEPTS
Auxiliary M, S, T function transfer



**CNC 8040** 

#### Transferring M, S, T using the AUXEND signal 6.8.1

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.

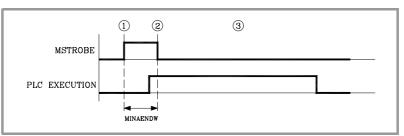


**FAGO** 

**CNC 8040** 

# 6.8.2 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".



**CNC 8040** 

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.

6.

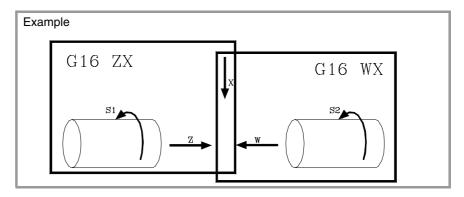
Main and second spindle

CONCEPTS



**CNC 8040** 

Use function G16 to select the work plane. See programming manual.



## 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 "5.2 General machine parameters" on page 66.

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



**CNC 8040** 

The setting of s.m.p. SPDLTYPE (P0) allows the following possibilities:

SPDLTYPE = 0 Analog spindle output..

SPDLTYPE = 1 2-digit BCD coded spindle output (S).

SPDLTYPE = 2 8-digit BCD coded spindle output (S).

When using BCD coded output, the spindle will operate in open loop and it will be controlled by means of functions M3, M4 and M5.

When using analog 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 output 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.



CONCEPTS
Main and second spindle



**CNC 8040** 

To do that, set s.m.p. SPDLTYPE (P0) with the right value.

SPDLTYPE = 1 2-digit BCD coded spindle output (S).

SPDLTYPE = 2 8-digit BCD coded spindle 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:

Ţ.					
Programmed S	S BCD	Programmed S	S BCD	Programmed S	S BCD
0	00	50-55	54	800-899	78
1	20	56-62	55	900-999	79
2	26	63-70	56	1000-1119	80
3	29	71-79	57	1120-1249	81
4	32	80-89	58	1250-1399	82
5	34	90-99	59	1400-1599	83
6	35	100-111	60	1600-1799	84
7	36	112-124	61	1800-1999	85
8	38	125-139	62	2000-2239	86
9	39	140-159	63	2240-2499	87
10-11	40	160-179	64	2500-2799	88
12	41	180-199	65	2800-3149	89
13	42	200-223	66	3150-3549	90
14-15	43	224-249	67	3550-3999	91
16-17	44	250-279	68	4000-4499	92
18-19	45	280-314	69	4500-4999	93
20-22	46	315-354	70	5000-5599	94
23-24	47	355-399	71	5600-6299	95
25-27	48	400-449	72	6300-7099	96
28-31	49	450-499	73	7100-7999	97
32-35	50	500-559	74	8000-8999	98
36-39	51	560-629	75	9000-9999	99
40-44	52	630-709	76		
45-49	53	710-799	77		

6.

CONCEPTS

Main and second spindle



**CNC 8040** 

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 analog output voltage 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 analog voltage 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 analog voltage for the spindle.
- From this instant on, the CNC outputs the spindle analog voltage 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 analog voltage 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.



Main and second spindle

CONCEPTS



**CNC 8040** 

## 6.9.3 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.

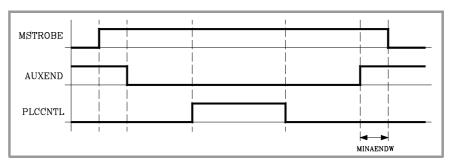
In turn, the PLC must "tell" the CNC the speed gear being selected. This will be indicated by means of the logic inputs for the spindle: "GEAR1 (M5458)", "GEAR2 (M5459)", "GEAR3 (M5460)" and "GEAR4 (M5461)".

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.

CONCEPTS

Main and second spindle



**CNC 8040** 

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 voltage 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 analog voltage for the spindle.
- 2. From this instant on, the CNC outputs the spindle analog voltage 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 analog voltage 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.



CONCEPTS
Main and second spindle



**CNC 8040** 

## 6.9.4 Spindle in closed loop

In order for the spindle to operate in closed loop by means of "spindle orientation (M19)", the following conditions must be met:

- The velocity command for the spindle must be analog (±10V). s.m.p. SPDLTYPE (P0) = 0.
- An encoder must be mounted onto the spindle. s.m.p. NPULSES (P13) must indicate the number of square pulses supplied by the spindle encoder.

Also, when switching from open to closed loop, either an "M19" or an "M19 S $\pm$ 5.5" must be executed.

The  $S\pm5.5$  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, it performs a home-switch search at the turning speed set by s.m.p. REFEED1 (P34).
  - It then searches for actual marker pulse (Io) of the spindle encoder at the turning speed set by s.m.p. REFEED2 (P35).
  - And, finally, it positions the spindle at the programmed S±5.5 point.
- 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).

And, then, it positions the spindle at the programmed S±5.5 point.

## Calculating spindle resolution

The CNC assumes that one encoder revolution represents 360°. 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  $0.001^{\circ}$ , a 90,000 line encoder is required and a 180,000 line encoder to obtain a resolution of  $0.0005^{\circ}$ .

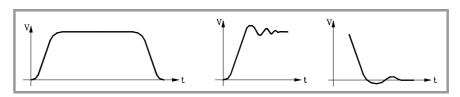
s.m.p. NPULSES (P13) must indicate the number of square pulses supplied by the spindle encoder.

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 brake down:





**CNC 8040** 

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 analog output 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 analog output dependent of 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 analog output applied depending on the fluctuations of following error.

The "AC-forward gain" sets the percentage of analog output 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 positional loop, the analog output of 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.

Analog output = 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 analog output corresponding to a feedrate resulting in  $1^{\circ}$  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 (rapid traverse G00) is 500 rpm and we would like to obtain 1º at a speed of 1000 º/min. (2.778 rpm)

Command from the drive: 9.5V for 500 rpm

Analog output corresponding to S = 1000 °/min. (2.778 rpm)

Analog velocity command = (S x 9,5 V) / "MAXGEAR1"

Analog voltage = (9.5 V / 500 rev/min) \*2.778 rev/min =52.778 mV.

Therefore "PROGAIN" = 53.



CONCEPTS
Main and second spindle



**CNC 8040** 

#### Bear in mind

When setting the proportional gain that:

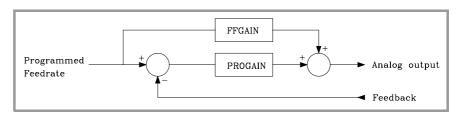
- The maximum amount of following error allowed by the CNC for the spindle is the value indicated by s.m.p. MAXFLWE1 (P21). When exceeded, the CNC issues the corresponding following error message.
- The amount of following error decreases as the gain increases, but it tends to make the system unstable.

## 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 set the percentage of analog output 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.



For example, if s.m.p. FFGAIN (P25) has been set to "80", the spindle analog voltage 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 voltage at the drive.
- 3. Set parameter MAXVOLT (P37) to a value 10 times the measured value. For example, If the measured voltage was 0,945 V, then set this parameter to 9.45 V, in other words: P37=9450.

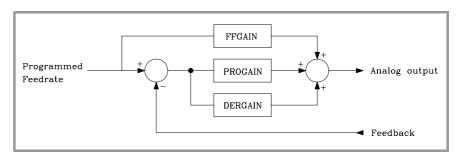
Next, set s.m.p. FFGAIN (P25) to the desired value.

## 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 analog voltage is due to fluctuations of following error, "ACFGAIN" (P46) = NO, it is called "derivative gain".



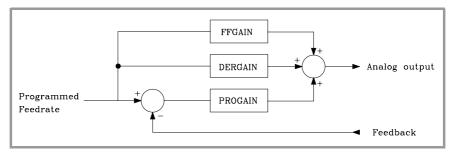


**CNC 8040** 

6. Main and second spindle

CONCEPTS

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.



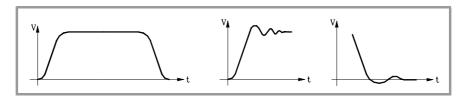
Best results are usually obtained when using it as AC-forward Gain, "ACFGAIN" (P42) = YES together with feed-forward gain.

This gain is only to be used when operating with acceleration / deceleration.

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 is no oscillations on following error, In other words, that it is not unstable.
- Check, with an oscilloscope, the tacho voltage or the analog voltage at the drive (velocity command), verify that it is stable (left graph) and that there are neither instabilities when starting up (center graph) nor when braking down (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 lo 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 the 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.



**CNC 8040** 

 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:

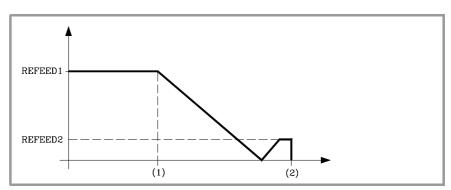
If the point whose known position is located at 12° mm from Machine Reference Zero and the CNC reads -123.5° as the coordinate value for this point, the coordinate of the Machine Reference Point with respect to Machine Reference Zero will be:

- 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 searching 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.



**CNC 8040** 

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 analog voltage 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 analog voltage output for the auxiliary spindle.
- 2. From then on, the CNC outputs the analog 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 analog voltage output.
- 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.



CONCEPTS
Auxiliary spindle controlled by PLC



**CNC 8040** 

## 6.11 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 axes 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 8040** 

### **CNC Treatment of emergency signals**

The emergency inputs of the CNC are:

/EMERGEN (M5000)

Physical input coming from the PLC.

/EMERGENCY STOP

Physical input coming from the outside. Pin 10 of connector X2

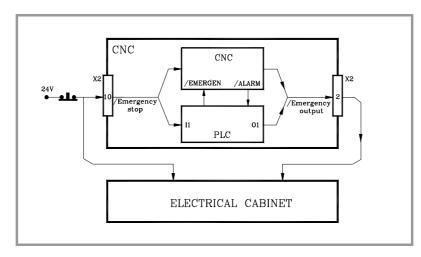
The emergency outputs of the CNC are:

/ALARM (M5507)

Physical output to the PLC.

/EMERGENCY OUTPUT

Physical output to the outside. Pin 2 of connector X2



There are to ways to cause an emergency at the CNC, by activating the physical input /EMERGENCY STOP or the general logic input "/EMERGEN" from the PLC.

Whenever any of these signals is activated, the CNC stops the axes feed and the spindle rotation and it displays the corresponding error message.

By the same token, when the CNC detects an internal malfunction or at an external device, it stops the axes feed and the spindle rotation displaying at the same time the corresponding error message.

In both cases, the CNC will activate the /EMERGENCY OUTPUT and /ALARM signals to indicate to the PLC and to the outside world that an emergency has occurred at the CNC.

Once the cause of the emergency has disappeared, the CNC will deactivate these signals to indicate to the PLC and to the outside world that everything is back to normal.



Treatment of emergency signals

CONCEPTS



**CNC 8040** 

## **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 born 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.



**CNC 8040** 

## 6.12 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.

#### 6.12.1 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 various 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.



**CNC 8040** 

## Cyclic channel. Read-only variables for the CNC-PLC

The plc.m.p. SRR700 (P28) through SRR739 (P67) indicate which drive and what type of information will be placed in CNC registers R700 through R739.

P28=>R700 P29=>R701 P30=>R702 P31=>R703 etc.

These parameters are set in 1.5 format. The units digit identifies the drive (node) supplying the data and the decimals indicate the identifier number (see table below).

For example, "P32=1.00040" indicates that PLC register R704 contains the "VelocityFeedback" supplied by the drive located in bus node 1.



To identify the units of the variables, see the drive manual.

Read-only registers R700 through R739 are updated at the beginning of the PLC scan, unless the MRD instruction is used.

The type of information available and its associated identifiers are:

Type of information	Identifier
Class2Diagnostics (Warnings)	00012
Class3Diagnostics (OperationStatus)	00013
VelocityFeedback	00040
PositionFeedbackValue1	00051
TorqueFeedback	00084
CurrentFeedback	33079
FagorDiagnostics	33172
AnalogInputValue	33673
AuxiliaryAnalogInputValue	33674
DigitalInputsValues	33675
PowerFeedback	34468
PowerFeedbackPercentage	34469

The bits of identifier 33172 "FagorDiagnostics" contain the following information:

bits	Meaning	ld 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





**CNC 8040** 

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).

6.

CONCEPTS
Digital servo (CAN or Sercos)



**CNC 8040** 

# FAGOR 🚄

**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x)

## 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, SETGY, SETGZ 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.

## 6.12.2 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.

**6.** 

CONCEPTS
Digital servo (CAN or Sercos)



**CNC 8040** 

## 6.13 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	<ul><li>·0· If it assumes the feedback of the main axis.</li><li>·1· if it has its own feedback.</li></ul>

- Act upon marks SWITCH1 through 4 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.



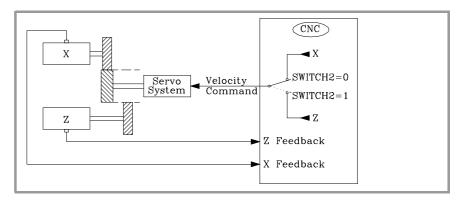
In versions older than V7.01 and V8.01, the sign of the velocity command of the associated axis corresponded to that of the main axis.

When updating the software version, the secondary axis may run away. Before installing the new version, make sure that parameter LOOPCHG of the associated axis has the same value as that of the main axis.

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 analog voltage is always output through the X axis connector (main).

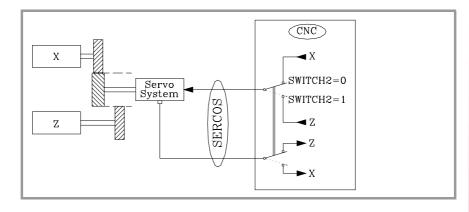
The mark for the secondary axis is SWITCH2 (M5155)

With SWITCH2=0 , analog voltage of the X axis and with SWITCH2=1 that of the Z axis.



**CNC 8040** 

## 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 analog voltage and feedback data transmitted via SERCOS correspond to

With SWITCH2=0, analog voltage and feedback data for the X axis.

With SWITCH2=1, analog voltage and feedback data for the Z axis.

## Cylindrical grinder (X and Z axes).

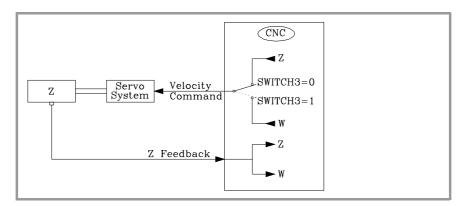
To make the reciprocating movement (back-and-forth table swing - Z axis) independent from the movement of the other axis (X), that movement should be controlled through the PLC execution channel.

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.

- Ζ 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.

6.

Axes (2) controlled by a single drive

CONCEPTS

**FAGOR** 

**CNC 8040** 

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 positioning 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)

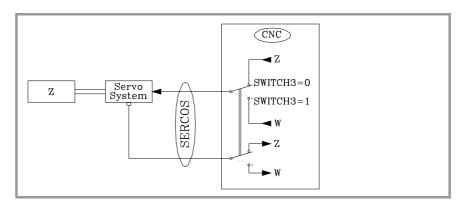
M40 AND NOT M41 = DRO2 Z axis as DRO

= 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

FAGOR ᢖ

**CNC 8040** 

CONCEPTS

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  $\cdot 10 \cdot$ .



**CNC 8040** 

**FAGO** 

# FAGOR

**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x)

#### **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
```

CONCEPTS

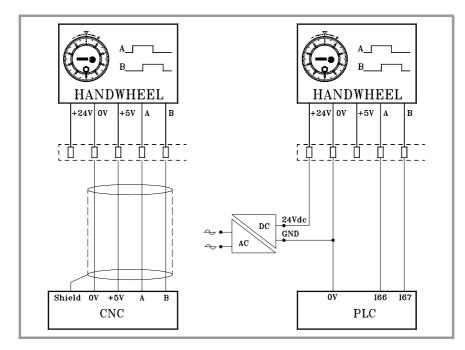
### 6.15 Fagor handwheels: HBA, HBE and LGB

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.

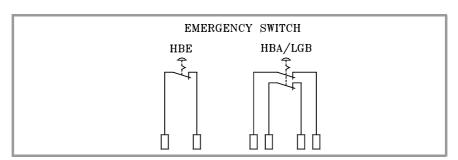
The signals from the 24V HBA and HBE models may also be taken to the PLC digital inputs located at the central unit (not remote).



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.



**FAGOF** 

**CNC 8040** 

There are 2 ways to use the "Enable Push Button".

178 Just press one of the buttons

I79 Both buttons must be pressed

The examples 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)

179 AND (173 OR 174) = HDWON

	173	174
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 ( $\cdot$ ) position: "NOT I73 AND NOT I74"
- position the CNC panel selector in the JOG area (not handwheel, not incremental) "SELECTOR > 7"

179 AND NOT 173 AND NOT 174 AND CPS SELECTOR GE 8



**CNC 8040** 

#### = JOGON

Axis selection. Inputs I70, I71, I72

NOT I70	AND	NOT I71	AND	NOT 172	= XSEL
NOT I70	AND	NOT I71	AND	172	= YSEL
NOT I70	AND	l71	AND	172	= ZSEL
NOT I70	AND	l71	AND	NOT I72	= 4SEL
170	AND	l71	AND	NOT I72	= 5SEL
170	AND	l71	AND	172	= 6SEL
I70	AND	NOT I71	AND	172	= 7SEL

	170	171	172
XSEL	0	0	0
YSEL	0	0	1
ZSEL	0	1	1
4SEL	0	1	0
5SEL	1	1	0
6SEL	1	1	1
7SEL	1	0	1

If handwheel jog (HDWON), R60 must be ready to store what will be written into the HBEVAR variable. The "a, b, c" bits indicate the x1, x10, x100 factor for each axis and bit 30 (\*) must be set to "1" in order for the CNC to read the handwheel pulses.

			С			В			Α			W			٧			U			Z			Υ			Χ	
* ,	^	С	b	t o																								

() = MOV 0 R60

**HDWON AND 4SEL** 

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 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)

				173	174	С	b	to
			x1	0	1	0	0	1
I73 AND	174	= RL1 R60 1 R60	x10	1	1	0	1	0
I73 AND	NOT I74	= RL1 R60 2 R60	x100	1	0	1	0	0

= MOV \$200 R60

And finally, the bit 30 (\*) of HBEVAR=1 is enabled, for the CNC to read the handwheel pulses.

()= OR R60 \$4000000 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 175 AND XSEL = AXIS+1



CONCEPTS

Fagor handwheels: HBA, HBE and LGB



**CNC 8040** 

JOGON AND 175 AND YSEL = AXIS+2

JOGON AND 175 AND ZSEL = AXIS+3

JOGON AND 175 AND 4SEL = AXIS+4

JOGON AND 175 AND 5SEL = AXIS+5

JOGON AND 175 AND 6SEL = AXIS+6

JOGON AND 175 AND 7SEL = AXIS+7

If JOG movement (JOGON) and [-] key pressed: "I77", then axis movement in the negative direction.

JOGON AND 177 AND XSEL = AXIS-1

JOGON AND 177 AND YSEL = AXIS-2

JOGON AND 177 AND ZSEL = AXIS-3

JOGON AND 177 AND 4SEL = AXIS-4

JOGON AND 177 AND 5SEL = AXIS-5

JOGON AND 177 AND 6SEL = AXIS-6

JOGON AND 177 AND 7SEL = AXIS-7

If JOG movement (JOGON) and [Rapid] key pressed: "I76", axis movement in rapid.

JOGON AND I76 = MANRAPID

Safety. When releasing the "Enable Push Button", the STOP command is sent out to the CNC (100 ms pulse) to stop the possible movement active at the time (for example: 10 mm in incremental). Only if the JOG mode is selected and NOT MDI

DFD I79 = TG1 17 100

MANUAL AND NOT MDI AND T17 = NOT /STOP

**END** 

In order to comply with the EN 61000-4-4 (IEC 1000-4-4) regulation on "immunity against rapid transients and blasts" use a 7x1x0.14 PVC shielded cable for the 5 V feedback cable.



**CNC 8040** 

CONCEPTS

### 6.16 Machine safety related functions

#### 6.16.1 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 maneuver 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
- 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.



FAGOR

**CNC 8040** 



**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x) END

```
PLC programming example.
```

```
PRG
REA
()=CNCRD(OPMODA,R100,M1000)
   Reading of the OPMODA variable.
BOR100 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.
;
```

### 6.16.2 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.

6.

Machine safety related functions

CONCEPTS



**CNC 8040** 

### 6.17 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.

X 52.417
Y 55.322
HOME x 52.417
T 55.322
T −5.022
S 1000 % 100
F 1000.000 % 100

PROGRAMMED DEPTH 0.000
Z TO GO:
T 5.022
T MOVE Z UP
S 1000
% 100
RRINGE 1

#### 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.

6.

CONCEPTS
Configuring a CNC as two and a half axes



**CNC 8040** 

### 6.17.1 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  $\cdot 0 \cdot$  (option available). Assigning the value of  $\cdot 1 \cdot$  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.



CONCEPTS

AGOR 🚄

CNC 8040

### 6.17.2 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 maneuver 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 BOR130 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.



**CNC 8040** 

```
CONCEPTS
Configuring a CNC as two and a half axes
```

**6.** 

```
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 BOR130 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
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
;
```



**CNC 8040** 

CONCEPTS
Tool change via PLC

### 6.18 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  $\cdot 0 \cdot$ .

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.



**CNC 8040** 

### INTRODUCTION TO THE PLC

7



It is recommended to save the PLC program and files into the memkey card" (CARDA) or in a peripheral or PC to avoid losing them.

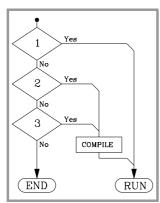
The PLC program (PLC\_PRG) may be edited at the front panel or copied from the memkey card (CARD A) or from a peripheral device or PC.

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).
- 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 memkey card (CARD A).

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.
- · Reading and writing internal CNC variables from the PLC.
- · Access to all PLC variables from any part program.
- Monitoring on the CNC screen of PLC variables.
- Access to all PLC variables from a computer, via DNC and through RS 232 C and RS 422 serial lines.



**CNC 8040** 

PLC Resources

#### 7.1 PLC Resources

#### Inputs (I)

These are elements which supply information to the PLC from signals received from the outside world. They are represented by the letter I and there are 512 inputs available.

#### Outputs (O)

These are elements which allow the PLC to activate or deactivate the different devices in the electrical cabinet. These are represented by the letter O and there are 512 outputs available.

#### Marks (M)

These are elements capable of memorizing in one bit (as if it were an internal relay) the status of the different internal variables of the CNC (information of the logic outputs received in the communication between the CNC and the PLC of the CNC) and the status of the different variables of the PLC, whether these are internal or established by the user. They are represented by the letter M, and there are 2000 user marks and other special marks.

#### Registers (R)

These are elements which allow a numerical value to be stored in 32 bits or facilitate CNC-PLC communication with the Logic CNC inputs and outputs. They are represented by the letter R and there are 256 user registers and other special registers.

#### Timers (T)

These are elements which, once activated, alter the status of their output for a specific time (time constant). They are represented by the letter T, and there are 256 timers.

#### Counters (C)

These are elements capable of counting up or down a specific amount of events. They are represented by the letter C and there are 256 counters.



**CNC 8040** 

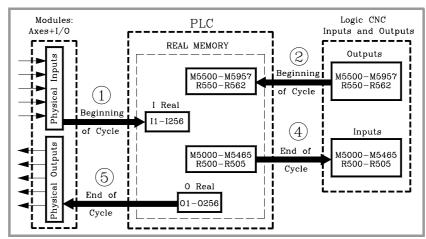
### 7.2 PLC program execution

The PLC executes the user program cyclically. In other words, once it executes the complete program, it restarts running this program from the first instruction.

This cyclic processing of the program is done as follows:

1. At the beginning of the cycle, PLC's "I" resources are assigned the current values of the physical inputs (connectors).

For example, if the physical input I10 is at 24V, the PLC sets the I10 resource to "1".



- 2. It allocates the current values of the logic CNC outputs (CNCREADY, START, FHOUT, etc.) to PLC resources M5500 thru M5957 and R550 thru R562.
- 3. It runs the program cycle.

The following sections indicate how the PLC program is structured and which are its execution modules. See "7.4 Modular structure of the program" on page 273.

- **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. It concludes this cycle scan and it gets ready for the next one.

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".



**CNC 8040** 

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"

110 = M100

M100 takes the value of resource I10

M100 AND I8 = M101

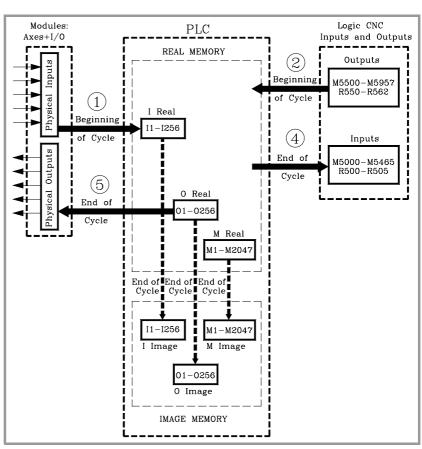
The value of M100 depends on the previous instruction.

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





**CNC 8040** 

The next example shows how the PLC acts when operating with real and image values.

PLC Program () = M1Assigns the value of "1" to mark M1.

> M1 = M2Assigns the value of M1 to M2.

> M2 = M3Assigns the value of M2 to M3.

M3 = O5Assigns the value of M3 to output O5.

			RE	Α		IMA					
		M1	M2	МЗ	O5	M1	M2	МЗ	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".

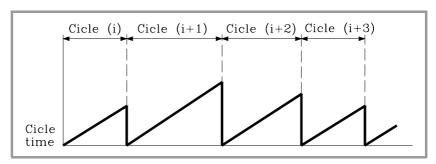
INTRODUCTION TO THE PLC PLC program execution



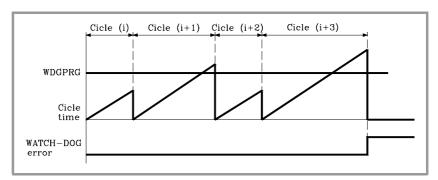
**CNC 8040** 

#### Cycle time 7.3

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.



**CNC 8040** 

### 7.4 Modular structure of the program

The program to be executed by the PLC consists of a series of modules which are appropriately defined by means of directing instructions.

The modules which can make up the program are:

- Main module (PRG)
- Periodic execution module (PE)
- First cycle module (CY1)

Each module must begin with the directing instruction which defines it (PRG, PE, CY1) and end with the directing instruction END.

Should the main program contain the main module only it is not necessary to place the instructions PRG and END.

### 7.4.1 First cycle module (CY1)

This module is optional and will only be executed when the PLC is turned on. It is used to initialize the different resources and variables with their initial values, before proceeding to execute the rest of the program.

This module operates by default with the real values of resources I, O, M.

It is not necessary for this to be at the beginning of the program, but must always be preceded by the instruction CY1.

#### 7.4.2 Main module (PRG)

This module contains the user program. It will be executed cyclically and will be given the task of analyzing and modifying CNC inputs and outputs. Its execution time will be limited by the value of plc.m.p. WDGPRG (P0)

This module operates by default with the image values of resources I, O, M.

There can only be one main program and this must be preceded by the instruction PRG, it is not necessary to define it if it starts on the first line.



**CNC 8040** 

### 7.4.3 Periodic execution module (PE t)

This module is optional and will be executed every period of time t indicated in the directing instruction defining the module.

This module may be used to process certain critical inputs and outputs which cannot be checked or updated properly in the body of the main program due to its extended execution time.

Another application for this module is for those cases where specific tasks need not be evaluated at every PLC program cycle. Those tasks would be programmed in the periodic module and they would be executed with the frequency established by the execution time assigned to this module (for example: if t= 30,000; every 30 seconds).

A "t" value between 1 and 65535 milliseconds may be programmed.

The execution time of this module will be limited by the value of plc.m.p. WDGPER (P1)

This module operates by default with the real values of resources I, O, M.

Example:

PE 10 Defines the beginning of the periodic module PE which will be executed every 10 milliseconds.

If this module is being executed with real values and acts on a physical output, this is updated at the end of the execution of the periodic module.

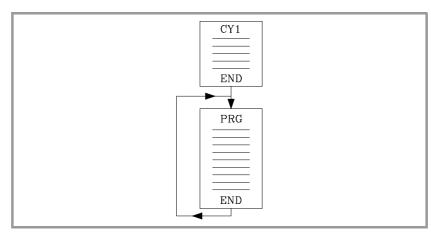


**CNC 8040** 

### 7.4.4 Priority of execution of the PLC modules

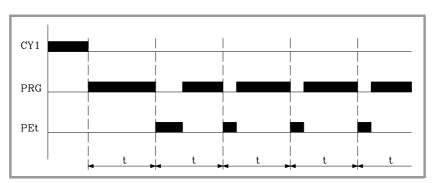
Every time the PLC program is started (command RUN) the first module to be executed is the first cycle module (CY1). Once execution has been completed, it will continue with the main module (PRG).

The main module will be executed cyclically until the execution of the PLC has stopped (command STOP).



The periodic module will be executed every time the time indicated in the directing instruction "PE t" elapses. This count starts when the execution of the main module (the first time) begins.

Every time this module is executed, the execution of the main module is interrupted, and its execution resumes when the execution of the periodic module finishes.



7.

INTRODUCTION TO THE PLC
Modular structure of the program



**CNC 8040** 



**CNC 8040** 

## **PLC RESOURCES**

8

### 8.1 Inputs

These are elements which supply information to the PLC from signals received 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

### 8.2 Outputs

These are elements which allow the PLC to activate or deactivate the different devices in 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).



**CNC 8040** 

PLC RESOURCES

Marks

These are elements capable of memorizing in one bit (as if they were an internal relay) information defined by the user, their value being inalterable even when the power supply to the system is turned off.

This will be programmed by the letter M followed by the number of the mark which it is wished to reference, for example, M1, M25, M102, etc.

The PLC controls the following marks:

User marks	M1 - M2000
Arithmetic flag marks	M2003
Clock marks	M2009 - M2024
Fixed status marks	M2046 and M2047
Marks associated with messages	M4000 - M4127
Marks associated with errors	M4500 - M4563
Screen marks	M4700 - M4955
CNC communication marks	M5000 - M5957

Marks M1 thru M2047 have image values unlike the remainder of the marks, and so the PLC will always work with their real values.

The arithmetic flag mark available at the PLC is:

M2003 Is the Zero flag and is set to 1 (high logic level) when the result of an AND, OR, XOR operation is 0.

The clock marks M2009 to M2024, make up internal clocks of different periods which can be used by the user.

The following table shows the available marks and the average period of each one.

M2009	100 ms.
M2010	200 ms.
M2011	400 ms.
M2012	800 ms.
M2013	1.6 s.
M2014	3.2 s.

6.4 s.
12.8 s.
1 s.
2 s.
4 s.
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 - M4127 or by means of their associated mnemonic MSG1 - MSG128:

M4000	M4001	M4002	 M4126	M4127
MSG1	MSG2	MSG3	 MSG127	MSG128

Likewise, 64 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.



**CNC 8040** 

They can be named by means of mark M4500-M4563 or by means of their associated mnemonic ERR1 - ERR64:

M4500	M4501	M4502	 M4562	M4563
ERR1	ERR2	ERR3	 ERR63	ERR64

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 one of the marks M4700-M4955 user pages 0-255 can be activated in the CNC. They can be named by means of mark M4700-M4955 or by means of their associated mnemonic PIC0 - PIC255:

M	1700	M4701	M4702	 M4954	M4955
Р	IC0	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 "11 Logic CNC inputs and outputs".

8.

PLC RESOURCES



**CNC 8040** 

PLC RESOURCES

### 8.4 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  $\pm 2147483647$ .

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 1001 1100



**CNC 8040** 

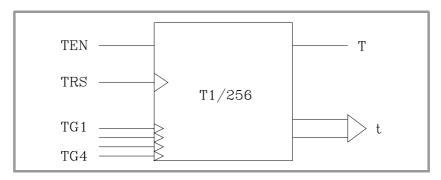
#### 8.5 Timers

These are elements capable of maintaining their output at a determined logic level during a preset time (time constant), after which the output changes status.

They do not have image values and are represented by the letter T, followed by the number of the timer it is required to reference, for example, T1, T25, T102, etc.

The time constant is stored in a 32-bit variable, and so its value can be between 0 and 4294967295 milliseconds, which is equivalent to 1193 hours (almost 50 days).

The PLC has 256 timers, each of which has T status output and TEN, TRS, TG1, TG2, TG3 and TG4 inputs. It is also possible to consult at any moment the time which has elapsed from the moment it was activated.

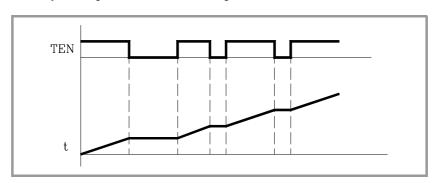


#### Enable input (TEN)

This input allows the timing of the timer to be stopped. It is referred to by the letter TEN followed by the number of the timer which is wished to reference, for example TEN 1, TEN 25, TEN 102, etc.

So that the time elapses within the timer this input must be at level "1". By default and every time a timer is activated the PLC will assign this input a logic level of "1".

If, once the timer is activated, TEN = 0 is selected, the PLC stops timing, and it is necessary to assign TEN = 1 for this timing to continue.



#### Example:

I2 = TEN 10 Input I2 controls the Enable input of timer T10.

8.

PLC RESOURCES
Timers



**CNC 8040** 

## FAGOR =

**CNC 8040** 

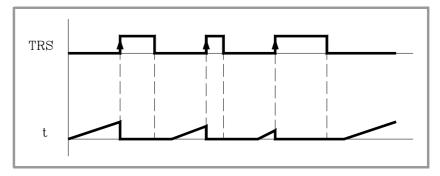
(SOFT M: V11.1x) (SOFT T: V12.1x)

#### 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). Additionally, the timer is deactivated it being necessary to activate its trigger input to activate it again.



#### 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 number of the timer which it is required to reference, 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 number of the timer which it is required to reference, 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 whether the time of T123 is greater than 1000, in which case it activates mark M100.

The PLC has a 32-bit variable to store the time of each timer.

8.

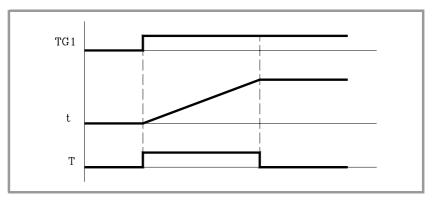
PLC RESOURCES



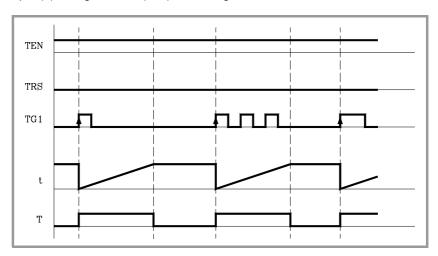
**CNC 8040** 

PLC RESOURCES

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 status (T=1) and timing t starts from a value of 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 alteration which may be produced in input TG1 (leading or trailing edge) during the timing operation will have no effect whatsoever.

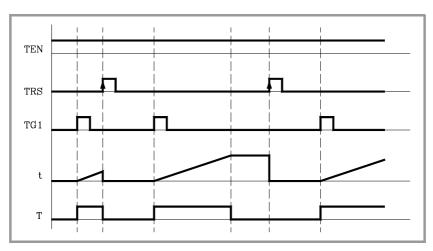
If, once the timing is complete it is required to activate the timer again, another leading edge must be produced at the TG1 input.



**CNC 8040** 

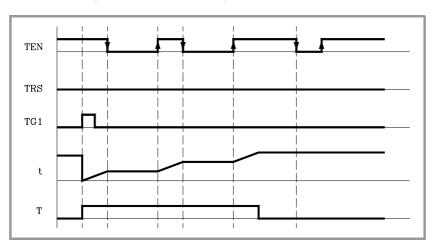
#### Operation of the TRS input in this mode

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 is activated, TEN = 0 is selected, the PLC stops timing, and it is necessary to assign TEN = 1 for this timing to continue.



8.

PLC RESOURCES
Timers

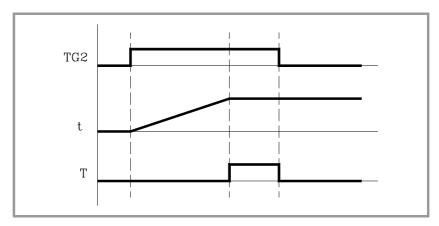


**CNC 8040** 

### 8.5.2 Delayed connection mode. TG2 input

This operating mode allows a delay to be made between the activation of the trigger input TG2 and the activation of the T status of the timer.

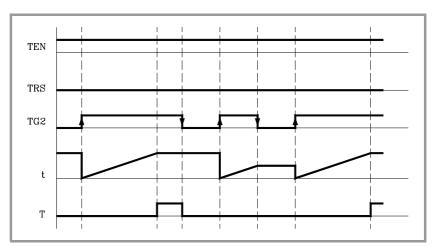
The duration of the delay is determined by the time constant.



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 moment, timing t will start from a value of 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).

8.

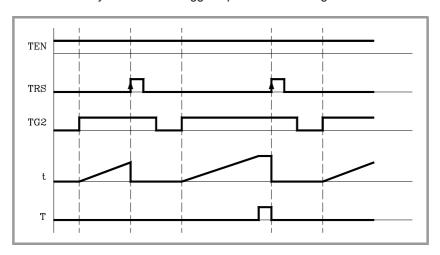
PLC RESOURCES



**CNC 8040** 

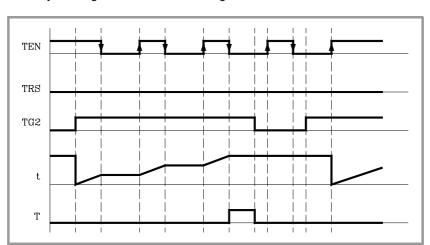
#### Operation of the TRS input in this mode

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 is activated, TEN = 0 is selected, the PLC stops timing, and it is necessary to assign TEN = 1 for this timing to continue.



8.

PLC RESOURCES

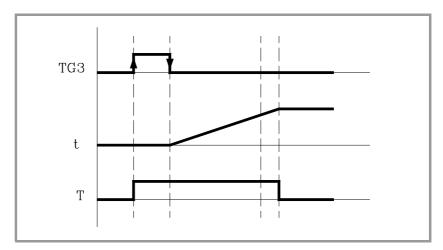


**CNC 8040** 

### 8.5.3 Delayed disconnection mode. TG3 input

This operating mode allows a delay to be made between the deactivation of the trigger input TG3 and the activation of the T status of the timer.

The duration of the delay is determined by the time constant.

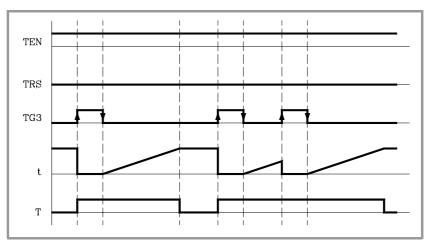


If the timer is initialized with values TEN=1 and TRS=0, the timer will be activated when a leading edge is produced at the TG3 input. At that moment, the timer status output will have a value of T=1.

The timer will wait a trailing edge of the TG3 input to start timing t from a value of 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.

8.

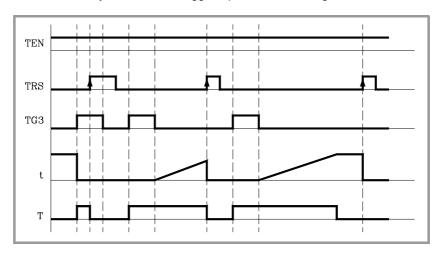
PLC RESOURCES



**CNC 8040** 

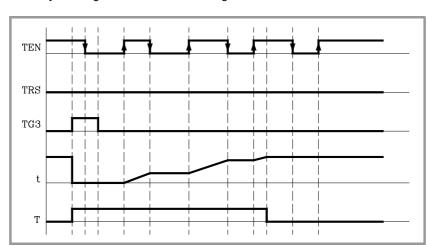
## Operation of the TRS input in this mode

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 is activated, TEN = 0 is selected, the PLC stops timing, and it is necessary to assign TEN = 1 for this timing to continue.



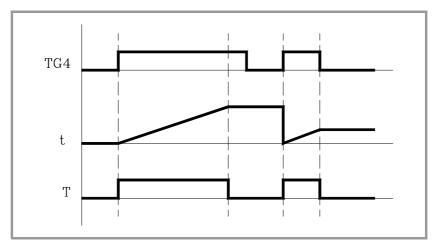
8.

PLC RESOURCES

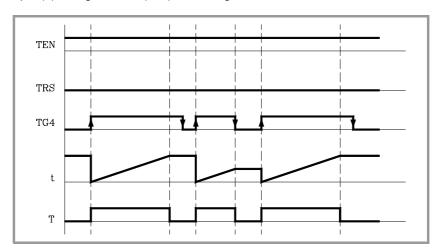


**CNC 8040** 

In this operating mode the timer status is kept at a high logic level (T=1) from the moment when the TG4 input is activated until the time indicated by the time constant has elapsed, or until a down flank is produced at the TG4 input.



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 status (T=1) and timing t starts from a value of 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.

8.

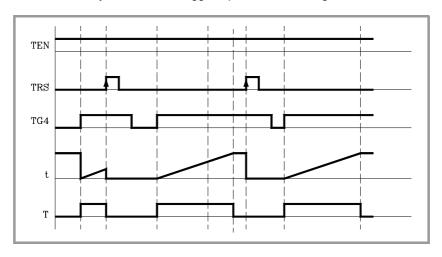
PLC RESOURCES



**CNC 8040** 

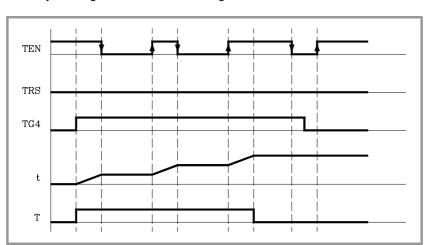
## Operation of the TRS input in this mode

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 is activated, TEN = 0 is selected, the PLC stops timing, and it is necessary to assign TEN = 1 for this timing to continue.



8.

PLC RESOURCES



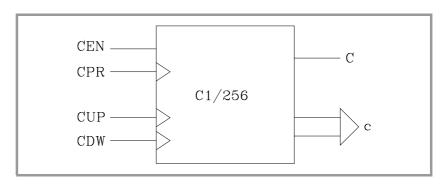
**CNC 8040** 

## 8.6 Counters

These are elements capable of counting up or down a specific 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 count of a counter is stored in a 32-bit variable, thus having a possible value of up to +2147483647.

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 which is required to reference, 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 which is required to reference, for example CDW 1, CDW 25, CDW 102, etc.

#### Example:

13 = 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 CPR followed by the number of the counter which is required to reference 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".

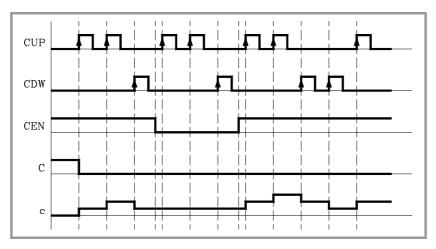
If CEN = 0 is selected the PLC stops the counter count, ignoring the inputs CUP and CDW until this input allows it (CEN = 1).

8.

PLC RESOURCES



**CNC 8040** 



Example:

I10 = CEN 12 Input I10 controls the enable input of counter C12.

### Preset input (CPR)

This input allows the counter to be preset with the desired value. It is referred to by the letters CPR followed by the number of the counter which is required to reference and the value to be assigned to the counter count.

For example CPR 1 100, CPR 25 224, CPR 102 0, CPR 200 500, etc.

The value of the count can be indicated by means of a numerical value or by assigning to it the internal value of an R register.

CPR 20 100 Presets the C20 counter to a value of 100.

CPR 22 R200 Presets the C22 counter with the value of the Register R200

when the instruction is executed.

The counter is preset with the value when a leading edge is produced 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 which is required to reference, for example C1, C25, C102, etc.

The logic status of the counter will be C=1 when the value of the count is zero and C=0 in the remainder of cases.

### 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 which is required to reference, for example C1, C25, C102, etc.

Although when written C123 it coincides with the status output, both are different and, are used in different types of instructions.

In binary type instructions function C123 makes reference to the counter's logic status.

C123 = M100 Assigns mark to M100 the status (0/1) of counter 123.

8.

PLC RESOURCES
Counters



**CNC 8040** 

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, in which case it activates mark M100.

The PLC has a 32-bit variable to store the count of each counter.

8.

PLC RESOURCES
Counters



**CNC 8040** 

## 8.6.1 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.

8.

PLC RESOURCES



**CNC 8040** 



**CNC 8040** 

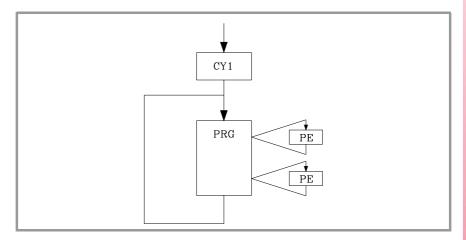
## PLC PROGRAMMING

The PLC program is structured by modules and it may consist of:

- Main module (PRG).
- Periodic execution module (PE).
- First cycle module (CY1).

Every time the PLC program starts running, the CNC will execute first, if it has been defined, the First Cycle module (CY1). Then it will execute the Main Program module (PRG) continuously until the PLC program is stopped.

The periodic execution modules (PE) will be executed every so often with the frequency established for each of them. This time period starts counting from the time the CY1 cycle is ended. The execution of a periodic module temporarily interrupts the execution of the main module.



When defining the PLC program, both the processing of the main module (PRG) and the periodic modules (PE) must be taken into consideration.

The main module (PRG) will be processed cyclically. See "7.2 PLC program execution" on page 269.

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).
- It ends the execution of the Periodic Module and resumes the execution of the main module.



**CNC 8040** 



**CNC 8040** 

The modules which make up the PLC program (main module "PRG", periodic modules "PE" and first cycle module "CY1") consist of a series of instructions which, depending on their functionality, can be divided into:

- · Directing instructions.
- Executable instructions.

The directing instructions provide the PLC with information on the type of module (PRG, CY1, ...) and on how it must be executed (REA, IMA, ...).

Executable instructions allow inquiries to be made on and/or alterations to the status of PLC resources and consist of:

Logic expressions (Boolean 0/1) I28 AND I30

Action instructions. = O25

Logic expressions consist of:

Consulting instructions I28, O25

Operators. AND

All comments must begin with ";". ";". Lines beginning with a ";" are considered comments and are not executed.

Programming example:

PRG ; Directing instruction

; Example Comment.

I100 = M102 ; Executable proposition.I28 AND I30 ; Logic expression.= O25 ; Action instruction.

i32 \ ; Consulting instruction (1st part of the expression)

AND I36 ; Consulting instruction (1st part of the expression).

= M300 ; Action instruction. END ; Directing instruction.

See "Summary of PLC commands" on page 485.



Empty lines are not allowed, they must contain at least one comment.

9.

PLC PROGRAMMING

Module structure



**CNC 8040** 

The directing instructions available at the PLC are:

PRG, PEt, CY1

Define the module type.

**PRG** Main module.

First cycle module. CY1

PΕ 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 of module CY1. **END** 

**PRG** Beginning of module PRG.

End of module PRG. **END** PE 100 Beginning of module PE.

**END** End of module PE.

Example of programming without using the directing instruction END:

CY1 Beginning of module CY1.

**PRG** Beginning of module PRG.

PE 100 Beginning of module PE.

End of modules CY1, PRG and PE.

L Label. Used to identify a program line, and is only used when references or program jumps are made.

> It will be represented with the letter L followed by three figures (1-256), it not being necessary to follow any order and numbers out of sequence are permitted.

> If there are 2 or more labels with the same number in a single program, the PLC will show the corresponding error when compiling it.

> Symbol definition. Allows a symbol to be associated with any PLC variable, it being possible to reference this variable throughout the program by means of the variable name or by means of the associated symbol.

Example:

**DEF EMERG I1** 

Assigns the EMERG symbol to input 11, so any reference throughout the program to EMERG will be interpreted by the PLC as a reference to I1.

It is also possible to associate a symbol to any number which can be given in decimal, with or without a sign, or hexadecimal format preceded with the "\$" sign.

PLC PROGRAMMING Directing instructions



**DEF** 

**CNC 8040** 

This option, among other applications, makes programming and later understanding of the PLC program much easier when trying to control the CNC by simulating its keyboard from the PLC program.

#### Example:

DEF HELP \$FFF2

Assigns the "HELP" symbol to the code for the HELP key.

() = MOV HELP R101

Assigns the code corresponding to the "HELP" key to register R101.

CNCWR (R101, KEY, M101)

Indicates to the CNC that the key whose code is stored in register R101 and corresponds to the HELP key.

The PLC allows up to 400 symbol definitions which must always be programmed at the beginning of the program, before any other instruction, be this directing or executing.

A symbol will be made up with up to 8 characters, and must not coincide with any of the words reserved for instructions, nor be formed by the characters space", equal "=", open and close parentheses "()", comma "," and semicolon ";".

Duplicate symbols cannot be defined; but several symbols may be assigned to the same resource.

### Example:

**DEF EMRGOUT 01** 

**DEF SALEMRG 01** 

The symbols associated to specialized marks and register (M> 2047 y R > =500) are pre-defined in the PLC and, therefore, it is not necessary to define them, nevertheless and if required, the PLC allows a different symbol to be assigned to them.

#### REA, IMA

Indicate to the PLC that the consultations defined below will be made on the real (REA) or image (IMA) values of I, O, M resources.

Counters, timers and registers do not have image values, so their real values will always be evaluated.

Action instructions (=O32) will always update the real values of PLC resources.

### Example:

IMA

Consultations will evaluate image values.

I1 AND I2 = 01

-----

REA

Consultations will evaluate real values.

IMA I3 AND REA M4 = 02

Evaluates the image of I3 and the real of M4.

IMA I5 REA = O3

Evaluates the image of I5 and the next ones in real.

9.

PLC PROGRAMMING
Directing instructions



**CNC 8040** 

### 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.

The data capture in the execution of the trace in this program takes place:

• At the beginning of each PRG cycle.

**END** 

- 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.



PLC PROGRAMMING
Directing instructions



**CNC 8040** 

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
0	1/512	Outputs
М	1/5957	Marks
Т	1/256	Timers
С	1/256	Counters
В	0/31 R 1/499	Register Bits

### Example:

112

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.)	l 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.



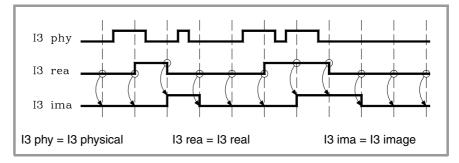
PLC PROGRAMMING
Consulting instructions



**CNC 8040** 

**CPS** 

The image values of the inputs, outputs and marks are updated after executing the program cycle.



#### Examples:

DFU I23 DFU B3R120 DFU AUXEND

## Comparison

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.



**CNC 8040** 

## 9.4 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 12 = 03

Output "O3" will be active when input I2 is not.

**AND** Logic function "AND".

14 AND 15 = 06

Output "O6" will be active when both inputs (I4, I5) are active.

**OR** Logic function "OR".

17 OR 18 = 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". In other words:

() = 02

Output O2 will always be high (=1).

9.

PLC PROGRAMMING
Operators and symbols



**CNC 8040** 

## 9.5 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.



**CNC 8040** 

## 9.5.1 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 upflank 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)) .

9.

PLC PROGRAMMING
Action instruction.



**CNC 8040** 

## 9.5.2 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	l 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 does 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.



**CNC 8040** 

These actions interrupt the sequence of a program, resuming it somewhere else in the program.

That area must be identified with a label (L 1/256).

A subroutine is any part of the program that starts with a label (L1/256) and end with the directing instruction END.

**= JMP** Unconditional Jump.

If the result of evaluating the logic expression is a "1", it causes jump to the indicated label. If the result is a "0", it goes on to the next program line.

Example:

I8 = JMP L12 If I8 = 1 it goes on to L12

M14 AND B7R120 = O8 If I8=1 it is not executed

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.



PLC PROGRAMMING
Action instruction.



**CNC 8040** 

## 9.5.4 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	l 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	l12	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	9999	with	16	9999999	with	28
99	with 8 bits		bits			bits	
999	with 12	99999	with bits	20	99999999	with bits	32

In these cases, it is recommended to make the move increasing the number of bits by using, if necessary, registers or marks in intermediate steps.

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.





**CNC 8040** 

#### = 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.

### = NGS

Register sign change.

Example: I16 = NGS R89

If input "I16 = 1", the PLC changes the sign of the contents of register R89.

= 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:

400	D.1/550	D.1/550	D.1/550	$\neg$
ADS	R1/559	R1/559	R1/559	
SBS	#	#		
MLS				
DVS				
MDS				

The operands may be: Registers, CNC-PLC communication registers and numbers (#) within  $\pm 2147483647$  or between 0 and \$FFFFFFF

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.



PLC PROGRAMMING
Action instruction.



**CNC 8040** 

= RR

= RL

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:

Examples with

()=XOR

	AND	R1/559	R1/559	R1/559
(	OR	#	#	
)	XOR			

Registers (R1/559) or numbers expressed in decimal, hexadecimal or binary format can be defined as first or second operand.

The destination register indicates where the result of the operation will be deposited and will be defined by means of a register (R1/559).

The mark M2003 is called zero flag and indicates whether the result of an AND, OR, XOR, operation equals zero, in which case it follows that M2003=1.

	R2	01 = B01	00 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

R200 = B1001 0010

B1010 B1110 R207

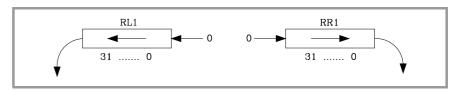
Used to rotate registers clockwise (RR) or counterclockwise (RL). There are two types of rotations: type 1 (RR1 or RL1) and type 2 (RR2 or RL2).

R207=B00000100

M2003=0

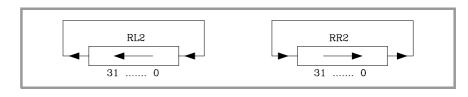
Rotation type 1 (RL1 or RR1):

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.



9.

PLC PROGRAMMING
Action instruction.



**CNC 8040** 

### Its programming format is:

	Origin	Nr of repetitions	Destination
RR1	R1/559	R1/559	R1/559
RR2		0/31	
RL1			
RL2			

The origin and destination registers must always be defined, even when they are both the same. The number of repetitions indicates the consecutive number of times the register will be rotated.

## Examples:

RR1 R100 1 R200

1 type 1 rotation to the right of the contents of R100 leaving the result in R200.

RL2 R102 4 R101

 $4\, type\mbox{-}2$  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

9.

PLC PROGRAMMING
Action instruction.



**CNC 8040** 

#### = 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
	0	1/512	1/512
	М	1/5957	1/5957
	Т	1/256	1/256
	С	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:

112 = ERA O5 12

If input I12 has a value of "1" the PLC will set to 0 outputs O5 thru O12.

123 = 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.

MMING

PLC PROGRAMMING
Action instruction.



**CNC 8040** 

### = 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:

115 = PAR R123 M222

If 115 = 1 the PLC checks the parity of register R123 and sets M222 = 1 if it is EVEN or M222 = 0 if it is ODD.



PLC PROGRAMMING
Action instruction.



**CNC 8040** 



**CNC 8040** 

# **CNC-PLC COMMUNICATION**

The exchange of information between the CNC and the PLC allows:

- 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.
- Reading and writing internal CNC variables from the PLC.
- · Access to all PLC variables from any part program.
- Monitoring on the CNC screen of PLC variables.
- Access to all PLC variables from a computer, via DNC through RS 232 C and RS 422 serial lines.



**CNC 8040** 

## 10.1 Auxiliary M, S, T functions

MBCD1 (R550)
MBCD2 (R551)
MBCD3 (R552)
MBCD4 (R553)
MBCD5 (R554)
MBCD6 (R555)
MBCD7 (R556)
MBCDP1 (R565)
MBCD2 (R566)
MBCD3 (R567)
MBCD4 (R568)
MBCD4 (R568)
MBCDP5 (R569)
MBCD6 (R570)
MBCD7 (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 \$FFFFFFF to those which are left free.

This way, if a block contains functions M100, M120 and M135, the CNC will transfer the following information:

MBCD1 (R550)	= \$100
MBCD2 (R551)	= \$120
MBCD3 (R552)	= \$135
MBCD4 (R553)	= \$FFFFFFF.
MBCD5 (R554)	= \$FFFFFFF.
MBCD6 (R555)	= \$FFFFFFF.
MBCD7 (R556)	= \$FFFFFFF.

Use one of the following methods to determine whether or not a specific "M" function has been programmed in a block which is being executed:

- 1. Check all MBCD registers one by one until the specific "M" function is found or until one of them contains the \$FFFFFFF 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.



**CNC 8040** 

If S output in 2-digit BCD is used the CNC will tell the PLC, by means of this register the selected spindle speed according to the following conversion table:

Programmed S	S BCD	Programmed S	S BCD		Programmed S	S BCD
0	00	50-55	54		800-899	78
1	20	56-62	55	ľ	900-999	79
2	26	63-70	56	Ī	1000-1119	80
3	29	71-79	57	Ī	1120-1249	81
4	32	80-89	58	Ī	1250-1399	82
5	34	90-99	59	ľ	1400-1599	83
6	35	100-111	60	Ī	1600-1799	84
7	36	112-124	61	Ī	1800-1999	85
8	38	125-139	62	ľ	2000-2239	86
9	39	140-159	63		2240-2499	87
10-11	40	160-179	64		2500-2799	88
12	41	180-199	65	ľ	2800-3149	89
13	42	200-223	66	ľ	3150-3549	90
14-15	43	224-249	67		3550-3999	91
16-17	44	250-279	68	ľ	4000-4499	92
18-19	45	280-314	69	ľ	4500-4999	93
20-22	46	315-354	70	ľ	5000-5599	94
23-24	47	355-399	71	ľ	5600-6299	95
25-27	48	400-449	72	ľ	6300-7099	96
28-31	49	450-499	73	ľ	7100-7999	97
32-35	50	500-559	74	ľ	8000-8999	98
36-39	51	560-629	75	ľ	9000-9999	99
40-44	52	630-709	76			
45-49	53	710-799	77			

10

CNC-PLC COMMUNICATION
Auxiliary M, S, T functions

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 \$FFFFFFF to this register.



**CNC 8040** 

### **TBCD (R558)**

The CNC tells the PLC by means of this 32-bit register, the pocket number in the magazine where the selected tool is.

If the g.m.p. RANDOMTC (P25) has been set so it is not a random magazine, the magazine pocket position coincides with the tool number.

This will be coded in BCD format (8 digits).

T 123 = 0000 0000 0000 0000 0001 0010 0011

If no T has been programmed in the block, the CNC will assign a value of \$FFFFFFF 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 \$FFFFFFF 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.



**CNC 8040** 

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.

**10.** 

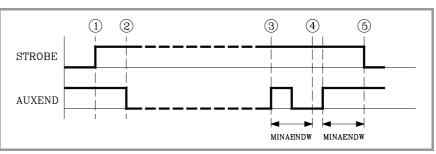
**CNC-PLC COMMUNICATION**Auxiliary M, S, T function transfer



**CNC 8040** 

## 10.2.1 Transferring M, S, T using the AUXEND signal

 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.

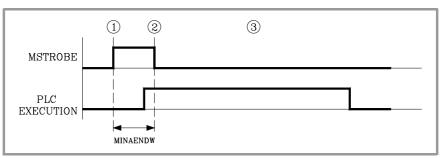


When the block being executed has several auxiliary functions (M, S, T), the CNC waits a time period set by g.m.p. MINAENDW (P30) between two consecutive transfers.



**CNC 8040** 

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.

**10.** 

**CNC-PLC COMMUNICATION**Auxiliary M, S, T function transfer



**CNC 8040** 

## 10.3 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

## Displaying messages

The PLC has 128 marks, with their corresponding mnemonic for displaying messages in the CNC.

M4000	MSG001
M4001	MSG002
M4002	MSG003

M4100	MSG101
M4101	MSG102
M4102	MSG103

M4125	MSG126
M4126	MSG127
M4127	MSG128

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 MSG128 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.



**CNC 8040** 

**CNC-PLC COMMUNICATION** 

# **Displaying errors**

The PLC has 64 marks, with their corresponding mnemonic, for displaying errors at the CNC.

M4500	ERR001		
M4501	ERR002		
M4502	ERR003		

M4530	ERR031		
M4531	ERR032		
M4532	ERR033		

M4561	ERR062
M4562	ERR063
M4563	ERR064

When one of these marks is activated (they are set high), they interrupt CNC partprogram 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			
M4701	PIC001			
M4702	PIC002			

)
2

M4953	PIC253	
M4954	PIC254	
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.



**CNC 8040** 

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.

10.

CNC-PLC COMMUNICATION
Access to the PLC from the CNC



**CNC 8040** 

The CNC allows the PLC to communicate with a computer via DNC through the RS232C and RS422 serial lines.

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.).
- Ftc.

The DNC manual can be applied for from the Commercial Department of Fagor Automation.

10.

CNC-PLC COMMUNICATION Access to the PLC from a PC, via DNC.



**CNC 8040** 

CNC-PLC COMMUNICATION
Access to the PLC from a PC, via DNC.



**CNC 8040** 

# LOGIC CNC INPUTS AND OUTPUTS

11

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



**CNC 8040** 

#### **General logic inputs** 11.1



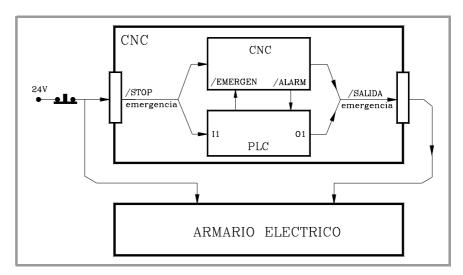
These inputs must always be defined in the PLC program.

/EMERGEN (M5000) /STOP (M5001)

/FEEDHOL (M5002) /XFERINH (M5003)

# **/EMERGEN (M5000)**

There are to ways to cause an emergency at the CNC, by activating the physical input /Emergency stop (pin 10 of connector X2) or the general logic input "/EMERGEN" from the PLC.



When the PLC sets the "/EMERGEN" input low (0V), the CNC stops the axes and the spindle and it displays the corresponding error message.

Also, the CNC activates the "/EMERGENCY OUTPUT" and "/ALARM" signals to let the outside world and the PLC know that an emergency has occurred at the CNC.

The CNC does not allow executing programs and it aborts any attempt to move the axes or the spindle while the "/EMERGEN" input is low (0V).

When the PLC brings the "/EMERGEN" input back high (24V), the CNC deactivates the "/EMERGENCY OUTPUT" and "/ALARM" signals to let the outside world and the PLC know that there is no longer an emergency at the CNC.

# Example

I-EMERG AND (rest of conditions) = /EMERGEN

If the external emergency input is activated or any other emergency occurs, the general logic input /EMERGEN of the CNC. When there is no emergency, this signal must remain high.



**FAGOR** 

**CNC 8040** 

#### /STOP (M5001)

When the PLC sets this signal low, the CNC stops the part program, and maintains spindle rotation.

In order to continue executing the program, as well as setting this signal at a high logic level, the general logic input CYSTART must be activated.

The treatment which this /STOP signal receives is similar to that given to the STOP key on the CNC front panel keeping all the keys enabled even when the /STOP signal is at low logic level (0).

#### Example

() = /STOP

There is always permission to execute the part program.

# /FEEDHOL (M5002)

When the PLC sets this signal low, the CNC stops the axes (maintaining spindle rotation). When the signal returns to the high logic level, the movement of the axes continues.

If the /FEEDHOL signal is activated (0V) in a block without motion, the CNC will continue the execution of the program until detecting a block with motion.

#### Example

() = /FEEDHOL

There is always permission to move the axes.

# /XFERINH (M5003)

If the PLC sets this signal low, the CNC prevents the following block from starting, but finishes the one it is executing. When the signal returns to high logic level, the CNC continues to execute the program.

#### Example

() = /XFERINH

There is always permission to execute the next block.

# CYSTART (M5007)

If the START key is pressed on the front panel of the CNC, this is indicated to the PLC by means of the general logic output START.

If the PLC program considers that there is nothing to prevent the part program form being executed, the CYSTART signal must be set at a high logic level, thus beginning the execution of the program.

The CNC will indicate by means of the general logic output INCYCLE that the program is being executed. As of that moment the CYSTART can return to low logic level.

# Example

START AND (rest of conditions) = CYSTART

When the cycle START key is pressed, the CNC activates the general logic output START. The PLC must check that the rest of the conditions (hydraulic, safety devices, etc.) are met before setting the general input CYSTART high in order to start executing the program

General logic inputs



**CNC 8040** 

# **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 jog mode.

#### **OVRCAN (M5010)**

If the PLC sets this signal at a high logic level, the CNC selects 100% feedrate override, irrespective of whether this is selected by the PLC, DNC, program or by the front panel switch.

While the OVERCAN signal is activated (logic 1), the CNC will apply in each mode 100 % of the feedrate corresponding to that mode.

# **LATCHM (M5011)**

This allows the type of JOG key operation to be selected in JOG mode.

If the PLC sets this signal low, the axes will only move while the corresponding JOG key is pressed.

If the PLC sets this signal at a high logic level, the axes will move from the moment the corresponding JOG key is pressed until the STOP key or other JOG key is pressed. In this case, the movement will be transferred to that indicated by the new key.

# ACTGAIN2 (M5013)

The axes and the spindle can have 2 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) = 0The CNC assumes the first set.

ACTGAIN2 (M5013) = 1 The CNC assumes the second set.

The change of gains and accelerations is always made at the beginning of the block

When working in round corner (G5), the change does not take place until G07 is programmed.



General logic inputs



**CNC 8040** 

#### 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.

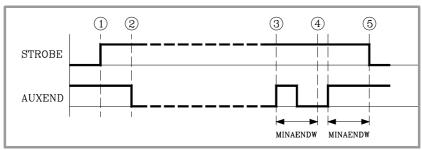
The treatment received by this signal is similar to that given to the RESET key on the front panel.

# **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:

 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.



**CNC 8040** 

# 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.

# 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 12** 

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 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.



**CNC 8040** 

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



**CNC 8040** 

# RETRACE (M5051)

The CNC takes into account this input when retracing is available, g.m.p. RETRACAC other than zero.

If while executing a part-program the PLC sets this signal high, retracing is activated. The CNC interrupts program execution and starts executing backwards what has been done so far.

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

Retracing executes backwards the current block plus up to 75 blocks that were already executed.

The retracing function ends in the following cases:

- When the previous 75 blocks are retraced.
- When retraced all the way to the beginning of the program.
- When 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.

Retracing cannot be activated while a canned cycle is active or when working in "look-ahead".

# **ACTLIM2 (M5052)**

The PLC sets this signal high to "tell" to the CNC to activate the second travel limits set by means of variables LIMPL(X-C) and LIMMI(X-C)

The second travel limit of each axis will be taken into account if the first one has been set using a.m.p. LIMIT+ (P5) and LIMIT- (P6).

# HNLINARC (M5053)

This signal is used when either the "path handwheel" or "path jog" work mode has been selected using general input "MASTRHND (M5054)". It allows selecting the type of movement.

M5053 = 0 Linear path. M5053 = 1 Arc path.

M5054 = 1

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.

"Path handwheel" or "path jog" function on.

M5054 = 0 Normal work mode with handwheels or jog keys.



#### 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"

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 has no effect when tangential control is active.

LOGIC CNC INPUTS AND OUTPUTS



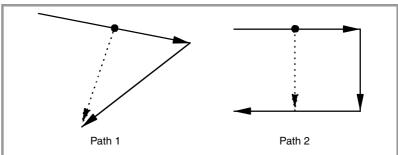
**CNC 8040** 

General logic inputs

#### 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.



**CNC 8040** 

# 11.2 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 4 (LIMIT+2, DECEL1,etc.) or using the axis name (LIMIT+X, DECELZ, etc.).

The marks of the axes that do not exist in the machine parameters assume the value of the M2045 mark, that's always set to 0.

When monitoring the PLC program, it shows the edited marks, either with a letter or with a number. However, the resource windows created from monitoring will replace the marks with axis names by the marks with axis numbers. For example:

SERVOXON by SERVO1ON

SERVOZON by SERVO2ON if there is no Y axis, but there are X and Z axes.



The use of mnemonics with the axis name is available from versions V9.0x and V10.0x on. If PLC programs older than this version have defined these marks as symbols, when compiling the program will issue an error on this line.

Example: DEF ENABLEX M333

# Mnemonics using numbers 1 through 4.

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: for the Z axis: LIMIT+3, LIMIT-3, DECEL3, etc. LIMIT+4, LIMIT-4, DECEL4, etc. for the U 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.

**FAGOR** 

**CNC 8040** 

Axis logic inputs

LIMIT+1 (M5100) | LIMIT-1 (M5101) LIMIT+2 (M5150) | LIMIT-2 (M5151) LIMIT+3 (M5200) | LIMIT-3 (M5201) LIMIT+4 (M5250) | LIMIT-4 (M5251)

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)

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)

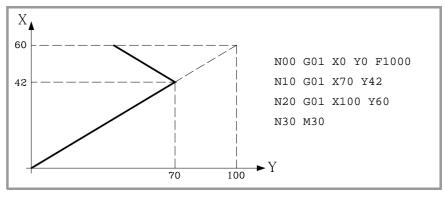
The PLC sets one of these signals at a high logic level in order to tell the CNC to prevent any movement of the corresponding axis. This movement will continue when the PLC sets this signal at the low logic level once more.

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)

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.



**CNC 8040** 

# SWITCH1 (M5105) SWITCH2 (M5155) SWITCH3 (M5205) SWITCH4 (M5255)

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 "6.13 Axes (2) controlled by a single drive" on page 250.

# DRO1 (M5106) DRO2 (M5156) DRO3 (M5206) DRO4 (M5256)

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)

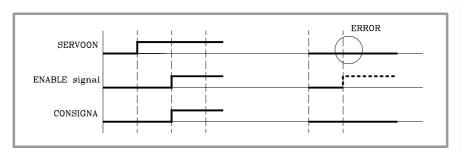
When one of these signals is set high, the CNC closes the positioning loop of the corresponding axis.

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 outputting the required analog voltage.

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.



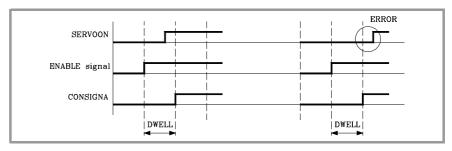
**CNC 8040** 

#### 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 analog output voltage.

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 analog voltage 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)

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)

> 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, it turns off the power circuit of the drive and the motor loses its 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.



Axis logic inputs



**CNC 8040** 

# SYNCHRO1 (M5112) SYNCHRO2 (M5162) SYNCHRO3 (M5212) SYNCHRO4 (M5262)

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)

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)

The SMOTIME (P58) filter set for each axis with parameter P58 can be canceled from the PLC.

This SMOTIME filter will be activated or deactivated at the beginning of the block. If one of these logic inputs is activated or deactivated while the CNC is overlapping blocks being executed in round corner, it will be ignored until that operation is finished.

# LIM1OFF (M5115) LIM2OFF (M5165) LIM3OFF (M5215) LIM4OFF (M5265)

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)

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.

11.

FAGOR

**CNC 8040** 

# 11.3 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 analog 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.

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.



**CNC 8040** 

LOGIC CNC INPUTS AND OUTPUTS

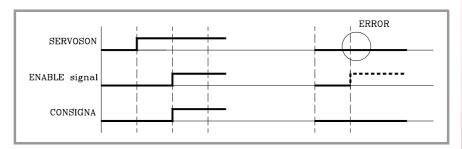
# SERVOSO2 (M5482) Second spindle

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 analog output voltage.



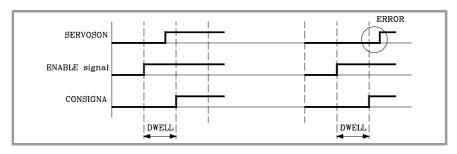
On the other hand, if the SERVOSON signal is low or if it changes to low during the rotation of the spindle, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.

# 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 analog output voltage.

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 analog voltage 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.



**CNC 8040** 

# 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:

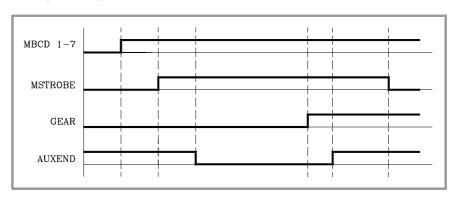
# The M41, M42, M43 or M44 function uses the "AUXEND" signal:

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 "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.



Spindle logic inputs



**CNC 8040** 

#### If the M41, M42, M43 or M44 function does not use the "AUXEND" signal:

- The CNC indicates to the PLC the selected gear M41, M42, M43 or M44 in one of the registers "MBCD1" through "MBCD7" and it activates the "MSTROBE" signal to let the PLC "know" that it must execute it.
- The CNC will keep the output "MSTROBE" active during the time indicated by means of g.m.p. MINAENDW (P30).
- After this time, the CNC will check whether the new spindle gear has been physically selected by verifying that the corresponding GEAR input (GEAR1... GEAR4) is set high.
- If it is not selected, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.

#### SPENAS (M5462) | 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, it turns off the power circuit of the drive and the motor loses its 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 0.00019/min.

Spindle logic inputs.



**CNC 8040** 

# **LOGIC CNC INPUTS AND OUTPUTS**

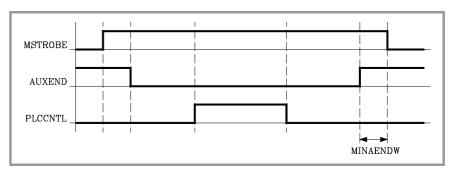
#### PLCCNTL (M5465) Main spindle

#### 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

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 output 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 send out the output 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 analog output which the CNC must send out when it is controlled by the PLC.

SANALOG=32767 corresponds to an analog output of 10 V.

(10/32767) 0.305185 millivolts of analog output correspond to SANALOG=1.

This way, for 4V of analog voltage, the following must be programmed:

SANALOG = (4 x 32767)/10 = 13107

For -4V of analog voltage, the following must be programmed:

SANALOG = (-4 x 32767)/10 = -13107



Spindle logic inputs



**CNC 8040** 

# 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.

LOGIC CNC INPUTS AND OUTPUTS Spindle logic inputs.



**CNC 8040** 

# 11.4 Logic inputs of the auxiliary spindle

# SPENAAS (M5449) | DRENAAS and (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, it turns off the power circuit of the drive and the motor loses its 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 analog voltage to output when the auxiliary spindle is controlled by the PLC or via Sercos.

A 10 V of analog voltage corresponds to SANALOAS=32767.

SANALOAS=1 corresponds (10/32767) 0.305185 millivolts of analog voltage.

This way, for 4V of analog voltage, the following must be programmed:

SANALOAS = (4x32767)/10 = 13107

For -4V of analog voltage, 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.



**CNC 8040** 

# LOGIC CNC INPUTS AND OUTPUTS Key inhibiting logic inputs.

# KEYDIS1 (R500) KEYDIS2 (R501) KEYDIS3 (R502) KEYDIS4 (R503) KEYDIS5 (R508)

Key inhibiting logic inputs.

11.5

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 521.

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	Register	Bit	Inhibited key
KEYDIS4	0	Handwheel x100	KEYDIS4	16	Feedrate override 60%
KEYDIS4	1	Handwheel x10	KEYDIS4	17	Feedrate override 70%
KEYDIS4	2	Handwheel x1	KEYDIS4	18	Feedrate override 80%
KEYDIS4	3	Jog 10000	KEYDIS4	19	Feedrate override 90%
KEYDIS4	4	Jog 1000	KEYDIS4	20	Feedrate override 100%
KEYDIS4	5	Jog 100	KEYDIS4	21	Feedrate override 110%
KEYDIS4	6	Jog 10	KEYDIS4	22	Feedrate override 120%
KEYDIS4	7	Jog 1	KEYDIS4	23	
KEYDIS4	8	Feedrate override 0%	KEYDIS4	24	
KEYDIS4	9	Feedrate override 2%	KEYDIS4	25	
KEYDIS4	10	Feedrate override 4%	KEYDIS4	26	
KEYDIS4	11	Feedrate override 10%	KEYDIS4	27	
KEYDIS4	12	Feedrate override 20%	KEYDIS4	28	
KEYDIS4	13	Feedrate override 30%	KEYDIS4	29	
KEYDIS4	14	Feedrate override 40%	KEYDIS4	30	
KEYDIS4	15	Feedrate override 50%	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%.



**CNC 8040** 

# 11.6 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.

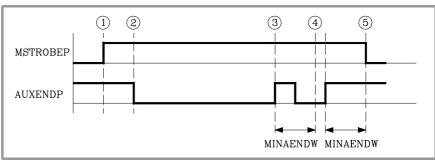
# AUXENDP (M5006)

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.



**CNC 8040** 

#### **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.

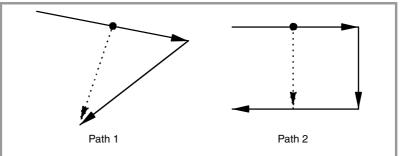
These marks do not affect the following functions.

- 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.

**FAGOR** 

**CNC 8040** 

# 11.7 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 (other 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)**

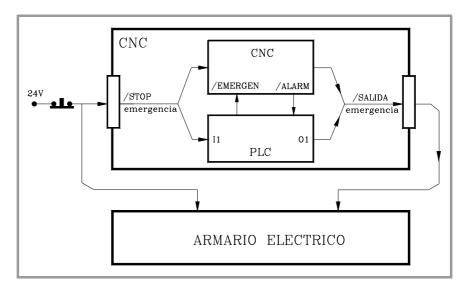
The CNC sets this signal high in order to tell the PLC that the positioning loop of the axes is open since an error has occurred.



**CNC 8040** 

# /ALARM (M5507)

The CNC sets this signal low in order to tell the PLC that an alarm or emergency condition has been detected. This signal will be set high once again, once the message from the CNC has been eliminated and the cause of the alarm has disappeared.



Likewise, while this signal is low, the CNC keeps the emergency output (pin 2 of connector X2) active (low).

# Example

/ALARM AND (other conditions) = O1

The emergency output O1 of the PLC must be normally high. If an alarm or an emergency is detected at the CNC, the emergency output O1 must be set low (0V).

# **MANUAL (M5508)**

The CNC sets this signal high to tell the PLC that the JOG (Manual) operating mode is selected.

# **AUTOMAT (M5509)**

The CNC sets this signal high to tell the PLC that the automatic operating mode is selected.

# MDI (M5510)

The CNC sets this signal high to tell the PLC that the MDI mode (manual data input) is selected in one of the operating modes (JOG, automatic, etc).

# **SBOUT (M5511)**

The CNC sets this signal high to tell the PLC that the single block execution mode is selected.



**CNC 8040** 

# **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.



**CNC 8040** 

# **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).

# **SELECTO (M5524) SELECT4 (M5528) SELECTOR (R564)**

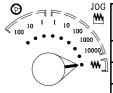
SELECT1 (M5525) SELECT2 (M5526) SELECT3 (M5527) SELECT5 (M5529) SELECT6 (M5530) SELECT7 (M5531)

The CNC uses these signals to indicate to the PLC the position selected at each keyboard switch.

SELECTOR indicates the position currently selected.

**SELECT** indicates the value applied by the CNC.

Usually, the two values coincide, except when a position has been selected which has been disabled with the KEYDIS4 input (R503). If while being the 60% and 120% inhibited, the 100% position is selected, SELECTOR will show the selected position (100%) and SELECT will show the value being applied (50%).



<i>3</i> 1		SELECTOR(3) SELECT3	SELECTOR(2) SELECT2	SELECTOR(1) SELECT1	SELECTOR(0) SELECT0
Handwheel x	100	0	0	0	0
Handwheel x	10	0	0	0	1
Handwheel x	:1	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 J	OG	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

**LOGIC CNC INPUTS AND OUTPUTS** 

**FAGOF** 

**CNC 8040** 

# MSTROBE (M5532)

The CNC sets this signal high to tell the PLC that it must execute the auxiliary M function or functions which are indicated in registers "MBCD1" to "MBCD7" (R550 to R556).

# SSTROBE (M5533)

This signal will be used when operating a spindle with BCD coded S signal. s.m.p. SPDLTYPE (P0).

The CNC sets this signal high to tell the PLC that it must execute the auxiliary S function which is indicated in the register "SBCD" (R557).

# TSTROBE (M5534)

The CNC sets this signal high to tell the PLC that it must execute the auxiliary S function which is indicated in the register "TBCD" (R558).

In this register the CNC will tell the PLC the position of the magazine where the selected tool is.

If the g.m.p. RANDOMTC (P25) has been set so it is not a random magazine, the magazine pocket position coincides with the tool number.

# **T2STROBE (M5535)**

This register is used when a special tool change has been made, family code or 200 or with machining centers with a non-random tool magazine (g.m.p. RANDOMTC (P25).

The CNC sets this signal high to tell the PLC that it must execute a second auxiliary T function indicated in the register "T2BCD" (R559).

In this register the CNC indicates to the PLC the position of the magazine in which the tool which was on the spindle must be placed.

# 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 having 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 the idle time, thus resulting in more 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.



**CNC 8040** 

#### 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 counterclockwise 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.



**CNC 8040** 

#### 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.



**CNC 8040** 

## 11.8 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 4 (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 by SERVO1ON** 

ENABLEZ by ENABLE2 if there is no Y axis, but there are X and Z axes.



The use of mnemonics with the axis name is available from versions V9.0x and V10.0x on. If PLC programs older than this version have defined these marks as symbols, when compiling the program will issue an error on this line.

Example: DEF ENABLEX M333

#### Mnemonics using numbers 1 through 4.

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:

#### 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.



**CNC 8040** 

FAGOR

#### ENABLE1 (M5600) ENABLE2 (M5650) ENABLE3 (M5700) ENABLE4 (M5750)

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)

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)

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 0.019
- 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)

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.	0	0
If correct, it activates the output "System OK".		
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).



**CNC 8040** 

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)

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.

11.

LOGIC CNC INPUTS AND OUTPUTS

Logic outputs of the axes

FAGOR

**CNC 8040** 

## 11.9 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).



**CNC 8040** 

#### 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.



**CNC 8040** 

## 11.10 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.	0	0
If correct, it activates the output "System OK".		
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).



**CNC 8040** 

## 11.11 Logic outputs of key status

## KEYBD1 (R560) KEYBD2 (R561) KEYBD3 (R562) KEYBD4 (R563)

These registers indicate whether or not one of the keys on the keyboard or on the operator panel is pressed. When one of these keys is pressed, the corresponding bit will be set high and it will return low when the key is released.

The appendix of this manual shows, for each keyboard, the code for each key. See "Logic outputs of key status" on page 509.

Register	Bit	Key pressed	Register	Bit	Key pressed
KEYBD4	0	Handwheel x100	KEYBD4	16	Feedrate override 60%
KEYBD4	1	Handwheel x10	KEYBD4	17	Feedrate override 70%
KEYBD4	2	Handwheel x1	KEYBD4	18	Feedrate override 80%
KEYBD4	3	Jog 10000	KEYBD4	19	Feedrate override 90%
KEYBD4	4	Jog 1000	KEYBD4	20	Feedrate override 100%
KEYBD4	5	Jog 100	KEYBD4	21	Feedrate override 110%
KEYBD4	6	Jog 10	KEYBD4	22	Feedrate override 120%
KEYBD4	7	Jog 1	KEYBD4	23	
KEYBD4	8	Feedrate override 0%	KEYBD4	24	
KEYBD4	9	Feedrate override 2%	KEYBD4	25	
KEYBD4	10	Feedrate override 4%	KEYBD4	26	
KEYBD4	11	Feedrate override 10%	KEYBD4	27	
KEYBD4	12	Feedrate override 20%	KEYBD4	28	
KEYBD4	13	Feedrate override 30%	KEYBD4	29	
KEYBD4	14	Feedrate override 40%	KEYBD4	30	
KEYBD4	15	Feedrate override 50%	KEYBD4	31	



**CNC 8040** 



**CNC 8040** 

# ACCESS TO THE INTERNAL CNC VARIABLES

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.



**CNC 8040** 

## 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  $\,$ 



**CNC 8040** 

#### 12.1 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 ( $\pm 393700787$ ).

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 ( $\pm 393700787$ ).

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 ( $\pm 999999999$ ).

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).

12.

**ACCESS TO THE INTERNAL CNC VARIABLES** Variables associated with tools.



**CNC 8040** 

#### 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).



**CNC 8040** 

#### **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  ${\sf R}$  value of the indicated tool offset (n).



**CNC 8040** 



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.



**CNC 8040** 

#### Variables associated with zero offsets. 12.2

These variables are associated with the zero offset table, due to which the values that will be assigned to or read from these fields will comply with the formats established for this table.

The possible zero offsets in addition to the additive offset indicated by the PLC, are G54, G55, G56, G57, G58 and G59.

The values are given in the units set by g.m.p. INCHES.

If, INCHES = 0, in ten-thousandths of a millimeter ( $\pm 999999999$ ).

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.

**ACCESS TO THE INTERNAL CNC VARIABLES** 

**FAGO** 

**CNC 8040** 

## 12.3 Variables associated with machine parameters

These variables associated with machine parameters are read-only variables.

Refer to the installation and start-up manual to know the format of the values returned. The values of 1/0 correspond to the parameters that are set as YES/NO, +/- or ON/OFF.

Values regarding position and feedrate values will be given in the units set by g.m.p. INCHES.

If, INCHES = 0, in ten-thousandths of a millimeter (±999999999).

If INCHES = 1, in hundred-thousandths of an inch ( $\pm 393700787$ ).

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.
1	[RESET] must be pressed.
	The rest of the parameters (those unmarked) will be updated automatically, only by changing them.

## Read-only variables

MPGn Returns

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.

CNC 8040

#### Variables associated with work zones 12.4

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 ( $\pm 999999999$ ).

If INCHES = 1, in hundred-thousandths of an inch ( $\pm 393700787$ ).

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). Upper limit of zone 1 along the selected axis (X-C). FZUP(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>
                               = CNCWR(R1, FZLOX, M1)
            = MOV 0 R1
             = MOV 1000000 R1
                               = CNCWR(R1, FZUPX, M1)
                               = CNCWR(R1, FZONE, M1)
             = MOV 1 R1
```

**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.

Lower limit of zone 4 along the selected axis (X-C). FOZLO(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). **ACCESS TO THE INTERNAL CNC VARIABLES** 

**FAGOR** 

**CNC 8040** 

#### Variables associated with feedrates 12.5

## 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.

Variables associated with feedrates



**CNC 8040** 

**ACCESS TO THE INTERNAL CNC VARIABLES** 

## 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

 $\boldsymbol{0}$  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.

FAGOR

**CNC 8040** 

#### Variables associated with coordinates 12.6

The values are given in the units set by g.m.p. INCHES.

If, INCHES = 0, in ten-thousandths of a millimeter ( $\pm 999999999$ ).

If INCHES = 1, in hundred-thousandths of an inch ( $\pm 393700787$ ).

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 ±360° (ORG\* ± 360º).

If ORG\* = 20º it displays between 20º and 380º / displays between -340º

and 20º

If ORG\* = -60° it displays between -60° and 300° / 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 ±360° (ORG\* ± 360º).

If ORG\* = 20° it displays between 20° and 380° / displays between -340°

and 20º.

If ORG\* = -60° it displays between -60° and 300° / 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.

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.

Variables associated with coordinates



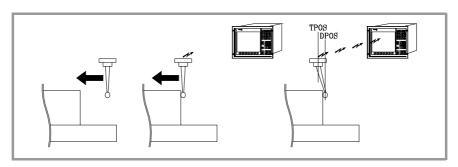
**CNC 8040** 

ATPOS(X-C)

#### DPOS(X-C)

The CNC updates this variable whenever probing operations "G75, G76" and probing cycles "PROBE 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).

Actual position of the probe when the CNC receives the probe TPOS(X-C) 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).



**FAGO**I

**CNC 8040** 

## 12.7 Variables associated with electronic handwheels

## Read-only variables

HANPF HANPS HANPT HANPFO They return the pulses of the first (HANPF), second (HANPS), third (HANPT) or fourth (HANPFO) handwheel received since the CNC was turned on. 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  $\cdot 0 \cdot$  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.

		С			В			Α			W			٧			U			Z			Υ			Χ		
C	С	b	t o	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 t

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 t

1 1 1 x1 factor

1 1 0 x10 factor



**HBEVAR** 

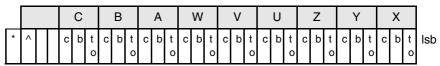
The screen always shows the value selected at the switch.

FAGOR 🚄

It must be used when having a Fagor HBE handwheel.

**CNC 8040** 

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).



(\*) 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.

С	b	t
		•

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:

С	b	t o	
1	1	1	x1 factor
1	1	0	x10 factor

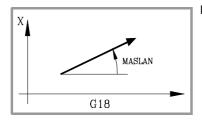
The HBE handwheel has priority. That is, regardless of the mode selected at the CNC switch (continuous or incremental JOG, handwheel), HBEVAR is set to other than "0", the CNC goes into handwheel mode.

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 "6.15 Fagor handwheels: HBA, HBE and LGB" on page 255.

#### **MASLAN**

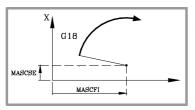
It must be used when the path-handwheel or the path-jog is selected.



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.



**CNC 8040** 

## 12.8 Variables associated with feedback

**ASIN(X-C)** "A" signal of the CNC's sinusoidal feedback for the X-C axis.

**BSIN(X-C)** "B" signal of the CNC's sinusoidal feedback for the X-C axis.

**ASINS** "A" signal of the CNC's sinusoidal feedback for the spindle.

**BSINS** "B" signal of the CNC's sinusoidal feedback for the spindle.

**SASINS** "A" signal of the CNC sinusoidal feedback for the second spindle.

**SBSINS** "B" signal of the CNC sinusoidal feedback for the second spindle.



**CNC 8040** 

#### 12.9 Variables associated with the main spindle

## Variables associated with the real speed

**SREAL** It returns the actual (real) turning speed of the main spindle. Its value is given in

0.0001 rpm units.

**FTEOS** It returns the theoretical turning speed of the main spindle.

## Variables associated with spindle speed

PLCS is a read-write variable; the rest are read-only.

**SPEED** It returns the turning speed of the main spindle currently selected at the CNC. Its value

is given in 0.0001 rpm units.

This turning speed may be indicated by program, by PLC or by DNC; the CNC selects one of them, the one indicated by DNC has the highest priority and the one indicated

by program has the lowest priority.

**DNCS** It returns the spindle speed limit selected via DNC. If it has a value of 0 it means that

it is not selected.

**PLCS** It returns the spindle speed limit selected via PLC. If it has a value of 0 it means that

it is not selected.

**PRGS** It returns the spindle speed limit selected by programa. If it has a value of 0 it means

that it is not selected.

## Variables associated with constant cutting speed (lathe model)

PLCCSS is a read-write variable, the rest are read-only.

**CSS** It returns the constant surface speed selected at the CNC.

> This constant surface speed may be indicated by program, by PLC or by DNC; the CNC selects one of them, the one indicated by DNC has the highest priority and the one indicated by program has the lowest priority.

The values are given in the units set by g.m.p. INCHES.

If INCHES = 0, in  $m/min (\pm 999999999)$ . If INCHES = 1, in ft/min ( $\pm 393700787$ ).

**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

**PRGCSS** It returns the constant surface speed selected by program. Its value is given in m/min

or ft/min.

**ACCESS TO THE INTERNAL CNC VARIABLES** 

**CNC 8040** 

## 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.

> 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.

> Maximum machining spindle speed. This variable is also updated (refreshed) when programming function G92 via MDI.

## Position related variables

**POSS** It returns the real position of the main spindle. Its value is given in 0.0001 degree units within ±999999999.

> It returns the real position of the main spindle in 360° module. Its value is given in 0.0001 degree units within 0 and 360°.

It returns the theoretical position of the main spindle (real position + lag). Its value is given in 0.0001 degree units within ±9999999999.

It returns the theoretical position of the main spindle (real position + lag) in 360° module. Its value is given in 0.0001 degree units within 0 and 360°.

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.

**PLCSL** 

**MDISL** 

**RPOSS** 

**TPOSS** 

**RTPOSS** 

**DRPOS** 



**CNC 8040** 

**FLWES** 

Following error (lag) of the main spindle. Its value is given in 0.0001 degree units within  $\pm 999999999^{\circ}$ .

**SYNCER** 

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".

12.

ACCESS TO THE INTERNAL CNC VARIABLES

Variables associated with the main spindle



**CNC 8040** 

## 12.10 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 ( $\pm 999999999$ ). If INCHES = 1, in ft/min ( $\pm 393700787$ ).

**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 second spindle



**CNC 8040** 

**ACCESS TO THE INTERNAL CNC VARIABLES** 

## 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 ±999999999.

**SRPOSS** It returns the real position of the second spindle in 360° module. Its value is given in 0.0001 degree units within 0 and 360°.

**STPOSS** It returns the theoretical position of the second spindle (real position + lag). Its value is given in 0.0001 degree units within ±9999999999.

**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 360°.

**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.

**CNC 8040** 

## 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  $\pm 999999999^{\circ}$ .

## 12.

ACCESS TO THE INTERNAL CNC VARIABLES

Variables associated with the second spindle



**CNC 8040** 

## 12.11 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.

**12.** 

**ACCESS TO THE INTERNAL CNC VARIABLES** Variables associated with the live tool



**CNC 8040** 

## 12.12 Variables associated with local and global parameters

The CNC offers two types of general purpose variables, local parameters P0 through P25 and global parameters P100 through P299.

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 allows reading or modifying the indicated global parameter (n) (P100-P299).

```
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.
```



**CNC 8040** 

## 12.13 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.

Sercos variables



**CNC 8040** 

## 12.14 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.

Bit	Meanin	ng
0	Turbo bo	pard.
4, 3, 2, 1	0100	8040 model
5	Sercos (	digital model).
6	Reserve	d.
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 boa	ard with 12-bit (=0) or 16-bit (=1) Digital/Analog converter.
14, 13, 12, 11	Reserve	d.
15	It has C	AN (digital module).
18,17,16	Keyboar	d type (technical service department).
20,19	CPU typ	e (technical service department).
23,22,21	000 010 110 111	Memkey card (4 Mb).  Memkey card (24 Mb).  Memkey card (512 Kb).  Memkey card (2 Mb).
26,25,24	000 001	Color LCD Monitor. Monochrome LCD monitor.
28,27	00 01	Turbo board at 25MHz. Turbo board at 40 MHz
30	Ethernet	t.
31	Compac	t flash.

## **HARCOA**

It indicates, with bits, the CNC's hardware configuration. The bit will be "1" when the relevant configuration is available.

Bit	Meaning
0	"Axes 2" board.
1	It has a connector for compact flash.

Bit  $\cdot 1 \cdot$  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.



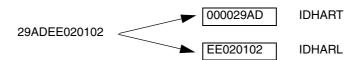
**CNC 8040** 

#### IDHARH IDHARL

They return, in BCD code, the hardware identification number corresponding to the memkey card. 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:

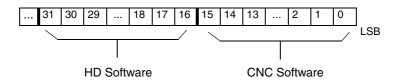


## **SOFCON**

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.



**CNC 8040** 

**FAGO** 

## 12.15 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.

## HARSWA

Bits	Board
31 - 28	
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	Turbo
3 - 0 (I SB)	CPU

#### **HARSWB**

Bits	Board
31 - 28	
27 - 24	
23 - 20	
19 - 16	
15 - 12	
11-8	Small sercos
7 -4	
3 - 0 (LSB)	HD

There may be two types of CAN boards (value of  $\cdot 0001 \cdot$  if it is an SJ1000 type and value of  $\cdot 0010 \cdot$  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		
13	Inside temperature	
12	I/O 3	(Board voltage)
11	I/O 2	(Board voltage)
10	I/O 1	(Board voltage)
9		
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)

## FAGOR

**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x) MEMTST

It returns the result of the memory test. Each data uses 4 bits that will be at "1" if the test is OK and will have a value other than "1" if there is an error.

Bits	Test
30	Test status
19 - 16	Caché

Bits	Test
15 - 12	Sdram
11-8	HD
7 -4	Flash
3 - 0 (LSB)	Ram

**NODE** It returns the number of the node assigned to 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.

They may be used to read the number of remote digital I/O available. **IOSREM** 

Bit	Meaning
0 - 15	Number of inputs.
16 - 31	Number of outputs.

**ACCESS TO THE INTERNAL CNC VARIABLES** Variables associated with telediagnosis



**CNC 8040** 

# **ACCESS TO THE INTERNAL CNC VARIABLES** Operating-mode related variables

#### **Operating-mode related variables** 12.16

#### 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.



**CNC 8040** 

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.

## 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).

21 = In graphic simulation mode.

30 = Cycle editing.

40 = In Jog mode (standard screen).

45 = In tool calibration mode.

60 = Managing parts. PPROG mode.



**CNC 8040** 

#### **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.
Bit6	Repositioning interrupted.
Bit 7	In tool inspection.
Bit 8	Block in execution via CNCEX1.
Bit 9	CNCEX1 block 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	
Dit 0	

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.

# FAGOR

**CNC 8040** 

#### 12.17 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".

**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 G	27 G26 G25
---------------------------------------	------------

**GGSC** 

It returns the status of functions G50 through G74. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G74	G73	G72	G71	G70	 G54	G53	G52	G51	G50
									1

**GGSD** 

It returns the status of functions G75 through G99. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G	99	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 G	3123 G122	G121	G120		G104	G103	G102	G101	G100	l
--------	-----------	------	------	--	------	------	------	------	------	---

Other variables

ACCESS TO THE INTERNAL CNC VARIABLES



**CNC 8040** 

Other variables

**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.

G22	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.

|--|

GGSK

It returns the status of functions G250 through G274. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G274 G273 G272 G271 G270 G254 G253 G252 G251 G250
---

**GGSL** 

It returns the status of functions G275 through G299. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

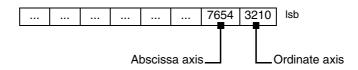
G299	G298	G297	G296	G295	 G279	G278	G277	G276	G275

GGSM

It returns the status of functions G300 through G320. The status of each one of the functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G320	G319	G318	G317	G316	 G304	G303	G302	G301	G300

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.



**CNC 8040** 

PLANE

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		
Absci	ssa axi	s	= 3 (0	011)		=> Z axis				
Ordinate axis			= 1 (0	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	LSB
		Axis 7	Axis 6	Axis 5	Axis 4	Axis 3	Axis 2	Axis 1	

The axis name corresponds to the number according to the programming order for them.

Example: If the CNC controls 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.



Other variables

**CNC 8040** 

Other variables

DATE

Returns the date in year-month-day format.

(CNCRD DATE, R101, M102); assigns the date to register R101. For example: if the date is April 25th 1992, R101 = 920425.

**CYTIME** 

It returns the amount of time (in hundredths of a second) elapsed executing the part. It ignores the time the execution has been interrupted. Possible values 0..4294967295.

The CNC will consider the execution of the program finished after executing its last block or after executing a block containing an M02 or M30 miscellaneous function.

**FIRST** 

Indicates whether it is the first time that a program has been run or not. It returns a value of 1 if it is the first time and 0 if not.

A first-time execution is considered as being one which is done:

- · After turning on the CNC.
- After pressing [SHIFT]+[RESET].
- Every time a new program is selected.

ANAIn

It returns the status in tenths of a volt (±5 V range) of the indicated analog input (n) and it is possible to select one of the 16 analog inputs (1 through 16).

In the case of analog CAN I/O, the value assigned will be expressed in 0.0001 volt units and within  $\pm 10$  V. It is possible to select one of the 16 analog inputs (1..16) available. See chapter "3 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.

	_	Pa	ir 3	Pa	ir 2	Pa	ir 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:

	Pa	ir 3	Pa	ir 2	Pa	ir 1	
					С	В	
	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  $\cdot 1 \cdot$  when a transmission is in progress.

Bit	Meaning
1	Transmission in progress through DNC1.
8	Transmission in progress through DNC2.

This variable, returns the time remaining to end the timing block in hundredths of a

(SOFT M: V11.1x) (SOFT T: V12.1x) **TIMEG** It shows the timing status of the timer programmed with G4 K in the CNC channel.

second.

**TIPPRB** 

It indicates the PROBE cycle being executed at the CNC.



**CNC 8040** 

404

**PANEDI** WGDRAW application. Number of the screen created by the user or the manufacturer

and is being consulted.

**DATEDI** WGDRAW 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.

#### Read-and-write variables

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.

> 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.

> 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.

**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x)

**TIMER** 

**PARTC** 

**KEY** 

Other variables

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.

In the case of analog CAN I/O:See chapter "3 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  $\cdot 1 \cdot$  thus selecting the first probe input. To select the second probe input, set it to a value of  $\cdot 2 \cdot$ .

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  $\cdot 1 \cdot$ , the programmed coordinates are assumed in diameter; when is set to  $\cdot 0 \cdot$ , 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  $\cdot 0 \cdot$ .

**PRBMOD** 

It indicates whether a probing error is to issued or not in the following cases, even if general machine parameter PROBERR (P119) =YES.

- When a G75 probing move finishes before the probe has touched part.
- When a G76 probing move finishes but the probe is still touching the part.

The PRBMOD variable takes the following values.

Value	Meaning
0	An error message is issued.
1	No error message is issued.

Default value 0.

The PRBMOD variable can be read and written from the CNC and the PLC an read from the DNC.



**CNC 8040** 

# AXES CONTROLLED FROM THE PLC

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.



**CNC 8040** 

#### **PLC** execution channel 13.1

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.

#### 13.1.1 Considerations

#### Screen editor

The a.m.p. "AXISTYPE" of each axis of the machine must be set properly indicating whether that axis is controlled by the CNC or from the PLC.

The axes of the PLC channel can only be governed from the PLC.

They may be edited and part-programs may be generated with axes of the PLC channel. This permits generating part-programs or subroutines associated with the PLC channel.

It issues an error message when trying to execute, from the CNC channel, a program block that includes a PLC axis.

When all the axes of the machine are set to be governed from the CNC, with the CNCEX action only blocks programmed in high level language may be executed through the PLC execution channel.

#### **Axis control**

To govern axes managed by PLC, use the following marks associated with "Feedhold" 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.

PLC execution channel



**CNC 8040** 

#### 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.



**CNC 8040** 

#### 13.1.2 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:

J	
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.

PLC execution channel



**CNC 8040** 

#### 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 "Feedhold" 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.



**CNC 8040** 

PLC execution channel

## **FAGOR**

**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x)

#### 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
- In that case, it is limited to 100%. In G75, G76, it is only applied when g.m.p. FOVRG75 (P126) = YES.

#### Blocks programmed in high-level language

The high-level instructions that can be used in the PLC execution channel are:

(IF condition <action1> ELSE <action2>)

(CALL (expression))

CNCEX ((CALL 100), M1000)

Sends the (CALL 100) command to the CNC so it executes (calls) subroutine 100

CNCEX ((P100=P100+2), M1000)

Sends the (P100=P100+2) command to the CNC to increment the value of parameter P100 in 2 units.

Programming high-level blocks has the following restrictions:

- The programmed blocks can only work with global parameters.
- Up to 5 nesting levels of standard subroutines are allowed (neither parametric nor global).

#### Example in mm:

Move the W axis to the coordinate indicated by register R101.

When the PLC works with integers (32 bits), the value of register R2 is given in tenths of microns (0.0001 mm).

CNCWR (R101, GUP 155, M101)

Assigns the value indicated in R101 to global parameter P155.

CNCEX ((P155=P155/10000), M101)

Converts the value of P155 into mm.

CNCEX (G1 WP155 F2000, M101)

Movement of the W axis

#### Interrupt block preparation

Same as in the CNC channel, blocks are also prepared in advance in the PLC

```
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>)
  18 is checked after moving the axis.
```

#### **Auxiliary M functions**

The M functions programmed in the PLC channel may be defined in the M function

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.
```



**FAGO** 

**CNC 8040** 

#### 13.1.3 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 partprogram 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)



**CNC 8040** 

**AXES CONTROLLED FROM THE PLC** PLC execution channel

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.

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.



**CNC 8040** 

#### 13.2 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.



**CNC 8040** 

#### Synchronize a PLC axis with a CNC axis 13.3

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.

**AXES CONTROLLED FROM THE PLC** 



**CNC 8040** 

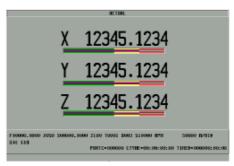


**CNC 8040** 

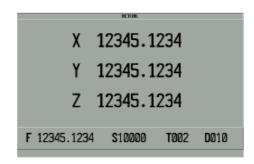
#### **CUSTOMIZABLE SCREENS**

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 Wgdraw 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.



**CNC 8040** 

#### 14.1 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:

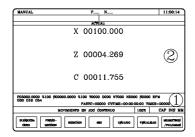
The configuration file is a CNC program edited in high level language (configuration language) which is described later on. It may be edited both at the CNC and at a PC.

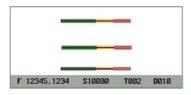
It may be in the user RAM and in the CARDA. If it is in both places, the one located in user RAM will be used. It is recommended to store it only in CARDA once it has been fully debugged.

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

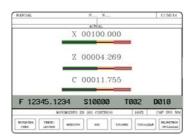




Standard screen

OEM screen

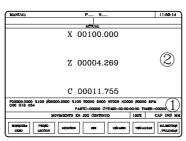
Both screens are overlapped; but "Disable 1" indicates that the area 1 of the standard screen is not displayed. Therefore:



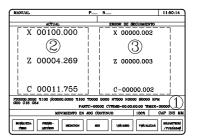
Next, all the screens that may be customized will be shown and the nomenclature to be used in the configuration file. Numbers 1, 2, 3, 4 and 5 indicate the areas each screen is divided into.



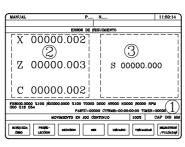
**CNC 8040** 

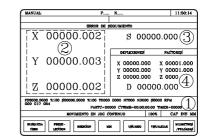


[JOG] JOG mode - Actual

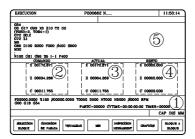


[JOGAFL] JOG mode - Actual and Following error



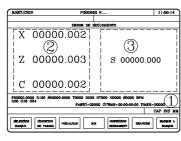


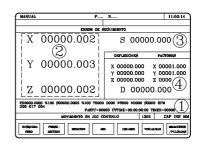
[JOGFLW] JOG mode - Following error



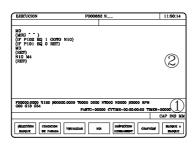
[STD] Execution mode - Standard

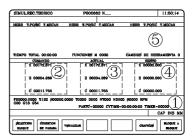
[POS] Execution mode - Position





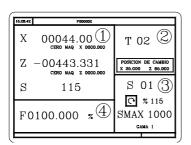
[FLW] Execution mode - Following error

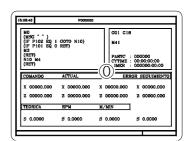




[PRG] Execution mode - Program

[SUB]
Execution mode - Subroutines





[STDCONV] Conversational mode - Standard

[AUXCONV] Conversational mode - Auxiliary for execution

CNC 8040

**FAGOR** 

#### 14.2 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)
;;----- \ensuremath{\mathbf{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)
```



CUSTOMIZABLE SCREENS
Configuration language



**CNC 8040** 

#### 14.3 Key words

#### ;(PRGSCRIPT 1)

Header of the configuration file and version used to edit it (in this case "1"). It must always be defined.

#### ;[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.

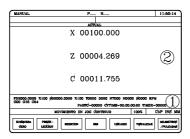
It is possible to define as many DISABLE instructions as screen areas are to be eliminated.

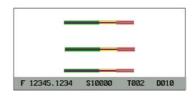
To eliminate the whole standard screen, program (DISABLE 0). In this case, only the OEM screen will be displayed.



**CNC 8040** 

#### Examples:





Standard screen

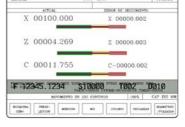
OEM screen

Without "Disable"

Both screens overlap.

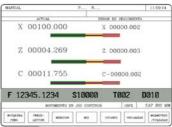
There are areas with information overlapped.

In this case, area 1.



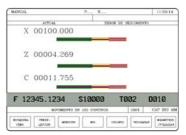
With (DISABLE 1)

Area 1 of the standard screen is not displayed.



With (DISABLE 1) and (DISABLE 3)

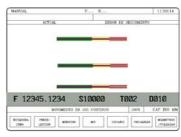
Areas 1 and 3 of the standard display are not displayed.



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 WGDRAW 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.



**CNC 8040** 

#### ;(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.

14.

CUSTOMIZABLE SCREENS
Key words

FAGOR

**CNC 8040** 

**CUSTOMIZABLE SCREENS** 

#### ;(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)



**CNC 8040** 

```
;(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

FAGOR 🚄

**CNC 8040** 

```
; (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)



**CNC 8040** 

; (END)

End of debug, end of the portion of the configuration file corresponding to the screen and end of the configuration file.



**CNC 8040** 

#### 14.5 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



**CNC 8040** 

#### CONFIGURABLE WORK MODE

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.

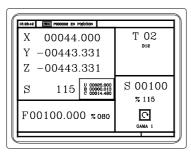
Since all the OEM screens have a subroutine associated, they can also create OEM cycles to:

- · Consult inputs and outputs.
- · Adjust the machine axes.
- · Manage tool magazines.
- · Manage external devices.
- etc.

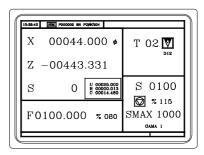
The OEM machining canned cycles may even be used to machine parts. A machining operation may be repeated as often as you wish, but it cannot be stored in memory.

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:







T model

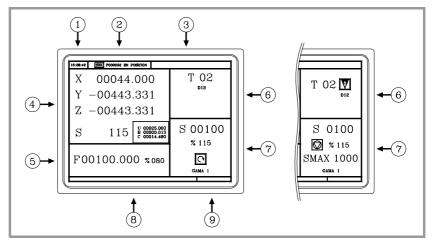
The way to operate with either model is very similar. If any of the features described here is not common to both models, it will clearly indicate which model it corresponds to.

These screens may be customized entirely or by areas. See "14.1 Configuration file" on page 420.



**CNC 8040** 

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 in a frame) 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. See Wgdraw manual.
- 9. Reserved.



**CNC 8040** 

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.

# Preset

It must be done on one axis at a time and proceeding as follows:



The CNC requests confirmation of the command.

# Axis feedrate (F)

To set the axis feedrate, key in:



# 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:



CONFIGURABLE WORK MODE



**CNC 8040** 

CONFIGURABLE WORK MODE

Tool control

#### **Tool control** 15.2

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



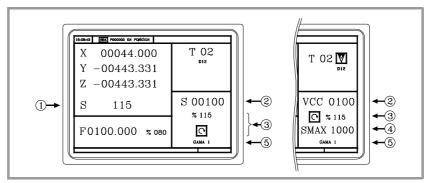
The CNC temporarily assumes the new offset for the current tool. The internal table is not modified, the tool's associated offset is still the one assigned to it when it was calibrated.



**CNC 8040** 

#### **Spindle control** 15.3

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).

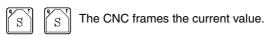
To select another speed, press:



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:

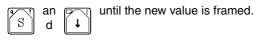


spindle to exceed these rpm. 5. Spindle gear currently selected.

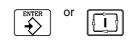
This value cannot be changed when using an automatic gear change.

To change gears, press

Value



Gear number



The CNC assumes this value and does not allow the

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 or program 999997 has not been defined.





**CNC 8040** 

# 15.4 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.

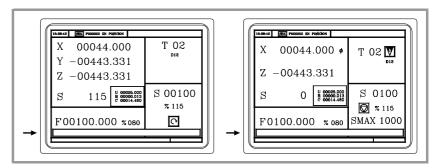
( ) = MOV 0 R100 = MOV 1 R101 = MOV \$F01E R102 Initializations.

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.



**CNC 8040** 

# 15.5 Screens, subroutines and cycles

User Fagor's Wgdraw (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. See Wgdraw manual.
- Program P999994 that contains the help texts used by all the screens. See Wgdraw 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 WGDRAW 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 Wgdraw 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.



**CNC 8040** 

#### **Associated keys** 15.6

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				\$	Screen	s			
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.



CONFIGURABLE WORK MODE
Associated keys



**CNC 8040** 

CONFIGURABLE WORK MODE

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.



**CNC 8040** 

# 15.7 OEM text in several languages.

**PLCERR** 

By default, the texts defined by the OEM are in a single language and stored in several programs:

PLCMSG Texts for PLC messages.

P999995 Texts and titles used by all the OEM screens.

P999994 Help texts of the OEM screens or cycles.

Texts for PLC errors.

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 memkey card. If it is in both places, it takes the one in user memory.

# **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

The text groups may be defined in the desired order, grouping them by subjects, languages, etc.



**CNC 8040** 

# **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

The texts of the [OEMMSG] are directed to:

;2 \$Text 2

• Texts of the user channel used with the program control instructions MSG, ERR, IB, SK that depend on the selected work language.

(MSG "HELLO") shows the text HELLO on the screen.

(MSG 200) displays the text 200 of the [OEMMSG] group in the

currently selected language.

(MSG \$C8) is 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 Wgdraw application.

The texts used by the application are created in the Wgdraw.txt program and must be copied to the CNC as program P999995 or as part of the MSGFILE program within the [CO999995] group.

**15.** 

CONFIGURABLE WORK MODE OEM text in several languages.



**CNC 8040** 



**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x) 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.

CONFIGURABLE WORK MODE

# 15.8 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 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.

Pogram that logs (stores) the errors occurred when interpreting a configuration file.

Help texts of the OEM screens or cycles. See Wgdraw manual.

P999999 Texts and titles used by all the OEM screens. See Wgdraw manual.

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.



**CNC 8040** 

# CONFIGURABLE WORK MODE Associated subroutines

# 15.9 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 field that may be edited, B (P1) that of the 2nd field and so on up to Z (P25) that indicates the value of the 26th field. The order is set when designing the screen with Wgdraw.

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.



**CNC 8040** 

# 15.10 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.

Configuration file



**CNC 8040** 

# ;(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 9300 + cycle number cycle is

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.

Configuration file



**CNC 8040** 

# ;(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 grey 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)

FAGOR 🚄

**CNC 8040** 

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 477.

# ;(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)

Configuration file



**CNC 8040** 

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.

**15.** 

CONFIGURABLE WORK MODE

Configuration file



**CNC 8040** 

# 15.11 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, the 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.

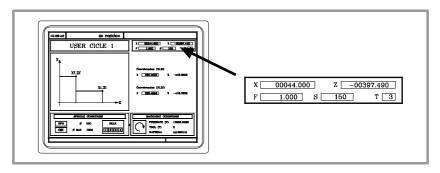


**CNC 8040** 

CONFIGURABLE WORK MODE

# 15.12 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 Wgdraw).

> Χ 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,





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 125 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.

**FAGO** 

**CNC 8040** 

# Select an option among the ones available

It is used to select a button in a group of Wgdraw buttons.



Use the keys





# Select one of the representations in an icon

It is used to select an icon in a multiple representation of Wgdraw.



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 [CYCLE START] key to call the associated subroutine (9001) to execute the cycle.



**CNC 8040** 

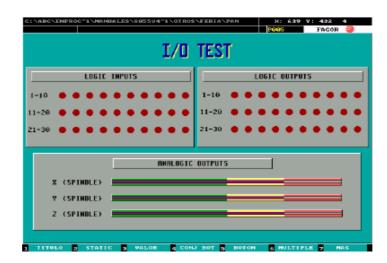
# 15.13 Example. Consult inputs and outputs

Screen 005 is used. 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.

15.

**CONFIGURABLE WORK MODE**Example. Consult inputs and outputs

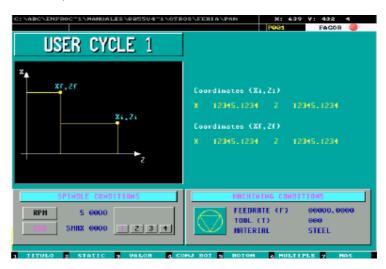


**CNC 8040** 

# 15.14 Example. Machining canned cycle

Screen 001 is used. 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.



**CNC 8040** 

# PLC PROGRAMMING EXAMPLE

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.



**CNC 8040** 

# PLC PROGRAMMING EXAMPLE Definition of symbols (mnemonics)

# 16.1 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 are not allowed; but, one resource may have more than one symbol.

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	170	Conditional mode. The CNC interrupts part-program execution when executing auxiliary function M01
DEF	SERVO-OK	l71	The servo drives are O.K.
DEF	O-EMERG	01	Emergency output. It must be normally high.

# Used in: Treatment of the axis travel limit switches.

DEF	I-LIMTX1	172	X axis positive overtravel limit switch
DEF	I-LIMTX2	173	X axis negative overtravel limit switch
DEF	I-LIMTY1	174	Y axis positive overtravel limit switch
DEF	I-LIMTY2	175	Y axis negative overtravel limit switch
DEF	I-LIMTZ1	176	Z axis positive overtravel limit switch
DEF	I-LIMTZ2	177	Z axis negative overtravel limit switch

# Used in: Treatment of the machine reference (home) switches.

DEF	I-REF0X	178	X axis home switch
DEF	I-REF0Y	179	Y axis home switch
DEF	I-REF0Z	180	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	02	Ways lubrication output

# FAGOR 🚄

**CNC 8040** 

# Used in: Coolant treatment.

DEF	I-COOLMA	182	The operator control the coolant. Manual mode.
DEF	I-COOLAU	183	The CNC controls the coolant. Automatic mode.
DEF	O-COOL	О3	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	184	Indicates that Gear 1 is selected
DEF	I-GEAR2	185	Indicates that Gear 2 is selected

# Used in: Keyboard simulation.

DEF	I-SIMULA	186	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
		ΨΟ.	Code of the Finey
DEF	KEY2	\$32	Code of the "2" key
DEF DEF		* -	•
	KEY2	\$32	Code of the "2" key



**CNC 8040** 

( ) = 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** 



PLC PROGRAMMING EXAMPLE
First cycle module.



**CNC 8040** 

PLC PROGRAMMING EXAMPLE

**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.



**CNC 8040** 



**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x)

#### --- 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 MSG128,. 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 ERR64 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

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.

Main module.



**CNC 8040** 

# ---- 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 analog signal to change gears.
- Move the gears.
- Verify that the gear change has been completed.
- Remove the oscillating analog signal.
- 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 analog signal to change gears.

#### DFU M-41 OR DFU M-42

When a range (gear) change is requested...

#### = MOV 2000 SANALOG

... A 0.610V analog signal 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 analog signal.

Return the control of the spindle back to the CNC.



**CNC 8040** 

# (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 analog voltage,...

#### = 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).

Manual lubrication.

This operation will last 5 minutes and it will be performed at operator's request.

#### **DFU I-LUBING = TG1 3 300000**

Whenever the operator requests the lubricating (lubing) operation, T3 must be activated in the mono-stable mode for 5 minutes (300000 milliseconds).

Lubrication every hour of axis motion.

This operation takes place when the axes of the machine have been moving for an accumulated time period of 1 hour. They will be lubricated for 5 minutes.

Timer T4 is used to keep track of the axis accumulated moving time and T5 to time the 5 minute lubrication period.

The first cycle module CY1 must activate timer T4 in the delayed activation mode with a time constant of 1 hour (3600 000 milliseconds).

() = TG2 4 3600000

# **ENABLE1 OR ENABLE2 OR ENABLE3 = TEN 4**

T4 only times when any of the axis is moving.

Main module.



**CNC 8040** 

#### 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".



**CNC 8040** 

#### ---- Coolant treatment ----

The CNC executes function M08 to turn the coolant on and function M09 to turn it off.

Also, in this case, the operator has a switch to select whether the coolant is activated manually by the operator or automatically by the CNC.

I-REFMAN The operator control the coolant. Manual mode.

I-REFAUT The operator control the coolant. Automatic mode.

O-REFRIG Coolant on/off output.

#### I-COOLMA OR (I-COOLAU AND M-08) = O-COOL

Coolant ON.

#### RESETOUT = NOT O-COOL = RES M-08

The coolant will be turned off when the CNC is reset to initial conditions (RESETOUT) or when executing functions M00, M02, M09 and M30.

This instruction does not contemplate functions M00, M02, M09 and M30 since the treatment of M, S, T functions turns mark M-08 off when activating any of them.

# ---- Treatment of the general CNC input AUXEND -----

It is advisable to have one single instruction to control each one of the logic CNC inputs, thus preventing undesired functioning.

When having several instructions which can activate or deactivate an input, the PLC will always assign the result of analyzing the last one of those instructions.

This example shows how to group in a single instruction all the conditions that activate or deactivate one logic CNC input.

# NOT T1 AND NOT M-41 AND NOT M-42 = AUXEND

Input AUXEND will remain low while:

- The "Treatment of the MSTROBE, TSTROBE, STROBE signals" is in progress (timer T1 active)
- A spindle gear change is being performed (M-41, M-42)

# ---- Keyboard simulation -----

With this example it is possible to simulate the theoretical path of part-program P12 whenever the operator requests it.

To do this, follow these steps:

- Indicate to the CNC that from now on the keys will come from the PLC.
- Simulate all the necessary steps sending the code of each one of the keys.
- Indicate to the CNC that from now on the keys will be coming from the CNC keyboard, not from the PLC.

In order to make sending the keys easier, a subroutine is used which utilizes the following parameters:

ENVIATEC (Send Key) Calling parameter that must be activated whenever a key

is to be sent.

CODTECLA (Code of the key) Calling parameter that must contain the code corresponding to the key being simulated.

ENVIOK (Sent OK) Outgoing parameter indicating that the key code has been sent successfully.



**CNC 8040** 

# 16-

AMMING EXAMPLE
Main module.



**CNC 8040** 

(SOFT M: V11.1x) (SOFT T: V12.1x)

#### 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

 $\dots$  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 program.



**CNC 8040** 



**CNC 8040** 

# APPENDIX

Α.	Technical specifications of the CNC	471
В.	Probe connection	475
C.	Summary of internal CNC variables	477
D.	Summary of PLC commands	485
E.	Summary of PLC inputs and outputs	489
F.	2-digit BCD code output conversion table	495
G.	Key codes	497
н.	Logic outputs of key status	509
I.	Key inhibiting codes	521
J.	Machine parameter setting chart	533
K.	M functions setting chart	545
L.	Leadscrew error compensation table	547
M.	Cross compensation table	549
N	Maintenance	551



**CNC 8040** 

# APPENDIX Technical specifications of the CNC

# **TECHNICAL SPECIFICATIONS OF THE 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.

Access to the inside of the unit is absolutely forbidden to non authorized 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 Vdc)

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.

Block processing time of 6.5 ms.

Configurable sample time: 2, 3, 4, 5 or 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.



**CNC 8040** 

#### Monochrome monitor

Technology: LCD STN.

Diagonal display area dimension: 10,4".

Resolution: 640 x 480 pixels.

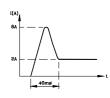
8 grey ranges.

Backlit with 1 cold-cathode fluorescent lamp.



Due to the current state of the LCD technology, all manufacturers accept the fact the LCD screens have a certain number of defective pixels.

#### Power supply



Nominal voltage: 20 V minimum and 30 V maximum.

Ripple: 4 V.

Nominal current: 2 A.

Current peak on power-up: 8 A.

The figure shows the shape of the supply current on power-up

#### PLC

Memory: 100 kbytes.

Programming in mnemonics.

1 millisecond time unit.

512 inputs.

512 outputs.

2047 user marks.

256 32-bit registers.

256 32-bit counters.

256 32-bit timers.

#### 5V probe input.

Typical value 0.25 mA.  $\cong$  Vin = 5V.

High threshold (logic level 1) VIH: from +2.4 Vdc up.

Low threshold (logic level 0) VIL: Below +0.9 Vdc

Maximum nominal voltage Vimax = +15 Vcc.

#### 24V probe input.

Typical value 0.30 mA.  $\cong$  Vin = 24V.

High threshold (logic level 1) VIH: from +12.5 Vdc up.

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: from +18 Vdc up.

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.



**CNC 8040** 

#### 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 K $\Omega$ .

Shielded cable should be used.

#### Ambient conditions

Relative humidity: 30-95 % non condensing

Operating temperature: between +5 °C (41 °F) and +40 °C (104 °F) with an average

under +35 °C (95 °F).

Storage temperature: between -25  $^{\circ}$ C (-13  $^{\circ}$ F) and +70  $^{\circ}$ C (158  $^{\circ}$ F). Maximum operating 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 the enclosure: 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.



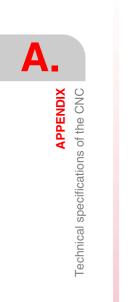
Neither attempt to recharge the battery nor expose it to temperatures over 100  $^{\circ}$ C (212  $^{\circ}$ F).

Do not short-circuit the terminals for risk of explosion or combustion.





**CNC 8040** 





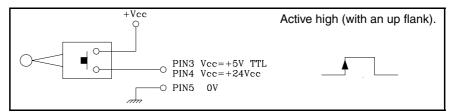
**CNC 8040** 

# PROBE CONNECTION

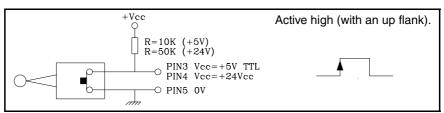
The CNC has two probe inputs (of 5Vdc and 24Vdc) at connector X3.

Depending on the type of connection applied the g.m.p. "PRBPULSE" (P39) must be set, indicating whether it operates with the leading edge or trailing edge of the signal which the probe provides.

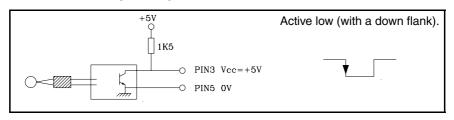
#### Probe with "normally open contact" output,



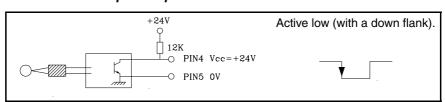
# Probe with "normally closed contact" output.



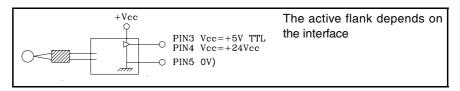
#### Interface with output in open collector Connection to +5 V.



#### Interface with output in open collector Connection to +24 V.



#### Interface with output in PUSH-PULL





APPENDIX
Probe connection



**CNC 8040** 



APPENDIX
Probe connection



**CNC 8040** 

# 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	12.1)
TOOL	R	R	R	Number of 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).		

#### Tool related variables (specific of the mill model).

TORn	R/W	R/W	-	Tool radius value of offset (n)
TOLn	R/W	R/W	-	Tool length value of offset (n)
TOIn	R/W	R/W	-	Tool radius wear of offset (n).
TOKn	R/W	R/W	-	Tool length wear of offset (n).

#### Tool related variables (specific of the lathe model).

TOXn	R/W	R/W	-	Tool length offset (n) along X axis.
TOZn	R/W	R/W	-	Tool length offset (n) along Z axis.
TOFn	R/W	R/W	-	Location code of offset (n).
TORn	R/W	R/W	-	Tool radius value of offset (n)
TOIn	R/W	R/W	-	Tool length wear of offset (n) along X axis.
TOKn	R/W	R/W	-	Tool length wear of offset (n) along Z axis.
NOSEAn:	R/W	R/W	-	Cutter angle of indicated tool.
NOSEWn	R/W	R/W	-	Width of indicated tool.
CUTAn	R/W	R/W	-	Cutting angle of indicated tool.

#### Variables associated with zero offsets..

Variable	CNC	PLC	DNC	( section 12.2)
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 in this value.
PORGF	R	-	R	Abscissa coordinate value of polar origin.
PORGS	R	-	R	Ordinate coordinate value of polar origin.
ORG(X-C)n	R/W	R/W	R	Zero offset (n) value of the selected axis.
PLCOF(X-C)	R/W	R/W	R	Value of the additive Zero Offset activated via PLC.
ADIOF(X-C)	R	R	R	Value for the selected axis of the zero offset with additive handwheel.

# Variables associated with machine parameters.

Variable	CNC	PLC	DNC	( section 12.3)
MPGn	R	R	-	Value assigned to general machine parameter (n).
MP(X-C)n	R	R	-	Value assigned to axis machine parameter (n) (X-C)
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.



**CNC 8040** 

#### Variables associated with work zones.

Variable	CNC	PLC	DNC		( section	12.4)
FZONE	R	R/W	R	Status of work zone 1.		
FZLO(X-C)	R	R/W	R	Work zone 1. Lower limit along the selected axis (X/C).		l
FZUP(X-C)	R	R/W	R	Work zone 1. Upper limit along the selected axis (X/C).		I
SZONE	R	R/W	R	Status of work zone 2.		l
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	12.5)
FREAL	R	R	R	Real feedrate of the CNC in mm/min or inch/min.		
FREAL(X-C)	R	R	R	Actual (real) CNC feedrate of the selected axis.		
FTEO/X-C)	R	R	R	Theoretical CNC feedrate of the selected axis.		

#### Variables associated with function G94.

FEED	R	R	R	Active feedrate at the CNC in mm/min or inch/min.
DNCF	R	R	R/W	Feedrate selected via DNC.
PLCF	R	R/W	R	Feedrate selected via PLC.
PRGF	R	R	R	Feedrate selected by program.

#### Variables associated with function G95.

FPREV	R	R	R	Active feedrate at CNC, in m/rev or inch/rev.
DNCFPR	R	R	R/W	Feedrate selected via DNC.
PLCFPR	R	R/W	R	Feedrate selected via PLC.
PRGFPR	R	R	R	Feedrate selected by program.

#### Variables associated with function G32.

PRGFIN	R	R	R	Feedrate selected by program, in 1/min.
--------	---	---	---	---

#### Variables associated with feedrate override (%)

FRO	R	R	R	Feedrate Override (%) active at the CNC.
PRGFRO	R/W	R	R	Override (%) selected by program.
DNCFRO	R	R	R/W	Override (%) selected via DNC.
PLCFRO	R	R/W	R	Override (%) selected via PLC.
CNCFRO	R	R	R	Override (%) selected from the front panel knob.
PLCCFR	R	R/W	R	Override (%) of the PLC execution channel.



**CNC 8040** 

#### Coordinate related variables.

Variable	CNC	PLC	DNC	( section 12.6)		
PPOS(X-C)	R	-	-	Programmed theoretical position value (coordinate).		
POS(X-C)	R	R	R	Machine coordinates. Real coordinates of the tool base.		
TPOS(X-C)	R	R	R	Machine coordinates. Theoretical coordinates of the tool base.		
APOS(X-C)	R	R	R	Part coordinates. Real coordinates of the tool base.		
ATPOS(X-C)	R	R	R	Part coordinates. Theoretical coordinates of the tool base.		
DPOS(X-C)	R	R	R	Theoretical position of the probe when the probe touched the part.		
FLWE(X-C)	R	R	R	Following error of the indicated axis.		
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 travelled 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 12.7)			
HANPF	R	R	-	Pulses received from 1st handwheel since the CNC was turned on.			
HANPS	R	R	-	Pulses received from 2nd handwheel since the CNC was turned on.			
HANPT	R	R	-	Pulses received from 3rd handwheel since the CNC was turned on.			
HANPFO	R	R	-	Pulses received from 4th handwheel since the CNC was turned on.			
HANDSE	R	R		For handwheels with a selector button, it indicates whether that button has			
				been pressed or not.			
HANFCT	R	R/W	R	Multiplying factor different for each handwheel (when having several).			
HBEVAR	R	R/W	R	HBE handwheel. Reading enabled, axis being jogged and multiplying factor			
				(x1, x10, x100).			
MASLAN	R/W	R/W	R/W	Linear path angle for "Path handwheel" or "Path Jog" mode.			
MASCFI	R/W	R/W	R/W	Arc center coordinates for "Path handwheel mode" or "Path jog".			
MASCSE	R/W	R/W	R/W	Arc center coordinates for "Path handwheel mode" or "Path jog".			

#### Variables associated with feedback.

Variable	CNC	PLC	DNC	( section 12.8)	
ASIN(X-C)	R	R	R	A signal of the CNC's sinusoidal feedback for the selected axis.	
BSIN(X-C)	R	R	R	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 12.9 )
SREAL	R	R	R	Real spindle speed	
FTEOS	R	R	R	Theoretical spindle speed.	

# Variables associated with spindle speed.

SPEED	R	R	R	Active spindle speed at the CNC.
DNCS	R	R	R/W	Spindle speed selected via DNC.
PLCS	R	R/W	R	Spindle speed selected via PLC.
PRGS	R	R	R	Spindle speed selected by program.

# Variables associated with constant cutting speed (lathe model).

CSS	R	R	R	Constant surface speed active at the CNC.
DNCCSS	R	R	R/W	Constant surface speed selected via DNC.
PLCCSS	R	R/W	R	Constant surface speed selected via PLC.
PRGCSS	R	R	R	Constant surface speed selected by program.



**CNC 8040** 

#### Variables associated with the spindle override.

SSO	R	R	R	Spindle Speed Override (%) active at the CNC.	
PRGSSO	R/W	R	R	R Override (%) selected by program.	
DNCSSO	R	R	R/W	Override (%) selected via DNC.	
PLCSSO	R	R/W	R		
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 $\pm 9999999999$ ) and from the CNC in degrees (within $\pm 99999.9999$ ).
RPOSS	R	R	R	Real Spindle position.
				Reading from the PLC in ten-thousandths of a degree (between 0 and 3600000) and from the CNC in degrees (between 0 and 360).
TPOSS	R	R	R	Theoretical spindle position.
				Reading from the PLC in ten-thousandths of a degree (within $\pm 9999999999$ ) and from the CNC in degrees (within $\pm 99999.9999$ ).
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.

#### 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	<b>12.10</b> )
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.

S	SSO	R	R	R	Spindle Speed Override (%) active at the CNC.
SI	PRGSO	R/W	R	R	Override (%) selected by program.
SI	DNCSO	R	R	R/W	Override (%) selected via DNC.
SI	PLCSO	R	R/W	R	Override (%) selected via PLC.
S	CNCSO	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.



**CNC 8040** 

#### 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 0 and 3600000) and from the CNC in degrees (between 0 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.

Variables related to the following error.

SFLWES	R	R	R	Spindle following error.

#### Variables associated with the live tool.

Variable	CNC	PLC	DNC	( section 12.11 )
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 12.12)
GUP n	-	R/W	-	Global parameter (P100-P299) (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 12.13)
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"



**CNC 8040** 

# Software & hardware configuration variables.

Variable	CNC	PLC	DNC	( section 12.14)
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	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.

# Variables associated with telediagnosis.

Variable	CNC	PLC	DNC	( section 12.15 )
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	12.16)
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 12.17)
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.
MSn	R	-	-	Status of the indicated M function (n)
GMS	-	-	R	Status of M functions: M (06, 8, 9, 19, 30, 4144).
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.

APPENDIX
Summary of internal CNC variables.



**CNC 8040** 



Variable	CNC	PLC	DNC	( section 12.17)
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).
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.
TIPPRB	R	R	R	PROBE cycle being executed.
PANEDI	R	R	R	WGDRAW application. Number of screen being executed.
DATEDI	R	R	R	WGDRAW application. Number of element (item) being executed.



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.



**CNC 8040** 



**CNC 8040** 

# **SUMMARY OF PLC COMMANDS**

#### PLC Resources.

Inputs:	l 1/512
Outputs:	O 1/512
User marks:	M 1/2000
Arithmetic flag marks:	M 2003
Clock marks:	M 2009/2024
Fixed status marks:	M 2046/2047
Marks associated with messages:	M 4000/4127
Marks associated with errors:	M 4500/4563
Screen marks:	M 4700/4955
CNC communication marks:	M 5000/5957
Timers:	T 1/256
Counters:	C 1/256
User registers	R 1/499
Registers for communication with the CNC	R 500/559

The value stored in each register will be considered by the PLC as a signed integer which could be referred to in the following formats:

Decimal Integer within ±2147483647.

Hexadecimal Number preceded by the \$ sign and between 0 and FFFFFFF.

Binary Number preceded by the letter B and made up of up to 32 bits (1

or 0).

## Directing instructions.

PRG	Main module.
CY1	First cycle module.
PE t	Periodic module. It will be executed every t time (in milliseconds).
END	End of module.
L 1/256	Label.
DEF	Symbol definition.
REA	All consultations will be performed on real values.
IMA	All consultations will be performed on 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.



APPENDIX
Summary of PLC commands



**CNC 8040** 

**FAGOR** 

# **CNC 8040**

(SOFT M: V11.1x) (SOFT T: V12.1x)

# Simple consulting Instructions.

I	1/512	Inputs
0	1/512	Outputs
М	1/5957	Marks
Т	1/256	Timers
С	1/256	Counters
В	0/31 R 1/499	Register Bits

#### Flank detection consulting Instructions.

DFU (Up flank detection)	l 1/512
DFD (Down flank detection)	O 1/512
( )	M 1/5957

# Comparison consulting 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.

# Conditioned binary actions instructions.

= SET If the logic expression is "1", this action assigns a "1" to the resource.

= RES If the logic expression is "1", this action assigns a "0" to the resource.

= CPL If the logic expression is "1", this action complements the logic state of the resource.

= SET = RES	I 1/512 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 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.

	Code	Code	Source code	Destinati on 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

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 Right-hand register rotation.
- = RR 1/2 Left-hand register rotation.

AND	R1/559	R1/559	R1/559
OR	#	#	
XOR			

	Code	Number of repetitions	Code
RR1 RR2 RL1 RL2	R1/559	R1/559 0/31	R1/559





# Specific action instructions.

= ERA Group erase

= CNCRD CNCRD (Variable, R1/559, M1/4955)

Read internal CNC variables.

CNCWR (R1/559, Variable, M1/5957) = CNCWR

Write internal CNC variables.

= PARPAR R1/559 M1/5957

Parity of register

ERA	I	1/512	1/512	
	0	1/512	1/512	
	М	1/5957	1/5957	
	Т	1/256	1/256	
	С	1/256	1/256	
	R	1/559	1/559	



**CNC 8040** 

# **SUMMARY OF PLC INPUTS AND OUTPUTS**

# **GENERAL LOGIC INPUTS.**

/EMERGEN	M5000	Stops the axes and the spindle. Displays the error.
/STOP	M5001	Stops execution of the part program, maintaining spindle rotation.
/FEEDHOL	M5002	Stops axis feed momentarily, maintaining spindle rotation.
/XFERINH	M5003	Prevents the next block from being executed, but finishes the one being executed.
CYSTART	M5007	Starts program execution
SBLOCK	M5008	The CNC changes to single block execution mode.
MANRAPID	M5009	Selects rapid travel for all the movements that are executed in JOG mode.
OVRCAN	M5010	Selects feedrate override at 100%.
LATCHM	M5011	The axes will move from the moment the corresponding JOG key is pressed until the STOP key is pressed.
ACTGAIN2	M5013	Indicates that the CNC assumes the 2nd set of gains.
RESETIN	M5015	Initial machining conditions selected by machine parameter.
AUXEND	M5016	Indicates that the execution of the M, S and T functions has completed.
TIMERON	M5017	Enables the timer:
TREJECT	M5018	Rejection of tool in use.
PANELOFF	M5019	Deactivation of keyboard.
PLCABORT	M5022	Possibility to abort the PLC channel
PLCREADY	M5023	PLC without errors.
INT1	M5024	Executes the interruption subroutine indicated in g.m.p. P35
INT2	M5025	Executes the interruption subroutine indicated in g.m.p. P36
INT3	M5026	Executes the interruption subroutine indicated in g.m.p. P37
INT4	M5027	Executes the interruption subroutine indicated in g.m.p. P38
BLKSKIP1	M5028	The "/ and /1" block skip condition is met.
BLKSKIP2	M5029	The "/2" block skip condition is met.
BLKSKIP3	M5030	The "/3" block skip condition is met.
M01STOP	M5031	Stops execution of the part program when the auxiliary M01 function is executed.
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.
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.



FAGOR =

**CNC 8040** 

	Axis 1	Axis 3	
	Axis 2	Axis 4	
LIMIT+	M5100	M5200	Travel limit overrun. Stops the axes and the spindle. Displays the error.
	M5150	M5250	
LIMIT-*	M5101	M5201	Travel limit overrun. Stops the axes and the spindle. Displays the error.
	M5151	M5251	
DECEL*	M5102	M5202	Home switch pressed.
	M5152	M5252	
INHIBIT*	M5103	M5203	Inhibits axis movement.
	M5153	M5253	
MIRROR*	M5104	M5204	Applies mirror image.
	M5154	M5254	
SWITCH*	M5105	M5205	Swaps commands (axes with 1 drive)
	M5155	M5255	
DRO*	M5106	M5206	DRO axis. (DRO*=1 and SERVOON*=0).
	M5156	M5256	
SERVO*ON	M5107	M5207	Servo signal. (=1) closes the position loop
	M5157	M5257	
AXIS+*	M5108	M5208	Moves the axis in JOG mode. Similar to JOG keys.
	M5158	M5258	
AXIS-*	M5109	M5209	Moves the axis in JOG mode. Similar to JOG keys.
	M5159	M5259	
SPENA*	M5110	M5210	With Sercos. Speed enable signal of the drive.
	M5160	M5260	
DRENA*	M5111	M5211	With Sercos. Drive enable signal of the drive.
	M5161	M5261	
SYNCHRO*	M5112	M5212	Couples the axis with the one indicated by SYNCHRO*.
	M5162	M5262	
ELIMINA*	M5113	M5213	It does not display the axis and cancels the feedback alarms.
	M5163	M5263	
SMOTOF*	M5114	M5214	Cancels the SMOTIME filter, a.m.p. SMOTIME (P58).
	M5164	M5264	
LIM*OFF	M5115	M5215	It ignores the software limits.
	M5165	M5265	
MANINT*	M5116	M5216	Activate the additive handwheel in each axis.
	M5166	M5266	

# SPINDLE LOGIC INPUTS.



**CNC 8040** 

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.

Main		Second		
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 analog voltage. 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	Analog voltage of 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



**CNC 8040** 



**CNC 8040** 

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.		
MANUAL	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. Mill 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).		
		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.		

# **AXIS LOGIC OUTPUTS.**

	Axis 1 Axis 2	Axis 3 Axis 4	
ENABLE*	M5600 M5650	M5700 M5750	Enables axis movement.
DIR*	M5601 M5651	M5701 M5751	Indicate axis moving direction.
REFPOIN*	M5602 M5652	M5702 M5752	Home search done.
DRSTAF*	M5603 M5653	M5703 M5753	With Sercos. They indicate servo drive status.
DRSTAS*	M5604 M5654	M5704 M5754	With Sercos. They indicate servo drive status.
ANT*	M5606 M5656	M5706 M5756	If distance < MINMOVE (P54), ANT*=1
INPOS*	M5607 M5657	M5707 M5757	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.
REVOK	M5956	REVOK2	M5981	Spindle rpm correspond to programmed speed.
INPOSS	M5957	INPOSS2	M5982	Spindle in position.



FAGOR

**CNC 8040** 

# LOGIC OUTPUTS OF THE AUXILIARY SPINDLE

DRSTAFAS	M5557	With Sercos. They indicate servo drive status.
DRSTASAS	M5556	

APPENDIX
Summary of PLC inputs and outputs

# **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	



**CNC 8040** 

# 2-DIGIT BCD CODE OUTPUT CONVERSION TABLE

			•
Programmed S	S BCD	Programmed S	S BCD
0	00	50-55	54
1	20	56-62	55
2	26	63-70	56
3	29	71-79	57
4	32	80-89	58
5	34	90-99	59
6	35	100-111	60
7	36	112-124	61
8	38	125-139	62
9	39	140-159	63
10-11	40	160-179	64
12	41	180-199	65
13	42	200-223	66
14-15	43	224-249	67
16-17	44	250-279	68
18-19	45	280-314	69
20-22	46	315-354	70
23-24	47	355-399	71
25-27	48	400-449	72
28-31	49	450-499	73
32-35	50	500-559	74
36-39	51	560-629	75
40-44	52	630-709	76
45-49	53	710-799	77

Programmed S	S BCD
800-899	78
900-999	79
1000-1119	80
1120-1249	81
1250-1399	82
1400-1599	83
1600-1799	84
1800-1999	85
2000-2239	86
2240-2499	87
2500-2799	88
2800-3149	89
3150-3549	90
3550-3999	91
4000-4499	92
4500-4999	93
5000-5599	94
5600-6299	95
6300-7099	96
7100-7999	97
8000-8999	98
9000-9999	99



**CNC 8040** 



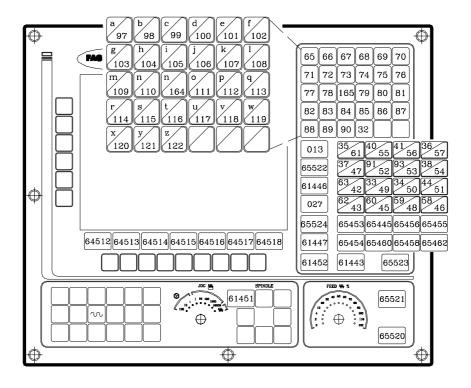
APPENDIX
2-digit BCD code output conversion table

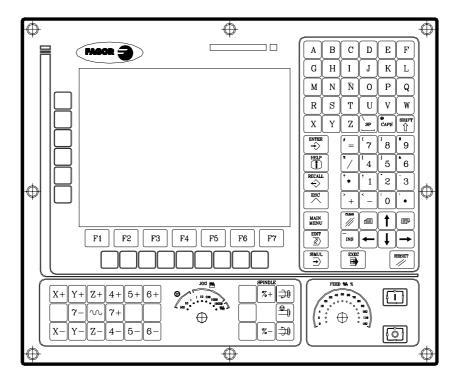


**CNC 8040** 

# **KEY CODES**

## Alphanumeric keyboard and monitor (M-T models)





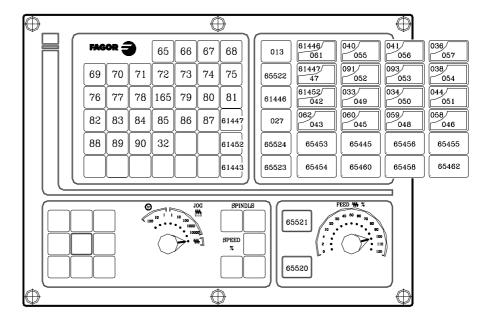


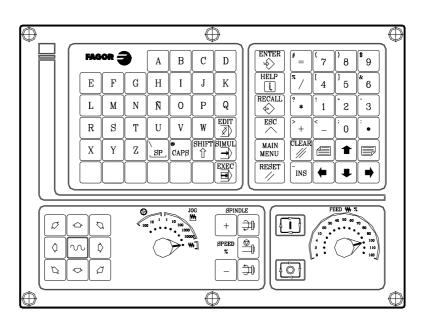
APPENDIX Key codes



**CNC 8040** 

#### Alphanumeric operator panel (M-T models)



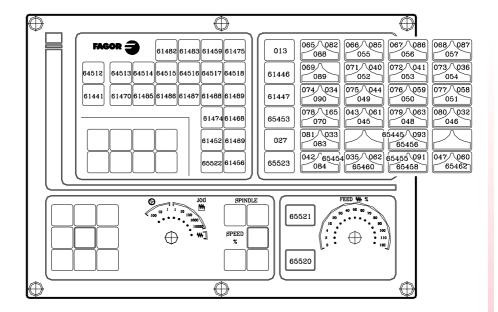






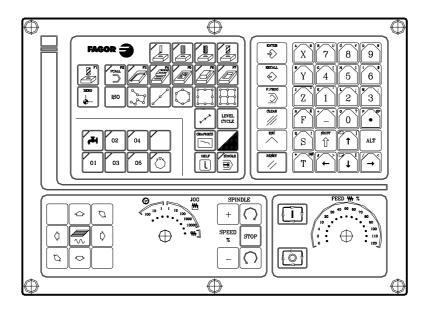
**CNC 8040** 

#### MC operator panel





APPENDIX
Key codes

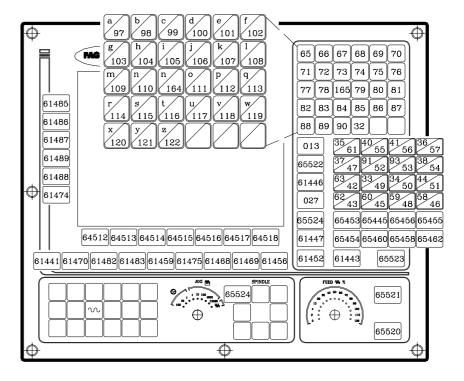


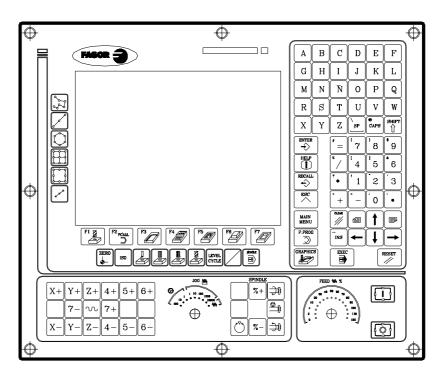


**CNC 8040** 



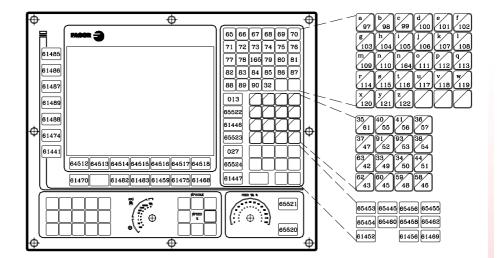
APPENDIX Key codes





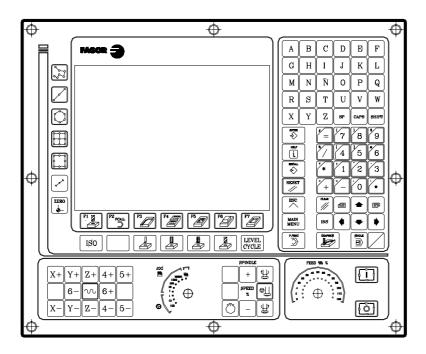


**CNC 8040** 





APPENDIX Key codes

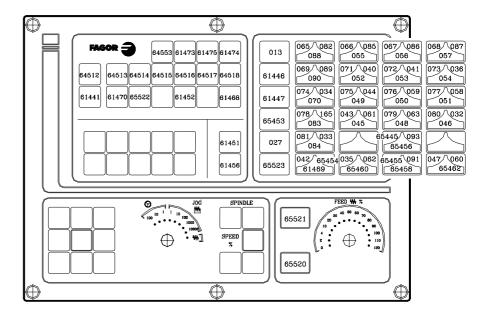


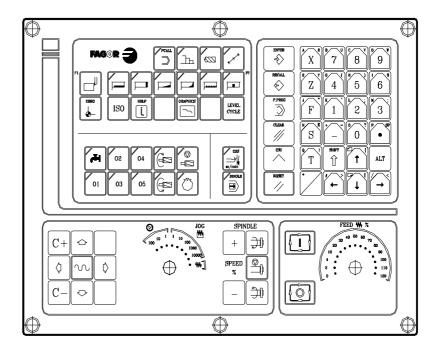


**CNC 8040** 

#### TC operator panel

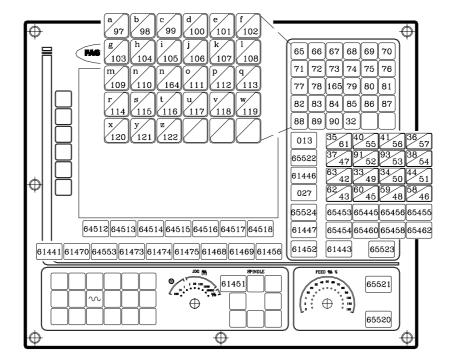


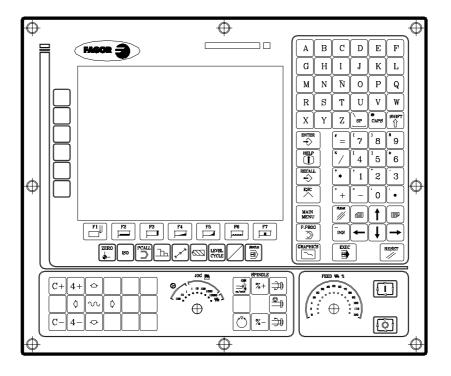






**CNC 8040** 





G.

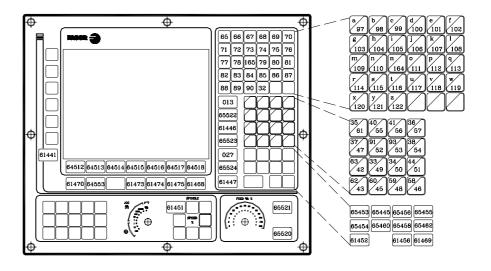
APPENDIX Key codes

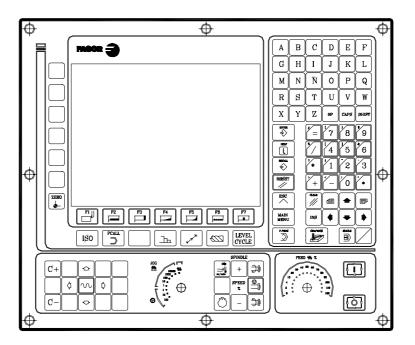


**CNC 8040** 



APPENDIX Key codes

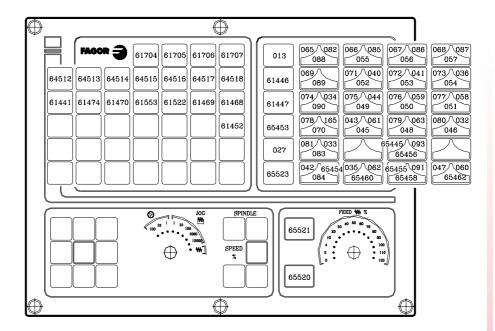


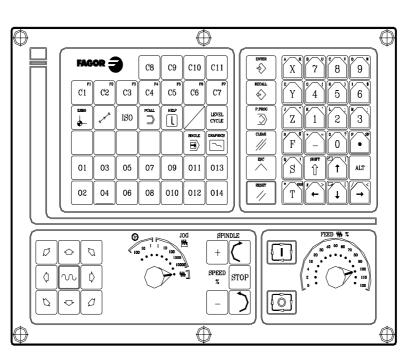




**CNC 8040** 

#### MCO/TCO operator panel





G.

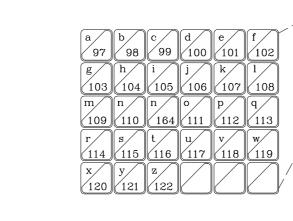
APPENDIX
Key codes

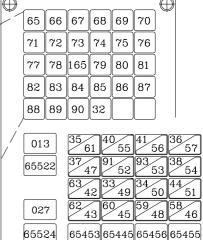


**CNC 8040** 

APPENDIX
Key codes

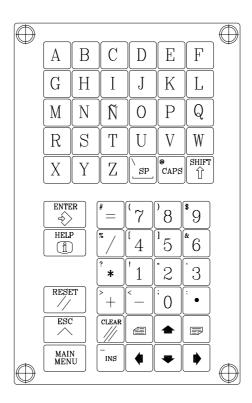
## Alphanumeric keyboard





61447

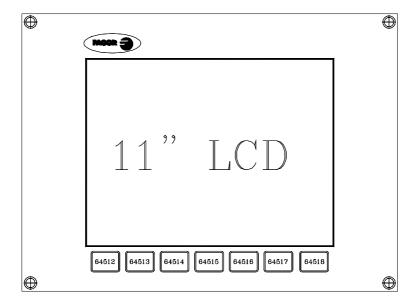
65454 65460 65458 65462





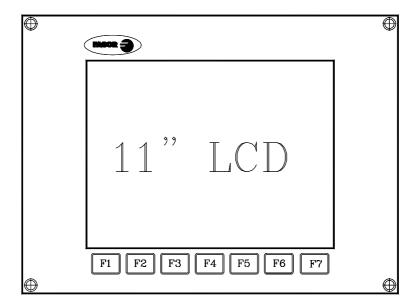
**CNC 8040** 

# 11" LCD Monitor





APPENDIX
Key codes





**CNC 8040** 

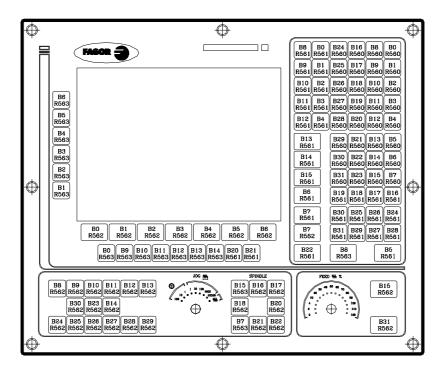


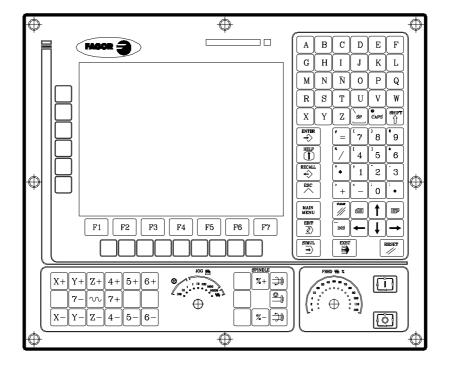


**CNC 8040** 

# LOGIC OUTPUTS OF KEY STATUS

#### Alphanumeric keyboard and monitor (M-T models)





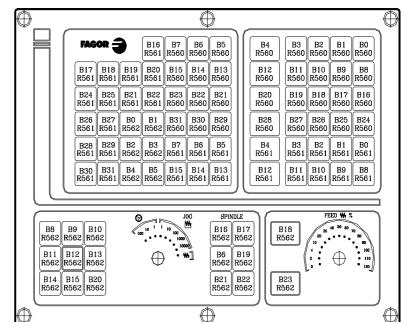


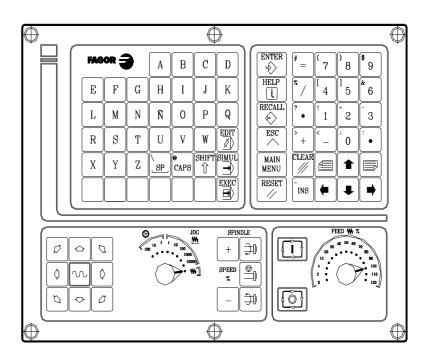
APPENDIX
Logic outputs of key status



**CNC 8040** 

#### Alphanumeric operator panel (M-T models)



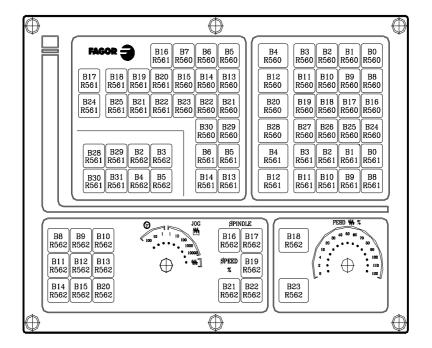


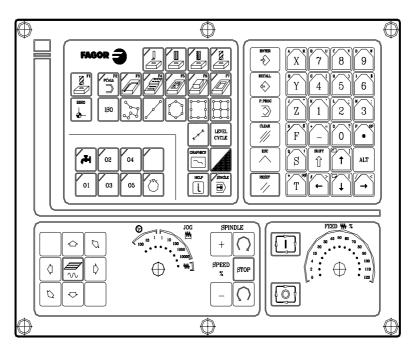


Logic outputs of key status



**CNC 8040** 







Logic outputs of key status

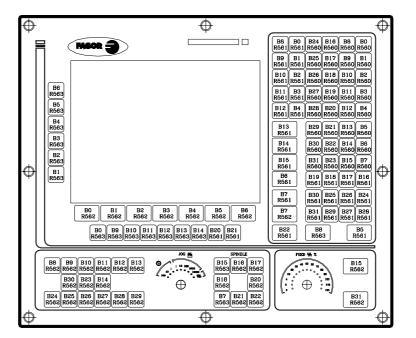


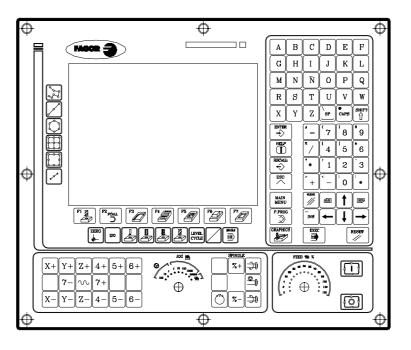
**CNC 8040** 



APPENDI

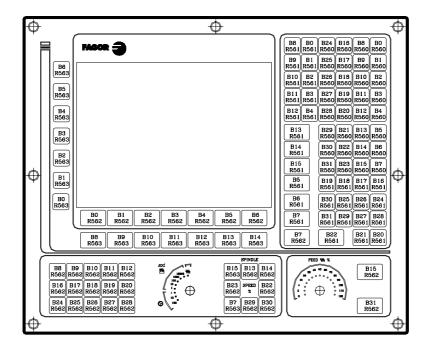
Logic outputs of key status

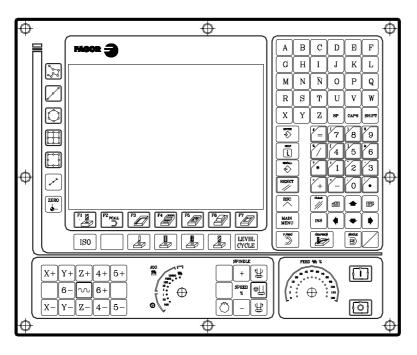






**CNC 8040** 





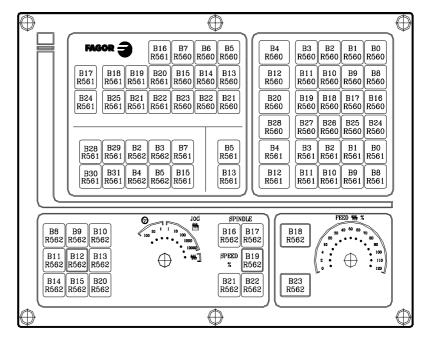


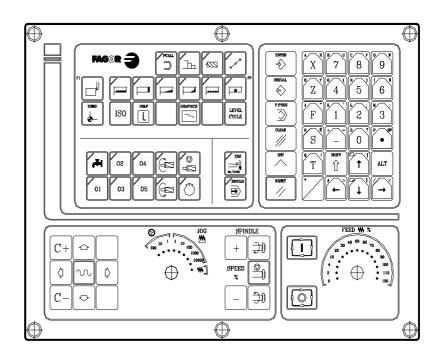
Logic outputs of key status



**CNC 8040** 

#### TC operator panel



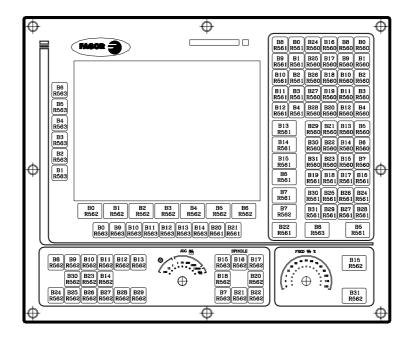


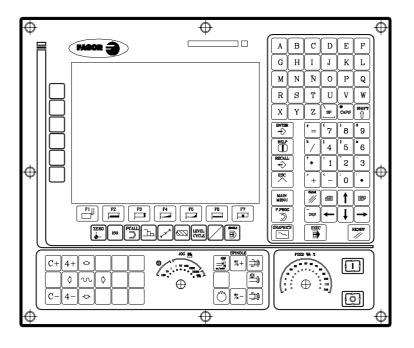


APPENDIX
Logic outputs of key status



**CNC 8040** 







Logic outputs of key status

**APPENDIX** 

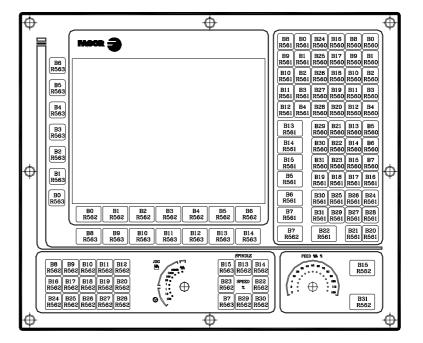


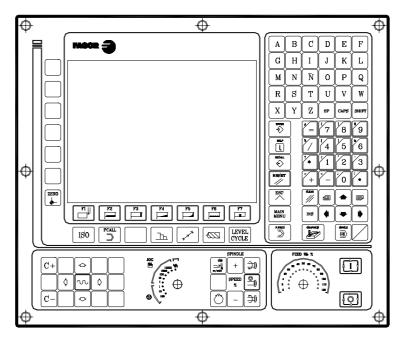
**CNC 8040** 



APPENDI

Logic outputs of key status

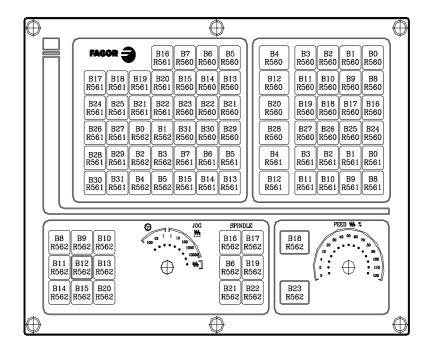


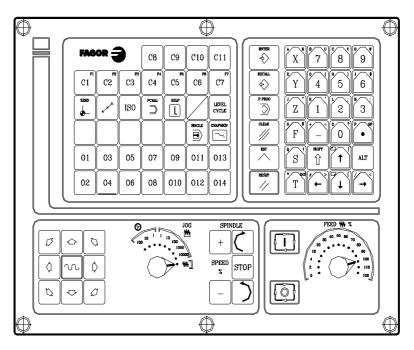




**CNC 8040** 

# MCO/TCO operator panel







Logic outputs of key status



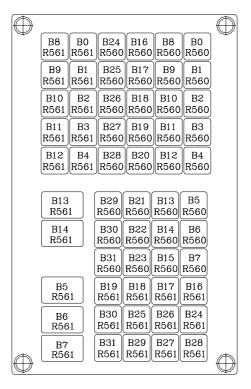
**CNC 8040** 

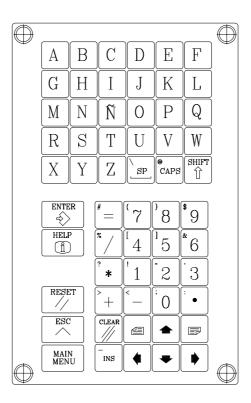
## Alphanumeric keyboard



APPENDI)

Logic outputs of key status

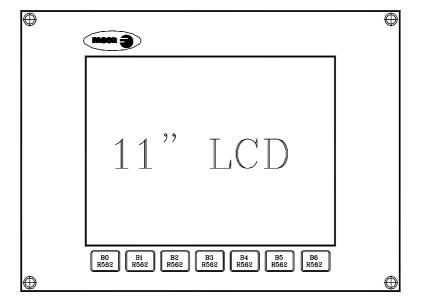




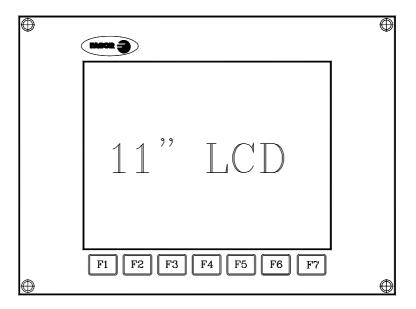


**CNC 8040** 

#### 11" LCD Monitor









**CNC 8040** 



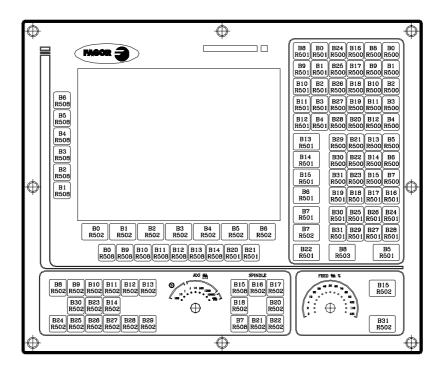
APPENDIX
Logic outputs of key status

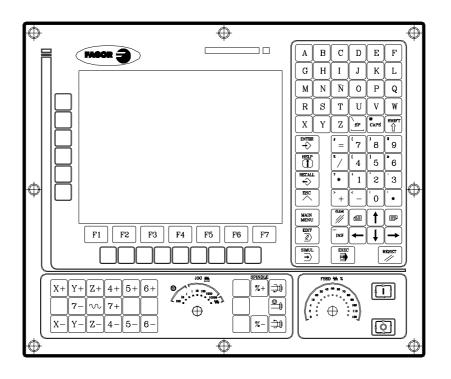


**CNC 8040** 

#### **KEY INHIBITING CODES**

#### Alphanumeric keyboard and monitor (M-T models)





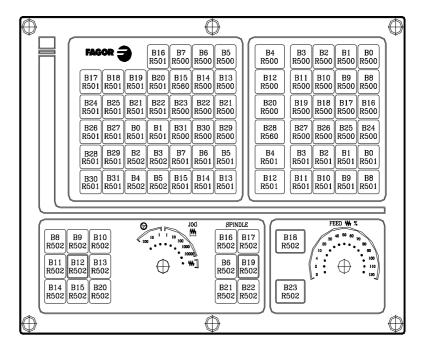


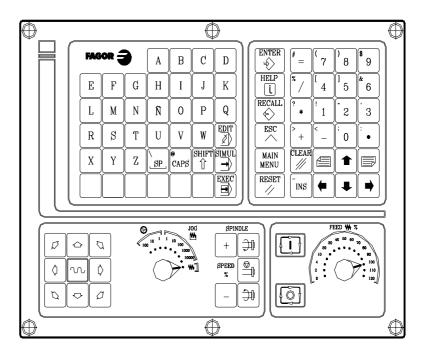
Key inhibiting codes

**FAGOR** 

**CNC 8040** 

# Alphanumeric operator panel (M-T models)



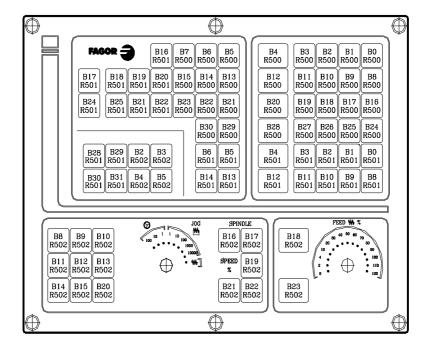


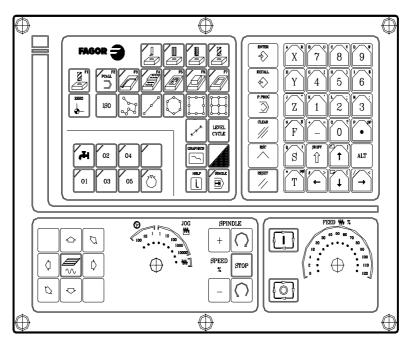




**CNC 8040** 

#### MC operator panel



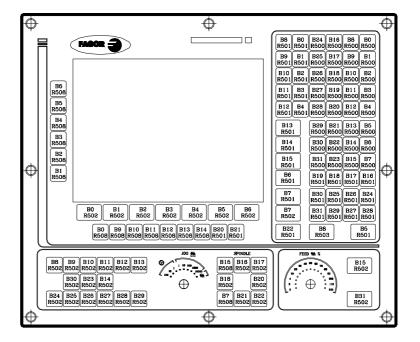


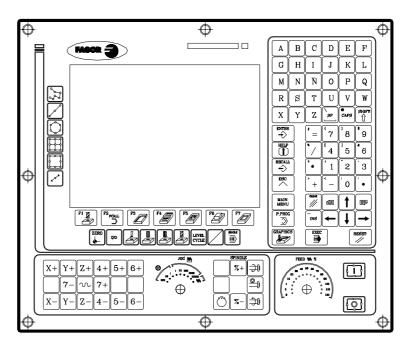


Key inhibiting codes



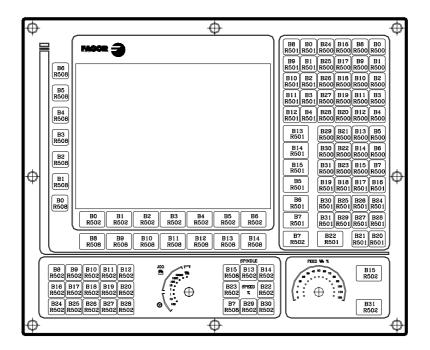
**CNC 8040** 

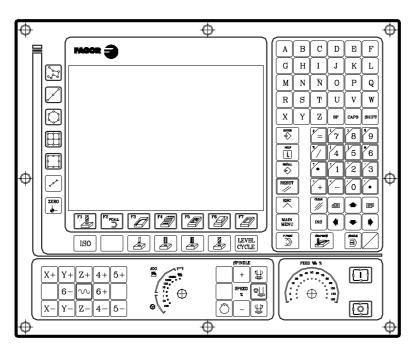






**CNC 8040** 





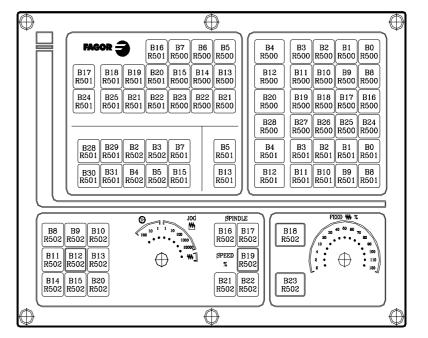


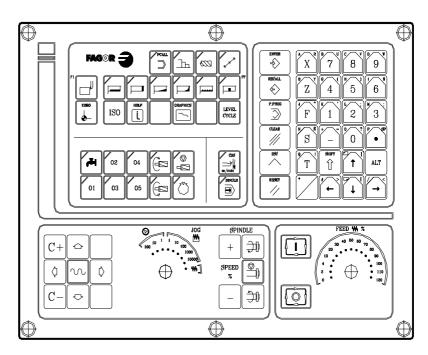
Key inhibiting codes



**CNC 8040** 

#### TC operator panel

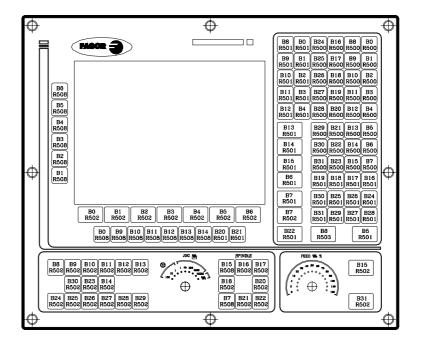


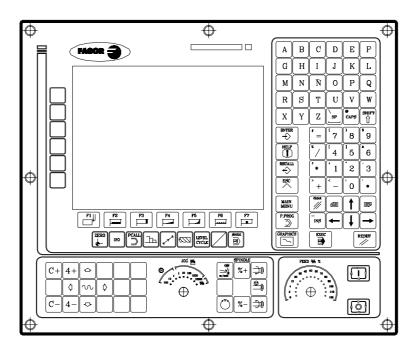






**CNC 8040** 

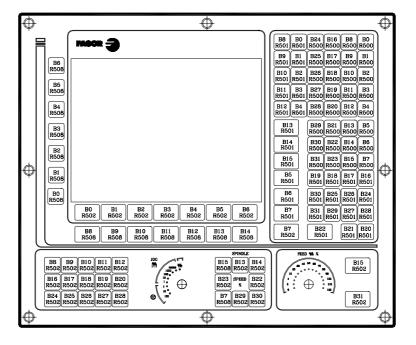


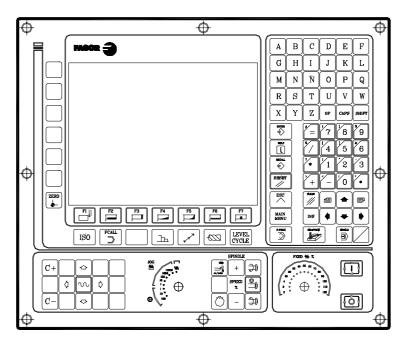




FAGOR

**CNC 8040** 

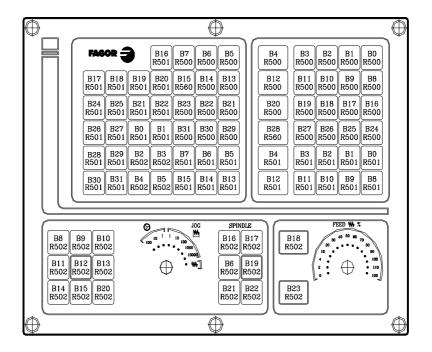


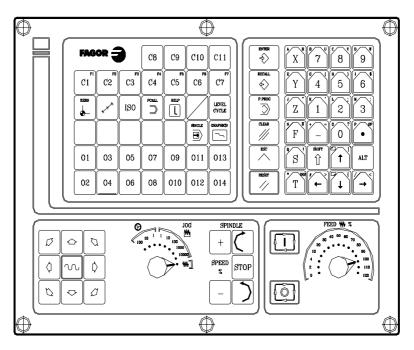




**CNC 8040** 

# MCO/TCO operator panel





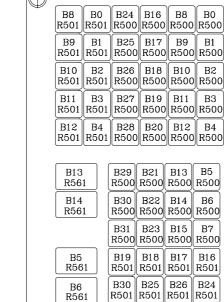


Key inhibiting codes



**CNC 8040** 

## Alphanumeric keyboard



B2

ВЗ

В6

B16

B24

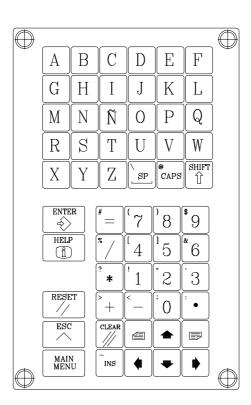
B28

B27 R501 R501 R501 R501

B29

B31

R561



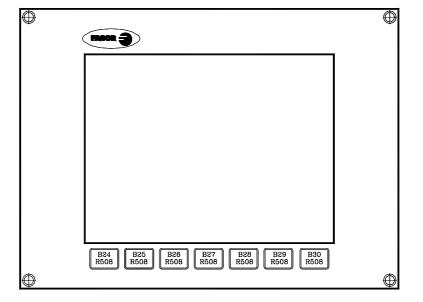


Key inhibiting codes

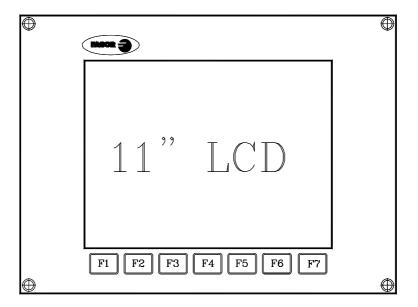


**CNC 8040** 

# 11" LCD Monitor









**CNC 8040** 



APPENDIX
Key inhibiting codes



**CNC 8040** 

# **MACHINE PARAMETER SETTING CHART**

# General machine parameters

P0     P50     P100     P1       P1     P51     P101     P1       P2     P52     P102     P1       P3     P53     P103     P1       P4     P54     P104     P1	51
P2         P52         P102         P1           P3         P53         P103         P1	
P3 P53 P103 P1	
P5         P55         P105         P1	
P6 P56 P106 P1	
P7 P57 P107 P1	
P8 P58 P108 P1	
P9         P59         P109         P1	
P10 P60 P110 P1	
P11         P61         P111         P1	
P12 P62 P112 P1	
P13 P63 P113 P1	
P14         P64         P114         P1	
P15         P65         P115         P1	
P16         P66         P116         P1	
P17         P67         P117         P1	
P18 P68 P118 P1	68
P19         P69         P119         P1	
P20         P70         P120         P1	
P21         P71         P121         P1	
P22 P72 P122 P1	
P23 P73 P123 P1	
P24         P74         P124         P1	
P25 P75 P125 P1	75
P26         P76         P126         P1	
P27         P77         P127         P1	
P28         P78         P128         P1	
P29 P79 P129 P1	
P30 P80 P130 P1	80
P31 P81 P131 P1	
P32 P82 P132 P1	
P33 P83 P133 P1	
P34 P84 P134 P1	84
P35         P85         P135         P1	
P36 P86 P136 P1	86
P37 P87 P137 P1	
P38 P88 P138 P1	88
P39         P89         P139         P1	
P40 P90 P140 P1	
P41         P91         P141         P1	
P42 P92 P142 P1	
P43         P93         P143         P1	
P44         P94         P144         P1	
P45         P95         P145         P1	
P46         P96         P146         P1	
P47         P97         P147         P1	
P48         P98         P148         P1	98
P49         P99         P149         P1	99

APPENDIX
Machine parameter setting chart



**CNC 8040** 

J.

APPENDIX
Machine parameter setting chart



**CNC 8040** 

P0	P50	P100	P150
P1	P51	P101	P151
P2	P52	P102	P152
P3	P53	P103	P153
P4	P54	P104	P154
P5	P55	P105	P155
P6	P56	P106	P156
P7	P57	P107	P157
P8	P58	P108	P158
P9	P59	P109	P159
P10	P60	P110	P160
P11	P61	P111	P161
P12	P62	P112	P162
P13	P63	P113	P163
P14	P64	P114	P164
P15	P65	P115	P165
P16	P66	P116	P166
P17	P67	P117	P167
P18	P68	P118	P168
P19	P69	P119	P169
P20	P70	P120	P170
P21	P71	P121	P171
P22	P72	P122	P172
P23	P73	P123	P173
P24	P74	P124	P174
P25	P75	P125	P175
P26	P76	P126	P176
P27	P77	P127	P177
P28	P78	P128	P178
P29	P79	P129	P179
P30	P80	P130	P180
P31	P81	P131	P181
P32	P82	P132	P182
P33	P83	P133	P183
P34	P84	P134	P184
P35	P85	P135	P185
P36	P86	P136	P186
P37	P87	P137	P187
P38	P88	P138	P188
P39	P89	P139	P189
P40	P90	P140	P190
P41 P42	P91	P141	P191
	P92	P142	P192
P43 P44	P93 P94	P143 P144	P193 P194
P45	P95	P145	P195
P45	P95	P145	P195
P47	P97	P147	P197
P47	P98	P147	P197
P49	P99	P149	P199
F48	୮୬୫	F149	רושש

P0	P50	P100	P150
P1	P51	P101	P151
P2	P52	P102	P152
P3	P53	P103	P153
P4	P54	P104	P154
P5	P55	P105	P155
P6	P56	P106	P156
P7	P57	P107	P157
P8	P58	P108	P158
P9	P59	P109	P159
P10	P60	P110	P160
P11	P61	P111	P161
P12	P62	P112	P162
P13	P63	P113	P163
P14	P64	P114	P164
P15	P65	P115	P165
P16	P66	P116	P166
P17	P67	P117	P167
P18	P68	P118	P168
P19	P69	P119	P169
P20	P70	P120	P170
P21	P71	P121	P171
P22	P72	P122	P172
P23	P73	P123	P173
P24	P74	P124	P174
P25	P75	P125	P175
P26	P76	P126	P176
P27	P77	P127	P177
P28	P78	P128	P178
P29	P79	P129	P179
P30	P80	P130	P180
P31	P81	P131	P181
P32	P82	P132	P182
P33	P83	P133	P183
P34	P84	P134	P184
P35	P85	P135	P185
P36	P86	P136	P186
P37	P87	P137	P187
P38	P88	P138	P188
P39	P89	P139	P189
P40	P90	P140	P190
P41	P91	P141	P191
P42	P92	P142	P192
P43 P44	P93 P94	P143 P144	P193 P194
P45 P46	P95	P145 P146	P195
	P96		P196
P47 P48	P97 P98	P147 P148	P197 P198
P49	P99	P149	P199



APPENDIX
Machine parameter setting chart



**CNC 8040** 

J.	•	
	APPENDIX	Machine parameter setting chart

D0 1	I DEO	L B400	D450
P0	P50	P100	P150
P1	P51	P101	P151
P2	P52	P102	P152
P3	P53	P103	P153
P4	P54	P104	P154
P5	P55	P105	P155
P6	P56	P106	P156
P7	P57	P107	P157
P8	P58	P108	P158
P9	P59	P109	P159
P10	P60	P110	P160
P11	P61	P111	P161
P12	P62	P112	P162
P13	P63	P113	P163
P14	P64	P114	P164
P15	P65	P115	P165
P16	P66	P116	P166
P17	P67	P117	P167
P18	P68	P118	P168
P19	P69	P119	P169
P20	P70	P120	P170
P21	P71	P121	P171
P22	P72	P122	P172
P23	P73	P123	P173
P24	P74	P124	P174
P25	P75	P125	P175
P26	P76	P126	P176
P27	P77	P127	P177
P28	P78	P128	P178
P29	P79	P129	P179
P30	P80	P130	P180
P31	P81	P131	P181
P32	P82	P132	P182
P33	P83	P133	P183
P34	P84	P134	P184
P35	P85	P135	P185
P36	P86	P136	P186
P37	P87	P137	P187
P38	P88	P138	P188
P39	P89	P139	P189
P40	P90	P140	P190
P41	P91	P141	P191
P42	P92	P142	P191
P43	P93	P143	P193
P44	P94	P143	P193
P45	P95	P145	P195
P45	P96	P145	P195
P47 P48	P97 P98	P147 P148	P197 P198
P49	P99	P149	P199



**CNC 8040** 

P0	P50	P100	P150
P1	P51	P101	P151
P2	P52	P102	P152
P3	P53	P103	P153
P4	P54	P104	P154
P5	P55	P105	P155
P6	P56	P106	P156
P7	P57	P107	P157
P8	P58	P108	P158
P9	P59	P109	P159
P10	P60	P110	P160
P11	P61	P111	P161
P12	P62	P112	P162
P13	P63	P113	P163
P14	P64	P114	P164
P15	P65	P115	P165
P16	P66	P116	P166
P17	P67	P117	P167
P18	P68	P118	P168
P19	P69	P119	P169
P20	P70	P120	P170
P21	P71	P121	P171
P22	P72	P122	P172
P23	P73	P123	P173
P24	P74	P124	P174
P25	P75	P125	P175
P26	P76	P126	P176
P27	P77	P127	P177
P28	P78	P128	P178
P29	P79	P129	P179
P30	P80	P130	P180
P31	P81	P131	P181
P32	P82	P132	P182
P33	P83	P133	P183
P34	P84	P134	P184
P35	P85	P135	P185
P36	P86	P136	P186
P37	P87	P137	P187
P38	P88	P138	P188
P39	P89	P139	P189
P40	P90	P140	P190
P41	P91	P141	P191
P42	P92	P142	P192
P43	P93	P143	P193
P44	P94	P144	P194
P45	P95	P145	P195
P46	P96	P146	P196
P47	P97	P147	P197
P48	P98	P148	P198
P49	P99	P149	P199



APPENDIX
Machine parameter setting chart



**CNC 8040** 

P0

# \_ axis machine parameters

P50

P95

P96

P97

P98

P99

1 0	1 30	100	1 130	1
P1	P51	P101	P151	
P2	P52	P102	P152	
P3	P53	P103	P153	
P4	P54	P104	P154	
P5	P55	P105	P155	
P6	P56	P106	P156	
P7	P57	P107	P157	
P8	P58	P108	P158	
P9	P59	P109	P159	
P10	P60	P110	P160	
P11	P61	P111	P161	
P12	P62	P112	P162	
P13	P63	P113	P163	
P14	P64	P114	P164	
P15	P65	P115	P165	
P16	P66	P116	P166	
P17	P67	P117	P167	
P18	P68	P118	P168	
P19	P69	P119	P169	
P20	P70	P120	P170	
P21	P71	P121	P171	
P22	P72	P122	P172	
P23	P73	P123	P173	
P24	P74	P124	P174	
P25	P75	P125	P175	
P26	P76	P126	P176	
P27	P77	P127	P177	
P28	P78	P128	P178	
P29	P79	P129	P179	
P30	P80	P130	P180	
P31	P81	P131	P181	
P32	P82	P132	P182	
P33	P83	P133	P183	
P34	P84	P134	P184	
P35	P85	P135	P185	
P36	P86	P136	P186	
P37	P87	P137	P187	
P38	P88	P138	P188	
P39	P89	P139	P189	
P40	P90	P140	P190	
P41	P91	P141	P191	
P42	P92	P142	P192	
P43	P93	P143	P193	
P44	P94	P144	P194	
				•

P145

P146

P147

P148

P149

P195 P196

P197

P198

P199

P100

P150





P45

P46

P47

P48

P49

CNC 8040

P150

P151

P152

P153

P190

P191

P192

P193

P194

P195

P196

P197

P198

P199

P50

P51

P52

P53

P90

P91

P92

P93

P94

P95

P96

P97

P98

P99

P0

Ρ1

P2

РЗ

P40

P41

P42

P43

P44

P45

P46

P47

P48 P49

. •	. 55			
P4	P54	P104	P154	
P5	P55	P105	P155	
P6	P56	P106	P156	
P7	P57	P107	P157	
P8	P58	P108	P158	
P9	P59	P109	P159	
P10	P60	P110	P160	
P11	P61	P111	P161	
P12	P62	P112	P162	
P13	P63	P113	P163	
P14	P64	P114	P164	
P15	P65	P115	P165	
P16	P66	P116	P166	
P17	P67	P117	P167	
P18	P68	P118	P168	
P19	P69	P119	P169	
P20	P70	P120	P170	
P21	P71	P121	P171	
P22	P72	P122	P172	
P23	P73	P123	P173	
P24	P74	P124	P174	
P25	P75	P125	P175	
P26	P76	P126	P176	
P27	P77	P127	P177	
P28	P78	P128	P178	
P29	P79	P129	P179	
P30	P80	P130	P180	
P31	P81	P131	P181	
P32	P82	P132	P182	
P33	P83	P133	P183	
P34	P84	P134	P184	
P35	P85	P135	P185	
P36	P86	P136	P186	
P37	P87	P137	P187	
P38	P88	P138	P188	
P39	P89	P139	P189	
D 40	500	5440	D.100	

P140

P141

P142

P143

P144

P145

P146

P147

P148

P149

P100

P101

P102

P103



**APPENDIX** 

Machine parameter setting chart



**CNC 8040** 

APPENDIX
Machine parameter setting chart



**CNC 8040** (SOFT M: V11.1x) (SOFT T: V12.1x)

P0	P50	P100	P150
P1	P51	P101	P151
P2	P52	P102	P152
P3	P53	P103	P153
P4	P54	P104	P154
P5	P55	P105	P155
P6	P56	P106	P156
P7	P57	P107	P157
P8	P58	P108	P158
P9	P59	P109	P159
P10	P60	P110	P160
P11	P61	P111	P161
P12	P62	P112	P162
P13	P63	P113	P163
P14	P64	P114	P164
P15	P65	P115	P165
P16	P66	P116	P166
P17	P67	P117	P167
P18	P68	P118	P168
P19	P69	P119	P169
P20	P70	P120	P170
P21	P71	P121	P171
P22	P72	P122	P172
P23	P73	P123	P173
P24	P74	P124	P174
P25	P75	P125	P175
P26	P76	P126	P176
P27	P77	P127	P177
P28	P78	P128	P178
P29	P79	P129	P179
P30	P80	P130	P180
P31	P81	P131	P181
P32	P82	P132	P182
P33	P83	P133	P183
P34	P84	P134	P184
P35	P85	P135	P185
P36	P86	P136	P186
P37	P87	P137	P187
P38	P88	P138	P188
P39 P40	P89 P90	P139 P140	P189 P190
P41 P42	P91 P92	P141 P142	P191 P192
P43	P93	P143	P192
P43	P94	P143	P193
P45	P95	P145	P195
P45	P96	P145	P196
P47	P97	P147	P197
P47	P98	P147	P197
P49	P99	P149	P199
1 40	ו טט	I   H J	1 133

	J
APPENDIX	Machine parameter setting chart

P0	P50	P100	P150
P1	P51	P101	P151
P2	P52	P102	P152
P3	P53	P103	P153
P4	P54	P104	P154
P5	P55	P105	P155
P6	P56	P106	P156
P7	P57	P107	P157
P8	P58	P108	P158
P9	P59	P109	P159
P10	P60	P110	P160
P11	P61	P111	P161
P12	P62	P112	P162
P13	P63	P113	P163
P14	P64	P114	P164
P15	P65	P115	P165
P16	P66	P116	P166
P17	P67	P117	P167
P18	P68	P118	P168
P19	P69	P119	P169
P20	P70	P120	P170
P21	P71	P121	P171
P22	P72	P122	P172
P23	P73	P123	P173
P24	P74	P124	P174
P25	P75	P125	P175
P26	P76	P126	P176
P27	P77	P127	P177
P28	P78	P128	P178
P29	P79	P129	P179
P30	P80	P130	P180
P31	P81	P131	P181
P32	P82	P132	P182
P33	P83	P133	P183
P34	P84	P134	P184
P35	P85 P86	P135	P185
P36		P136	P186
P37 P38	P87 P88	P137 P138	P187 P188
P39 P40	P89 P90	P139 P140	P189 P190
P41 P42	P91 P92	P141 P142	P191 P192
P43			
P43	P93 P94	P143 P144	P193 P194
P45			
P45 P46	P95 P96	P145 P146	P195 P196
P47	P97	P147	P197
P47	P98	P147	P197
P49	P99	P149	P199
Г49	୮୪୪	F149	F 188



**FAGOR** 

## Machine parameters for the second spindle

P0	P50	P100	P150
P1	P51	P101	P151
P2	P52	P102	P152
P3	P53	P103	P153
P4	P54	P104	P154
P5	P55	P105	P155
P6	P56	P106	P156
P7	P57	P107	P157
P8	P58	P108	P158
P9	P59	P109	P159
P10	P60	P110	P160
P11	P61	P111	P161
P12	P62	P112	P162
P13	P63	P113	P163
P14	P64	P114	P164
P15	P65	P115	P165
P16	P66	P116	P166
P17	P67	P117	P167
P18	P68	P118	P168
P19	P69	P119	P169
P20	P70	P120	P170
P21	P71	P121	P171
P22	P72	P122	P172
P23	P73	P123	P173
P24	P74	P124	P174
P25	P75	P125	P175
P26	P76	P126	P176
P27	P77	P127	P177
P28	P78	P128	P178
P29	P79	P129	P179
P30	P80	P130	P180
P31	P81	P131	P181
P32	P82	P132	P182
P33	P83	P133	P183
P34	P84	P134	P184
P35	P85	P135	P185
P36	P86	P136	P186
P37	P87	P137	P187
P38	P88	P138	P188
P39	P89	P139	P189
P40	P90	P140	P190
P41	P91	P141	P191
P42	P92	P142	P192
P43	P93	P143	P193
P44	P94	P144	P194
P45	P95	P145	P195
P46	P96	P146	P196
P47	P97	P147	P197
P48	P98	P148	P198
 D40	D00	D4.40	D400

P149

P199





**CNC 8040** 

P49

P99

	U
APPENDIX	arameter setting chart

D0	DEO	D100	D150
P0	P50	P100	P150
P1	P51	P101	P151
P2	P52	P102	P152
P3	P53	P103	P153
P4	P54	P104	P154
P5	P55	P105	P155
P6	P56	P106	P156
P7	P57	P107	P157
P8	P58	P108	P158
P9	P59	P109	P159
P10	P60	P110	P160
P11	P61	P111	P161
P12	P62	P112	P162
P13	P63	P113	P163
P14	P64	P114	P164
P15	P65	P115	P165
P16	P66	P116	P166
P17	P67	P117	P167
P18	P68	P118	P168
P19	P69	P119	P169
P20	P70	P120	P170
P21	P71	P121	P171
P22	P72	P122	P172
P23	P73	P123	P173
P24	P74	P124	P174
P25	P75	P125	P175
P26	P76	P126	P176
P27	P77	P127	P177
P28	P78	P128	P178
P29	P79	P129	P179
P30	P80	P130	P180
P31	P81	P131	P181
P32	P82	P132	P182
P33	P83	P133	P183
P34	P84	P134	P184
P35	P85	P135	P185
P36	P86	P136	P186
P37	P87	P137	P187
P38	P88	P138	P188
P39	P89	P139	P189
P40	P90	P140	P190
P41	P91	P141	P191
P42	P92	P142	P192
P43	P93	P143	P193
P44	P94	P144	P194
P45	P95	P145	P195
P46	P96	P146	P196
P47	P97	P147	P197
P48	P98	P148	P198
P49	P99	P149	P199
1 43	ГЭЭ	1 143	1 133



# J.

×	- 5
END!	char
_	$\overline{c}$
<b></b> .	_
	C
ΔPP	D
4	A C
◂	ď
	U,
	parameter
	Œ
	+
	7
	- ⊱
	~
	- 20
	π
	C
	ine
	.=

P0	P8	P16	P24	
P1	P9	P17	P25	
P2	P10	P18	P26	
P3	P11	P19	P27	
P4	P12	P20	P28	
P5	P13	P21	P29	
P6	P14	P22	P30	
P7	P15	P23	P31	

# Machine parameters for serial line 2

P0	P8	P16	P24	
P1	P9	P17	P25	
P2	P10	P18	P26	
P3	P11	P19	P27	
P4	P12	P20	P28	
P5	P13	P21	P29	
P6	P14	P22	P30	
P7	P15	P23	P31	

#### Ethernet related machine parameters.

P0	P8	P16	P24	
P1	P9	P17	P25	
P2	P10	P18	P26	
P3	P11	P19	P27	
P4	P12	P20	P28	
P5	P13	P21	P29	
P6	P14	P22	P30	
P7	P15	P23	P31	

## PLC machine parameters

P0	P22	P44	P66	
P1	P23	P45	P67	
P2	P24	P46	P68	
P3	P25	P47	P69	
P4	P26	P48	P70	
P5	P27	P49	P71	
P6	P28	P50	P72	
P7	P29	P51	P73	
P8	P30	P52	P74	
P9	P31	P53	P75	
P10	P32	P54	P76	
P11	P33	P55	P77	
P12	P34	P56	P78	
P13	P35	P57	P79	
P14	P36	P58	P80	
P15	P37	P59	P81	
P16	P38	P60	P82	
P17	P39	P61	P83	
P18	P40	P62	P84	
P19	P41	P63	P85	
P20	P42	P64	P86	
P21	P43	P65	P87	



**CNC 8040** 

# **M FUNCTIONS SETTING CHART**

M		Setting bits							
function		7	6	5	4	3	2	1	0
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								

M	Associated subroutine		;	Set	tin	g t	oits	•	
function		7	6	5	4	3	2	1	0
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								
М	S								



APPENDIX
M functions setting chart



**CNC 8040** 



APPENDIX

M functions setting chart



**CNC 8040** 

## **LEADSCREW ERROR COMPENSATION TABLE**

Axis				
Point	Position	Error	Error (-)	
Р		E	Е	
Р		Е	E	
Р		Е	Е	
Р		Е	Е	
Р		Е	Е	
Р		Е	Е	
Р		Е	E	
Р		Е	E	
Р		E	E	
Р		Е	E	
Р		E	E	
Р		E	E	
Р		E	E	
Р		Е	E	
Р		E	Е	
Р		Е	E	
Р		E	E	
Р		Е	E	

	Axis				
Point	Position	Error	Error (-)		
Р		E	E		
Р		E	E		
Р		Е	Е		
Р		E	E		
Р		E	Е		
Р		E	E		
Р		Е	E		
Р		E	E		
Р		E	Е		
Р		E	Е		
Р		Е	Е		
Р		E	Е		
Р		Е	Е		
Р		E	Е		
Р		E	E		
Р		E	E		
Р		E	E		
Р		E	E		

Axis				
Point	Position	Error	Error (-)	
Р		Е	Е	
Р		E	E	
Р		Е	Е	
Р		E	E	
Р		E	Е	
Р		E	E	
Р		Е	Е	
Р		Е	E	
Р		Е	Е	
Р		Е	E	
Р		Е	Е	
Р		Е	E	
Р		Е	Е	
Р		Е	E	
Р		Е	Е	
Р		Е	E	
Р		Е	Е	
Р		E	Е	

	Axis					
Point	Point Position Error Error					
Р		E	E			
Р		E	E			
Р		E	E			
Р		E	E			
Р		E	E			
Р		E	E			
Р		E	E			
Р		E	E			
Р		Е	Е			
Р		E	E			
Р		E	E			
Р		E	E			
Р		Е	Е			
Р		E	E			
Р		Е	Е			
Р		E	Е			
Р		Е	Е			
Р		E	Е			
		-	-			

APPENDIX
Leadscrew error compensation table



**CNC 8040** 



# **CROSS COMPENSATION TABLE**

Moving axis		MOVAXIS (P32)
Point	Position	Error
Р		E
Р		E
Р		E
Р		E
Р		Е
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E

Axis to be c	COMPAXIS (P33)	
Point	Position	Error
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E
Р		E

Moving axis		MOVAXIS (P55)
Point	Position	Error
Р		E
Р		E
Р		Е
Р		E
Р		Е
P		E
Р		Е
Р		E
Р		Е
Р		E
Р		E
Р		E
Р		Е
P		E
Р		Е
Р		E
Р		Е
Р		E

Point         Position         Error           P         E         E           P         E	Axis to be compensated		COMPAXIS (P56)
P       E         P       E	Point	Position	Error
P	Р		E
P E P E P E P E P E P E P E P E P E P E	Р		E
P E E E E E E E E E E E E E E E E E E E	Р		E
P       E         P       E         P       E         P       E         P       E         P       E         P       E         P       E         P       E         P       E         P       E         P       E         P       E         P       E         P       E         P       E         P       E         P       E         P       E	Р		E
P E E E E E E E E E E E E E E E E E E E	Р		E
P	Р		E
P	Р		E
P	Р		E
P	Р		
P	Р		E
P E E E E E E E E E E E E E E E E E E E	Р		E
P	Р		E
P E E F			
P E E	Р		E
P E			
1	Р		E
Р Е	·		E
	Р		E



APPENDIX
Cross compensation table



**CNC 8040** 



Moving axis		MOVAXIS (P58)	Axis to be compensated		COMPAXIS (P59)
Point	Position	Error	Point	Position	Error
P P		пп	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
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



**CNC 8040** 

#### **MAINTENANCE**

#### Cleaning

The dirt accumulated in the unit could act as a layer that hampers the dissipation of the heat generated by the internal electronic circuitry with the risk of overheating and damaging the CNC.

Also, accumulated dirt

To clean the operator panel and the monitor's front panel, a soft cloth should be used soaked in de-ionized water and home non-abrasive dishwasher soap (liquid, never powder) or with 75° alcohol.

Do not use air at high pressure to clean the unit because it could cause grease to accumulate which in turn may cause electrostatic discharges.

The plastics used on the front panel of the units are resistant to:

- · Grease and mineral oil.
- · 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 open this unit. Only personnel authorized by Fagor Automation may open this unit.

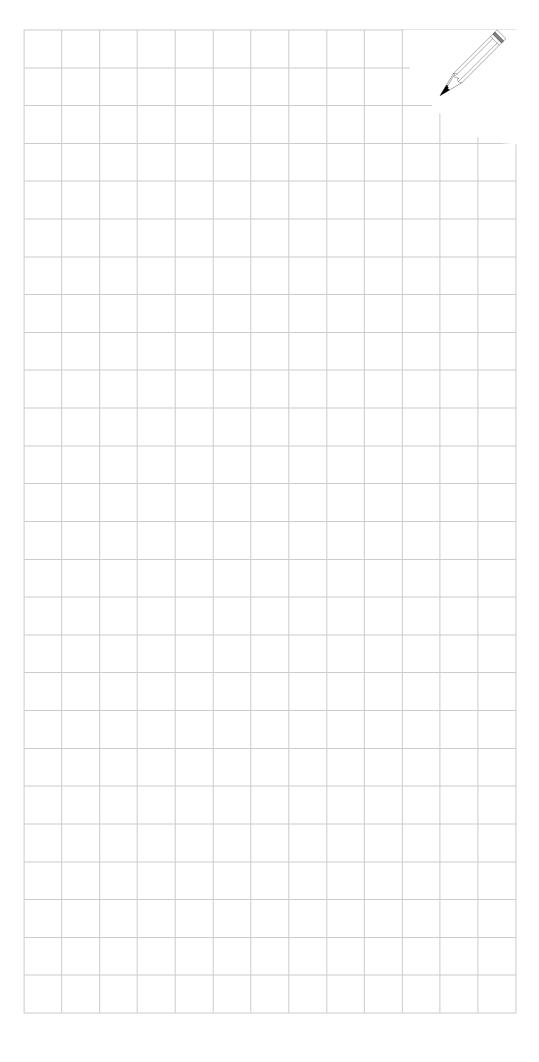
Do not handle the connectors with the unit connected to mains. Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.



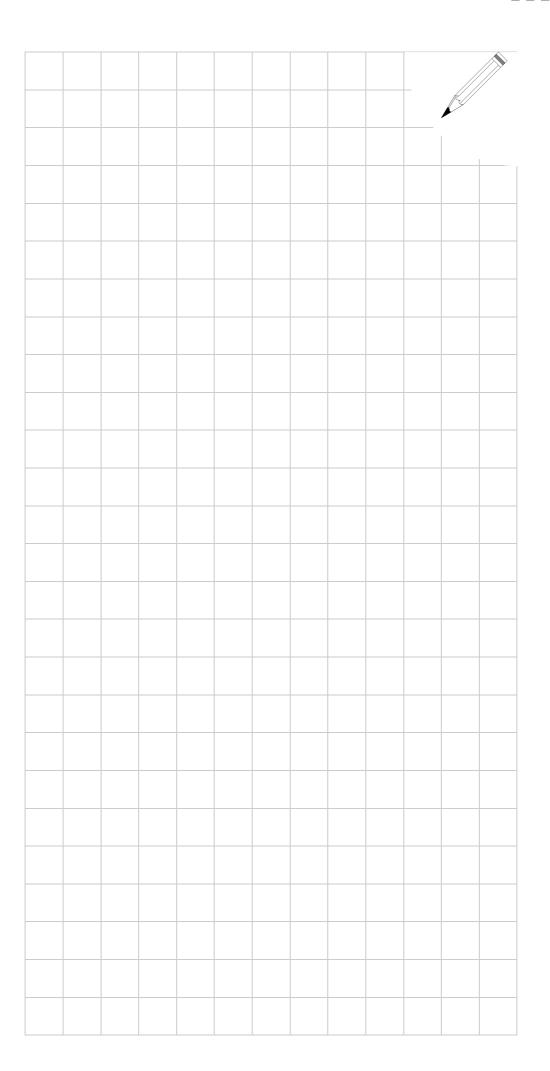
**APPENDIX**Maintenance



**CNC 8040** 



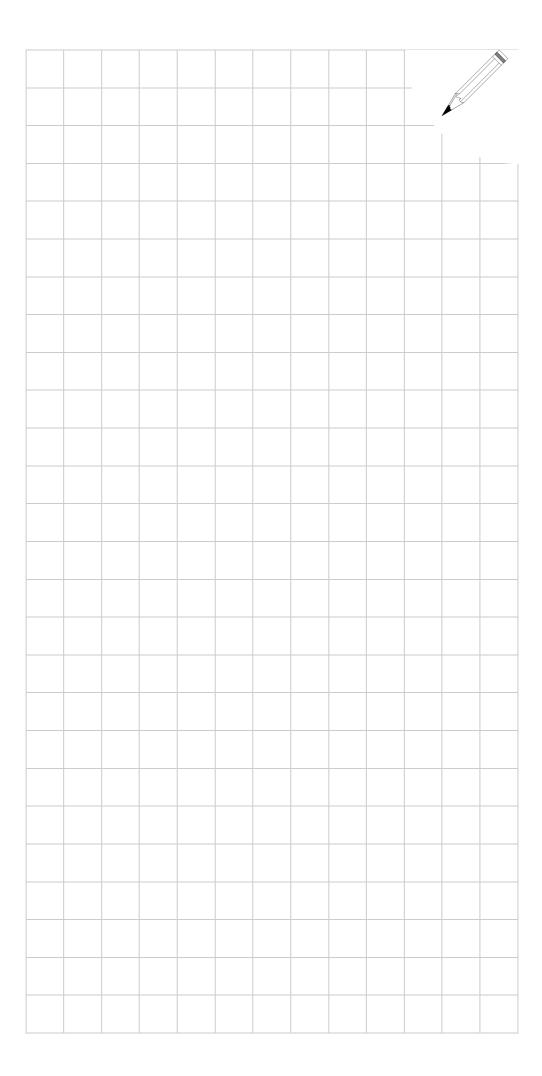




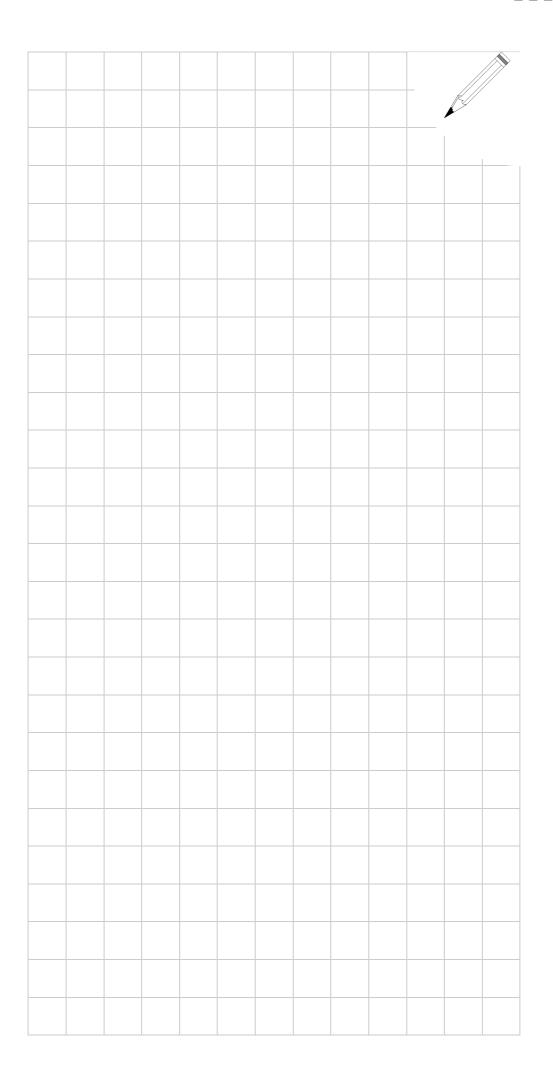
N.



**CNC 8040** 







N.



**CNC 8040** 

