FAGOR 8050 CNC

INSTALLATION
MANUAL

Ref. 9701 (in)

FAGOR AUTOMATION S. Coop. keeps informed all those customers who request it about new features implemented onto the FAGOR 8050 CNC.

This way, the customer may request anynew features he may wish to integrate into his own machine.

To do this, simply send us your full company address as well as the reference numbers (model and serial number) of the various CNC models you have.

Please note that some of the features described in this manual might not be implemented in the software version that you just obtained.

The features dependent on the software version are:

Mill Model	**Lathe Model**
Tool life monitoring	**Tool life monitoring**
Probing canned cycle	**Probing canned cycles**
DNC	**DNC**
Profile editor	**Profile Editor**
Software for 4 or 6 axes	**Software for 4 or 6 axes**
Irregular pockets (with islands)	**C axis**
Digitizing	
Solid graphics	
Rigid Tapping	
Tracing	

The information described in this manual may be subject to variations due to technical modifications.

When purchasing a **FAGOR 8050 GP CNC** the following considerations must taken:

* This model is based on the FAGOR 8050 M CNC (Mill model).
* It is missing some of the features available at the FAGOR 8050 M CNC.

The list below indicates those features missing with respect to the Mill model CNC as well as the software options available for this model (GP).

Features not available	Software options
Electronic threading (G33)	Software for 4 or 6 axes
Tool magazine management	DNC
Machining canned cycles (G8x)	Rigid tapping (G84)
Multiple machining cycles (G6x)	Tool radius compensation (G40, G41, G42)
Probing canned cycles	Profile editor
Tool life monitoring	
Irregular pockets (with islands)	
Digitizing	
Solid Graphics	
Tracing	

INDEX

Section | Page

Section Page

Chapter 2 POWER AND MACHINE CONNECTION

Chapter 3 MACHINE PARAMETERS

Chapter 4 CONCEPT SUBJECTS

Chapter 5 INTRODUCTION TO THE PLC

Section | Page

Chapter 6 PLC RESOURCES

Chapter 7 PLC PROGRAMMING

Chapter 8 CNC-PLC COMMUNICATION

Chapter 9 CNC LOGIC INPUTS AND OUTPUTS

Section Page

Chapter10 ACCESS TO INTERNAL CNC VARIABLES

Chapter 11 AXES CONTROLLED FROM THE PLC

Chapter 12 PLC PROGRAMMING EXAMPLE

APPENDICES

NEW FEATURES AND MODIFICATIONS

(Mill model)

Date: **June 1992** ***Software Version:*** **7.01 and newer**

FEATURE	**AFFECTED MANUAL AND CHAPTERS**	
GP Model	All Manuals	1st page
Reception of Autocad drawings	Dedicated Manual. Supplied with the software	
Auxiliary Spindle / Live tool	Installation Manual Programming Manual	Chap. 3, Chap. 9, Appendix Chap. 5, Chap. 13
Tracing	Installation Manual Program. Manual	Chap. 1, Chap. 3 Chap. 5, Chap. 14, Chap. 16, Appen.
Profile Editor	Operating Manual	Chap. 4
Interactive Editor	Operating Manual	Chap. 4
"TEACH-IN" Editing	Operating Manual	Chap. 4
Software for 4 or 6 axes	Installation Manual Programming Manual	Chap.4, Chap. 9, Chap. 10, Appen. Chap.3, Chap. 13
Axes Controlled from the PLC	Installation Manual	Chap. 3, Chap. 11
Storing of EEPROM memory contents into an EPROM memory	Operating Manual	Chap.7
Tool calibration with a probe in JOG mode	Installation Manual Operating Manual	Chap. 3 Chap. 5
Interruption Subroutines (4 inputs)	Installation Manual	Chap. 3, Chap. 9, Appendix
Logic Analyzer for the PLC	Installation Manual Operating Manual	Chap. 7 Chap. 9
AC-forward	Installation Manual	Chap.3
PLC Monitoring in JOG mode	Operating Manual	Chap. 5
Execution time Estimates	Operating Manual	Chap. 3
Part program storing in EEPROM memory	Installation Manual Operating Manual	Chap. 3 Chap. 7, Chap. 12
Three cross compensation tables	Installation Manual Operating Manual	Chap. 3, Appendix Chap. 11
Axes jogging when setting leadscrew and cross compensation tables	Operating Manual	Chap. 11
Subroutine associated with the tools	Installation Manual	Chap. 3
Possibility to FIND TEXT in the BLOCK SELECTION option	Operating Manual	Chap. 3
More double and triple size characters	Operating Manual	Chap. 10
Programming of the ERROR instruction by parameter	Programming Manual	Chap. 14
Variables to access the rotation center: ROTPF and ROTPS	Programming Manual	Chap. 13, Appendix

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Variables to access the tracing probe: DEFLEX, DEFLEY and DEFLEZ	Installation Manual Programming Manual	Chap. 10, Appendix Chap. 13, Appendix
General logic output indicating the status of the axes positioning loop: LOPEN	Installation Manual	Chap. 9, Appendix
PLC. Initialize a group of registers	Operating Manual	Chap. 9
PLC. New instructions	Installation Manual	Chap. 7
PLC. 200 symbols	Installation Manual	Chap. 7
New possibilities in irregular pockets (with islands)	Programming Manual	Chap. 11
Connector X7 of the AXES module	Installation Manual	Chap. 1
Support of the FAGOR Floppy disc unit	Installation Manual	Chap. 1, Chap. 3
Make the tool change cycle more flexible	Installation Manual	Chap. 3
Improved error processing	Operating Manual	Chap. 1

Date: **April 1993** ***Software Version:*** **7.06 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Limitless rotary axes	Installation Manual	Chap. 3
Positioning axes in G01	Programming Manual	Chap. 6
Reference point shift	Installation Manual	Chap. 3, Chap. 4
Work zone variables (R/W) from PLC	Installation Manual Programming Manual	Chap. 10, Appendix Appendix
Possibility to abort the PLC channel	Installation Manual	Chap. 9 Appendix
Movement until contact	Installation Manual Programming Manual	Chap. 3, Cap. 11 Chap. 6, Appendix
Boring Mill graphics	Installation Manual	Chap. 3
"WBUF" programmable without parameters	Programming Manual	Chap. 14

Date: **July 1993** ***Software Version:*** **7.07 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
The GP model offers optional Tool radius compensation (G40, G41, G42)		
Logic outputs of the key status	Installation manual	Chap. 9

Date: **January 1994** ***Software Version:*** **9.01 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Tool base or tool tip position display	Installation manual	Chap. 3
Measurement in graphics via cursor	Operating manual	Chap. 3
Two ways for tool calibration (manual and probe)	Operating manual	Chap. 5
Treatment of coded Io signals	Installation manual	Chap. 3
Possibility to store PLC errors and messages in EEPROM memory	Installation manual Operating manual	Chap. 3 Chap. 7
"Program in EEPROM" indicator	Operating manual	Chap. 7
"Program in execution" indicator	Operating manual	Chap. 7
G50. Controlled corner rounding	Installation manual Programming manual	Chap. 3, Chap. 11 Chap. 5, 7, Appendix
Feedrate per revolution (G95) for axes controlled via PLC	Installation manual	Chap. 11
Concentric roughing of irregular pockets (with islands)	Programming manual	Chap. 11
G93 when defining the profile of an irregular pocket	Programming manual	Chap. 11
Manual; one, two and three-dimensional tracing and digitizing cycles	Installation manual Programming manual	Chap. 9, Appendix Chap. 5, 16, Appendix
New tracing/digitizing cycles	Programming manual	Chap. 16
Display of deflection and correction factor for the tracing probe	Operating manual	Chap. 3, 5
Infinite program execution from a PC	Operating manual	Chap. 8
Multi-disk infinite program in Floppy Disk Unit	Operating manual	Chap. 8
Multi-disk digitizing in Floppy Disk Unit	Operating manual	Chap. 8.

Date: **May 1994** ***Software Version:*** **9.03 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Anticipation time, for punching	Installation manual	Chap. 3, 9, Appendix
Variables: TPOS(X-C), TPOSS, FLWES	Installation manual	Chap. 10, Appendix
M19 speed modification via PLC	Installation manual	Chap. 9, Appendix.
G75 and G76 moves at 100% of F	Programming manual	Chap. 10

Date: **December 1994** ***Software Version:*** **9.06 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Third work zone	Installation manual Programming manual	Chap. 10, Appendix Chap. 3, 13, Appendix
For easier operation without monitor, the default values of parameters: PROTOCOL (1) and POWDNC (yes) have been changed	Installation manual	Chap. 3

Date: **February 1995** ***Software Version:*** **9.07 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
If while searching "coded" home, the DECEL* signal of the axis goes high, the homing direction is reversed.	Installation manual	Chap. 4
A "T" function with associated subroutine may be programmed in a motion block.	Installation manual	Chap. 3
The TAFTERS parameter indicates whether the "T" function is executed before or after its associated subroutine.	Installation manual	Chap. 3
Function G53 without motion information cancels the active zero offset.	Programming manual	Chap. 4
The "M" function table allows interrupting block preparation until the "M" starts or ends.	Installation manual Operating manual	Chap. 3 Chap. 11

Date: **October 1995** ***Software Version:*** **9.09 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
M19TYPE (spindle parameter) indicates whether or not the spindle is homed every time it switches from open loop to closed loop.	Installation manual	Chap. 3
Variables POSS and TPOSS always active (whether in open loop or closed loop).	Installation manual Programming manual	Chap. 10 Chap. 13
Leadscrew compensation tables allow slopes of up to ±45º.	Installation manual Operating manual	Chap. 3 Chap. 11

Date: **April 1996** ***Software Version:*** **9.10 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
New spindle related variables RPOSS and RTPOSS	Installation manual Programming manual	Chap. 10 and Appendix Chap. 13 and Appendix

Date: **July1996** ***Software Version:*** **9.11 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Machine parameter EXTMULT to be used when the feedback system has coded marker pulses (Io).	Installation manual	Chap. 3

Date: **May 1996** ***Software Version:*** **11.01 and newer**

FEATURE	**AFFECTED MANUAL AND CHAPTERS**	
CPU TURBO	Installation manual	Chap. 1 and 3
Look-Ahead	Programming manual	Chap. 5, 7 and Appendix
3D Irregular pockets (with islands)	Programming manual	Chap. 11
Possibility to choose beginning and end of tool radius compensation.	Installation manual Programming manual	Chap. 3 Chap. 8
Anticipation signal for each axis	Installation manual	Chap. 3, 9 and Appendix
High-level block execution from PLC	Installation manual	Chap. 11
Non-rollover rotary axis now possible	Installation manual	Chap. 3
Line graphics on GP models		
Optional Profile Editor on GP models		

NEW FEATURES AND MODIFICATIONS
(Lathe Model)

Date: **June 1992** ***Software Version:*** **6.01 and newer**

FEATURE	**AFFECTED MANUAL AND CHAPTERS**	
Reception of Autocad drawings	Dedicated Manual. Supplied with the software	
C axis	Installation Manual Programming Manual Operating Manual	Chap. 9, Appendix Chap. 5, Chap. 6, Appendix Chap. 3, Chap. 6
Auxiliary Spindle / Live tool	Installation Manual Programming Manual	Chap. 3, Chap. 9, Appendix Chap. 5, Chap. 11
Profile Editor	Operating Manual	Chap. 4
Interactive Editor	Operating Manual	Chap. 4
"TEACH-IN" editing	Operating Manual	Chap. 4
Software for 2, 4 or 6 axes	Installation Manual Programming Manual	Chap. 4, Chap. 9, Chap. 10, Appen. Chap. 3, Chap. 11
Axes Controlled from the PLC	Installation Manual	Chap. 3, Chap. 11
Storing of EEPROM memory contents in EPROM memory	Operating Manual	Chap.7
Tool calibration with a probe in JOG mode	Installation Manual Operating Manual	Chap. 3 Chap. 5
Interruption Subroutines (4 inputs)	Installation Manual	Chap. 3, Chap. 9, Appendix
Logic analyzer for the PLC	Installation Manual Operating Manual	Chap. 7 Chap. 9
AC- forward	Installation Manual	Chap.3
PLC Monitoring in JOG mode	Operating Manual	Chap. 5
Execution time estimates	Operating Manual	Chap. 3
Part-program storing in EEPROM memory	Installation Manual Operating Manual	Chap. 3 Chap. 7, Chap. 12
Three cross compensation tables	Installation Manual Operating Manual	Chap. 3, Appendix Chap. 11
Axes jogging when setting leadscrew and cross compensation tables	Operating Manual	Chap. 11
Subroutine associated with the tools	Installation Manual	Chap. 3
Possibility to FIND TEXT in the BLOCK SELECTION option	Operating Manual	Chap. 3

FEATURE	AFFECTED MANUAL AND CHAPTERS	
More double and triple size characters	Operating Manual	Chap. 10
Possibility to select colors for solid graphics	Operating Manual	Chap. 3
Programming of the ERROR instruction by parameter	Programming Manual	Chap.12
General logic output indicating the status of the axes positioning loop: LOPEN	Installation Manual	Chap. 9, Appendix
PLC. Initialize a group of registers	Operating Manual	Chap. 9
PLC. New instructions	Installation Manual	Chap. 7
PLC. 200 symbols	Installation Manual	Chap. 7
Finishing pass (G05 or G07) in canned cycles	Programming Manual	Chap. 9
Connector X7 of the AXES module	Installation Manual	Chap. 1
Support for the FAGOR floppy disc unit	Installation Manual	Chap. 1, Chap. 3
Make the tool change cycle flexible	Installation Manual	Chap. 3
Improved error processing	Operating Manual	Chap. 1

Date: **April 1993** ***Software Version:*** **6.06 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Limitless rotary axes	Installation Manual	Chap. 3
Positioning axes in G01	Programming Manual	Chap. 6
Reference point shift	Installation Manual	Chap. 3, Chap. 4
Work zone variables (R/W) from PLC	Installation Manual Programming Manual	Chap. 10, Appendix Appendix
Possibility to abort the PLC channel	Installation Manual	Chap. 9 Appendix
Movement until contact	Installation Manual Programming Manual	Chap. 3, Cap. 11 Chap. 6, Appendix
INCH/MM in Geometry table	Operating Manual	Chap. 6
"WBUF" programmable without parameters	Programming Manual	Chap. 12

Date: **July 1993** ***Software Version:*** **6.07 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Logic outputs of the key status	Installation manual	Chap. 9

Date: **January 1994** ***Software Version:*** **8.01 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Tool base or tool tip position display	Installation manual	Chap. 3
Measurement in graphics via cursor	Operating manual	Chap. 3
Two ways for tool calibration (manual and probe)	Operating manual	Chap. 5
Treatment of coded Io signals	Installation manual	Chap. 3
Possibility to store PLC errors and messages in EEPROM memory	Installation manual Operating manual	Chap. 3 Chap. 7
"Program in EEPROM" indicator	Operating manual	Chap. 7
"Program in execution" indicator	Operating manual	Chap. 7
G50. Controlled corner rounding	Installation manual Programming manual	Chap. 3, Chap. 11 Chap. 5, 7, Appendix
Feedrate per revolution (G95) for axes controlled via PLC	Installation manual	Chap. 11
G93 when defining the profile of a canned cycle	Programming manual	Chap. 9
Infinite program execution from a PC	Operating manual	Chap. 8

Date: **May 1994** ***Software Version:*** **8.02 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Withdrawal mode selection in canned cycles: G68, G69, G81, G82, G84 and G85	Programming manual	Chap. 9
Excess material on X and Z on G66, G68: and G69 canned cycles	Programming manual	Chap. 9
Axis selection in G66 canned cycle	Programming manual	Chap. 9
G75 and G76 moves at 100% of F	Programming manual	Chap. 10

Date: **July 1994** ***Software Version:*** **8.03 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Anticipation time, for punching	Installation manual	Chap. 3, 9, Appendix
Variables: TPOS(X-C), TPOSS, FLWES	Installation manual	Chap. 10, Appendix
M19 speed modification via PLC	Installation manual	Chap. 9, Appendix.

Date: **October1994** ***Software Version:*** **8.04 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS
Permanent "C" axis	Installation manual Chap. 3 Programming manual Chap. 6

Date: **January 1995** ***Software Version:*** **8.06 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS
Third work zone	Installation manual Chap. 10, Appendix Programming manual Chap. 3, 11, Appendix
If while searching "coded" home, the DECEL* signal of the axis goes high, the homing direction is reversed.	Installation manual Chap. 4

Date: **March 1995** ***Software Version:*** **8.07 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS
A "T" function with associated subroutine may be programmed in a motion block.	Installation manual Chap. 3
The TAFTERS parameter indicates whether the "T" function is executed before or after its associated subroutine.	Installation manual Chap. 3
Function G53 without motion information cancels the active zero offset.	Programming manual Chap. 4
The "M" function table allows interrupting block preparation until the "M" starts or ends.	Installation manual Chap. 3 Operating manual Chap. 11
For easier operation without monitor, the default values of parameters: PROTOCOL (1) and POWDNC (yes) have been changed.	Installation manual Chap. 3

Date: **July 1995** ***Software Version:*** **8.08 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS
M19TYPE (spindle parameter) indicates whether or not the spindle is homed every time it switches from open loop to closed loop.	Installation manual Chap. 3
Variables POSS and TPOSS always active (whether in open loop or closed loop).	Installation manual Chap. 10 Programming manual Chap. 11
Leadscrew compensation tables allow slopes of up to ±45º.	Installation manual Chap. 3 Operating manual Chap. 11

Date: **April 1996** ***Software Version:*** **8.09 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS
New spindle related variables RPOSS and RTPOSS	Installation manual Chap. 10 and Appendix Programming manual Chap. 13 and Appendix

Date: **July1996** ***Software Version:*** **8.10 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
Machine parameter EXTMULT to be used when the feedback system has coded marker pulses (Io).	Installation manual	Chap. 3

Date: **Sep. 1996** ***Software Version:*** **10.01 and newer**

FEATURE	AFFECTED MANUAL AND CHAPTERS	
CPU TURBO	Installation manual	Chap. 1 and 3
Look-Ahead	Programming manual	Chap. 5, 7 and Appendix
Possibility to choose beginning and end of tool radius compensation.	Installation manual Programming manual	Chap. 3 Chap. 8
Anticipation signal for each axis	Installation manual	Chap. 3, 9 and Appendix
High-level block execution from PLC	Installation manual	Chap. 11
Possibility of non-rollover rotary axes	Installation manual	Chap. 3
Feedrate per revolution in JOG mode.	Installation manual	Chap. 3
Possibility of sharing the electronic handwheel with any axis.	Installation manual	Chap. 3
RESET effective without prior CYCLE STOP.	Installation manual	Chap. 3
New canned cycles with live tool: G60, G61, G62 and G63.	Programming manual	Chap. 9 and Appendix
Possibility of final roughing pass in cycles: G68, G69, G81, G82, G84 and G85	Programming manual	Chap. 9
G83. Tapping canned cycle	Programming manual	Chap. 9
Possibility of setting the grooving pass in canned cycles: G88 and G89.	Programming manual	Chap. 9
Possibility of defining a profile in another program. For canned cycles: G66, G67 and G69.	Programming manual	Chap. 9
2 Spindles	Installation manual Programming manual	Chap. 3, 9, 10 and Appendix Chap. 5, 11 and Appendix
Canned cycles in any plane.	Programming manual	Chap. 9
Tool compensation in any plane	Installation manual Programming manual	Chap. 3 and 4 Chap. 8
Change PC directory from the CNC via DNC	Operating manual	Chap. 7

INTRODUCTION

Atention:

Before starting up the CNC, carefully read the instructions of Chapter 2 in the Installation Manual.

The CNC must not be powered-on until verifying that the machine complies with the "89/392/CEE" Directive.

DECLARATION OF CONFORMITY

Manufacturer: Fagor Automation, S. Coop.

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

We hereby declare, under our resposibility that the product:

Fagor 8050 CNC

meets the following directives:

SAFETY:

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

ELECTROMAGNETIC COMPATIBILITY:

EN 50081-2 Emission

EN 55011	Radiated. Class A, Group 1.
EN 55011	Conducted. Class A, Group 1.

EN 50082-2 Immunity

EN 61000-4-2	Electrostatic Discharges.
EN 61000-4-4	Bursts and fast transients.
EN 61000-4-11	Voltage fluctuations and Outages.
ENV 50140	Radiofrequency Radiated Electromagnetic Fields.
ENV 50141	Conducted disturbance induced by radio frequency fields.

As instructed by the European Community Directives on Low Voltage: 73/23/EEC, on Machine Safety 89/392/CEE and 89/336/CEE on Electromagnetic Compatibility.

In Mondragón, on April 1st, 1996

Fagor Automation, S. Coop. Ltda.
Director Gerente

Fdo.: Julen Busturia

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 45º C (113º 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 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 the unit in the right place
It is recommended, whenever possible, to instal the CNC away from coolants, chemical product, blows, etc. that could damage it.

This unit complies with the European directives on electromagnetic compatibility.

Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as.

- Powerful loads connected to the same AC power line as this equipment.
- Nearby portable transmitters (Radio-telephones, Ham radio transmitters).
- Nearby radio / 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 all the interference generating elements (relay coils, contactors, motors, etc.) uncoupled.

Use the proper power supply

Use an external regulated 24 Vdc power supply for the 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 (See chapter 2).

Ambient conditions

The working temperature must be between +5° C and +45° C (41ºF and 113º F)
The storage temperature must be between -25° C and 70° C. (-13º F and 158º F)

Monitor enclosure

Assure that the Monitor is installed at the distances indicated in chapter 1 from the walls of the enclosure.

Use a DC fan to improve enclosure ventilation.

Main AC Power Switch

This switch must be easy to access and at a distance between 0.7 m (27.5 inches) and 1.7 m (5.6 ft) off the floor.

Protections of the unit itself

Power Supply Module

It carries two fast fuses of 3.15 Amp./ 250V. to protect the mains AC input

Axes module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against reverse connection of the power supply.

Input / Output Module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against a voltage overload (greater than 33Vdc) and against reverse connection of the power supply.

Input / Output and Tracing Module

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.
They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against a voltage overload (greater than 33Vdc) and against reverse connection of the power supply.

Fan Module

It carries 1 or 2 external fuses depending on model
The fuses are fast (F), of 0.4 Amp./ 250V. to protect the fans.

Monitor

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

Precautions during repair

Do not manipulate the inside of the unit

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

Do not manipulate the connectors with the unit connected to AC power.

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

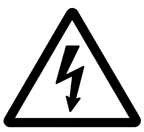
Safety symbols

Symbols which may appear on the manual

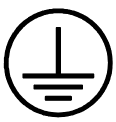
WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

Symbols that may be carried on the product

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.



"Electrical Shock" symbol
It indicates that point may be under electrical voltage



"Ground Protection" symbol
It indicates that point must be connected to the main ground point of the machine as protection for people and units.

WARRANTY TERMS

WARRANTY

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 CONTRACTS

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 Monitor 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.- When sending it to a Fagor Automation office for repair, attach a label indicating the owner of the unit, person to contact, type of unit, serial number, symptom and a brief description of the problem.

3.- Wrap the unit in a polyethylene roll or similar material to protect it.

    When sending the monitor, especially protect the CRT glass

4.- Pad the unit inside the cardboard box with poly-utherane foam on all sides.

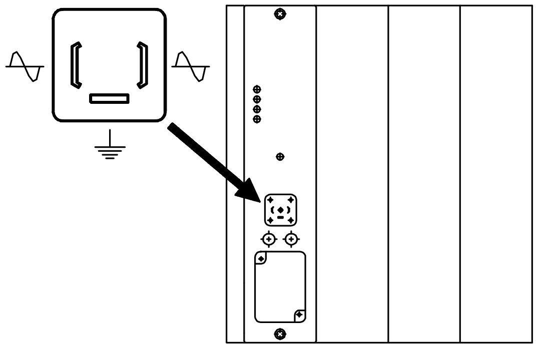
5.- Seal the cardboard box with packing tape or industrial staples.

ADDITIONAL REMARKS

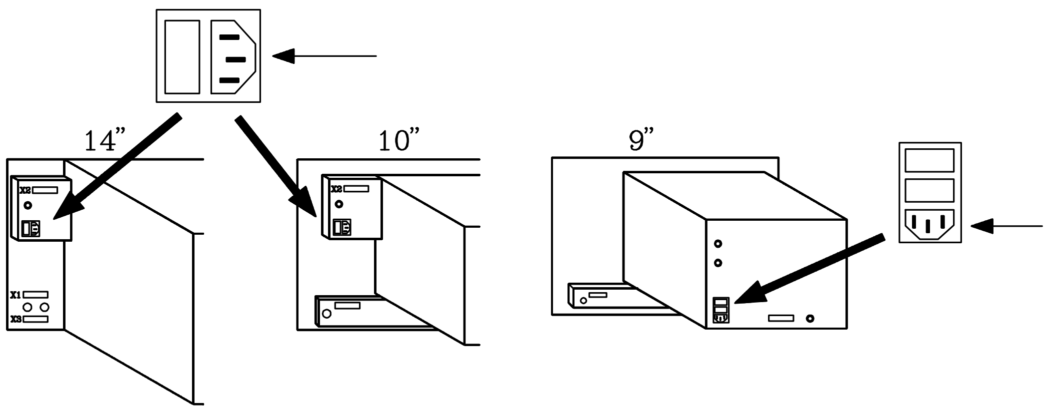
* Mount the CNC away from coolants, chemical products, blows, etc. which could damage it.

* Before turning the unit on, verify that the ground connections have been properly made. See Section 2.2 of this manual.

* To prevent electrical shock at the Central Unit, use the proper mains AC connector at the Power Supply Module. Use 3-wire power cables (one for ground connection)

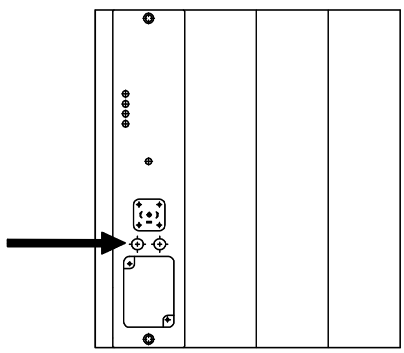


* To prevent electrical shock at the Monitor, use the proper mains AC connector at the Power Supply Module. Use 3-wire power cables (one for ground connection)



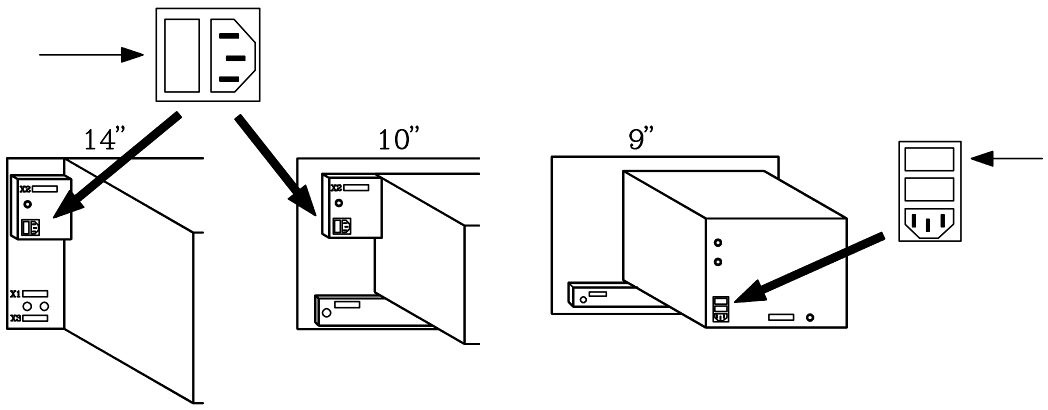
* Before turning the unit on, verify that the external AC line fuse, of each unit, is the right one.

Central Unit (Power Supply Module)
Must be 2 fast fuses (F) of 3.15 Amp./ 250V.



Monitor

Depends on the type of monitor. See identification label of the unit itself.



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

FAGOR DOCUMENTATION
FOR THE 8050 CNC

8050 CNC OEM Manual	Is directed to the machine builder or person in charge of installing and starting-up the CNC. It is common to CNC models: 8050-M and 8050-T and it has the Installation manual inside
8050-M CNC USER Manual	Is directed to the end user or CNC operator. It contains 2 manuals: Operating Manual describing how to operate the CNC. Programming Manual describing how to program the CNC.
8050-T CNC USER Manual	Is directed to the end user or CNC operator. It contains 2 manuals: Operating Manual describing how to operate the CNC. Programming Manual describing how to program the CNC.
8050 DNC Software Manual	Is directed to people using the optional 8050 DNC communications software.
8050 DNC Protocol Manual	Is directed to people wishing to design their own DNC communications software to communicate with the 8050.
AUTOCAD 8050 Manual	Is directed to people wishing to design their own customized CNC screens and symbols on AUTOCAD. This manual indicates how to set up the Autocad program for the CNC to correctly interpret the designed screens and symbols.
FLOPPY DISK Manual	Is directed to people using the Fagor Floppy Disk Unit and it shows how to use it.

MANUAL CONTENTS

The Installation manual is common to CNC models: 8050-M and 8050-T and it is divided into the following sections:

1. CONFIGURATION OF THE FAGOR 8050 CNC

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

It can control machine movements and devices.

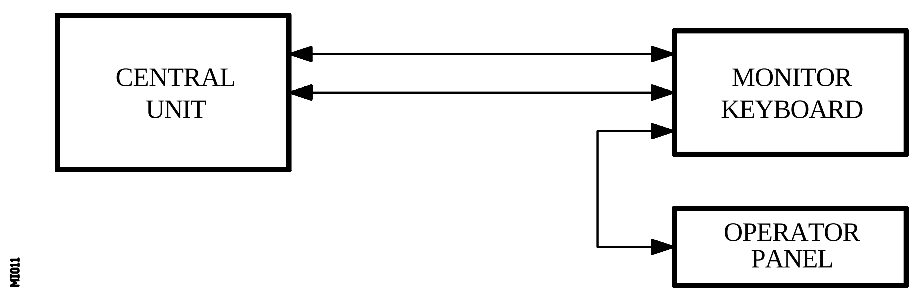
1.1 STRUCTURE OF THE 8050 CNC.

The FAGOR 8050 CNC is composed by the following modules:

- **CENTRAL UNIT**
- **MONITOR/KEYBOARD**
- **OPERATOR PANEL**

The OPERATOR PANEL module is connected to the MONITOR/KEYBOARD module by means of the provided cable. These two modules are placed next to each other and will be connected to the CENTRAL UNIT module by means of the provided two cables. This module may be installed at a different location away from the other two modules but no farther than 25m (82ft.). These cables are:

- Video signal cable
- Keyboard signal cable



1.2 CENTRAL UNIT

The CENTRAL UNIT is located in the electrical cabinet, it has a modular structure and its basic configuration is as follows:

- **POWER SUPPLY MODULE**
- **CPU MODULE**
- **AXES MODULE**
- **FAN MODULE**

This configuration may be expanded by adding 1, 2 or 3 units of another module called:

- **INPUT/OUTPUT MODULE**
- **INPUT/OUTPUT AND TRACING MODULE (I/O TRACING)**

When shorter CNC block processing and sampling times are required, optional CPU-PLC and CPU-TURBO boards should be used.

These boards may be inserted into the "AXES" module, into the "I/O" module or into the "I/O AND TRACING" module. One board per module.

These boards free the main CPU of the CNC from the following tasks:

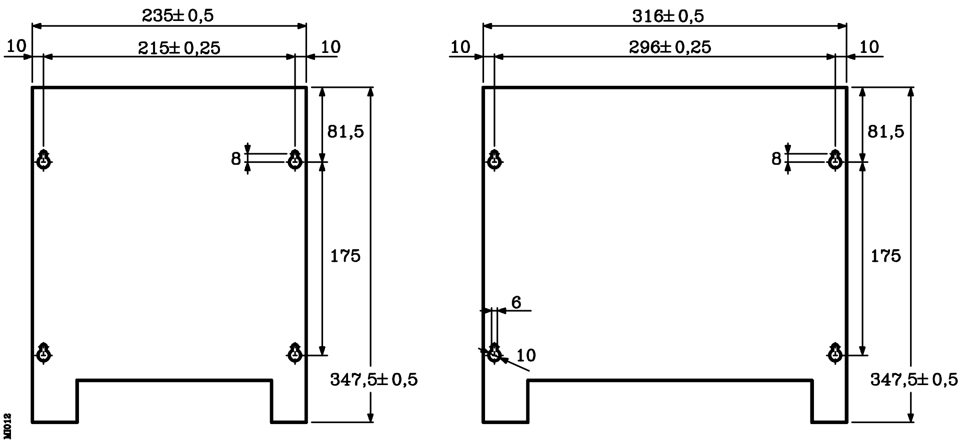
* The CPU-PLC board manages the PLC program.
* The CPU-TURBO manages the positioning loop of the axes, their interpolation, the LOOK-AHEAD function, etc.

The modules forming the CENTRAL UNIT are installed on a RACK PANEL and they are attached to it by two screws on the back of each module.

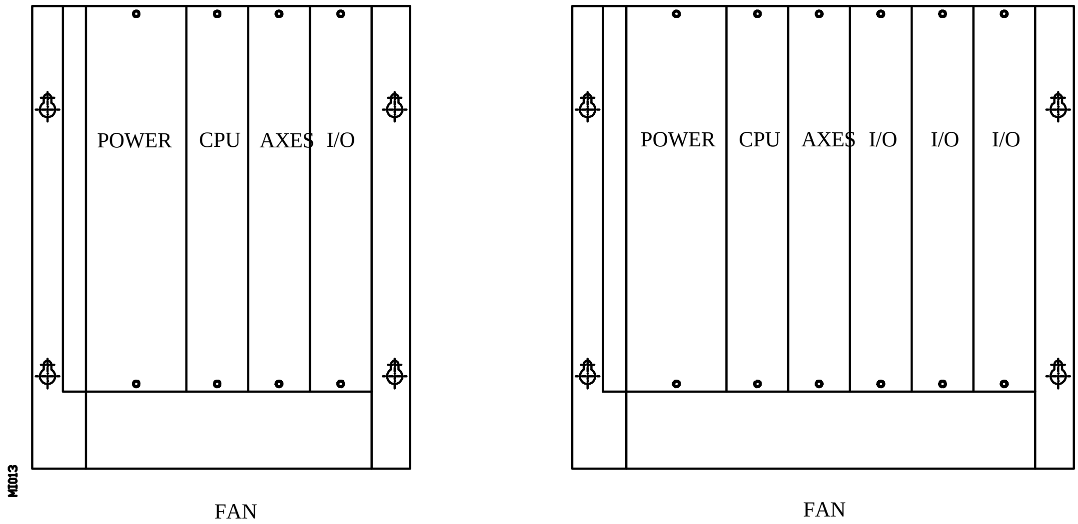
The data exchange between the various modules is carried out via the connection bus on the RACK PANEL. Each module will be connected to this bus when its mounting screws are fastened.

There are two types of RACK PANELS depending on the desired configuration: one conceived for up to 5 modules (FAN MODULE included) and the other one for up to 7 modules (FAN MODULE included).

All dimensions are in millimeters.



The POWER SUPPLY module must be installed as first module from the left side and the CPU MODULE must go next to it (to its right). The rest of the modules (AXES, I/O and I/O TRACING) do not require a specific order and may be interchanged at will.



The FAN MODULE must be installed underneath the other ones and attached to the RACK by two nuts. The FAN MODULE provided will be different depending on the type of configuration ordered.

Each module has a logic address (device select code), regardless of the physical position it occupies, that identifies it to the CNC itself.

The factory-set logic address for each module is as follows:

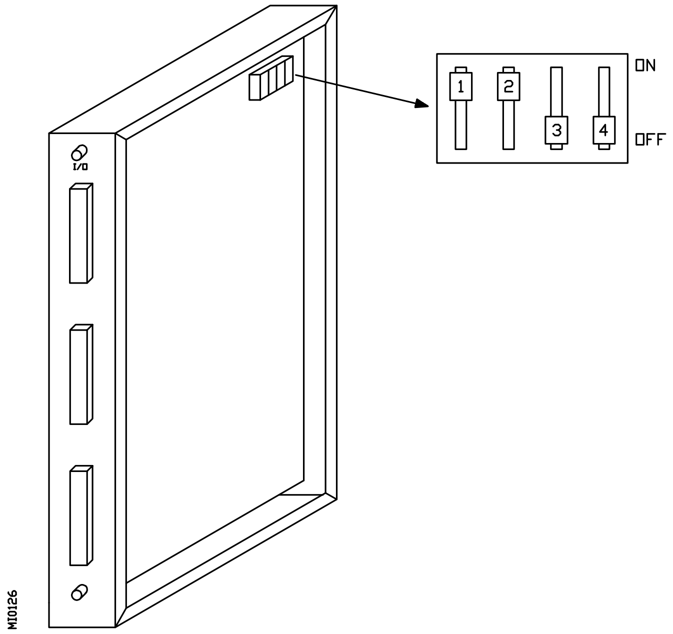
Module	Logic address (device select code)
AXES	2
I/O 1	3
I/O 2	4
I/O 3	5
I/O TRACING	6

The following options also have their own logic address (device select code):

CPU-PLC	1
CPU-TURBO	7

However, it is possible to change the assignment of these addresses.

To do this, remove the module cover located to its right and access the microswitches located at one of the corners of the PC board.



The logic address (device select code) is defined in binary and can be selected between 1 and 14. Addresses 0 and 15 are reserved.

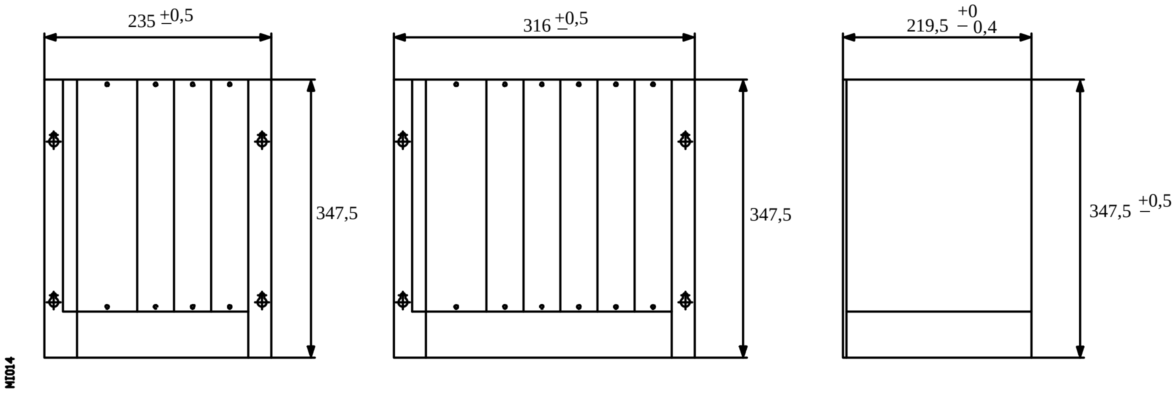
Logic Address	**Switch position**			
	1	**2**	**3**	**4**
0	OFF	OFF	OFF	OFF
1	OFF	OFF	OFF	ON
2	OFF	OFF	ON	OFF
3	OFF	OFF	ON	ON
4	OFF	ON	OFF	OFF
5	OFF	ON	OFF	ON
6	OFF	ON	ON	OFF
7	OFF	ON	ON	ON
8	ON	OFF	OFF	OFF
9	ON	OFF	OFF	ON
10	ON	OFF	ON	OFF
11	ON	OFF	ON	ON
12	ON	ON	OFF	OFF
13	ON	ON	OFF	ON
14	ON	ON	ON	OFF
15	ON	ON	ON	ON

When having several I/O modules, I/O(1) will be the one with the lowest address, I/O(2) will be the next one up and I/O(3) will have the highest address of all three.

1.2.1 DIMENSIONS AND INSTALLATION

The CENTRAL UNIT is attached to the electrical cabinet by means of the two screw holes provided for that purpose. Make sure that the FAN module is installed underneath it.

All dimensions are in millimeters.



1.2.2 POWER SUPPLY MODULE

This module converts the A.C. power voltage into the different D.C. voltages required by the other modules.

The CNC central unit must be powered by a separate shielded transformer of 110 VA with an output between 100V AC and 240V AC + 10% -15%.

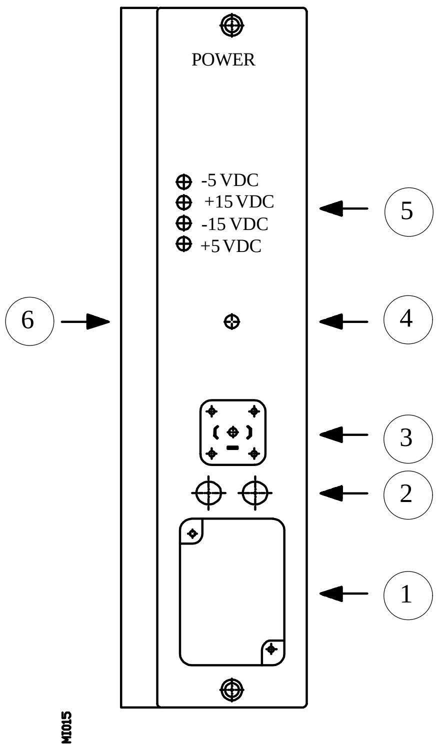
In case of a power surge (outside of the 100V to 240V range), the power supply will shut off, it is recommended to wait for about 3 minutes before powering the CNC up again.

For further information, please refer to the appendix corresponding to the technical specifications of the FAGOR 8050 CNC.

Atention:

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

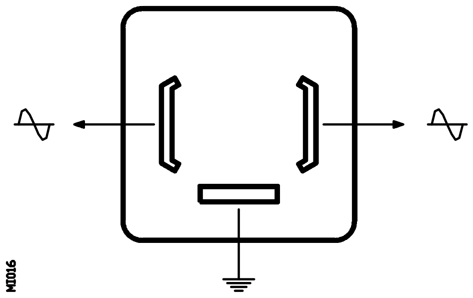
1.2.2.1 ELEMENT DESCRIPTION



1.- **Lithium battery**. Maintains the RAM data when the system's power disappears.

2.- **Main AC fuses (2)**. It has two 3.15Amp./250V fast fuses (F), one per line to protect the Main AC input.

3.- **A.C. power connector**. To power the central unit. It must be connected to the transformer and to ground.



4.- **Ground terminal**. It must be connected to general machine ground point. Metric 6 mm.

5.- **LEDs.**. When on, they indicate that the various supply voltages required by other modules are O.K.

6.- **Heat-sink.**

<u>Atention:</u>

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

1.2.3 CPU MODULE

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

The EPROMS containing the system's software are located in a removable cartridge in order to ease future replacements. Note: Optional CPU PLC will contain its own EPROMs

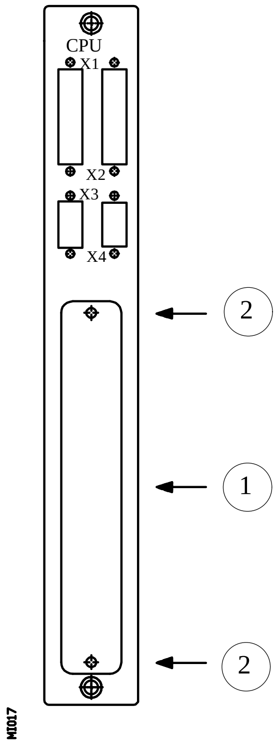
Also the interconnection connectors for the CENTRAL UNIT and the MONITOR/ KEYBOARD are located in this module.

<u>Atention:</u>

When replacing the CPU module, the CNC must be powered back on IMMEDIATELY in order to prevent the lithium battery from excessive drainage.

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

1.2.3.1 ELEMENT DESCRIPTION



X1 SUB-D type 25-pin female connector to connect the CENTRAL UNIT with the KEYBOARD.

X2 SUB-D type 25-pin male connector to connect the CENTRAL UNIT with the MONITOR.

X3 SUB-D type 9-pin female connector to connect the SERIAL PORT RS 232 C.

X4 SUB-D type 9-pin male connector to connect the SERIAL PORT RS 422.

1.- EPROM Cartridge. Is removable and contains the system's software.

2.- knurled screws. To fasten the EPROM cartridge to the module.

<u>Atention:</u>

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

1.2.3.2 CONNECTORS AND CONNECTIONS

Connector X1

SUB-D type 25-pin female connector to connect the CENTRAL UNIT with the KEYBOARD.

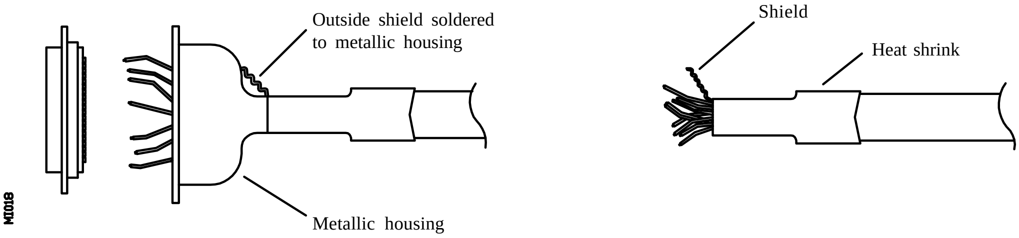
FAGOR AUTOMATION provides the cable necessary for this connection. This cable has two 25-pin male connectors of the SUB-D type.

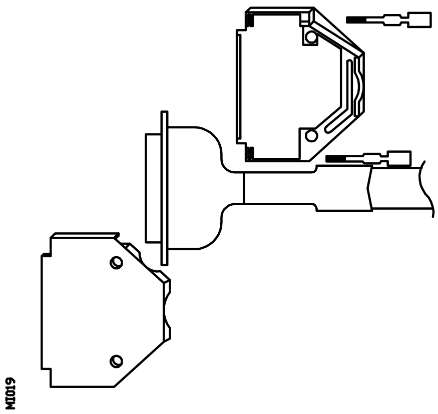
Both connectors have a latching system by means of two screws UNC4.40.

PIN	CABLE COLOR	SIGNAL
1	green	GND
2	green-brown	C9
3	blue	C11
4	white	C13
5	black	C15
6	brown-red	C1
7	red	C3
8	pink-brown	C5
9	gray-brown	C7
10	red-blue	D1
11	brown-blue	D3
12	yellow-white	D5
13	yellow-brown	D7
14	white-black	C8
15	white-green	C10
16	white-red	C12
17	white-gray	C14
18	white-blue	C0
19	white-pink	C2
20	pink	C4
21	gray	C6
22	blown	D0
23	yellow	D2
24	gray-pink	D4
25	purple	D6
Metal housing	Shield	Chassis

The cable has 25 wires of 0.14 mm² (25 x 0.14mm²) section, global shielding and it is covered by acrylic rubber. The maximum length permitted is 25m (82ft).

The cable shield is soldered to the metallic housing of the connectors. This shield is connected to pin 1 at both the CENTRAL UNIT and KEYBOARD connectors,





Connector X2

25-pin male connector of the SUB-D type to connect the CENTRAL UNIT with the MONITOR.

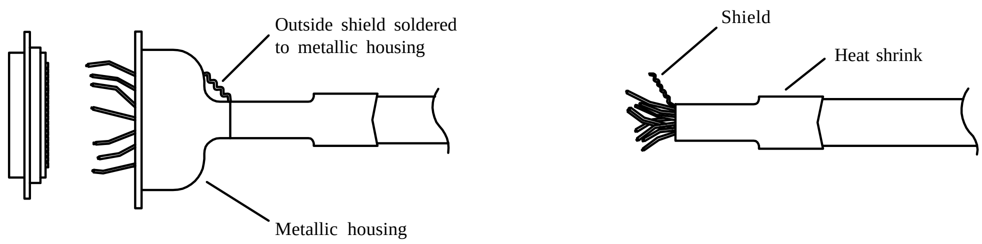
FAGOR AUTOMATION provides the cable necessary for this connection. This cable has two 25-pin female connectors of the SUB-D type.

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

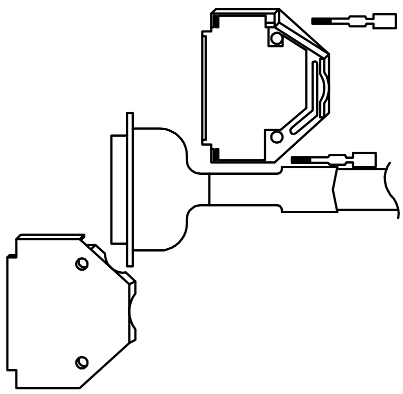
PAIR	CABLE COLOR	PIN	SIGNAL
	Shield	Metal housing	CHASSIS
1	blue white	4 17	VD4 $\overline{\text{VD4}}$
2	red yellow	5 18	VD3 $\overline{\text{VD3}}$
3	green pink	7 20	HSINC $\overline{\text{HSINC}}$
4	orange white-black	8 21	VSINC $\overline{\text{VSINC}}$
5	red gray	9 22	VD0 $\overline{\text{VD0}}$
6	green white	10 23	VD1 $\overline{\text{VD1}}$
7	blue gray	11 24	VD2 $\overline{\text{VD2}}$
8	black brown	12 25	BLANK $\overline{\text{BLANK}}$

The cable has 8 twisted-pair wires of 0.34 mm² (8 x 2 x 0.34mm²) section with global shielding and it is covered with acrylic rubber. It has a specific impedance of 120 ohms and the maximum permitted length is 25m (82ft).

The cable shield is soldered to the metallic housing of the connectors. This shield is connected to pin 1 at both the CENTRAL UNIT and MONITOR connectors,



MI018



MI019

Connector X3

9-pin female connector of the SUB-D type to connect the RS 232 C serial port.

The cable shield must be soldered to pin 1 at the CNC end and to the metallic housing at the peripheral end.

PIN	SIGNAL	FUNCTION
1	FG	Shield
2	TxD	Transmit Data
3	RxD	Receive Data
4	RTS	Request To Send
5	CTS	Clear To Send
6	DSR	Data Send Ready
7	GND	Ground
8		
9	DTR	Data Terminal Ready

SUGGESTIONS FOR THE RS 232C INTERFACE

* Connect/disconnect peripheral

> ⚠ The CNC must be powered off when connecting or disconnecting any peripheral through connector X3 (connector for the RS232C interface).

* 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 7 0.14 mm^2 section.

* Transmission speed (baudrate)

The most common baudrate used with peripherals is 9600 baud; but the CNC can operate at up to 19200 baud.

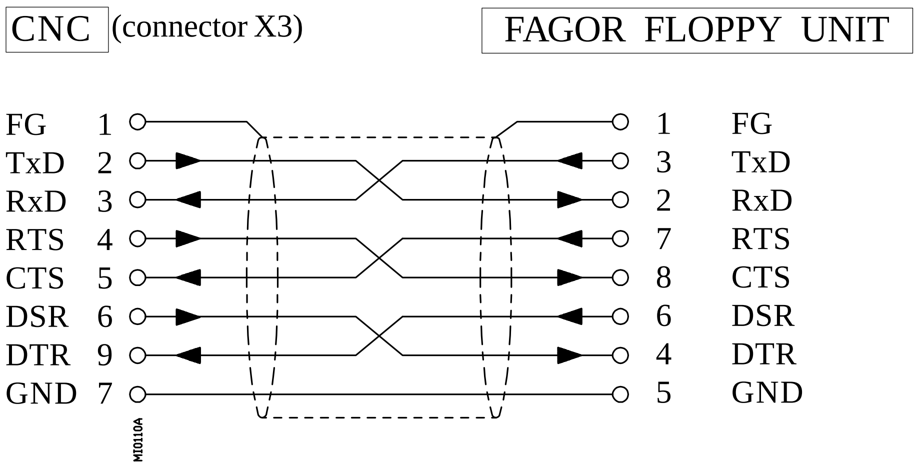
All unused wires should be grounded to avoid erroneous control and data signals.

* 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 CONNECTIONS FOR THE RS 232 C INTERFACE

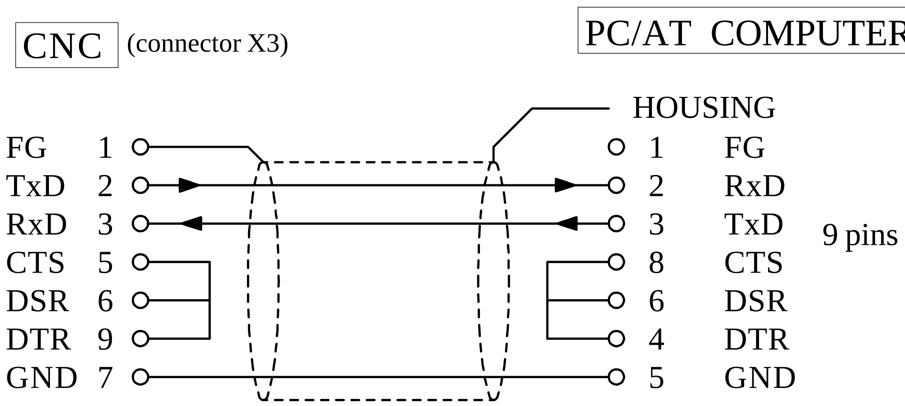
*** Recommended connection for FAGOR floppy disc unit.**

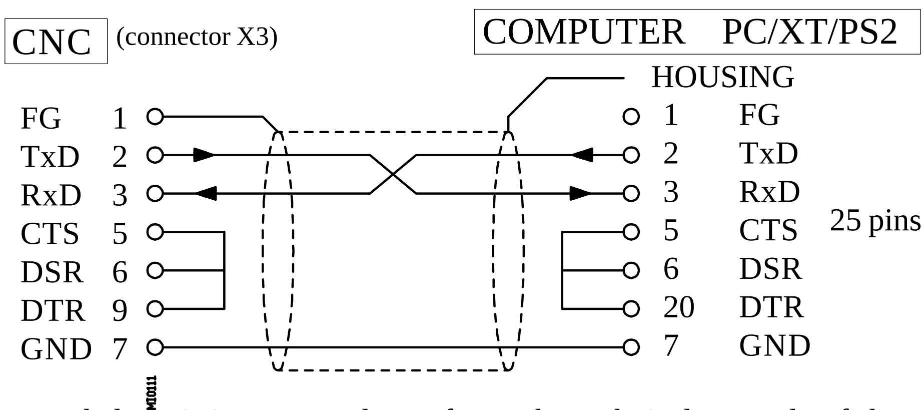


*** Simplified connection**

To be used when the peripheral or the computer meets one of the following requirements:

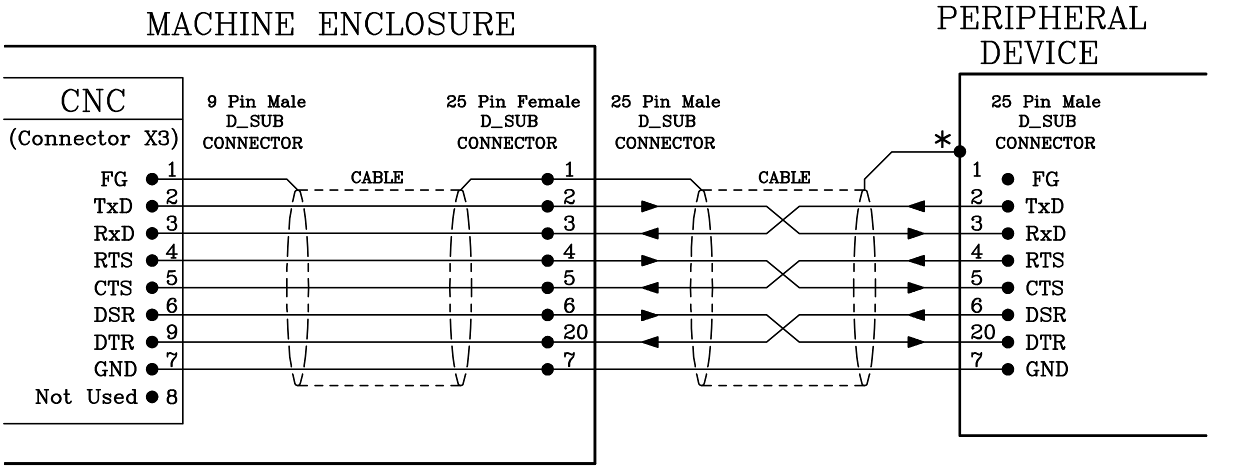
Does not have the RTS signal
It is connected via DNC
The receiver can receive data at the selected baudrate



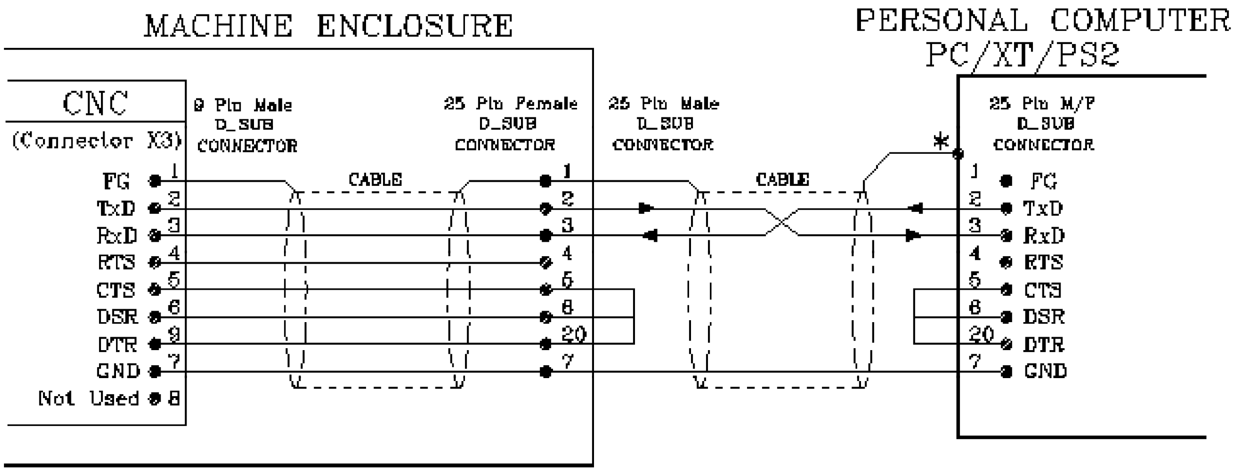


Nevertheless, it is suggested to refer to the technical manuals of the peripheral equipment if there is any discrepancy.

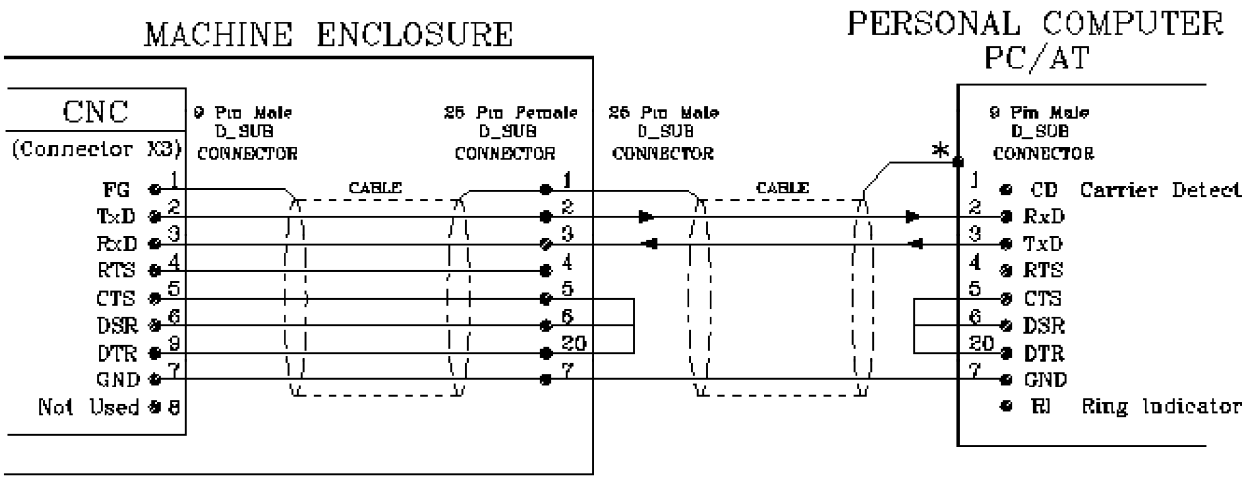
RS232C Connections



NOTE: VERIFY THE GENDER OF THE SERIAL PORT CONNECTOR USED ON YOUR PERIPHERAL DEVICE BEFORE MAKING THE CABLE ASSEMBLY



NOTE: VERIFY THE GENDER OF THE SERIAL PORT CONNECTOR USED ON YOUR PERSONAL COMPUTER BEFORE MAKING THE CABLE ASSEMBLY



* It is recommended to connect the cable shield to the Peripheral device or Personal Computer chassis in order to improve transmissions

Connector X4

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

The cable shield must be soldered to pin 1 at the CNC end and to the metallic housing at the peripheral end.

PIN	SIGNAL	FUNCTION
1	FG	Shield
2		
3	$\overline{\text{TxD}}$	Transmit data
4	$\overline{\text{RxD}}$	Receive data
5	RxD	Receive data
6		
7	GND	Ground signal
8	TxD	Transmit data
9		

CONSIDERATIONS REGARDING THE RS422 INTERFACE

It uses two separate wires for the signals. This offers the following advantages:

- It increases noise immunity.
- The data transmission distance, at the same baudrate, is greater.
- The problems due to signal interference and different voltage references are minimized.

The RS422 standard defines the electrical interface to be used and it can be used in conjunction with the RS449 standard.

A terminating resistor must be installed between pins 3 and 8 (data transmit) and between pins 4 and 5 (Receive Data). These resistors must be installed on both connectors. Their values must match the cable´s impedance.

Typical value: 120 Ohm 1/4W

*** Transmission speed (baudrate)**

The common baudrate between the CNC and a peripheral device or computer is 9600 Baud. However the CNC can operate at up to 19200 Baud.

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

RECOMMENDED CABLE FOR THE RS422

TECHNICAL CHARACTERISTICS
BUÑOFLES COMPUTER PAR 3x2x0.34 mm2 Individual shield + Overall shield (polyester/aluminum)

SPECIFICATIONS	
Conductor	Type: 7x0.25 mm (twisted-pair) Material: Tin-plated copper Resistance: 52 Ohms/Km (32 Ohms/mile)
Insulation	Material: Solid polyethylene
Shields	Shields: Polyester/aluminum band Stranded tin-plated copper wire 7x0.25 mm
Outside cover	Material: metal gray PVC
Capacitance	Between conductors: 91,7 pF/m at 1 KHz Between a conductor and the rest connected to shield: 180 pF/m at 1 KHz
Impedance	50 Ohms

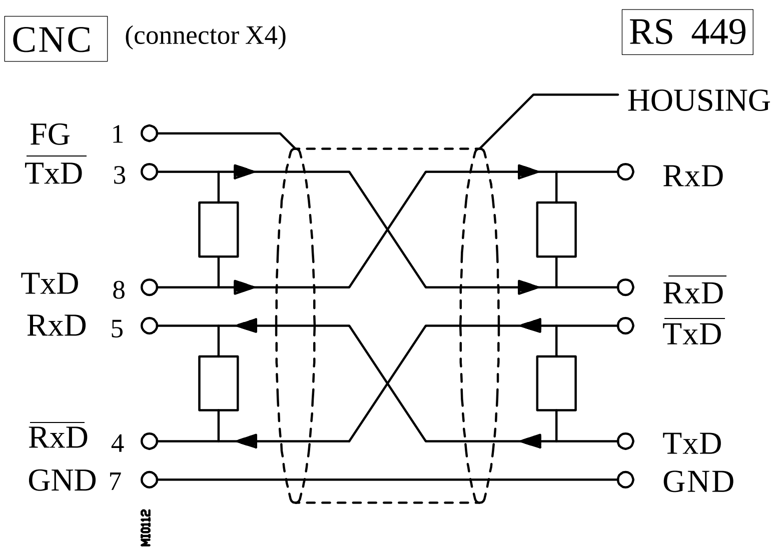
RECOMMENDED CONNECTIONS

* Connect/disconnect peripheral

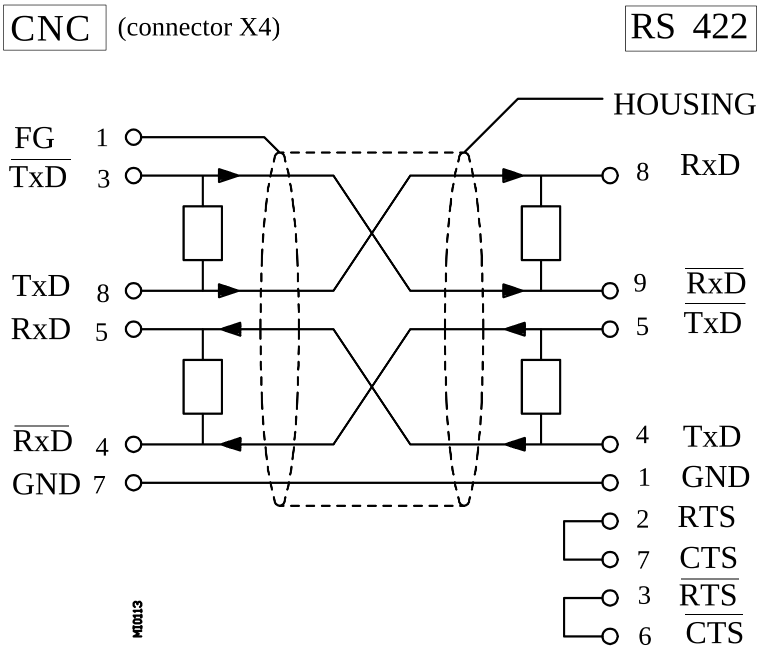
The CNC must be powered off when connecting or disconnecting any peripheral through connector X4 (RS422).

* Connection with serial port RS449

It is suggested to refer to the technical manual of the peripheral or computer to properly identify the signals and their corresponding pins.

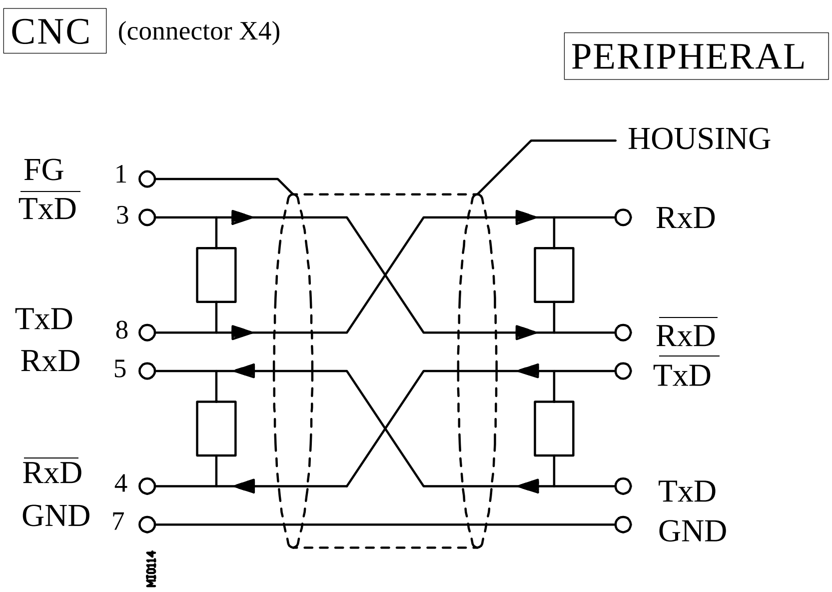


* **Connection with an RS422 interface board from METRABYTE**

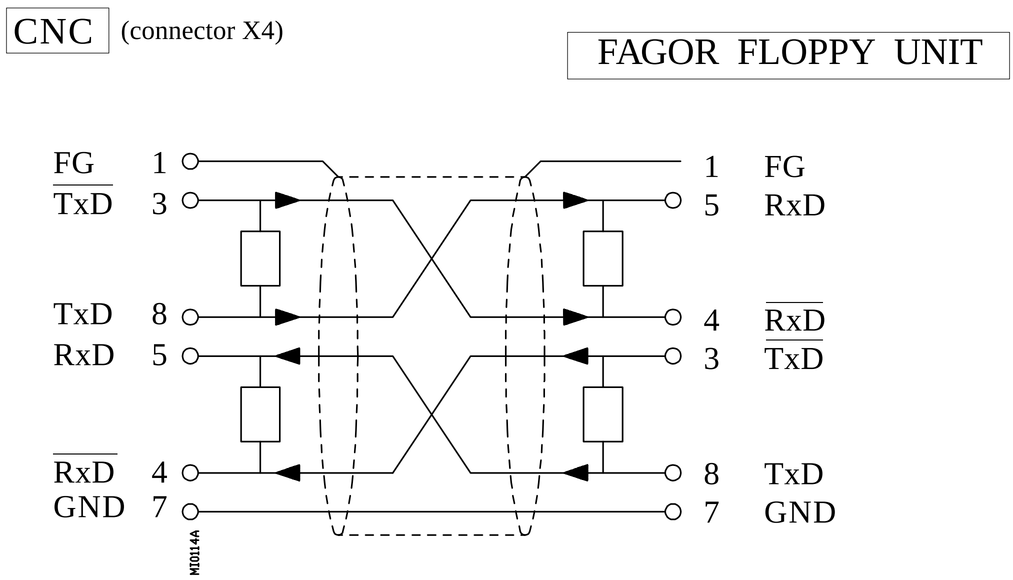


* **Connection with other peripherals**

It is suggested to refer to the technical manual of the peripheral or computer to properly identify the signals and their corresponding pins.



*** Connection for FAGOR floppy disc unit.**



1.2.4 AXES MODULE

This module processes the control and feedback signals of up to 6 axes, plus spindle encoder and electronic handwheel.

This module also has a powerful PLC (Programmable Logic Controller) which, thanks to its own CPU, can execute in real time the logic program created by the user.

This module offers the following features to communicate with the outside world:

4 **Feedback inputs** admitting single-ended and double-ended (differential) squarewave signals as well as single-ended sinewave signals.

4 **Feedback inputs** admitting single and double-ended (differential) squarewave signals.

8 **Analog outputs** for the servo drives.

24 **Digital outputs** ,optocoupled, commanded by the PLC.

40 **Digital inputs** ,optocoupled, read by the PLC.

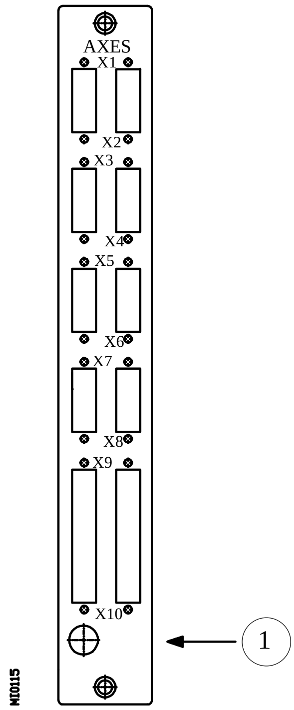
8 **Analog inputs** to use at will on control and supervision systems.

1 **Digital probe input**.

Atention:

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

1.2.4.1 ELEMENT DESCRIPTION.



X1, X2, X3 and X4. SUB-D type 15-pin female connectors for feedback systems of each axis. The accept sinewave signals.

X5 and X6. SUB-D type 15-pin male connectors for feedback system of the axes. Up to 2 axes may be connected per connector. They do not accept sinewave signals.

X7. SUB-D type 15-pin male connector to connect up to 8 analog inputs (range ±5V) and a probe input (TTL or 24V).

X8. SUB-D type 15-pin female connector to connect up to 8 analog outputs (range ±10V).

X9. SUB-D type 37-pin male connector for the 32 PLC digital inputs.

X10. SUB-D type 37-pin female connector for the 8 digital inputs of the PLC and its 24 digital outputs.

1.- 3,15Amp./250V. Fast fuse (F) for internal protection of the PLC inputs and outputs.

Atention:

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

1.2.4.2 *CONNECTORS AND CONNECTIONS*

Connectors X1, X2, X3, X4

They are 25-pin female connectors of the SUB-D type and they are used for the feedback system connections of the axes.

It is required to set general machine parameters AXIS1, AXIS2, AXIS3 and AXIS4 to indicate which axis has been connected to each one of them.

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

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

The appendix at the end of this manual shows the characteristics of the square and sinusoidal feedback inputs and those of the differential feedback alarm signals.

PIN	SIGNAL AND FUNCTION	
1 2 3 4	A $\overline{A}$ B $\overline{B}$	Differential squarewave feedback signals (double ended)
5 6	Io $\overline{Io}$	Machine reference signals (Home marker pulses)
7 8	Ac Bc	Depending on machine parameter, they could be sinewave feedback signals or differential alarm signals generated by certain transducers
9 10 11 12 13 14	+5V +5V 0V 0V -5V -5V	Feedback system power supply
15	CHASSIS	Shield

When using a FAGOR 100P model handwheel, the axis selecting signal must be connected to pin 5 of this connector.

Connectors X5, X6

They are 15-pin male connectors of the SUB-D type utilized for feedback system connections.

It is possible to connect up to 2 axes to each one of them. It is required to set global machine parameters AXIS5, AXIS6, AXIS7 and AXIS8 to indicate which axes have been connected to each connector.

The cables must have global shielding. The rest of the specifications depend on the feedback system utilized and the cable length required.

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

PIN	SIGNAL AND FUNCTION	
1 2 3 4	A $\overline{A}$ B $\overline{B}$	Differential squarewave feedback signals (double ended)
5 6	Io $\overline{Io}$	Machine reference signals (Home marker pulses)
7 8	+5V 0V	Feedback system power supply
9 10 11 12	A $\overline{A}$ B $\overline{B}$	Differential squarewave feedback signals (double ended)
13 14	Io $\overline{Io}$	Machine reference signals (Home marker pulses)
15	CHASSIS	Shield

When using a FAGOR 100P model handwheel, the axis selecting signal must be connected to pin 5 of this connector.

The appendix at the end of this manual shows the characteristics of the square and sinusoidal feedback inputs and those of the differential feedback alarm signals.

Connector X7

It is a 15-pin male connector of the SUB-D type utilized for touch probe input and for the analog inputs.

It is possible to connect up to 8 analog inputs which could be used for system supervision, etc. Their analog value must be within ± 5V.

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

Refer to the appendix at the end of this manual for more details on the characteristics of these probe inputs and recommended interface connections.

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

The meaning or use of **pin 11** of this connector has been changed for those models corresponding to version **09A** and newer (See serial number on the module's top cover). The previous models used this pin as 0V input for the analog signals. But from this version **09A** on, it is used as the +5V output to feed the probe.

For models prior to version 09A

PIN	SIGNAL AND FUNCTION	
1	I01	Analog inputs, range ±5V
2	I02	
3	I03	
4	I04	
5	I05	
6	I06	
7	I07	
8	I08	
9	0V	
10	0V	
11	0V	
12	PROBE 5	Probe input 5V TTL
13	PROBE 24	Probe input 24V Vcc
14	PROBE 0	Probe input 0V
15	CHASSIS	Shield

For models with version 09A and newer

PIN	SIGNAL AND FUNCTION	
1	I01	Analog inputs, range ±5V
2	I02	
3	I03	
4	I04	
5	I05	
6	I06	
7	I07	
8	I08	
9	0V	
10	0V	
11	+5V	+5V output to feed the probe
12	PROBE 5	Probe input 5V TTL
13	PROBE 24	Probe input 24V Vcc
14	PROBE 0	Probe input 0V
15	CHASSIS	Shield

Atention:

When using pin 11 as +5V power supply output for the probe, pin 14 (PROBE 0) must be connected to either pin 9 or 10 (0V) of this connector.

The machine manufacturer must comply with the EN 60204-1 (IEC-204-1) regulation regarding the protection against electrical shock derived from defective input/output connection with the external power supply when this connector is not connected before turning the power supply on.

Connector X8

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

Each one of the outputs (O1 thru O8) correspond to the feedback inputs X1 thru X6. The name of the axis connected to each one of them is determined by setting global machine parameters AXIS1 thru AXIS8.

All shields must only be connected to ground at the CNC end leaving the other end free.

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

PIN	SIGNAL AND FUNCTION	
1 2 3 4 5 6 7 8	O01 O02 O03 O04 O05 O06 O07 O08	Servo analog outputs Range ±10V
9 10 11 12 13 14	GND GND GND GND GND GND	Reference signal for the analog outputs
15	CHASSIS	Shield

Connector X9

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

Since the response time of the EMERGENCY signal must be very short, the FAGOR 8050 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 0V of the power supply used for these inputs must be connected to pins 18 and 19 of the connector.

PIN	SIGNAL AND FUNCTION	
1		
2	I01	$\overline{\text{EMERGENCY STOP}}$
3	I03	
4	I05	
5	I07	
6	I09	
7	I11	
8	I13	
9	I15	
10	I17	
11	I19	
12	I21	
13	I23	
14	I25	
15	I27	
16	I29	
17	I31	
18	0V	External power supply
19	0V	
20		
21	I02	
22	I04	
23	I06	
24	I08	
25	I10	
26	I12	
27	I14	
28	I16	
29	I18	
30	I20	
31	I22	
32	I24	
33	I26	
34	I28	
35	I30	
36	I32	
37	CHASSIS	Shield

Connector X10

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

When an error is issued, the CNC, besides indicating it to the PLC, activates output O01 of this connector. This way, regardless of how this signal is treated by the PLC program, the electrical cabinet can process this signal immediately.

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

PIN	SIGNAL AND FUNCTION	
1	24V	External power supply
2	O01	$\overline{\text{EMERGENCY OUTPUT}}$
3	O03	
4	O05	
5	O07	
6	O09	
7	O11	
8	O13	
9	O15	
10	O17	
11	O19	
12	O21	
13	O23	
14	I33	
15	I35	
16	I37	
17	I39	
18	0V	External power supply
19	0V	
20	24V	External power supply
21	O02	
22	O04	
23	O06	
24	O08	
25	O10	
26	O12	
27	O14	
28	O16	
29	O18	
30	O20	
31	O22	
32	O24	
33	I34	
34	I36	
35	I38	
36	I40	
37	CHASSIS	Shield

Atention:

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).

The machine manufacturer must comply with the EN 60204-1 (IEC-204-1) regulation regarding the protection against electrical shock derived from defective input/output connection with the external power supply when this connector is not connected before turning the power supply on.

1.2.5   *I/O MODULE*

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

Each module provides:

64 Optocoupled digital inputs.
32 Optocoupled digital outputs.

The maximum configuration possible with one axis module and three I/O modules is:

232 Digital inputs (40 + 3 * 64)
120 Digital outputs (24 + 3* 32)

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

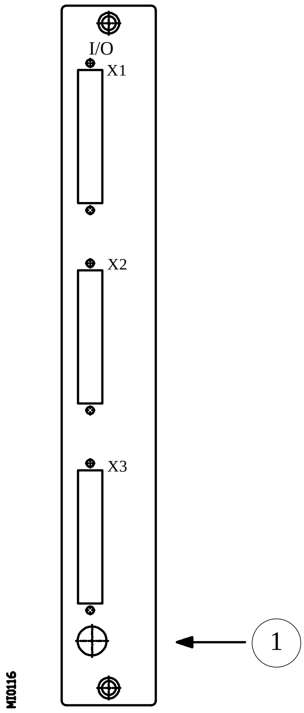
AXES module	I1 -I40	O1 -O24
I/O 1 module	I65 -I128	O33-O64
I/O 2 module	I129-I192	O65-O96
I/O 3 module	I193-I256	O97-O128

The PLC can control up to 256 inputs and 256 outputs although it can only communicate to the outside world through the ones indicated above.

Atention:

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

1.2.5.1 *ELEMENT DESCRIPTION*



X1 and X2. 37-pin male connector of the SUB-D type for 64 digital inputs of the INTEGRATED PLC.

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

1.- 3.15Amp./250V Fast fuse (F) for internal circuitry protection of the INTEGRATED PLC (PLCI) inputs and outputs.

Atention:

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

1.2.5.2 CONNECTORS AND CONNECTIONS

Connectors X1, X2

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

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

Connector X1

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

Connector X2

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

Connector X3

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

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

PIN	SIGNAL AND FUNCTION	
1	24V	External power supply
2	O33	
3	O35	
4	O37	
5	O39	
6	O41	
7	O43	
8	O45	
9	O47	
10	O49	
11	O51	
12	O53	
13	O55	
14	O57	
15	O59	
16	O61	
17	O63	
18	0V	External power supply
19	0V	
20	24V	External power supply
21	O34	
22	O36	
23	O38	
24	O40	
25	O42	
26	O44	
27	O46	
28	O48	
29	O50	
30	O52	
31	O54	
32	O56	
33	O58	
34	O60	
35	O62	
36	O64	
37	CHASSIS	Shield

Atention:

Te machine manufacturer must comply with the EN 60204-1 (IEC-204-1) regulation regarding the protection against electrical shock derived from defective input/output connection with the external power supply when this connector is not connected before turning the power supply on.

1.2.6 FAN MODULE

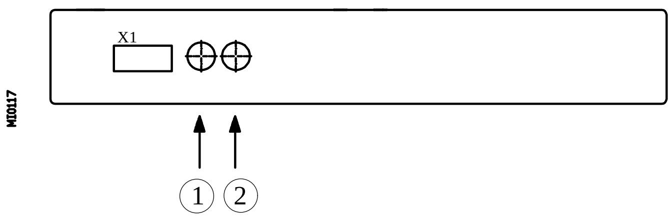
This module, whose purpose is to maintain the equipment at the proper operating temperature, may have one or two fans depending on the selected configuration.

It must be powered by a 24V (+15% -25%) D.C. power supply separate from the rest of the equipment.

Atention:

Do not maipulate inside this unit
Only personnel authorized by Fagor Automation may manipulate inside this module.

1.2.6.1 ELEMENT DESCRIPTION



X1 **Power supply connector**

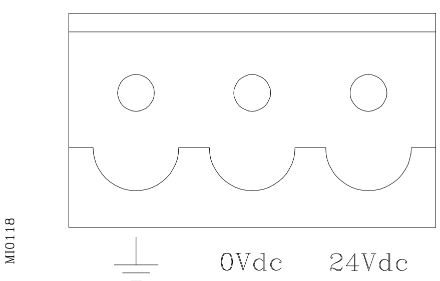
1.- **3,15Amp./250V fast fuse (F)** to protect the fan.

2.- **3,15Amp./250V fast fuse (F)** to protect the other fan.

1.2.6.2 CONNECTORS

Connector X1

3-pin male connector of the WEIDMÜLLER type used for the PLC power supply.



1.2.7 I/O AND TRACING MODULE

This module is used to expand the number of digital inputs and outputs of the basic configuration and it allows the possibility to use the **Renishaw SP2** probe, for tracing parts.

The internal adaptor that this module has for the Renishaw SP2 probe multiplies the signals received by a factor of 2, thus obtaining a resolution of 1 micron (40 millionths of an inch).

This module also provides:

32 Optocoupled digital inputs.
32 Optocoupled digital outputs.

The maximum configuration possible with one **AXES** module, the **I/O AND TRACING** module and two **I/O** modules is:

200 Digital inputs (40 + 32 + 2 x 64)
120 Digital outputs (24 + 32 + 2 x 32)

The numbering of the various inputs and outputs of each module depends on the logic address assigned to the module in such way that the first group of I/Os corresponds to the lowest address and the last one to the highest address. for example:

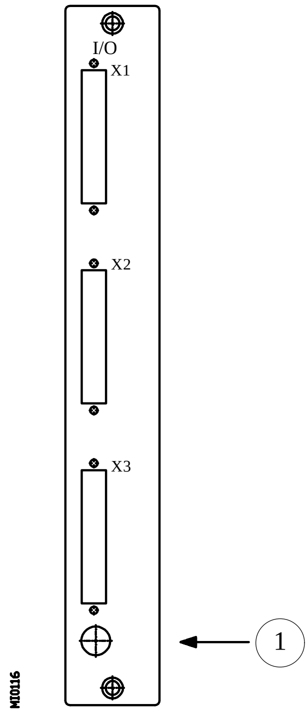
MODULE	LOGIC ADDRESS	I/Os	
AXES	2	I1-I40	O1-O24
I/O TRACING	3	I65-I96	O33-O64
I/O (1)	4	I129-I192	O65-O96
I/O (2)	5	I193-I256	O97-O128

The PLC can control up to 256 inputs and 256 outputs although it can only communicate to the outside world through the ones indicated above.

Atention:

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

1.2.7.1 ELEMENT DESCRIPTION



X1 . 25-pin female connector of the SUB-D type to connect the Renishaw **SP2** probe.

X2. 37-pin male connector of the SUB-D type for 32 digital inputs of the INTEGRATED PLC.

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

1.- 3.15Amp./250V Fast fuse for internal circuitry protection of the INTEGRATED PLC (PLCI) inputs and outputs.

Atention:

Do not manipulate the connectors with the unit connected to main AC power

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

1.2.7.2 *CONNECTORS AND CONNECTIONS*

Connector X1

25-pin female connector of the SUB-D type used to connect the Renishaw **SP2** probe.

FAGOR AUTOMATION provides the union cable required for this connection. It consists of a cable hose, one SUB-D type 25-pin male connector and the corresponding connector for the Renishaw **SP2**.

The male connector has a latching system by means of two UNC4.40 screws.

The cable used has 12 conductors of $0.14mm^2$ section, global shield and it is covered by acrylic rubber.

CNC PIN	COLOR	SIGNAL	FUNCTION	RENISHAW PIN
1	Brown	Xa	Sin X	N
2		-	Reserved	
3		-	Reserved	
4	Yellow	Ya	Sin Y	G
5		-	Reserved	
6		-	Reserved	
7	Green	Za	Sin Z	M
8		-	Reserved	
9		-	Reserved	
10	Pink	OVER 1	Overtravel Alarm	K
11	Pink/Grey	OVER 2	Overtravel Alarm	V
12		-	Reserved	
13		-	Reserved	
14	Purple	Xb	Cos X	E
15		-	Reserved	
16	Grey	Yb	Cos Y	C
17		-	Reserved	
18	White	Zb	Cos Z	H
19		-	Reserved	
20		-	Reserved	
21		-	Reserved	
22	Red	+15V	Power Supply	U
23	Black	0V	Power Supply	A
24	Blue	-15V	Power Supply	B
25	Shield	Chassis	Shield	R

Connector X2

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

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

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

Connector X3

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

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

PIN	SIGNAL AND FUNCTION	
1	24V	External power supply
2	O33	
3	O35	
4	O37	
5	O39	
6	O41	
7	O43	
8	O45	
9	O47	
10	O49	
11	O51	
12	O53	
13	O55	
14	O57	
15	O59	
16	O61	
17	O63	
18	0V	External power supply
19	0V	
20	24V	External power supply
21	O34	
22	O36	
23	O38	
24	O40	
25	O42	
26	O44	
27	O46	
28	O48	
29	O50	
30	O52	
31	O54	
32	O56	
33	O58	
34	O60	
35	O62	
36	O64	
37	CHASSIS	Shield

1.3 *MONITOR / KEYBOARD*

This module, installable on the pendant, allows the operator to access all the information provided by the MONITOR as well as operate the CNC by means of the keyboard and the operator panel.

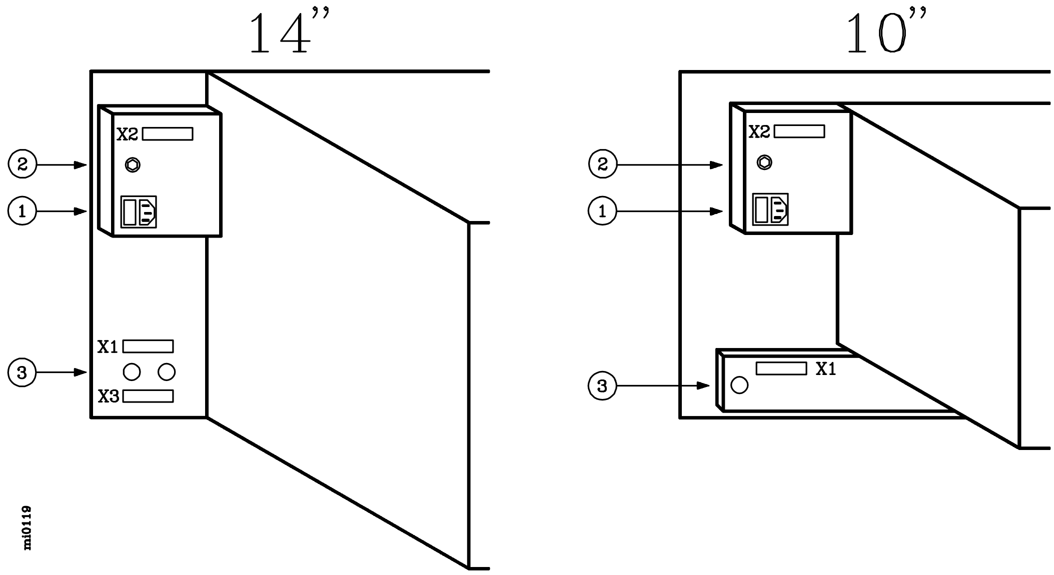
This module contains the connectors for the interconnections between the CENTRAL UNIT and the MONITOR/KEYBOARD unit.

Atention:

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

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

1.3.1 ELEMENT DESCRIPTION



X1 **25-pin female connector of the SUB-D type** to connect the keyboard signals.

X2 **25-pin male connector of the SUB-D type** to connect the video signals.

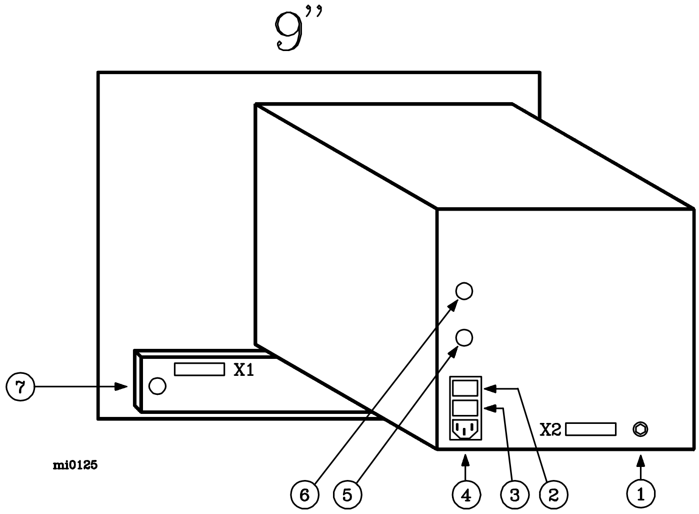
X3 **15-Pin female connector of the SUB-D type** to connect the KEYBOARD with the OPERATOR PANEL.

1.- **A.C. power connector:** Use the connector provided for this purpose connecting it to 220V A.C. and ground.

2.- **Ground terminal**: To connect the general machine ground connections. Metric screw 6mm.

3.- **Buzzer**

9 " monitor



X1 **SUB-D type 25-in female connector** to connect the KEYBOARD and the CENTRAL UNIT.

X2 **SUB-D type 25-in male connector** to connect the MONITOR and the CENTRAL UNIT.

1.- **Ground terminal:** Used for the general ground connection. Metric 6mm.

2.- **A.C. power fuses (2).** One for each line.

3.- **On/off power switch.**

4.- **A.C. power connector:** for A.C. power and ground connection.

5.- **MONITOR contrast adjusting knob.**

6.- **MONITOR brightness adjusting knob.**

7.- **Buzzer.**

1.3.2 CONNECTORS AND CONNECTIONS

Connectors X1, X2

They are described in the section for the CPU module of the CENTRAL UNIT.

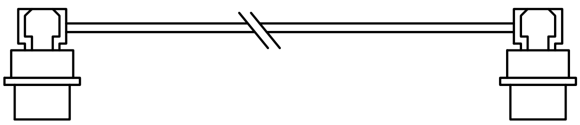
Connector X3 on 14" monitors

15-pin male connector of the SUB-D type used to connect the KEYBOARD with the OPERATOR PANEL.

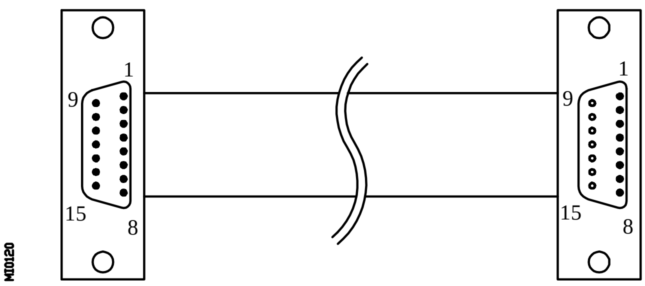
FAGOR AUTOMATION provides the 15-conductor ribbon-cable required for this connection. It is 250mm long (about 10 inches).

When a greater separation between the Monitor/keyboard and the Operator panel is desired, this cable must be replaced with a 15-conductor cable of 15 0.14mm^2 section, global shielding and covered with acrylic rubber. The maximum total length of the cable connecting the Central Unit and the Keyboard must not exceed the 25m (82ft).

PIN	SIGNAL
1	
2	uC13
3	uC12
4	jC11
5	jC10
6	JC9
7	D7
8	D6
9	D5
10	D4
11	D3
12	D2
13	D1
14	D0
15	D14

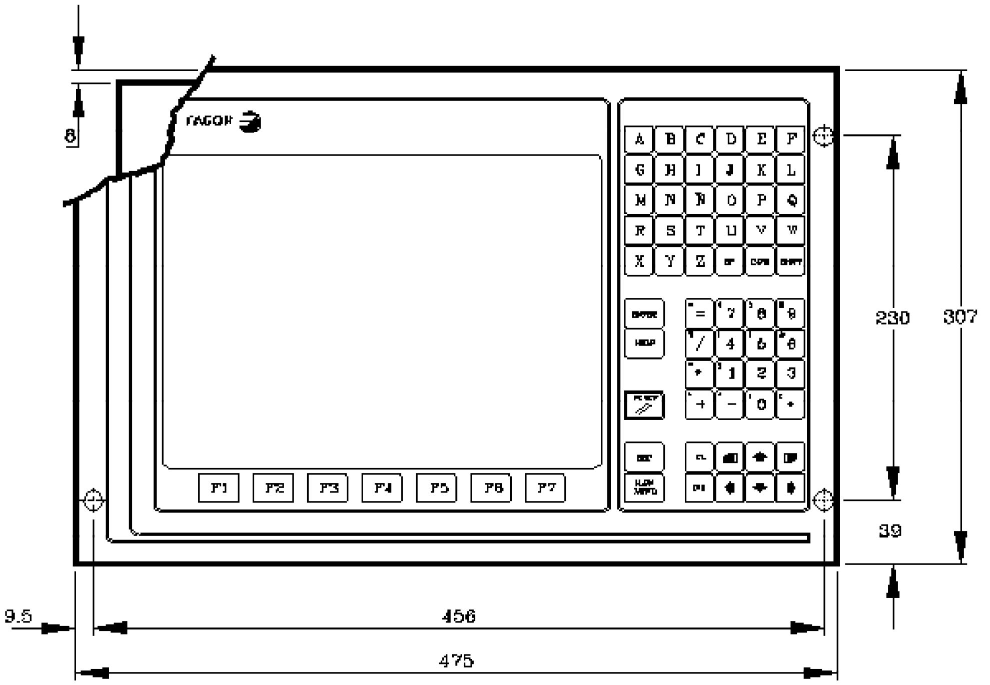


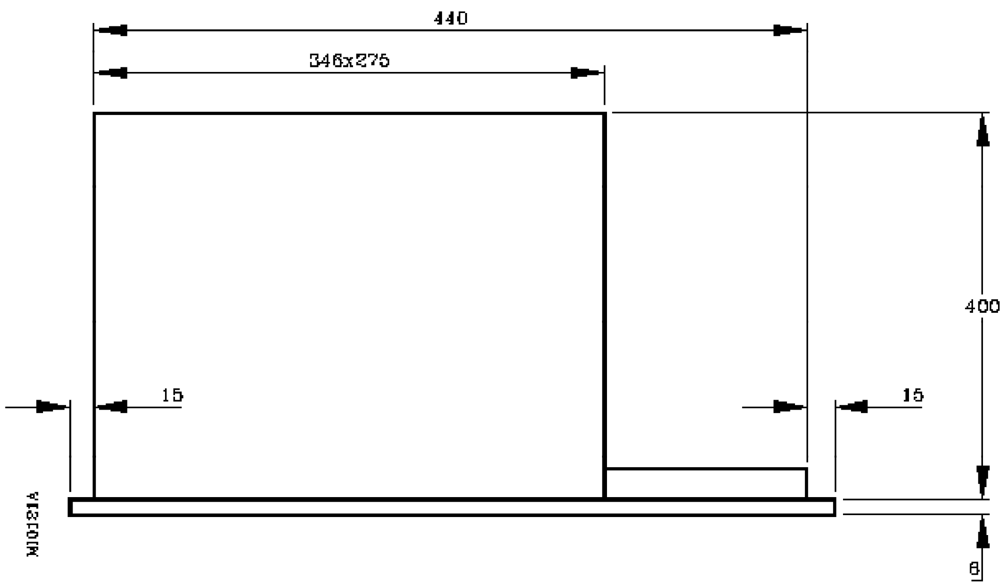
KEYBOARD side OPERATOR PANEL side



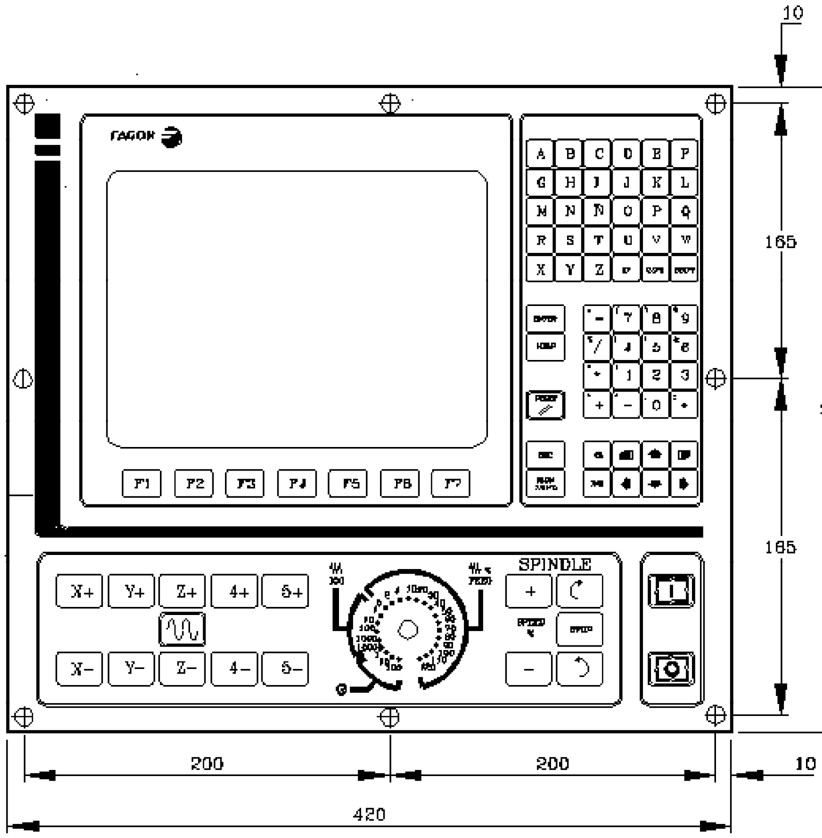
1.3.3 *DIMENSIONS OF THE MONITOR/KEYBOARD (mm)*

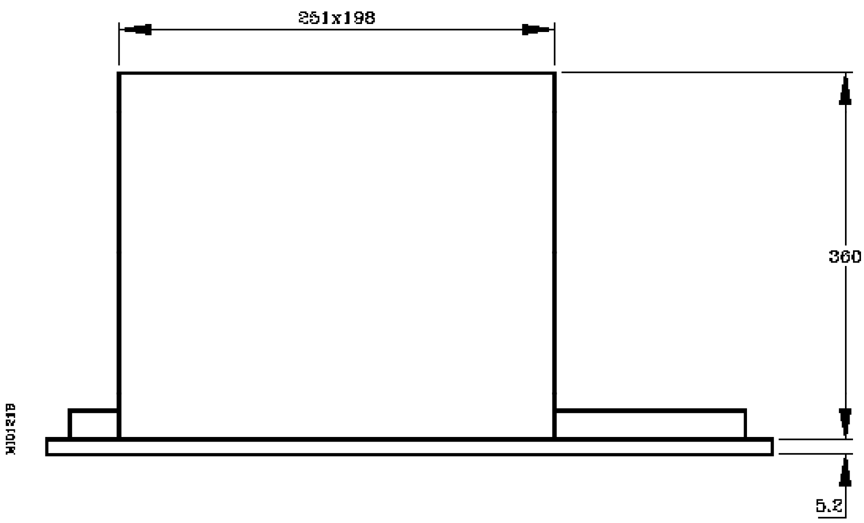
(14" monitor)



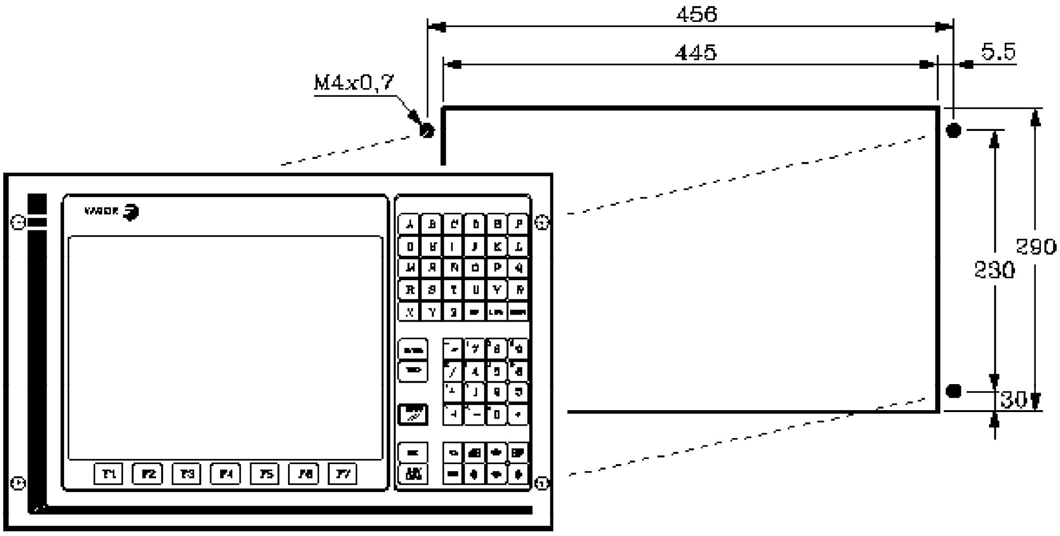


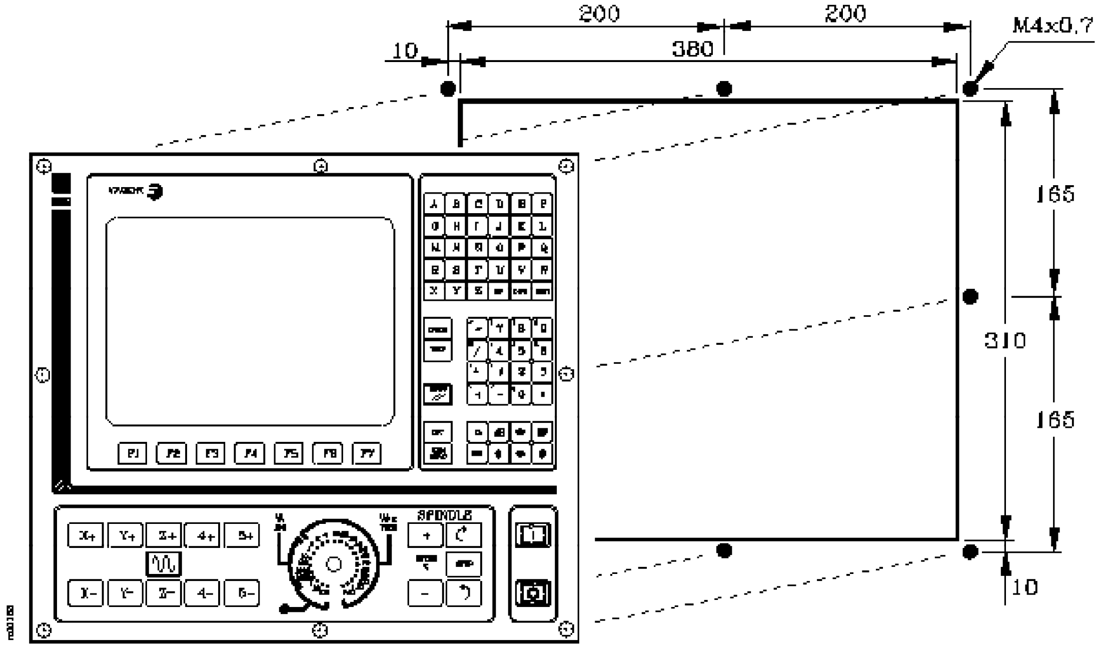
(9" and 10" monitors)



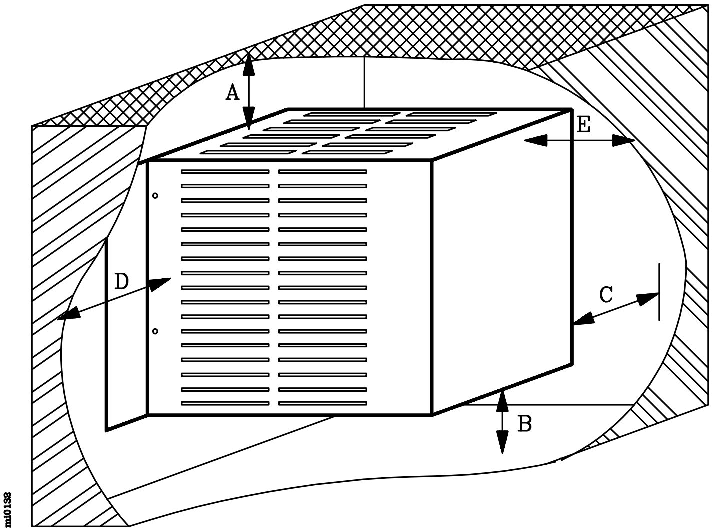


1.3.4 MONITOR/KEYBOARD ENCLOSURES (mm)





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



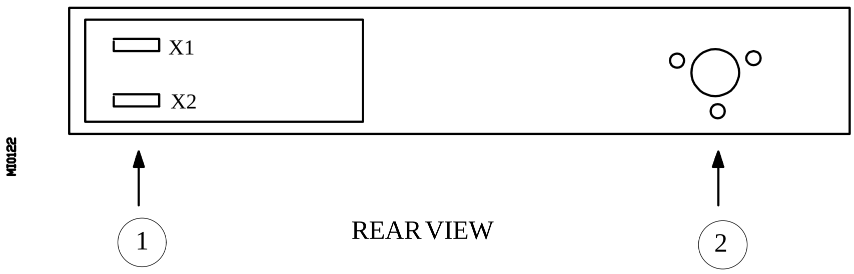
Monitor	A	B	C	D	E
9", 10"	100 mm	100 mm	100 mm	100 mm	150 mm
14"	100 mm	100 mm	100 mm	150 mm	50 mm

A fan must be used to improve ventilation inside the enclosure. **This fan must be D.C.** powered since A.C. powered fans generate magnetic fields which could distord the image on the CRT.

1.4 OPERATOR PANEL

This module, which is connected to the MONITOR/KEYBOARD by a ribbon cable, contains the keys to Jog the axes of the machine (Feedrate Override Knob, normal speed and fast jog keys), Cycle Start and Cycle Stop keys as well as either an Emergency Stop button or an electronic Handwheel (optional).

1.4.1 ELEMENT DESCRIPTION



X1 **15-pin female connector of the SUB-D type** to connect the OPERATOR PANEL with the MONITOR/KEYBOARD

X2 **Not used.**

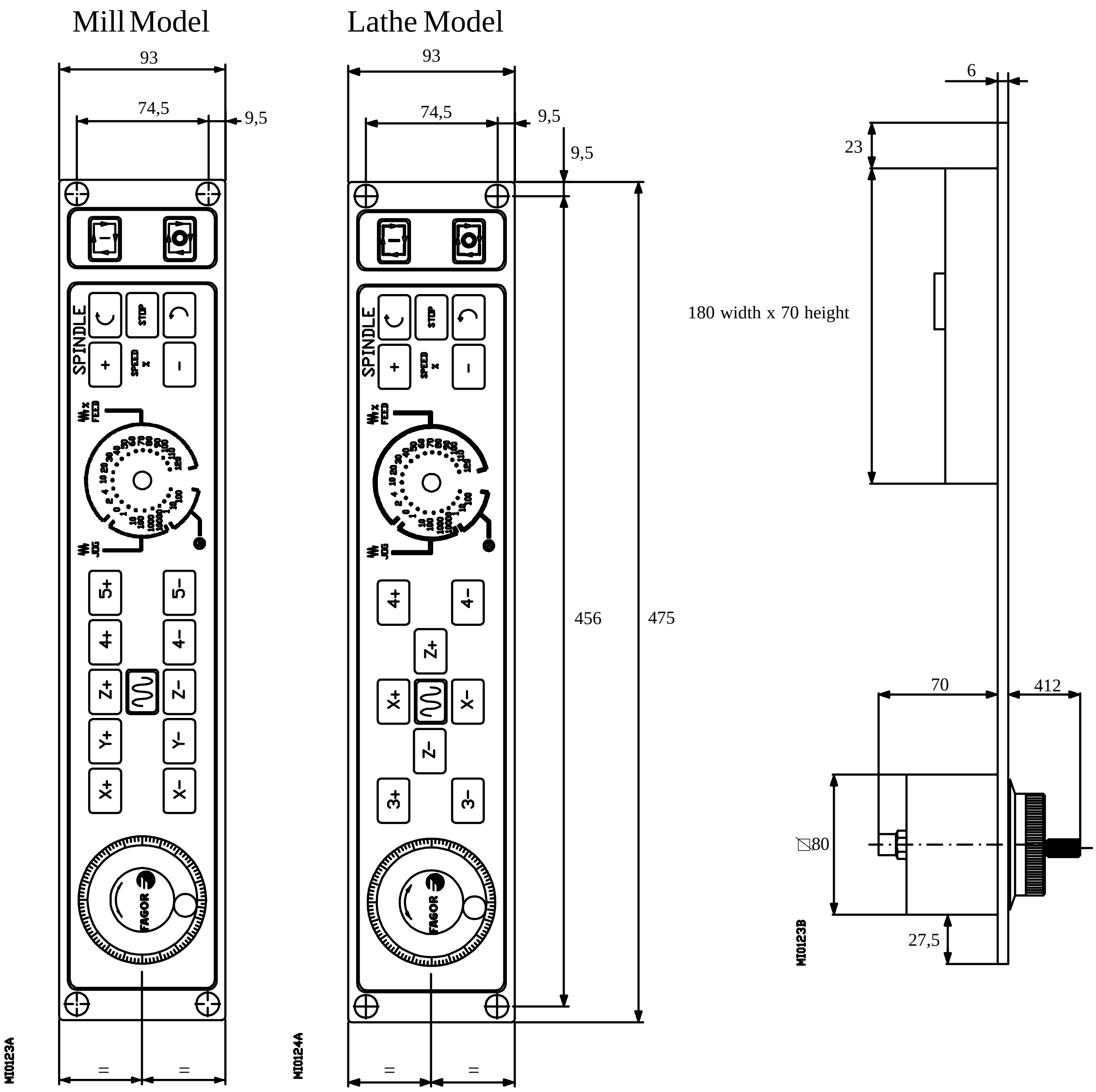
2.- **Optional connection** for the E-STOP button or the electronic Handwheel

1.4.2 CONNECTORS AND CONNECTIONS

Connector X1

It is described in the section for the MONITOR/KEYBOARD.

1.4.3 DIMENSIONS OF THE OPERATOR PANEL (mm)



2. *POWER AND MACHINE CONNECTION*

Atention:

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.

2.1 POWER CONNECTION

The CENTRAL UNIT of the FAGOR 8050 CNC has a three-prong connector located in the POWER SUPPLY MODULE to be connected to A.C. power and ground.

This connection must be done through an independent shielded 110VA transformer with an A.C. output voltage between 100V and 240V +10% -15%.

The FAN MODULE, located in the central unit, must be powered with a 24V D.C. power supply independent of the rest of the equipment.

The MONITOR/KEYBOARD must be powered with 220V A.C.

2.2 MACHINE CONNECTION

2.2.1 GENERAL CONSIDERATIONS

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

* D.C. relay coils.

  Diode type 1N4000.

* A.C. relay coils.

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

  R 220 Ohms/1W
  C 0,2 µF/600V

* A.C. motors.

  RC connected between phases with values:

  R 300 Ohms/6W
  C 0,47µF/600V

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.

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.

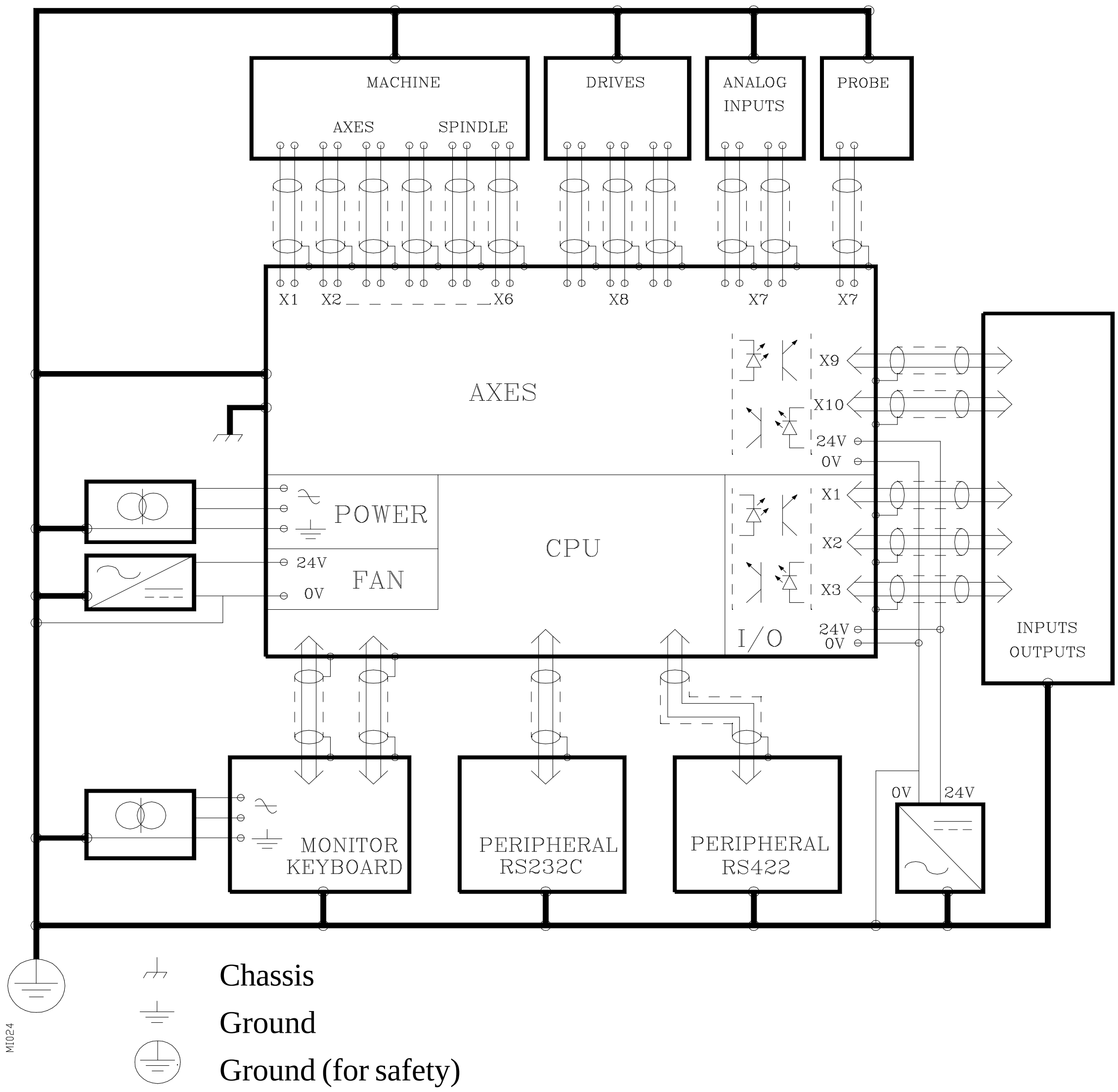
Even when a proper **ground** connection reduces the effects of electrical interference (noise), the signal cables require additional protection.
This is generally achieved by using twisted-pair cables which are also covered with anti-static shielding mesh-wire. This shield must be connected to a specific point avoiding **ground loops t**hat could cause undesired effects. This connection is usually done at one of CNC's ground point.

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

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

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

Ground connection diagram



2.2.2 DIGITAL OUTPUTS.

The 8050 CNC system offers a number of optocoupled digital outputs which can be used to activate relays, deacons, etc.

These digital outputs, corresponding to the PLC, are located at the AXES MODULE and at the I/O MODULE. Each of these modules provides:

* 24 outputs on the AXES module (connector X10).
* 32 outputs on each of the I/O modules (connector X3).

All these outputs, with galvanic protection by optocouplers, can commutate D.C. voltages supplied by the electrical cabinet of the machine.

The electrical characteristics of these outputs are:

Nominal voltage value	+24 V D.C.
Maximum voltage value	+30 V.
Minimum voltage value	+18 V.
Output voltage	2V less than power supply voltage Vcc.
Maximum output current	100 mA.

All outputs are protected by means of:

Galvanic isolation by optocouplers
External 3A fuse for:
protection against external power supply surges (over 33VCC) and
protection against reversal connection of the power supply.

2.2.3 DIGITAL INPUTS.

The digital inputs offered by the 8050 CNC system are used to read external devices, etc.

These digital inputs, corresponding to the PLC, are located on the AXES module and on the I/O module. Each one of them contains:

* 40 inputs on the AXES module (connectors X9, X10).
* 64 inputs on each of the I/O modules (connectors X1, X2).

All of them offer galvanic isolation, between the CNC circuitry and the outside world, by optocouplers .

The electrical characteristics of these inputs are:

Nominal voltage value	+24V D.C.
Maximum voltage value	+30V.
Minimum voltage value	+18V.
High threshold voltage (logic level 1)	+18V.
Low threshold voltage (logic level 0)	+5V.
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.

Atention:

The external 24V 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.

2.2.4 ANALOG OUTPUTS.

The 8050 CNC system offers 8 analog outputs which could be used to command servo drives, spindle drives and other devices.

These analog outputs are located on the AXES module (connector X8).

The electrical characteristics of these outputs are:

Analog voltage range:... ±10V
Minimum impedance of the connected drive: 10KW
Maximum cable length without shield: .... 75 mm (3 inches)

It is highly recommended to use shielded cable connecting the shield to the corresponding pin of the X8 connector at the AXES module.

<u>Atention:</u>

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

2.2.5 ANALOG INPUTS.

The 8050 CNC offers 8 analog inputs used for supervision, control, etc. of external devices.

These analog inputs are located in the AXES module (connector X7).

The electrical characteristics of these inputs are:

Analog voltage range: ....±5V
Input impedance: 20KW
Maximum cable length without shield: ...75mm (3 inches)

It is highly recommended to use shielded cable connecting the shield to the corresponding pin of the X7 connector at the AXES module.

2.3 START UP

2.3.1 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.

Verify that the 24V power supply used for the digital inputs and outputs of the PLC is REGULATED and that its 0V are connected to the Main Ground Point.

Verify the connection of the feedback system cables to the CNC.

DO NOT connect or disconnect these cables to/from the CNC when the CNC is on.

Look for shortcircuits in all connectors (inputs, outputs, axes, feedback, etc.) BEFORE supplying power to them.

2.3.2 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.

2.3.3 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 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.

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

Connect the A.C. power to the CNC. If there is any problem, the CNC will display the corresponding error. Install the machine parameters and PLC program.

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

2.3.4 MACHINE PARAMETER SETTING

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 chapter dedicated to 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.

Once the new parameter values are entered, key in **SHIFT** and then **RESET** or turn the CNC off and back on so these new values are assumed by the CNC.

2.3.5 ADJUSTMENT OF THE MACHINE PARAMETERS FOR THE AXES

Once the active axes have been assigned by means of general machine parameters "AXIS1" thru "AXIS8", 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 starting the adjustment of the axes, it is a good idea to move them close to the middle of their travels placing the travel-limit switches (controlled by the electrical cabinet) close to these points in order to avoid any 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.

* Move the axis being adjusted in the JOG mode.

  In case of run-away, 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** (axis feedback counting direction) and **LOOPCHG** (sign of the analog output) will have to be changed.

2.3.6 MACHINE REFERENCE POINT ADJUSTMENT FOR EACH AXIS (HOME)

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

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

* This adjustment should be done one axis at a time.

* Indicate in the Axis Machine Parameter for the axis **REFPULSE** the type of marker pulse Io being used for Home Search.

* Set Axis Machine Parameter **REFDIREC** to indicate the direction of the axis when searching Home.

* Set General Machine Parameters **REFEED1** and **REFEED2** to indicate the feedrates for Home search.

* Set Axis Machine Parameter **REFVALUE** to 0.

* Once the JOG mode has been selected at the CNC, position the axis so the Home search can be carried out in the desired direction and execute the home search from this JOG mode. When the search is completed, the CNC will assign a 0 position value to this point (Machine Reference **Point**).

* If the Machine Reference **Zero** desired is in a different physical location from the Machine Reference **Point** (location of the marker pulse), proceed as follows:

  move the axis to a known position and the value displayed by the CNC will be the value to be assigned to Axis Machine Parameter **REFVALUE**.

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

  Example:

  After homing and moving the axis to a known point, if this known point is 3 inches away from the desired Machine **Zero** and the CNC shows it to be -1.7 inches away from the Machine Reference **Point** (marker pulse location); the distance from the Reference **Point** to the Reference **Zero** will be:

  "REFVALUE" = 3 - (-1.7) = 4.7 inches.

  Assign this new value to **REFVALUE** and press SHIFT RESET or turn the CNC off and back on so it assumes this new value.

  If REFVALUE is other than 0, it is necessary to search Home once again in order for this axis to assume the correct reference values.

2.3.7 SOFTWARE TRAVEL LIMITS FOR THE AXES (soft limits)

Once Home Search has been carried out on all the axes, the soft limits for the CNC have to be established.

This is achieved a single axis at a time and in the following manner:

* Jog the axis in the positive direction to a point close to the travel limit switch keeping a safety distance from it.

* Assign the position value displayed by the CNC to Axis Machine Parameter "**LIMIT+**".

* Repeat those steps in the negative direction assigning the displayed value to Axis Machine Parameter "**LIMIT-**".

* 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.

2.3.8 ADJUSTMENT OF THE DRIFT (OFFSET) AND MAXIMUM FEEDRATE (G00)

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

Drift adjustment (offset)

* 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 0mV D.C.

* Take the wire jumper out.

Adjustment of the maximum feedrate

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 Axis Machine Parameter or the Spindle parameter "**MAXVOLT**".

Also, the maximum feedrate must be indicated in the Axis Machine Parameter "**G00FEED".**

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

Example:

A motor can turn at 3000 rpms and it is attached to a 5 pitch screw (5 turns/inch).

The maximum feedrate will be:

3000 rev./min. x 1 inch./5 rev. = 600 inch/min

This will be the value to be assigned to Axis Machine Parameter "**G00FEED**".

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. This program could be:

```
N10 G00 G90 X200
X-200
(GOTO N10)
```

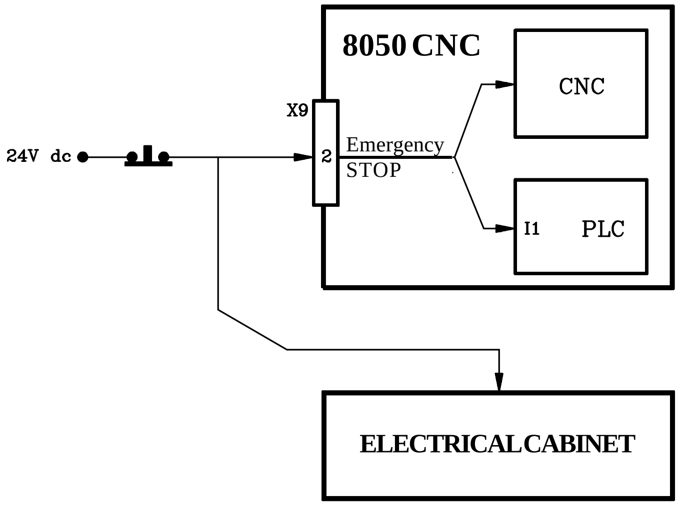
If the Tach in use provides 20V per 1000 rpms, its voltage should be:

$$\frac{20\text{ V.}}{1000\text{ rpm}} \text{ x } 3000\text{ rpm} = 60\text{ V.}$$

2.3.9 *CONNECTION OF THE EMERGENCY INPUT AND OUTPUT*

The EMERGENCY INPUT of the CNC, pin 2 of connector X9 of the AXES module, corresponds with the I01 input of the PLC and must be supplied with 24V.

Since the CNC also processes this signal directly, if the 24V disappears, the CNC will display 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 logic level 0 in order to avoid a premature activation of the electrical cabinet.

If this process is successful, the CNC will release control of the EMERGENCY OUTPUT to the PLC. Otherwise, it will keep it active (low) and it will display the corresponding error message.

Once this process is over, the PLC will execute the PLC program stored in memory if any or, if none is available, it will “wait” for one to be entered and executed.

After completing the execution of the initial cycle (CY1) of the PLC program or, in its absence, the first cycle it will assign the value of the PLC output O1 to the physical EMERGENCY output.

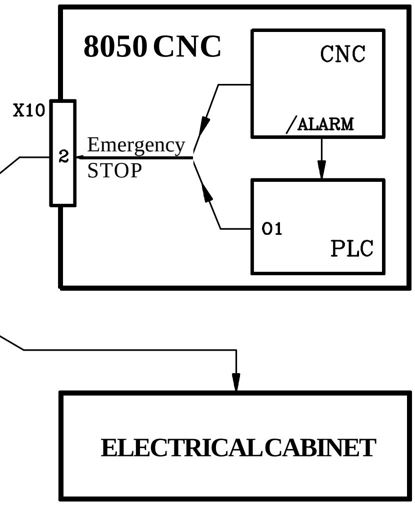
It is recommended to program the CY1 cycle of the PLC program assigning a value of 1 to O1 when everything checks out fine and a value of 0 when there is an error.

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.
* Travel limit switch of any axis has be pressed.
* There is a malfunction on a drive or it is locked without analog signal.

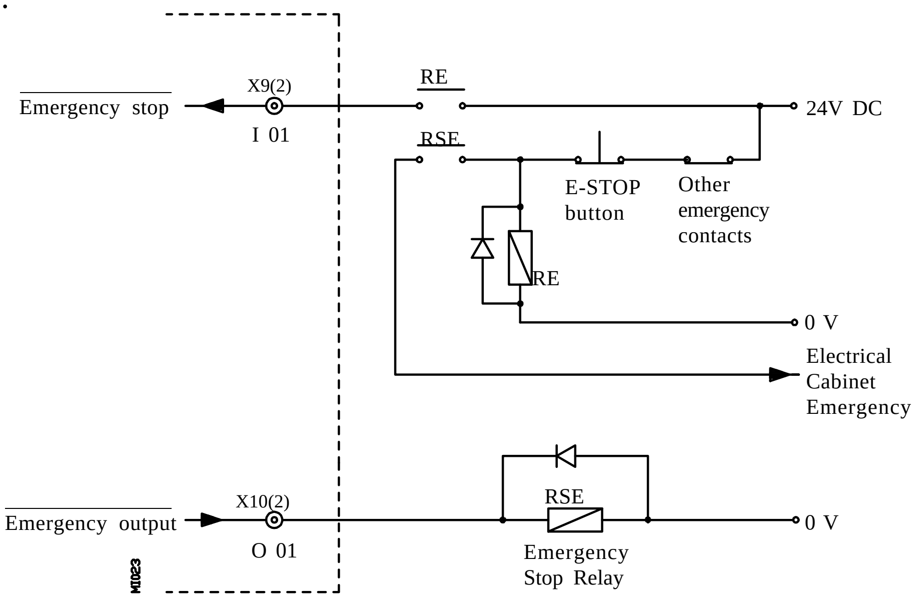
When the CNC detects an error, it will activate the CNC to PLC signal "**/ALARM**", and it will activate the Emergency output (logic level 0) at pin 2 of connector X10 of the AXES module.

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

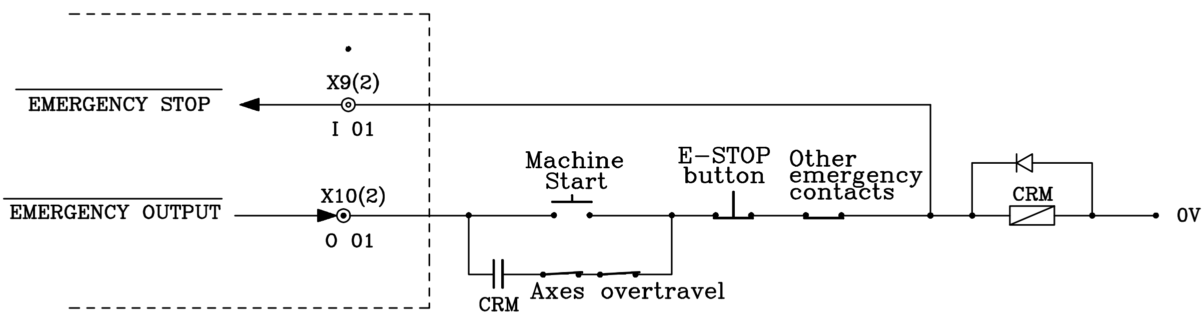


The recommended connection diagram can be one of the following:

European Style:



USA Style:



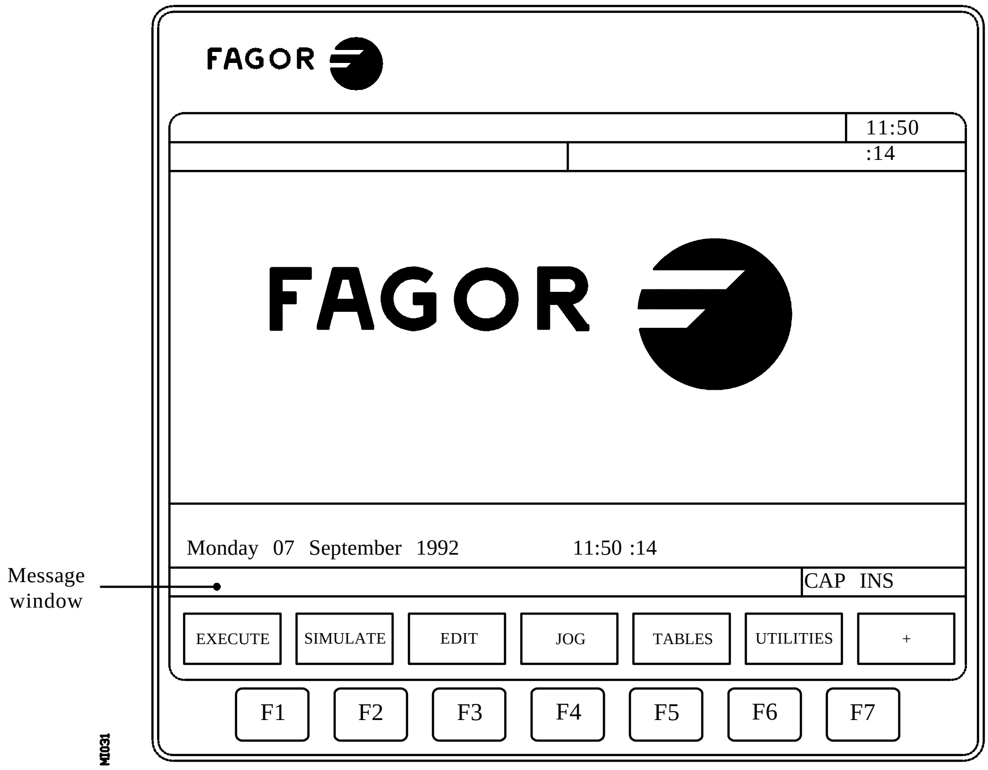
3. *MACHINE PARAMETERS*

<u>Atention:</u>

It is highly recommended to save the machine parameters in a peripheral device or computer in order to avoid losing them by replacing modules, checksum errors, operator errors, etc.

3.1 INTRODUCTION

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. This screen will be defined using the GRAPHIC EDITOR (refer to the operating manual of the 8050 CNC).

If any error occurs, it will be displayed in the message 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 thru 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 offers access to the following tables:

General Machine Parameters
Machine Parameters for the Axes (one table per axis)
Spindle Parameters
Parameters for serial ports (RS-422 and RS-232-C)
PLC parameters
Miscellaneous Functions M
Leadscrew Error Compensation (one table per axis)
Cross compensation

On all of them, it is possible to move the cursor line by line using the arrow keys or page by page using the Pagedown and Pageup keys.

3.2 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 right and left arrow keys .

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

* **CL** erases characters.

* **INS** switches between insert and 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.

* The **ESC** key cancels the editing of the line.

* When pressing **ENTER** the edited parameter goes into the table.

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

* **EDIT** a parameter. The CNC will indicate the proper format by means of the softkeys.

* **MODIFY** a parameter. Position the cursor over the desired parameter to be modified and press this softkey.

  Make the changes using the keys described above.

  And press ENTER. The new parameter value will appear in its previous location in the table.

* **FIND** 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** the table assuming the default values.

* **LOAD** the table receiving its values via any of the two serial ports of the CNC: RS-232-C or RS-422.

* **SAVE** the table values out to a peripheral device via any of the two serial ports of the CNC: RS-232-C or RS-422.

* Display the parameter values in millimeters or inches by pressing the **MM/INCH** key. Only those parameters affected by this conversion will be altered. The general machine parameter "INCHES" will not be changed.

3.3 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.

The FAGOR 8050 CNC offers the following machine parameter groups:

* General Parameters.
* Axes Parameters.
* Spindle Parameters.
* Parameters for the two communication channels, RS-422 and RS-232-C.
* PLC Parameters.
* M Miscellaneous Functions (Auxiliary functions).
* Leadscrew error compensation.
* Cross Compensation.

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.

3.3.1 *GENERAL MACHINE PARAMETERS*

AXIS1 (P0)

Indicates the axis whose feedback is connected to connector X1 and whose analog signal comes out of output O01 of connector X8 of the AXES module. The possible values are:

0 = Free. Not associated with any axis.
1 = X axis
2 = Y axis
3 = Z axis
4 = U axis
5 = V axis
6 = W axis
7 = A axis
8 = B axis
9 = C axis
10 = Main spindle
11 = Handwheel
12 = Handwheel with axis selector button
13 = Auxiliary spindle / Live tool
14 = Second main spindle (only on Lathe models)

By default, this parameter is set to 1 (X axis).

AXIS2 (P1)

Indicates the axis whose feedback is connected to connector X2 and whose analog signal comes out of output O02 of connector X8 of the AXES module.

By default, the CNC assigns a value of 2 (Y axis) if Mill model and a value of 3 (Z axis) if Lathe model.

AXIS3 (P2)

Indicates the axis whose feedback is connected to connector X3 and whose analog signal comes out of output O03 of connector X8 of the AXES module.

By default, the CNC assigns a value of 3 (Z axis) if Mill model and a value of 10 (spindle) if Lathe model.

AXIS4 (P3)

Indicates the axis whose feedback is connected to connector X4 and whose analog signal comes out of output O04 of connector X8 of the AXES module.

By default, the CNC assigns a value of 4 (U axis) if Mill model and a value of 11 (handwhell) if Lathe model.

AXIS5 (P4)

Indicates the axis whose feedback is connected to connector X5 (pins 1 thru 6) and whose analog signal comes out of output O05 of connector X8 of the AXES module. It uses the same definition code as AXIS1.

By default, the CNC assigns a value of 5 (V axis) if Mill model and a value of 0 (free) if Lathe model.

AXIS6 (P5)

Indicates the axis whose feedback is connected to connector X5 (pins 9 thru 14) and whose analog signal comes out of output O06 of connector X8 of the AXES module. It uses the same definition code as AXIS1.

By default, the CNC assigns a value of 10 (Spindle) if Mill model and a value of 0 (free) if Lathe model.

AXIS7 (P6)

Indicates the axis whose feedback is connected to connector X6 (pins 1 thru 6) and whose analog signal comes out of output O07 of connector X8 of the AXES module. It uses the same definition code as AXIS1.

By default, the CNC assigns a value of 11 (Handwheel) if Mill model and a value of 0 (free) if Lathe model.

AXIS8 (P7)

Indicates the axis whose feedback is connected to connector X6 (pins 9 thru 14) and whose analog signal comes out of output O08 of connector X8 of the AXES module. It uses the same definition code as AXIS1.

By default, the CNC assigns a value of 0 (Free).

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:

0 = millimeters (G71)
1 = inches (G70)

By default, the CNC assigns a value of 0 (mm) to this parameter.

IMOVE (P9) (**I**nitial **MOVE**ment)

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:

0 = G00 (rapid traverse)
1 = G01 (linear interpolation)

By default the CNC assigns a value of 0 (G00) to this parameter.

ICORNER (P10) (**I**nitial **CORNER**)

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:

0 = G07 (square corner)
1 = G05 (round corner)

By default, the CNC assigns a value of 0 (G07) to this parameter.

IPLANE (P11) (**I**nitial **PLANE**)

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

0 = G17 (XY plane)
1 = G18 (ZX plane)

By default, the CNC assigns a value of 0 (G17) to this parameter if Mill model and a value of 1 (G18) if Lathe model.

ILCOMP (P12) (**I**nitial **L**ength **COMP**ensation)

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:

0 = G44 (tool length compensation OFF)
1 = G43 (tool length compensation ON)

By default, the CNC assigns a value of 0 (G44) to this parameter.

ISYSTEM (P13) (**I**nitial **SYSTEM**)

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

0 = G90 (absolute programming)
1 = G91 (incremental programming)

By default, the CNC assigns a value of 0 (G90) to this parameter.

IFEED (P14) (**I**nitial **FEED**)

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. The code is:

0 = G94 (mm/min or inch/min)
1 = G95 (mm/rev or inch/rev)

By default, the CNC assigns a value of 0 (G94) to this parameter.

THEODPLY (P15) (**THEO**retical **D**is**PL**a**Y**)

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

0 = real position values
1 = theoretical position values

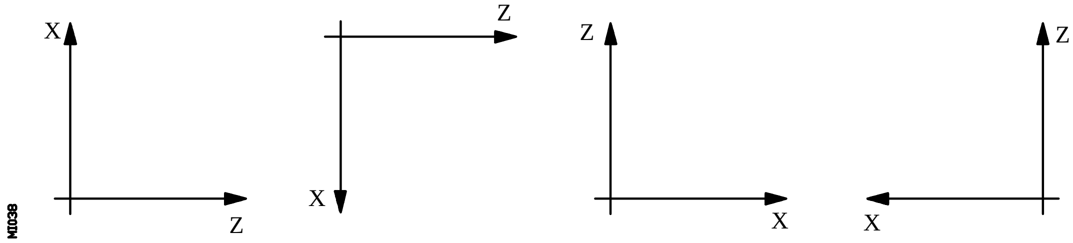
By default, the CNC will assume a value of 1 (theoretical display).

GRAPHICS (P16)

On MILL MODEL 8050 CNCs, 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.

Possible values:
- 0 = Mill graphics
- 1 = Mill graphics with added W axis
- 2 = Boring Mill graphics
- 3 = Boring Mill graphics with added W axis

On LATHE MODEL 8050 CNCs, this parameter indicates the axes coordinates system to be used for the graphic representation.



GRAPHICS=0   GRAPHICS=1   GRAPHICS=2   GRAPHICS=3

RAPIDOVR (P17) (**RAPID OV**e**R**ride)

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

The feedrate can be overridden from the Feedrate Override Knob on the operator panel, from the PLC, via DNC or by program.

YES = It is possible to vary the % of G00 (rapid).
NO = It is NOT possible to vary it, remaining set at 100%.

By default, the CNC assumes YES for this parameter (Feedrate override active).

MAXFOVR (P18) (**MAX**imum **F**eed **OV**e**R**ride)

Indicates the maximum value of the FEEDRATE OVERRIDE applicable to the programmed feedrate.

The programmable feedrate F can be varied between 0% and 120% from the override knob on the operator panel or between 0% and 255% from the PLC, via DNC or by program.

The resulting feedrate will be limited to the value indicated in the axis machine parameter “MAXFEED”.

Possible values: Integers between 0 and 255

By default, the CNC assumes a value of 120 for this parameter.

CIRINLIM (P19) (**CIR**cular **IN**terpolation **LIM**it)

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.

Example:

If “CIRINLIN” = 1500 and an arc of a radius = 0.5mm at F=10000mm/min. The result is:

Theoretical angular feedrate = 10000 mm/min : 0.5 mm.= 20000 $min.^{-1}$

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

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

Possible values: between 0 and 65535. A value of 0 means that no feedrate limitation is to be applied.

By default, the CNC assumes a value of 0 for this parameter (not limited).

CIRINERR (P20) (**CIR**cular **IN**terpolation **ERR**or)

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: 0.0001 thru 99999.9999 millimeters
0.00001 thru 3937.00787 inches.

By default, the CNC assumes a value of 0.01 mm for this parameter.

PORGMOVE (P21) (**P**olar **OR**i**G**in **MOVE**ment)

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

YES = it does.
NO = The polar origin is not affected by G02 or G03.

By default, the CNC assumes NO for this parameter.

BLOCKDLY (P22) (**BLOCK D**e**L**a**Y**)

Indicates the dwell in milliseconds (0 thru 65535) between blocks when executing moves in G07 (square corner).

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

By default, the CNC assumes a value of 0 (no dwell) for this parameter.

NTOOL (P23) (**N**umber of **TOOL**s)

Indicates the number of tools in the tool magazine.

Possible values: Integers between 0 and 255.

By default, the CNC assumes a value of 100 for this parameter.

NPOCKET (P24) (**N**umber of **POCKET**s)

Indicates the number of pockets in the tool magazine.

Possible values: integers 0 thru 255.

By default, the CNC assumes a value of 100 for this parameter if Mill model and a value of 0 if Lathe model.

RANDOMTC (P25) (**RANDOM T**ool **C**hanger)

Indicates whether the tool magazine is RANDOM or not.

On a RANDOM magazine, the tool may occupy any position (pocket).
On a NON-RANDOM magazine, the tool always occupies its own pocket.

YES = It is a RANDOM tool magazine.
NO = It is Not a RANDOM tool magazine.

By default, the CNC assumes that it is NOT random.

If this machine parameter is set for **RANDOM** magazine, machine parameter **"TOFFM06" (P28)** must be set for machining center.

TOOLMONI (P26) (**TOOL MONI**tor)

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

0 = Minutes.
1 = Number of operations.

By default, the CNC assumes a value of 0 (minutes) to this parameter.

NTOFFSET (P27) (**N**umber of **T**ool **OFFSET**s)

Indicates the number of tool offsets available in the tool offset table.

Possible values: Integers 0 thru 255.

By default, the CNC assumes a value of 100 for this parameter.

TOFFM06 (P28) (**T**ool **OFF**set with **M06**)

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.

YES = It is a machining center.
NO = It is not a machining center.

By default, the CNC assumes that it is NOT a machining center.

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

Note: Parameters 23 through 28 as well as 60 and 61 relate to the tool changer.

NMISCFUN (P29) (**N**umber of **MISC**ellaneous **FUN**ctions)

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

Possible values: Integers 0 thru 255.

By default, the CNC assumes a value of 32 for this parameter.

MINAENDW (P30) (**MIN**imum **A**ux **END W**idth)

Indicates the minimum time period that the AUX END signal must remain activated so the CNC will interpret it as a valid signal.

AUX END 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 AUX END signal, the time period indicated in this parameter will be the duration of the MSTROBE signal.

Possible values: Integers 0 thru 65535 milliseconds.

By default, the CNC will assume a value of 100 for this parameter.

The application of this parameter is described in the section: “TRANSFERRING AUXILIARY M,S,T FUNCTIONS” of the chapter: “CONCEPT SUBJECT”.

NPCROSS (P31) (**N**umber of **P**oints **CROSS** compensation)

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: Integers 0 thru 255.

By default, the CNC assumes a value 0 for this parameter (no cross compensation being applied).

MOVAXIS (P32) (**MOV**ing **AXIS**)

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

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

By default the CNC assumes a 0 value for this parameter (none).

COMPAXIS (P33) (**COMP**ensated **AXIS**)

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:

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

By default, the CNC assumes a value of 0 for this parameter (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 this compensation onto the W axis.

Note: Parameters P54 through P59 are associated with the 2nd and 3rd cross compensation tables.

REFPSUB (P34) (**REF**erence **P**oint **SUB**routine)

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: integers 0 thru 9999. 0 means that there is no associated subroutine to be executed.

By default, the CNC assumes a value of 0 for this parameter.

INT1SUB (P35) (**INT1 SUB**routine)
INT2SUB (P36) (**INT2 SUB**routine)
INT3SUB (P37) (**INT3 SUB**routine)
INT4SUB (P38) (**INT4 SUB**routine)

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: Integers between 0 and 9999. If set to "0", no subroutine will be executed.

By default, these parameters are set to "0" (no subroutine is associated).

PRBPULSE (P39) (**PR**o**B**e **PULSE**)

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.

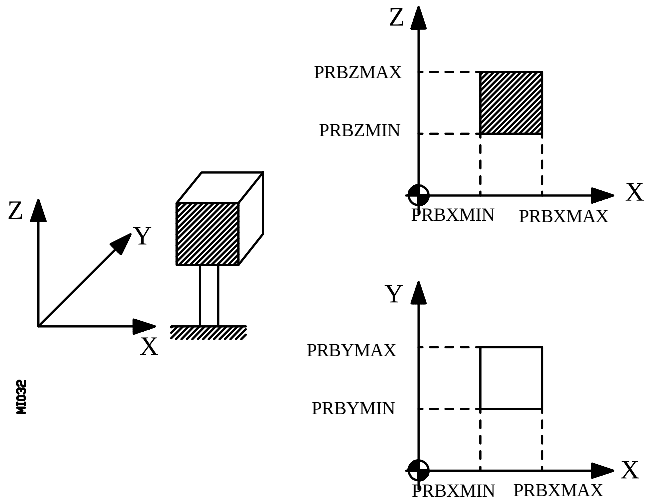
Sign **+** = leading edge (24V. or 5V positive pulse.)
Sign **-** = trailing edge (0V negative pulse.)

By default, the CNC assumes a + (positive pulse) value for this parameter

PRBXMIN (P40) (**PR**o**B**e **X MIN**imum value)
PRBXMAX (P41) (**PR**o**B**e **X MAX**imum value)
PRBYMIN (P42) (**PR**o**B**e **Y MIN**imum value)
PRBYMAX (P43) (**PR**o**B**e **Y MAX**imum value)
PRBZMIN (P44) (**PR**o**B**e **Z MIN**imum value)
PRBZMAX (P45) (**PR**o**B**e **Z MAX**imum value)

Indicate the position of the table-top probe used for tool calibration.

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



PRBXMIN indicates minimum X position value of the probe.
PRBXMAX indicates maximum X position value of the probe.
PRBYMIN indicates minimum Y position value of the probe.
PRBYMAX indicates maximum Y position value of the probe.
PRBZMIN indicates minimum Z position value of the probe.
PRBZMAX indicates maximum Z position value of the probe.

Possible values: ±99999.9999 millimeters.
±3937.00787 inches.

By default, the CNC assumes a value of 0 for this parameter.

PRBMOVE (P46) (**PR**o**B**e **MOVE**ment)

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

Possible values: Between 0.0001mm and 99999.9999mm.
Between 0.00001 inch and 3937.00787 inches.

The default value for this parameter is 50 mm (about 2 inches).

USERDPLY (P47) (**USER D**is**PL**a**Y**)

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

Possible values: integers 0 thru 65535. 0 means that there is no program associated with the USER channel for the EXECUTE mode.

By default, the CNC assumes a value of 0 to this parameter.

USEREDIT (P48) (**USER EDIT**or)

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

Possible values: Integers 0 thru 65535. 0 means that there is no program associated with the USER channel in the EDIT mode.

By default, the CNC assumes a value of 0 for this parameter.

USERMAN (P49) (**USER MAN**ual)

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

Possible values: integers 0 thru 65535. 0 means that there is no program associated with the USER channel for the JOG mode.

By default, the CNC assumes a value of 0 to this parameter.

USERDIAG (P50) (**USER DIAG**nosis)

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

Possible values: integers 0 thru 65535. 0 means that there is no program associated with the USER channel for the DIAGNOSIS mode.

By default, the CNC assumes a value of 0 to this parameter.

ROPARMIN (P51) (**R**ead **O**nly **PAR**ameter **MIN**imum)
ROPARMAX (P52) (**R**ead **O**nly **PAR**ameter **MAX**imum)

Indicate the upper limit “ROPORMAX” and the lower limit “ROPORMIN” of the global arithmetic parameter group (P100-P299) to be write protected.

Possible values: Integers 0 thru 9999.

If a value less than 100 is assigned to this parameter, the CNC will assume a value of 100.
If a value greater than 299 is assigned to this parameter, the CNC will assume a value of 299.

By default, the CNC assumes a value of 0 for these parameters (none of the arithmetic parameters are protected).

PAGESMEM (P53) (**PAGES MEM**ory)

Indicates the percentage (%) of **EEPROM** memory dedicated to store the customized Pages (screens) and Symbols created with the graphic editor.

This **EEPROM** memory stores the following information:

* The user pages and symbols as mentioned above.

* The **PLC** program. the percentage (%) of **EEPROM** memory used to store this program is established by **PLC** machine parameter "**PLCMEM**".

* The part-programs of the CNC and the PLC errors in the remaining free memory space of the **EEPROM**.

Possible values: Integers between 0 and 100.

The default value for this parameter is 50.

Atention:

Changing this parameter can cause items previously stored in EEPROM memory to be lost.

To avoid this, proceed as follows:

1.- Transfer the data from the EEPROM to a peripheral or PC.
2.- Change the setting of "PAGESMEM" and "PLCMEM".
3.- Press [**SHIFT**] [**RESET**].
4.- Re-enter all the data back into the EEPROM.

NPCROSS2 (P54) (**N**umber of **P**oints **CROSS** compensation **2**)

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: Integers 0 thru 255.

By default, the CNC assumes a value 0 for this parameter (no cross compensation being applied).

MOVAXIS2 (P55) (**MOV**ing **AXIS 2**)

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

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

By default the CNC assumes a 0 value for this parameter (none).

COMAXIS2 (P56) (**COM**pensated **AXIS 2**)

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:

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

By default, the CNC assumes a value of 0 for this parameter (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 this compensation on to the B axis.

NPCROSS3 (P57) (**N**umber of **P**oints **CROSS** compensation **3**)

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: Integers 0 thru 255.

By default, the CNC assumes a value 0 for this parameter (no third cross compensation being applied).

MOVAXIS3 (P58) (**MOV**ing **AXIS 3**)

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

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

By default the CNC assumes a 0 value for this parameter (none).

COMAXIS3 (P59) (**COM**pensated **AXIS 3**)

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:

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

By default, the CNC assumes a value of 0 for this parameter (none).

Example:

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

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

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

TOOLSUB (P60) (**TOOL SUB**routine)

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: Integers between 0 and 9999. If a value of 0 is assigned to this parameter, the CNC will assume that no subroutine is to be executed.

By default, this parameter is set to 0 (no associated subroutine).

When set with a value other than 0, the **T** function must be written alone in the block or only with the desired tool offset **D**.

CYCATC (P61) (**CYC**lic **A**utomatic **T**ool **C**hanger)

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.

A non-cyclic tool changer can perform several tool searches in a row without having to program an M06.

NO = The tool changer being used is **not cyclic** (no M06 required).
YES = The tool changer being used **is cyclic** (requires M06).

By default, the CNC takes the "**YES**" value for this parameter (cyclic).

TRMULT (P62) (**TR**acing **MULT**iplier**)**

It is only used on Mill model CNCs and has no effect on CNC version 9.01 and later.

Indicates the multiplier factor used for the tracing operation. It admits any integer value between 0 and 9999 where the value of 1000 corresponds to a factor of 1.

It is recommended to set it with a value of 0 when the tracing axis uses the feed-forward gain and no derivative gain is used.

When this parameter is set to a value other than 0, the derivative gain may be used for the control loop of the tracing axis.

If the tracing axis is associated with another axis, like a gantry axis or slaved axis, this parameter must be set to 0.

By default, the CNC sets this parameter to a value of 0.

TRPROG (P63) (**TR**acing **PRO**portional **G**ain)

Indicates the value of the Proportional Gain used for tracing.
It admits any integer value between 0 and 9999 where a value of 1000 corresponds to a factor of 1.

By default, the CNC sets this parameter to a value of 250.

TRDERG (P64) (**TR**acing **DER**ivative **G**ain)

Indicates the value of the Derivative Gain used for tracing. It admits any integer value between 0 and 9999 where a value of 1000 corresponds to a factor of 1.

By default, the CNC sets this parameter to a value of 250.

MAXDEFLE (P65) (**MAX**imum **DEFLE**ction)

Indicates the maximum deflection allowed for the probe while tracing.

The CNC will correct the probe position everytime this parameter value is reached.

Possible values: between 0 and 99999.9999 mm
between 0 and 3937.00787 inches

By default, the CNC sets this parameter to a value of 4mm.

The value assigned to this parameter must be less than or equal to the value that the probe has for "Measuring range".

MINDEFLE (P66) (**MIN**imum **DEFLE**ction)

Indicates the minimum deflection used by the probe while tracing.

Possible values: between 0 and 99999.9999 mm
between 0 and 3937.00787 inches

By default, the CNC sets this parameter to a value of 0.2 mm.

The value assigned to this parameter must be less than the value assigned to the general machine parameter "**MAXDEFLE**".

TRFBAKAL (P67) (**TR**ace **F**eed**BA**c**K AL**arm)

Indicates whether the feedback alarm of the tracing probe must be activated or not.

OFF = Feedback alarm deactivated.
ON = Feedback alarm activated.

By default, the CNC sets this parameter to **ON**.

TIPDPLY (P68) (**TIP D**is**PL**a**Y**)

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

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.

0 = The CNC displays the tool base position.
1 = The CNC displays the tool tip position.

By default, the CNC sets this parameter to "0" (tool base) on a Mill model and to "1" (tool tip) on a Lathe model.

ANTIME (P69) (**AN**ticipation **TIME**)

It is used on punch presses having an eccentric cam as their punching system.

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

This reduces the idle time, thus resulting in more punches per minute.

It is given in milliseconds with any integer value between 0 and 65535.

If the whole movement lasts less than the value of this parameter (ANTIME), the anticipation signal (ADVINPOS) will be activated immediately.

If the value of ANTIME is "0", the ADVINPOS signal will never be activated.

By defalut, the CNC sets this parameter to "0".

PERCAX (P70) (**PER**manent **C AX**is)

It is used on the Lathe model CNC.

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

YES = The "C" axis is only deactivated by the typical spindle related "M" functions (M03, M04, M05, etc.).

NO = Besides being canceled by those functions, it is also deactivated by turning the CNC off and back on, by an emergency or reset, by executing functions M02 or M30.

By default, this parameter is set to "NO".

TAFTERS (P71) (**T**ool **AFTER S**ubroutine)

General machine parameter "TOOLSUB" (P60) indicates the number of the subroutine associated with the tool.

This parameter (P71), determines whether the tool selection is carried out before or after executing that subroutine.

YES = The tool is selected after executing the subroutine.
NO = The tool is selected before executing the subroutine.

The default value for this parameter is "**NO**"

LOOPTIME (P72)

It sets the sampling period used by the CNC and, consequently, its block processing time.

Possible values:	0	4 msec. period (standard)
	1	(reserved)
	2...6	period in milliseconds.

Atention:

Sampling periods shorter than 4 msec. are not allowed when **not** using the CPU-TURBO option.

IPOTIME (P73) (**I**nter**PO**lation **TIME**)

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

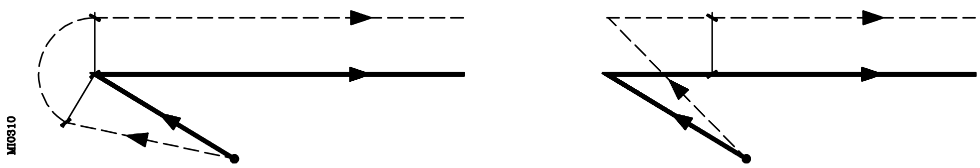
For instance, a 2 msec sampling and interpolation time results in block processing time of 4.5 msec. for a 3-axis linear interpolation with no tool compensation.

Possible values:	0	IPOTIME = LOOPTIME
	1	IPOTIME = Double LOOPTIME value

COMPTYPE (P74) (**COMP**ensation **TYPE**)

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

If COMPTYPE = 0 it approaches the starting point going around the corner.
If COMPTYPE = 1 it goes directly to the perpendicular of that point (without going around the corner.



COMPTYPE=0 COMPTYPE=1

By default, this parameter is set to 0.

FPRMAN (P75) (Feed Per Revolution in MANual)

It is only used on Lathe model CNCs and it indicates whether feedrate per revolution is permitted or not.

Possible values: YES / NO.

By default, this parameter is set to NO.

MPGAXIS (P76) (Manual Pulse Generator AXIS) (handwheel)

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:

0 =	Shared.	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.

The default value for this parameter is "0" (shared).

DIRESET (P77) (DIrect RESET)

It is only used on the Lathe model CNC and it indicates whether it is effective with or without prior CYCLE STOP.

NO = The CNC accepts the RESET only when the STOP condition is met. That is, when it is not execution.
YES = The CNC accepts the RESET any time.

The default value of this parameter is 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.

PLACOM (P78)

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

0 = Tool compensation in the ZX plane only.
1 = Tool compensation in all planes.

By default, the CNC sets this parameter to "0".
When "PLACOM = 1", the CNC interprets the tool table as follows:

	ZX plane	WX plane
Parameters Z and K, with abscissa axis	Z axis	W axis
Parameters X and I, with ordinate axis	X axis	X axis.

3.3.2 *MACHINE PARAMETERS FOR THE AXES*

AXISTYPE (P0)

Defines the axis type and whether it is commanded from the CNC or the PLC

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.

By default, the CNC assumes a value of 0 for this parameter (regular linear axis).

Atention:

By default, rotary axes are Rollover and are displayed between 0º and 359.9999º. If rollover is not desired, set machine parameter ROLLOVER (P55)=NO".

Absolute programming (G90) must always be done using positive values.

Positioning-only and/or Hirth axes follow the shortest path when programmed in absolute (G90). In other words, if its current position is 10º, and its target position is 350º, the axis will go through, 10º, 9º, 8º, ..., 352º, 351º, 350º.

DFORMAT (P1) (**D**isplay **FORMAT**)

Indicates the display format used for this axis. The code is:

0 = format 5.3 when in degrees or mm; format 4.4 when in inches.
1 = format 5.4 when in degrees or mm; format 4.5 when in inches.
2 = format 5.2 when in degrees or mm; format 4.3 when in inches.
3 = the axis is not displayed.

Also, on the Lathe model CNC, this parameter indicates the programming and display units (radius or diameter) for each axis position values.

Codes 0, 1 and 2 indicate operation in radius with the selected display format and the other codes indicate operation in diameter with the selected display format.

4 = format 5.3 when in mm; format 4.4 when in inches.
5 = format 5.4 when in mm; format 4.5 when in inches.
6 = format 5.2 when in mm; format 4.3 when in inches.

By default, the CNC assumes a value of 0 for this parameter.

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. The code is:

0 = It is NOT GANTRY.
1 = Associated with X axis.
2 = Associated with Y axis.
3 = Associated with Z axis.
4 = Associated with U axis.
5 = Associated with V axis.
6 = Associated with W axis.
7 = Associated with A axis.
8 = Associated with B axis.
9 = Associated with C axis.

It is possible to have more than one GANTRY pair of axis. The CNC assume the default value of 0 for this parameter.

Example:

If the X and U axes form a GANTRY pair, the U axis being the slave axis. The corresponding parameters will be programmed:

Parameter GANTRY for X axis = 0
Parameter GANTRY 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) (**SYNCHRO**nization)

It is possible to couple or decouple each one of the axes by PLC program using the logic inputs of the CNC: “SYNCHRO1”, “SYNCHRO2”, “SYNCHRO3”, “SYNCHRO4”, “SYNCHRO5”. Each axis will be synchronized to the axis indicated in its machine parameter “SYNCHRO”.

Indicates which axis will this one (slave) be coupled to when it so requested by the PLC.

0 = Coupled to none.
1 = Coupled to X axis.
2 = Coupled to Y axis.
3 = Coupled to Z axis.
4 = Coupled to U axis.
5 = Coupled to V axis.
6 = Coupled to W axis.
7 = Coupled to A axis.
8 = Coupled to B axis.
9 = Coupled to C axis.

By default, the CNC assumes a value of 0 for this parameter.

This way, to couple the V axis to the X axis, the following machine parameters must be defined:

SYNCHRO of the X axis = 0 (X is the master)
SYNCHRO of the V axis = X (V is the slave)

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) (**D**igital **R**ead**O**ut **AXIS**)

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

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

By default the CNC assumes the axis to be normal (NO)

LIMIT+ (P5)
LIMIT- (P6)

Indicate the software travel limits (positive and negative). They must indicate the distance from the machine reference zero (home) to these limits.

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

On linear axes, a "0" value means that there are no travel limits.

On rotary axes:

* When both parameters are set to "0", the axis may be moved indefinitely in any direction (rotary tables, indexers, etc.)

  When working with positioning only or HIRTH axes, they should be programmed using incremental position values in order to avoid errors. For example, the "C" axis with P5=0, P6=720 and the axis positioned at 700 (the screen displays 340) if G90 C10 is programmed, this axis tries to go via the shortest path (701, 702,...) but it issues an error message since it exceeds the travel limit.

* On positioning only and HIRTH axes, if the travel is limited to less than a revolution, they cannot be moved via the shortest path.

* When the travel is limited to less than a revolution, it is possible to display positive and negative position values (for example: P5=-120, P6=120) as well as programming them in G90.

By default, the CNC assumes a value of 8000 mm for "LIMIT+" and -8000 mm for "LIMIT-".

PITCH (P7)

Indicates the distance per revolution of the rotary encoder or the grating pitch of the linear feedback device in use. With a FAGOR scale, assign a value of 20 µm or 100 µm.

On a rotary axis, it must indicate the number of degrees per revolution of the encoder. For example, if the encoder is mounted on the motor and the axis has a gear reduction factor of 1/10, this parameter must be set to 360º/10 = 36.

Possible values: 0.0001 thru 99999.9999 millimeters.
0.00001 thru 3937.00787 inches.
0.0001 thru 99999.9999°

By default, the CNC assumes a value of 5 for this parameter.

NPULSES (P8) (**N**umber of **PULSES**)

Indicates the number of pulses per revolution provided by the encoder. When using a linear scale, set this parameter to 0 and the CNC will always apply 1 pulse per each grating pitch.

Possible values: integers 0 thru 65535.

By default, the CNC assumes a value of 1250 for this parameter.

DIFFBACK (P9) (**DIF**ferential **F**eed**BACK**)

Indicates Whether the feedback device outputs differential signals (double-ended) or not.

NO = No differential signals
YES = It uses differential signals

By default, the CNC assumes YES for this parameter.

SINMAGNI (P10) (**SIN**usoidal **MAGNI**fication)

Indicates the multiplying factor (x1, x4, x20, etc.) that the CNC will apply to the sine-wave feedback signal for this axis.

Possible values: Integers between 0 and 255.

When using square-wave feedback signals, set this parameter to 0 and the CNC will always apply a x4 multiplying factor.

By default, the CNC assumes a value of 0 (squarewave feedback signals) for this parameter.

The counting resolution of the CNC is determined using parameters P7, P8 and P10 as shown in the examples on the following pages:

METRIC, INCH & ROTARY AXIS FEEDBACK CALCULATION EXAMPLES

TYPE OF FEEDBACK	PITCH	NPULSES	DIFFBACK	SINMAGNI (XFACTOR)	COUNTING* RESOLUTION	MAXSPEED** LIMIT
METRIC 8mm pitch ballscrew with 2000 line squarewave encoder and 1:1 gear ratio.	8.000mm	2000	YES	0 (x4)	0.001mm	24M/min.
FAGOR Sinewave C scale	0.020mm	0	NO	20 (x20)	0.001mm	30M/min.
FAGOR Squarewave CX scale (Special case; These scales already apply an x5 factor to the squarewave signals before they leave the reader head!).	0.020mm	5	YES	0 (x4)	0.001mm	24M/min.
FAGOR Sinewave F scale	0.100mm	0	NO	100 (x100)	0.001	30M/min.
INCH .2" pitch ballscrew with 500 line squarewave encoder and 1:1 gear ratio.	0.2"	500	YES	0 (x4)	0.0001"	2400"/min.
.25" pitch ballscrew with 1250 line squarewave encoder and 1:1 gear ratio.	0.25"	1250	YES	0 (x4)	0.00005"	1200"/min.
ROTARY AXIS 4° per encoder turn with 1000 line squarewave encoder and 90:1 gear ratio.	4.000°	1000	YES	0 (x4)	0.001°	24,000°/min. (66 rpm)
360° per encoder turn with 3600 line sinewave encoder and 1:1 gear ratio.	360.000°	3600	NO	20 (x20)	0.005°	300,000°/min (833RPM)

* COUNTING RESOLUTION = PITCH ÷ NPULSES ÷ XFACTOR

** MAXSPEED LIMIT:
= MAX SIGNAL FREQUENCY * XFACTOR * 60 * COUNTING RESOLUTION

MAXIMUM SIGNAL FREQUENCY: WILL BE LIMITED BY ONE OF THE 6 ITEMS LISTED BELOW.

FAGOR 8050 CNC = 425KHZ SQUARE FEEDBACK!
= 50KHZ SINEWAVE FEEDBACK!

FAGOR ENCODERS = MAXIMUM MECHANICAL SPEED IS 12,000 RPM!
=MAXIMUM SIN OR SQR WAVE SIGNAL SPEED IS 100KHZ!

FAGOR SCALES = MAXIMUM MECHANICAL SPEED IS 30M/MIN (1,181 INCHES/MIN)!
= MAXIMUM SIN OR SQR WAVE SIGNAL SPEED IS 100KHZ!

MAXIMUM SIGNAL FREQUENCY:
FAGOR ENCODERS = MAXIMUM MECHANICAL SPEED IS 12,000 RPM!
= MAXIMUM SIN OR SQR WAVE SPEED IS 100KHZ!

! Denotes speed limiting item.

ENCODER LINE COUNT	50	100	200	250	400	500
MAX MECHANICAL SPEED	12,000 RPM!	12,000 RPM!	12,000 RPM!	12,000 RPM!	12,000 RPM!	12,000 RPM!
MAX SIGNAL SPEED	10KHZ	20KHZ	40KHZ	50KHZ	80KHZ	100KHZ!
DIFFERENTIAL OUTPUT	YES	YES	YES	YES	YES	YES

! Denotes speed limiting item.

ENCODER LINE COUNT	600	635	1000	1024	1250	1270
MAX MECHANICAL SPEED	10,000 RPM	9,448 RPM	6,000 RPM	5,859 RPM	4,800 RPM	4,724 RPM
MAX SIGNAL SPEED	100KHZ!	100KHZ!	100KHZ!	100KHZ!	100KHZ!	100KHZ!
DIFFERENTIAL OUTPUT	YES	YES	YES	YES	YES	YES

! Denotes speed limiting item.

ENCODER LINE COUNT	1500	2000	2500	3000	SW3600	5000
MAX MECHANICAL SPEED	4,000 RPM	3,000 RPM	2,400 RPM	2,000 RPM	1,666 RPM	1,200 RPM
MAX SIGNAL SPEED	100KHZ!	100KHZ!	100KHZ!	100KHZ!	100KHZ!***	100KHZ!
DIFFERENTIAL OUTPUT	YES	YES	YES	YES	NO	YES

*** NOTE:
THE 8050 CNC CAN ONLY COUNT 50KHZ (833 RPM) OF SW3600 SINEWAVE SIGNALS.

MAXIMUM SIGNAL FREQUENCY:
FAGOR SCALES = MAXIMUM MECHANICAL SPEED IS 30M/min (1,181 inches/min)
= MAXIMUM SIN OR SQR WAVE SPEED IS 100KHZ!

! Denotes speed limiting item.

SCALE MODEL	MVS	CVS	FS	MVX	CVX	FT
MAX MECHANICAL SPEED	30M/min! (1181"/min)	30M/min! (1181"/min)	30M/min! (1181"/min)	24M/min (944"/min)	24M/min (944"/min)	24M/min (944"/min)
MAX SIGNAL SPEED	25KHZ	25KHZ	5KHZ	100KHZ!	100KHZ!	100KHZ!
GRATING PITCH	0.020mm	0.020mm	0.100mm	0.020mm	0.020mm	0.100mm
A.S.I.C. MULTIPLIER	N/A	N/A	N/A	x5	x5	x25
DIFFERENTIAL OUTPUT	NO	NO	NO	YES	YES	YES

NOTE: SOME FAGOR SCALES AND ENCODERS ARE INTENTIONALLY NOT LISTED HERE.

FBACKAL (P11) (**F**eed**BACK AL**arm)

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

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

OFF = Feedback alarm Cancelled.
ON = Feedback alarm Active.

By default, the CNC keeps the feedback alarm active (ON).

FBALTIME (P12) (**F**eed**B**ack **AL**arm **TIME**)

It indicates the maximum time period given to the axis to respond to the analog voltage output by the CNC.

The CNC calculates the number of feedback pulses that it must receive in each sampling time period according to the corresponding analog voltage output.

It is assumed that the axis is performing properly when the feedback pulses received are within 50% and 200% of those expected (calculated) by the CNC.

If at some point the feedback pulses received are not within this range, the CNC will keep checking them for a period of time indicated in this parameter to “see” that the axis performance is back to normal (between 50% and 200%). If this has not happened in this time period, the CNC will issue the corresponding error message.

This parameter may have an integer value between 0 and 65535 milliseconds.

The default value for this parameter is "0". (no monitoring).

AXISCHG (P13) (**AXIS CH**an**G**e)

Indicates the counting direction of the feedback signals. If correct, leave it as is. If not, change it from NO to YES or viceversa.

If this parameter is modified, parameter “LOOPCHG” (P26) must also be modified so the axis does not run away.

Possible values: YES and NO.

By default, the CNC assumes NO for this parameter.

BACKLASH (P14)

Indicates the amount of backlash. Enter 0 when using linear scales.

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

By default, the CNC assumes a value of 0 for this parameter.

LSCRWCOM (P15) (**L**ead**SCR**e**W COM**pensation)

Indicates whether the CNC should apply leadscrew error compensation or not.

OFF = Not applied.
ON = Leadscrew error compensation applied.

By default, the CNC does NOT apply leadscrew error compensation (OFF).

NPOINTS (P16) (**N**umber of **POINTS**)

Indicates the number of leadscrew error compensation points available in the table. The values in this table will be applied if parameter “LSCRWCOM” (P15) is ON.

Possible values: Integer values between 0 and 255.

By default, the CNC assumes a value of 30 for this parameter.

DWELL (P17)

Indicates the dwell from the moment the ENABLE signal is activated until the analog signal is sent out.

It is given in milliseconds and it admits any integer value between 0 and 65535.

By default, the CNC assumes a value of 0 for this parameter (no dwell).

ACCTIME (P18) (**ACC**eleration **TIME**)

Indicates the time it takes the axis to reach the maximum feedrate selected by axis-machine-parameter “G00FEED” (acceleration stage). The deceleration time will be the same.

It is given in milliseconds and it admits any integer value between 0 and 65535.

By default, the CNC assumes a value of 0 for this parameter (no acceleration/deceleration control).

INPOSW (P19) (**IN POS**ition **W**idth)

Indicates the width of the IN POSITION zone (dead band) where the CNC considers the axis to be in position.

Possible values: 0 thru 99999.9999 millimeters.
0 thru 3937.00787 inches.
0 thru 99999.9999°

By default, the CNC assumes a value of 0.01 mm for this parameter.

INPOTIME (P20) (**IN PO**sition **TIME**)

Indicates the time period that the axis must remain in the “IN POSITION” zone in order to consider it to be in position.

This prevents the CNC from considering the axis to be in position and executing the next block on those machines where the axis could just overshoot the “IN POSITION” zone.

It is given in milliseconds and it admits any integer value between 0 and 65535.

By default, the CNC assumes a value of 0 for this parameter.

MAXFLWE1 (P21) (**MAX**imum **F**o**L**lo**W**ing **E**rror)

Indicates the maximum following error allowed when this axis moves.

Possible values: 0 thru 99999.9999 millimeters or degrees.
0 thru 3937.00787 inches.

By default, the CNC assumes a value of 30 mm for this parameter.

MAXFLWE2 (P22) (**MAX**imum **F**o**L**lo**W**ing **E**rror)

Indicates the maximum following error allowed when this axis is stopped.

Possible values: 0 thru 99999.9999 millimeters or degrees.
0 thru 3937.00787 inches.

By default, the CNC assumes a value of 0.1 mm for this parameter.

PROGAIN (P23) (**PRO**portional **GAIN**)

Indicates the value of the Proportional Gain. It is given in millivolts/mm and it admits integer values between 0 and 65535.

Its value represents the analog voltage corresponding to a following error of 1mm (0.040 inches).

By default, the CNC assumes a value of 1000 mV/mm.

Example 1 in metric:

Machine parameter “G00FEED” = 20000 mm/min and the feedrate for the desired following error of 1mm (0.040") is 1000mm/min.

The analog voltage for 20000 mm/min is 9.5 V.

Analog corresponding to F = 1000 mm/min:

$$\text{Analog Voltage} = \frac{9.5\ \text{V.}}{20000\ \text{mm/min.}} \times 1000\ \text{mm/min.} = 475\ \text{mV.}$$

Therefore, “PROGAIN” = 475

This formula also works for feedrates in inches.

Example 2 in inches:

Axis parameter “G00FEED” is set for 500 inches/min and the desired Following Error is 0.001 inch for a feedrate of 1 inch/min (unity gain).

Drive analog signal: 9.5V for F(G00) = 500 inches/min

Analog voltage corresponding to 0.3937 inch (1mm) = Pg:

$$Pg = \frac{9{,}500\text{mV} \times 1\ \text{inch/min}}{500\ \text{inch/min}} \times 39.37 = 748\ \text{mV}$$

Therefore, “PROGAIN” = 748

Example 3 in inches: "MAXVOLT" is set for 9.5V (9500mV)
"G00FEED" is set for 500 inches/min

A gain of 1 is desired: 1mm (0.03937 inches) of following error at 1m/min. (39.37inches/min.).

Then:

$$\frac{9500\text{mV}}{500\text{inches/min}} \times 39.37\text{inches/min} = 748\ \text{mV/mm}\ (0.03937")\ \text{following error.}$$

Example 4 in inches: All variables are the same as in Example 3 except that the gain has been increased by 20% to 1.2: 1mm (0.03937 inches) of following error at 1.2m/min (47.24inches/min). Then:

$$\frac{9500\text{mV}}{500\text{inches/min}} \times 47.24\text{inches/min} = 898\ \text{mV/mm}\ (0.03937")\ \text{following error.}$$

DERGAIN (P24) (**DER**ivative **GAIN**)

Indicates the value of the Derivative Gain. It is given in mV/10msec. and it admits values between 0 and 65535.

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 the Proportional Gain.

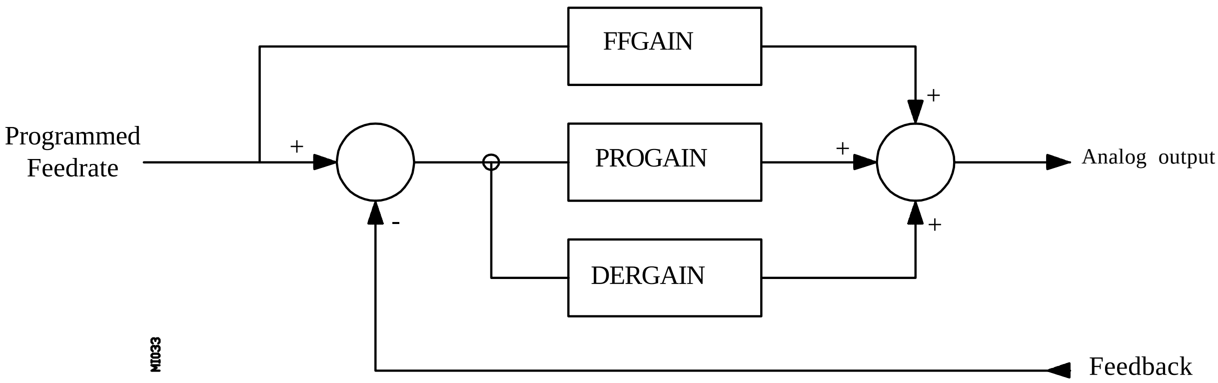
It is a good idea to also use the acc./dec. machine parameter "**ACCTIME**" for this axis (with a value other than 0) if this gain is to be applied.

By default, the CNC will assume a value of 0 (derivative gain not applied).

FFGAIN (P25) (**F**eed-**F**orward **GAIN**)

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 allows to improve the positioning loop minimizing the following error and it should be used when the "ACCTIME" machine parameter for this axis is active (acc/dec. being applied).



Possible values: Integers between 0 and 100.

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

By default, the CNC assumes a value of 0 for this parameter (no Feed-Forward gain applied).

LOOPCHG (P26) (**LOOP CH**an**G**e)

Indicates the sign of the analog output. If correct, leave it as is. If not, change it from YES to NO or viceversa.

Possible values: YES and NO.

By default, the CNC assumes NO.

MINANOUT (P27) (MINimum ANalog OUTput)

Indicates the minimum analog output for this axis.

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

MINANOUT	Minimum Analog
1	0.3 mV.
...	.....
3277	1 V.
...	.....
32767	10 V.

By default, the CNC assumes a value of 0 for this parameter.

SERVOFF (P28) (SERVo OFFset)

Indicates the offset value applied to the analog output for the drive.

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

SERVOFF	Offset
-32767	-10 V.
...	.....
-3277	-1 V.
...	.....
1	0.3 mV.
...	.....
3277	1 V.
...	.....
32767	10 V.

By default, the CNC assumes a value of 0 for this parameter (no offset).

BAKANOUT (P29) (**BA**c**K**lash **AN**alog **OUT**put)

Additional analog **pulse** to compensate for backlash when changing movement direction.

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.

BAKANOUT	Additional analog
1	0.3 mV.
...	.....
3277	1 V.
...	.....
32767	10 V.

Every time the axis changes direction, the CNC will apply the analog voltage corresponding to the move plus the one corresponding to The value selected in this machine parameter. This additional analog voltage will be applied for a period of time indicated in the axis machine parameter "**BA**c**KTIME**"

By default, the CNC assumes a value of 0 for this parameter (no additional analog pulse).

BAKTIME (P30) (**BA**c**K**lash peak **TIME**)

Indicates the duration of the additional analog pulse for backlash (P29).

It is given in milliseconds and it admits integer values between 0 and 65535.

By default, the CNC assumes a value of 0 for this parameter.

DECINPUT (P31) (**DEC**eleration **INPUT**)

Indicates whether or not this axis has a home switch for machine reference search.

NO = It doesn't have a home switch.
YES = it has a home switch.

By default, the CNC assumes that it has a home switch (YES).

REFPULSE (P32) (**REF**erence **PULSE**)

Indicates the type of flank of the marker pulse Io used for searching home.

+ = Up flank (change from 0V to 5V).
- = Down flank (change from 5V to 0V).

The CNC assumes "+" for this parameter.

REFDIREC (P33) (**REF**erencing **DIREC**tion)

Indicates the direction of the the home search in this axis.

+ = Positive direction.
- = Negative direction.

By default, the CNC assumes “+” (positive direction).

REFEED1 (P34) (**RE**ferencing **FEED**rate **1**)

Indicates the axis feedrate when searching home until it hits the home switch.

Possible values: 0.0001 thru 199999.9999 mm/min or degrees/min.
0.00001 thru 7874.01574 inch/min.

By default, the CNC assumes a feedrate of 1000mm/min (about 40 inch/min)

REFEED2 (P35) (**RE**ferencing **FEED**rate **2**)

Indicates the axis feedrate when searching home after hitting the home switch until it finds the marker pulse Io).

Possible values: 0.0001 thru 99999.9999 mm/min or degrees/min.
0.00001 thru 3937.00787 inch/min.

By default, the CNC assumes a feedrate of 100mm/min. (about 4 inch/min)

REFVALUE (P36) (**REF**erence **VALUE**)

Indicates the position value of the machine reference **point** (physical location of the marker pulse) with respect to machine reference **zero**.

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

The machine reference point is set by the manufacturer to synchronize the coordinate system. The machine positions the axis at this point instead of moving it to the machine reference 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 to be assigned to this machine reference point is "0".

By default, the CNC assumes a value of 0 for this parameter.

MAXVOLT (P37) (**MAX**imum **VOLT**age)

Indicates the maximum analog voltage corresponding to the maximum feedrate of the axis (axis machine parameter “G00FEED”).

It is given in millivolts and it admits integer values between 0 and 9999.

By default, the CNC assumes a value of 9500 (9.5 V).

G00FEED (P38) (**G00 FEED**rate)

Indicates the maximum feedrate G00 (rapid traverse) of this axis.

Possible values: 0.0001 thru 199999.9999 mm/min or degrees/min.
0.00001 thru 7874.01574 inch/min.

By default, the CNC assumes a value of 10000 mm/min.(about 400 inch/min).

UNIDIR (P39) (**UNI**directional positioning **DIR**ection)

Indicates the direction of the unidirectional approach in G00 moves.

+ = Positive direction.
- = Negative direction.

By default, the CNC assumes a positive direction “+”.

OVERRUN (P40)

Indicates the distance to be kept between the unidirectional approach point and the one programmed. If it is a Lathe model, this distance must be in radius.

Possible values: 0.0001 thru 99999.9999 millimeters or degrees.
0.00001 thru 3937.00787 inches.

By default, the CNC assumes a value of 0 for this parameter. Unidirectional approach not wanted.

UNIFEED (P41) (**UNI**directional positioning **FEED**rate)

Indicates the feedrate of the unidirectional approach from the approach point to the programmed point.

Possible values: 0.0001 thru 99999.9999 mm/min.or degrees/min.
0.00001 thru 3937.00787 inch/min.

By default, the CNC assumes a value of 0 for this parameter.

MAXFEED (P42) (**MAX**imum **FEED**rate)

Indicates the maximum programmable feedrate (F0).

Possible values: 0.0001 thru 199999.9999 mm/min. or degrees/min
0.00001 thru 7874.01574 inch/min.

By default, the CNC assumes a value of 5000 mm/min. (about 200 inch/min).

JOGFEED (P43) (**JOG**ging **FEED**rate)

Indicates the feedrate assumed in the JOG mode if no feedrate is active.

Possible values: 0.0001 thru 199999.9999 mm/min. or degrees/min.
0.00001 thru 7874.01574 inch/min.

By default, the CNC assumes a value of 1000mm/min (about 40 inch/min).

PRBFEED (P44) (**PR**o**B**ing **FEED**rate)

Indicates the probing feedrate when **calibrating** a tool in "**JOG**" mode.

Possible values: Between 0.0001 and 99999.999 mm/min.
Between 0.00001 and 3937.00787 inch/min.

The default value for this parameter is 100mm/min. (about 4 inch/min).

MAXCOUPE (P45) (**MAX**imum **COUP**ling **E**rror)

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

This value is only assigned to the slave axis.

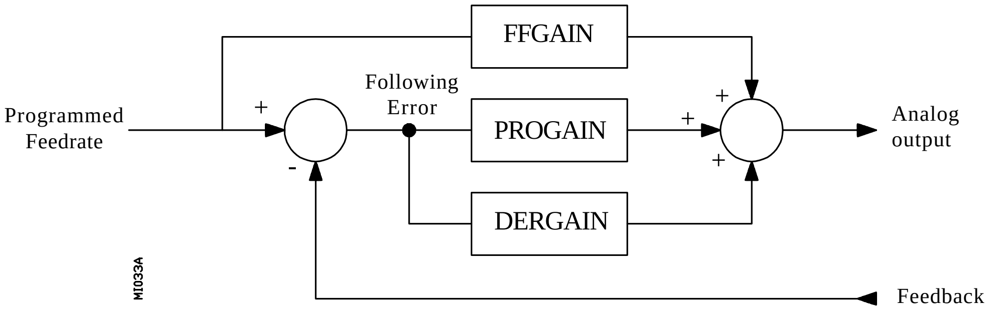
Possible values: 0.0001 thru 99999.9999 millimeters.
0.00001 thru 3937.00787 inches.

By default, the CNC assigns a value of 1 mm to this parameter.

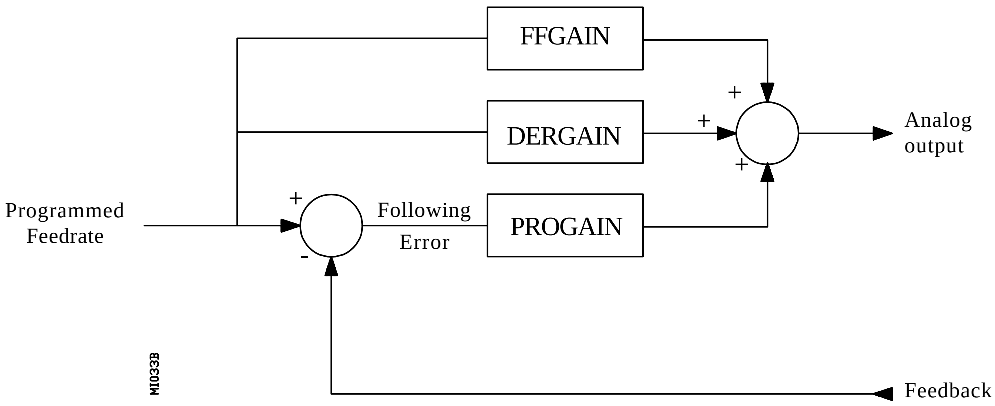
ACFGAIN (P46) (AC-Forward **GAIN**)

Indicates whether or not the value assigned to machine parameter "**DERGAIN**" (P24) is applied onto the variations of the programmed feedrate (AC-forward).

NO = It is applied onto the variations of the **following erro**r (derivative gain).



YES = It is applied onto the variations of the programmed feedrate caused by the acc./dec. (AC-forward).



By default, the CNC sets this parameter to NO.

When set to "YES", it can assist in adjusting DERGAIN for high % FFGAIN applications.

REFSHIFT (**P47**) (**REF**erence point **SHIFT**)

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 (shift) between those two reference points (previous and current)

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

By default, the CNC sets this parameter to "0".

If this parameter is set to a value other than "0"; when searching home, the CNC will move the axis the distance indicated by this parameter once the marker pulse has been detected in order to position the axis exactly at the **previous physical machine reference point**.

This additional movement is carried out at the feedrate indicated by machine parameter for the axes: "REFEED2".

STOPTIME (**P48**) (**STOP TIME**)
STOPMOVE (**P49**) (**STOP MOVE**ment)

These parameters are used in conjunction with parameter "**STOPAOUT (P50)**" when function G52 (move to hardstop) is active.

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.

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: 0.0001 thru 99999.9999 mm
0.00001 thru 3937.00787 inches.

The default value for these parameters is "0".

STOPAOUT (**P50**) (**STOP A**nalog **OUT**put)

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.

It is expressed in D/A converter units and it admits integers between 0 and 32767 in such a way that a value of 32767 corresponds to 10V.

STOPAOUT	Additional analog voltage
1 ... 3277 ... 32767	0.3mV ... 1V ... 10V

The CNC's default value for this parameter is "0".

INPOSW2 (P51) (IN POSition **W**idth **2**)

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 thus going on to execute the next block.

Possible values: Between 0 and 99999.9999 millimeters or degrees.
Between 0 and 3937.00787 inches.

By default, the CNC sets this parameter to a value of "0.01mm" and it is recommended to set it to a value 10 times the "INPOSW" value.

I0TYPE(P52) (I0 TYPE)

It indicates the type of Io signal (marker pulse) provided by the feedback device.

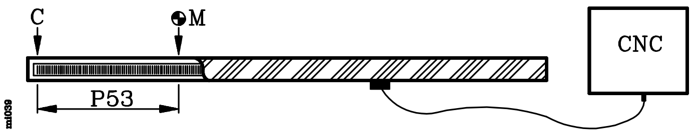
0 = Normal Io
1 = A type of coded Io
2 = B type of coded Io

ABSOFF (P53) (ABSolute OFFset)

The CNC takes this parameter into consideration when parameter "I0TYPE" (P52) is set with a value other than "0".

Thes scales having a coded Io (semi-absolute reference mark) indicate the machine position with respect to the "zero" of the scale.

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 scale "point "C").



Possible values: Between 0 and 99999.9999 millimeters.
Between 0 and 3937.00787 inches

The default value for this parameter is "0".

MINMOVE (P54) (MINinum MOVEment)

This parameter has to do with the axis logic outputs "ANT1" thru "ANT6".

If the axis move is smaller than the value indicated by this machine parameter "MINMOVE", the corresponding axis logic output "ANT1 thru "ANT6" goes high.

Possible values: Between 0 and 99999.9999 millimeters or degrees.
Between 0 and 3937.00787 inches

The default value for this parameter is "0".

ROLLOVER (P55)

This machine parameter is taken into account when the axis has been set as rotary, "AXISTPYE (P0) = 2 or 3". It indicates whether the rotary axis is also rollover or not.

Possible values: YES / NO

The default value for this parameter is "YES".

SERCOSID (P56)

Not being used at this time, it is reserved for future applications.

It must always be set to "0".

EXTMULT (P57) (EXTernal **MULT**iplier**)**

This parameter is to be used when utilizing a semi-absolute feedback system (with coded Io).

It indicates the relationship between the mechanical pitch or that of the glass graduation with the electrical pitch or the period of the feedback signal supplied to the CNC.

$$\text{EXTMULT} = \frac{\text{Glass graduation pitch (mechanical pitch)}}{\text{feedback signal period (electrical pitch)}}$$

For instance, the FAGOR "FOT" linear scale has a glass graduation pitch of 100 µm while its output signals have an electrical pitch of 20 µm.

$$\text{EXTMULT} = \frac{100}{20} = 5$$

Values to be allocated for semi-absolute Fagor scales (with coded Io):

COS, COVS, MOVS, FOS .... EXTMULT = 1
COC, COVC, MOVC, FOC .... EXTMULT = 1
COX, COVX, MOVX, FOT .... EXTMULT = 5

The default value for this parameter is "0".

3.3.3 SPINDLE MACHINE PARAMETERS

3.3.3.1 MACHINE PARAMETERS FOR MAIN SPINDLE

SPDLTYPE (P0) (**SP**in**DL**e **TYPE**)

Indicates the type of spindle output being used.

0 = ±10V DC.
1 = 2-digit BCD code.
2 = 8-digit BCD code.

By default, the CNC assumes ±10V D.C. (value 0).

DFORMAT (P1) (**D**isplay **FORMAT**)

Indicates the display format for the spindle.

0 = In 4 digits.
1 = In 5 digits.
2 = In format 4.3.
3 = In format 5.3.
4 = Not displayed.

By default, the CNC will assume a value of 0 for this parameter.

MAXGEAR1 (P2) (**MAX**imum speed of **GEAR 1**)

Indicates the maximum spindle speed assigned to the 1st range (M41).

It is given in revolutions per minutes (r.p.m.) between 0 and 65535.

By default, the CNC assumes a value of 1000 r.p.m. for this parameter.

MAXGEAR2 (P3) (**MAX**imum speed of **GEAR 2**)

Indicates the maximum spindle speed assigned to the 2nd range (M42).

It is given in revolutions per minutes (r.p.m.) between 0 and 65535.

By default, the CNC assumes a value of 2000 r.p.m. for this parameter.

MAXGEAR3 (P4) (**MAX**imum speed of **GEAR 3**)

Indicates the maximum spindle speed assigned to the 3rd range (M43).

It is given in revolutions per minutes (r.p.m.) between 0 and 65535.

By default, the CNC assumes a value of 3000 r.p.m. for this parameter.

MAXGEAR4 (P5) (**MAX**imum speed of **GEAR 4**)

Indicates the maximum spindle speed assigned to the 4th range (M44).

It is given in revolutions per minutes (r.p.m.) between 0 and 65535.

By default, the CNC assumes a value of 4000 r.p.m. for this parameter.

The value assigned to "MAXGEAR1" must be the lowest speed range and the one assigned to "MAXGEAR4" that of the highest speed range. When not all four ranges are required, use the lower ones starting from "MAXGEAR1" and assign the highest one to the unused ranges.

AUTOGEAR (P6) (**AUTO**matic **GEAR** change)

Indicates whether the change of range is generated automatically or not by the CNC activating the M functions M41, M42, M43 and M44.

NO = It is NOT done automatically.
YES = It is done automatically.

By default, the CNC assumes NO for this parameter.

POLARM3 (P7) (**POLAR**ity for **M3**)
POLARM4 (P8) (**POLAR**ity for **M4**)

Indicates the sign of the spindle analog for M03 and M04.

+ = Positive analog.
- = Negative analog.

If the same value is assigned to both parameters, the CNC will output a single polarity (0V to 10V) signal with the indicated sign.

By default, the CNC assigns "+" to "POLARM3" and "-" to "POLARM4".

SREVM05 (P9) (**S**pindle **REV**erse needs **M05**)

This parameter is used with a Mill model CNC.

Indicates whether it is necessary or not to stop the spindle (M05) when reversing rotation direction during a tapping canned cycle (G84).

NO = M5 is NOT necessary.
YES = M5 is necessary.

By default, the CNC assumes YES.

MINSOVR (P10) (**MIN**imum **S**pindle **OV**e**R**ride)
MAXSOVR (P11) (**MAX**imum **S**pindle **OV**e**R**ride)

Indicate the minimum and maximum % applicable to the programmed spindle speed.

Possible values: Integers between 0 and 255.

By default, the CNC assigns a value of 50% to “MINSOVER” and 120% to “MAXSOVR”.

The resulting speed will be limited by the value indicated in the spindle machine parameter: “MAXVOLT1”, “MAXVOLT2”, “MAXVOLT3”,” MAXVOLT4" corresponding to the selected range.

SOVRSTEP (P12) (**S**pindle **OV**e**R**ride **STEP**)

Indicates the incremental step of the spindle speed every time the override keys at the operator panel are pressed.

Possible values: Integer values between 0 and 255.

By default, the CNC assumes a value of 5 for this parameter.

NPULSES (P13) (**N**umber of **PULSES**)

Indicates the number of pulses per revolution provided by the spindle encoder. 0 means that there is no spindle encoder.

Possible values: Integer values between 0 and 65535.

By default, the CNC assumes a value of 1000 for this parameter.

DIFFBACK (P14) (**DIF**ferential **F**eed**BACK**)

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

NO = They are NOT differential signals.
YES = They are differential signals.

By default, the CNC assumes YES.

FBACKAL (P15) (**F**eed**BACK AL**arm)

Indicates whether the feedback alarm is OFF or ON.

OFF = Alarm off.
ON = Alarm on.

By default, the alarm is ON.

AXISCHG (P16) (**AXIS CH**an**G**e)

Indicate the counting direction of spindle encoder. If correct, leave it as is; if not, change it from YES to NO or viceversa.
If this parameter is changed, parameter “LOOPCHG” (P26) must also be changed so the spindle does not “run away”.

Possible values: YES and NO.

By default, the CNC assumes NO.

DWELL (P17)

Indicates the dwell from the moment the “ENABLE” signal is activated until the analog voltage is sent out.

It is given in milliseconds and it admits integer values between 0 and 65535.

By default, the CNC assumes a value of 0 (no dwell).

ACCTIME (P18) (**ACC**eleration **TIME**)

This parameter is used when working with the spindle in closed loop and it indicates the acceleration time given to reach the maximum speed (MAXVOLT1 thru MAXVOLT4) in each range. This value also represents the deceleration time.

Possible integer values between 0 and 65535 milliseconds.

The default value for this parameter is 0 (no acceleration or deceleration).

INPOSW (P19) (**IN POS**ition **W**idth)

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: 0 thru 99999.99999 degrees.

By default, the CNC assumes a value of 0.01 degrees.

INPOTIME (P20) (**IN PO**sition **TIME**)

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.

It is given in milliseconds and it admits any integer value between 0 and 65535.

By default, the CNC assumes a value of 0 for this parameter.

MAXFLWE1 (P21) (**MAX**imum **F**o**L**lo**W**ing **E**rror)

Indicates the maximum following error allowed for the spindle when moving in closed loop (M19).

Possible values: 0 thru 99999.99999 degrees.

By default, the CNC assumes a value of 30 degrees.

MAXFLWE2 (P22) (**MAX**imum **F**o**L**lo**W**ing **E**rror)

Indicates the maximum following error allowed for the spindle when stopped in closed loop (M19).

Possible values: 0 thru 99999.99999 degrees.

By default, the CNC assumes a value of 0.1 degrees.

PROGAIN (P23) (**PRO**portional **GAIN**)

Indicates, the value of the Proportional Gain in millivolts per degree and it admits integer values between 0 and 65535.

Its value represents the analog voltage corresponding to a following error of 1 degree when working in closed loop (M19).

This value is taken for the 1st range of spindle speeds and the CNC calculates the rest of the values for the other ranges.

By default, the CNC assumes a value of 1000mv/degree.

Example:

Spindle machine parameter “MAXGEAR1” = 500 rev/min. The desired speed for a 1 degree of following error is S = 1000°/min (2.778 rev/rpm).

Analog voltage for spindle drive: 9.5 V. for 500 rpm.

Analog voltage corresponding to S = 1000°/min (2.778 rpm):

$$\text{Analog} = \frac{9.5\ \text{V.}}{500\ \text{rev/min.}} \times 2.778\ \text{rev/min.} = 52{,}782\ \text{mV.}$$

Therefore, “PROGAIN” = 53

DERGAIN (P24) (**DER**ivative **GAIN**)

Indicates the value of the Derivative Gain. It is given in mV/10msec. and it admits values between 0 and 65535.

Its value represents the analog voltage (in millivolts) corresponding to a change in following error of 1mm (0.040 inches) in 10 milliseconds. This analog voltage will be added to the one calculated for the the Proportional Gain.

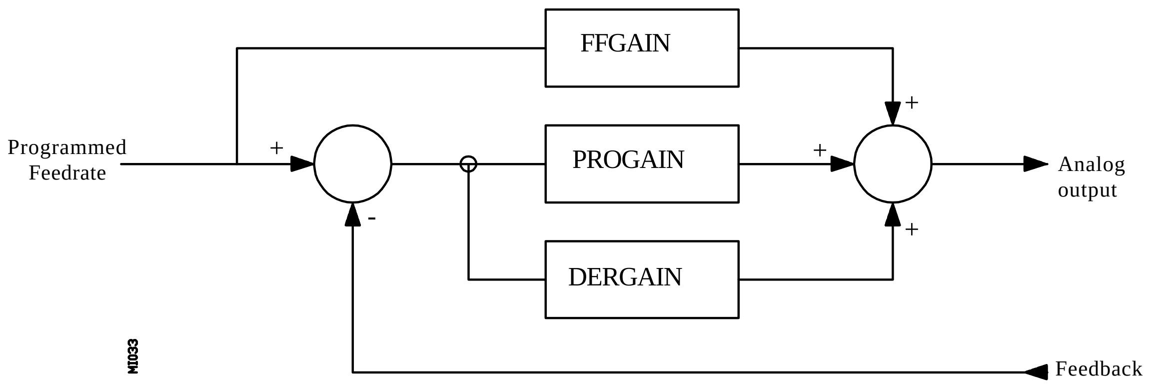
It is a good idea to also use the acc./dec. machine parameter “ACCTIME” for the spindle (with a value other than 0) if this gain is to be applied.

By default, the CNC will assume a value of 0 (derivative gain not applied).

FFGAIN (P25) (**F**eed-**F**orward **GAIN**)

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 allows to improve the positioning loop minimizing the following error, but it should not be used when the “ACCTIME” machine parameter for the spindle is active (acc/dec. being applied).



Possible values: Integers between 0 and 100.

Usually, a value between 40% and 60% is assigned depending mainly on the type of machine and its characteristics.

By default, the CNC assumes a value of 0 for this parameter (no Feed-Forward gain applied).

LOOPCHG (P26) (**LOOP CH**an**G**e)

Indicates the sign of the analog output. If correct, leave it as is; if not, change it from YES to NO or viceversa.

Possible values: YES and NO.

By default, the CNC assumes NO.

MINANOUT (P27) (**MIN**imum **AN**alog **OUT**put)

Indicates the minimum value for the spindle analog output.

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.

MINANOUT	Minimum Analog
1	0.3 mV.
...	.....
3277	1 V.
...	.....
32767	10 V.

By default, the CNC assumes a value of 0 for this parameter.

SERVOFF (P28) (**SERV**o **OFF**set)

Indicates the analog offset value for the spindle drive.

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.

SERVOFF	Offset
-32767	-10 V.
...	.....
-3277	-1 V.
...	.....
1	0.3 mV.
...	.....
3277	1 V.
...	.....
32767	10 V.

By default, the CNC assumes a value of 0 (no offset being applied).

LOSPDLIM (P29) (**LO**wer **SP**in**D**le **LIM**it)
UPSPDLIM (P30) (**UP**per **SP**in**D**le **LIM**it)

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.

It is given in % and it admits integer values between 0 and 255.

By default, the CNC assigns a value of 50 to "LOSPDLIM" and a value of 150 to "UPSPDLIM".

DECINPUT (P31) (**DEC**eleration **INPUT**)

Indicates whether or not the spindle has a home switch to synchronize the spindle when working in M19.

NO = It has NO home switch.
YES = It has a home switch.

By default, the CNC assumes YES.

REFPULSE (P32) (**REF**erence **PULSE**)

Indicates the type of marker pulse Io to synchronize the spindle when working in M19.

+ = Positive pulse (5V).
- = Negative pulse (0V).

By default, the CNC assumes "+".

REFDIREC (P33) (**REF**erencing **DIREC**tion)

Indicates the rotating direction when synchronizing the spindle during M19.

+ = Positive direction.
- = Negative direction.

By default, the CNC assumes "+".

REFEED1 (P34) (**RE**ferencing **FEED**rate **1**)

Indicates the spindle's positioning speed when in M19 and the synchronizing speed until it finds the home switch.

Possible values: 0.00001 thru 199999.99999 degrees/min.

By default, the CNC assumes a value of 9000 degrees/min.

REFEED2 (P35) (**RE**ferencing **FEED**rate **2**)

Indicates the synchronizing speed of the spindle after hitting the home switch and until it finds the marker pulse.

Possible values: 0.00001 thru 99999.99999 degrees/min.

By default, the CNC assumes a value of 360 degrees/min.

REFVALUE (P36) (**REF**erence **VALUE**)

Indicates the position value assigned to the reference point of the spindle (home or marker pulse).

Possible values: ±99999.99999 degrees.

By default, the CNC assumes a value of 0.

MAXVOLT1 (P37) (**MAX**imum **VOLT**age gear **1**)

Indicates the analog voltage corresponding to the maximum speed of range 1.

It is given in millivolts and it admits any integer between 0 and 9999.

By default, the CNC assumes a value of 9500 (9.5 V).

MAXVOLT2 (P38) (**MAX**imum **VOLT**age gear **2**)

Indicates the analog voltage corresponding to the maximum speed of range 2.

It is given in millivolts and it admits any integer between 0 and 9999.

By default, the CNC assumes a value of 9500 (9.5 V).

MAXVOLT3 (P39) (**MAX**imum **VOLT**age gear **3**)

Indicates the analog voltage corresponding to the maximum speed of range 3.

It is given in millivolts and it admits any integer between 0 and 9999.

By default, the CNC assumes a value of 9500 (9.5 V).

MAXVOLT4 (P40) (**MAX**imum **VOLT**age gear **4**)

Indicates the analog voltage corresponding to the maximum speed of range 4.

It is given in millivolts and it admits any integer between 0 and 9999.

The default setting of this parameter is 9500 (9.5V).

GAIN UNIT (P41)

It indicates the units of the “PROGAIN” (P23) and “DERGAIN” (P24) parameters for the spindle.

0 = millivolts/degree
1 = millivolts/0.01 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. Therefore, the adjustment of machine parameters “PROGAIN” and “DERGAIN” will be more critical (sensitive).

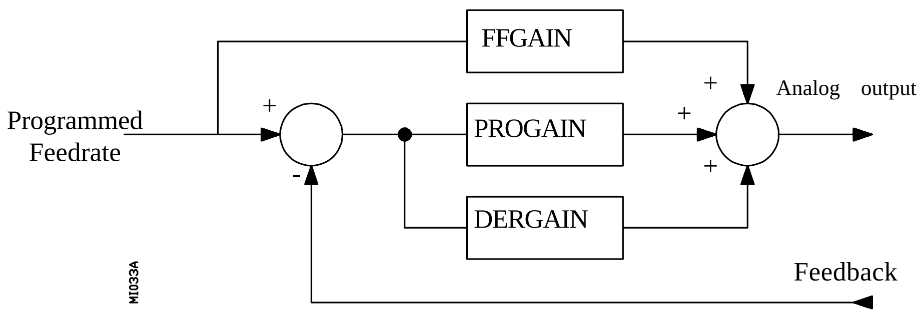
The default value for this parameter is 0 (mV/degree).

By default, the CNC assumes a value of 9500 (9.5 V).

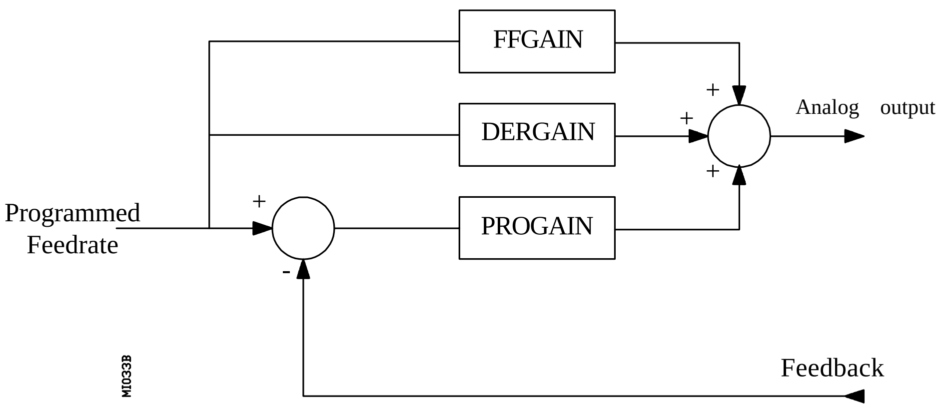
ACFGAIN (P42) (**AC-F**orward **GAIN**)

Indicates whether the "DERGAIN (P24)" value is to be applied onto the variations of following error(derivative gain) or onto the variations of programmed feedrate (AC-forward).

NO = It is applied onto the variations of following error (derivative gain).



YES = It is applied onto the variations of programmed feedrate due to ACC/DEC (AC-forward).



The default value of this parameter is "NO".

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.

0 = The spindle must be homed every time when switching from open to closed loop.
1 = It is enough to home the spindle once on power-up.

The default value for this parameter is "0".

3.3.3.2 MACHINE PARAMETERS FOR THE SECOND SPINDLE

These parameters are available on the lathe model CNC.

SPDLTYPE (P0) (**SP**in**DL**e **TYPE**)

Indicates the type of spindle output being used.

0 = ±10V DC.
1 = 2-digit BCD code.
2 = 8-digit BCD code.

By default, the CNC assumes ±10V D.C. (value 0).

DFORMAT (P1) (**D**isplay **FORMAT**)

Not being used at this time. Reserved for future applications.

MAXGEAR1 (P2) (**MAX**imum speed of **GEAR 1**)

Indicates the maximum spindle speed assigned to the 1st range (M41).

It is given in revolutions per minutes (r.p.m.) between 0 and 65535. By default, the CNC assumes a value of 1000 r.p.m. for this parameter.

MAXGEAR2 (P3) (**MAX**imum speed of **GEAR 2**)

Indicates the maximum spindle speed assigned to the 2nd range (M42).

It is given in revolutions per minutes (r.p.m.) between 0 and 65535. By default, the CNC assumes a value of 2000 r.p.m. for this parameter.

MAXGEAR3 (P4) (**MAX**imum speed of **GEAR 3**)

Indicates the maximum spindle speed assigned to the 3rd range (M43).

It is given in revolutions per minutes (r.p.m.) between 0 and 65535. By default, the CNC assumes a value of 3000 r.p.m. for this parameter.

MAXGEAR4 (P5) (**MAX**imum speed of **GEAR 4**)

Indicates the maximum spindle speed assigned to the 4th range (M44).

It is given in revolutions per minutes (r.p.m.) between 0 and 65535. By default, the CNC assumes a value of 4000 r.p.m. for this parameter.

The value assigned to “MAXGEAR1” must be the lowest speed range and the one assigned to “MAXGEAR4” that of the highest speed range. When not all four ranges are required, use the lower ones starting from “MAXGEAR1” and assign the highest one to the unused ranges.

AUTOGEAR (P6) (**AUTO**matic **GEAR** change)

Indicates whether the change of range is generated automatically or not by the CNC activating the M functions M41, M42, M43 and M44.

NO = It is NOT done automatically.
YES = It is done automatically.

By default, the CNC assumes NO for this parameter.

POLARM3 (P7) (**POLAR**ity for **M3**)
POLARM4 (P8) (**POLAR**ity for **M4**)

Indicates the sign of the spindle analog for M03 and M04.

\+ = Positive analog.
\- = Negative analog.

If the same value is assigned to both parameters, the CNC will output a single polarity (0V to 10V) signal with the indicated sign.

By default, the CNC assigns "+" to "POLARM3" and "-" to "POLARM4".

SREVM05 (P9)
MINSOVR (P10)
MAXSOVR (P11)
SOVRSTEP (P12)

Not being used at this time. Reserved for future applications.

NPULSES (P13) (**N**umber of **PULSES**)

Indicates the number of pulses per revolution provided by the spindle encoder. 0 means that there is no spindle encoder.

Possible values: Integer values between 0 and 65535.

By default, the CNC assumes a value of 1000 for this parameter.

DIFFBACK (P14) (**DIF**ferential **F**eed**BACK**)

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

NO = They are NOT differential signals.
YES = They are differential signals.

By default, the CNC assumes YES.

FBACKAL (P15) (**F**eed**BACK AL**arm)

Indicates whether the feedback alarm is OFF or ON.

OFF = Alarm off.
ON = Alarm on.

By default, the alarm is ON.

AXISCHG (P16) (**AXIS CH**an**G**e)

Indicate the counting direction of spindle encoder. If correct, leave it as is; if not, change it from YES to NO or viceversa.
If this parameter is changed, parameter “LOOPCHG” (P26) must also be changed so the spindle does not “run away”.

Possible values: YES and NO.

By default, the CNC assumes NO.

DWELL (P17)

Indicates the dwell from the moment the “ENABLE” signal is activated until the analog voltage is sent out.

It is given in milliseconds and it admits integer values between 0 and 65535.

By default, the CNC assumes a value of 0 (no dwell).

ACCTIME (P18) (**ACC**eleration **TIME**)

This parameter is used when working with the spindle in closed loop and it indicates the acceleration time given to reach the maximum speed (MAXVOLT1 thru MAXVOLT4) in each range. This value also represents the deceleration time.

Possible integer values between 0 and 65535 milliseconds.

The default value for this parameter is 0 (no acceleration or deceleration).

INPOSW (P19) (**IN POS**ition **W**idth)

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: 0 thru 99999.99999 degrees.

By default, the CNC assumes a value of 0.01 degrees.

INPOTIME (P20) (**IN PO**sition **TIME**)

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.

It is given in milliseconds and it admits any integer value between 0 and 65535.

By default, the CNC assumes a value of 0 for this parameter.

MAXFLWE1 (P21) (**MAX**imum **F**o**L**lo**W**ing **E**rror)

Indicates the maximum following error allowed for the spindle when moving in closed loop (M19).

Possible values: 0 thru 99999.99999 degrees.

By default, the CNC assumes a value of 30 degrees.

MAXFLWE2 (P22) (**MAX**imum **F**o**L**lo**W**ing **E**rror)

Indicates the maximum following error allowed for the spindle when stopped in closed loop (M19).

Possible values: 0 thru 99999.99999 degrees.

By default, the CNC assumes a value of 0.1 degrees.

PROGAIN (P23) (**PRO**portional **GAIN**)

Indicates, the value of the Proportional Gain in millivolts per degree and it admits integer values between 0 and 65535.

Its value represents the analog voltage corresponding to a following error of 1 degree when working in closed loop (M19).

This value is taken for the 1st range of spindle speeds and the CNC calculates the rest of the values for the other ranges.

By default, the CNC assumes a value of 1000mv/degree.

Example:

Spindle machine parameter "MAXGEAR1" = 500 rev/min. The desired speed for a 1 degree of following error is S = 1000°/min (2.778 rev/rpm).

Analog voltage for spindle drive: 9.5 V. for 500 rpm.

Analog voltage corresponding to S = 1000°/min (2.778 rpm):

$$\text{Analog} = \frac{9.5\text{ V.}}{500\text{ rev/min.}} \times 2.778\text{ rev/min.} = 52{,}782\text{ mV.}$$

Therefore, "PROGAIN" = 53

DERGAIN (P24) (**DER**ivative **GAIN**)

Indicates the value of the Derivative Gain. It is given in mV/10msec. and it admits values between 0 and 65535.

Its value represents the analog voltage (in millivolts) corresponding to a change in following error of 1mm (0.040 inches) in 10 milliseconds. This analog voltage will be added to the one calculated for the the Proportional Gain.

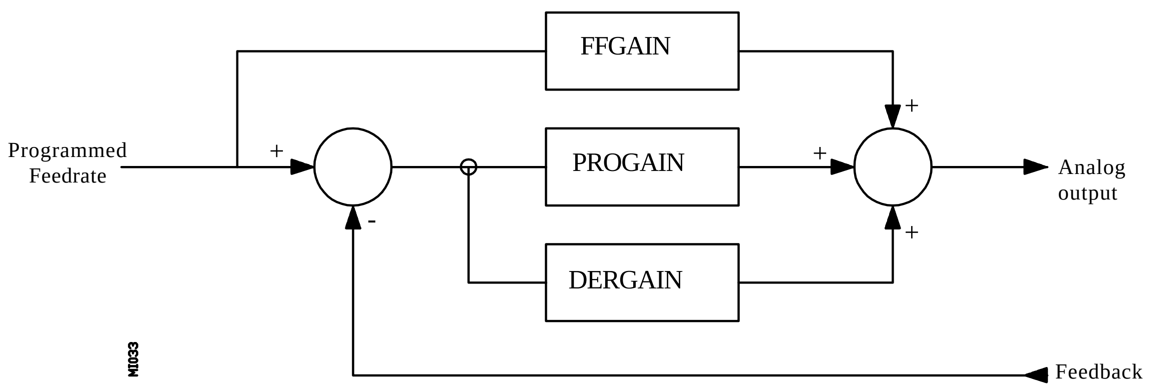
It is a good idea to also use the acc./dec. machine parameter "ACCTIME" for the spindle (with a value other than 0) if this gain is to be applied.

By default, the CNC will assume a value of 0 (derivative gain not applied).

FFGAIN (P25) (**F**eed-**F**orward **GAIN**)

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 allows to improve the positioning loop minimizing the following error, but it should not be used when the "ACCTIME" machine parameter for the spindle is active (acc/dec. being applied).



Possible values: Integers between 0 and 100.

Usually, a value between 40% and 60% is assigned depending mainly on the type of machine and its characteristics.

By default, the CNC assumes a value of 0 for this parameter (no Feed-Forward gain applied).

LOOPCHG (P26) (**LOOP CH**an**G**e)

Indicates the sign of the analog output. If correct, leave it as is; if not, change it from YES to NO or viceversa.

Possible values: YES and NO.

By default, the CNC assumes NO.

MINANOUT (P27) (**MIN**imum **AN**alog **OUT**put)

Indicates the minimum value for the spindle analog output.

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.

MINANOUT	Minimum Analog
1	0.3 mV.
...	.....
3277	1 V.
...	.....
32767	10 V.

By default, the CNC assumes a value of 0 for this parameter.

SERVOFF (P28) (**SERV**o **OFF**set)

Indicates the analog offset value for the spindle drive.

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.

SERVOFF	Offset
-32767	-10 V.
...	.....
-3277	-1 V.
...	.....
1	0.3 mV.
...	.....
3277	1 V.
...	.....
32767	10 V.

By default, the CNC assumes a value of 0 (no offset being applied).

LOSPDLIM (P29) (**LO**wer **SP**in**D**le **LIM**it)
UPSPDLIM (P30) (**UP**per **SP**in**D**le **LIM**it)

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.

It is given in % and it admits integer values between 0 and 255.

By default, the CNC assigns a value of 50 to "LOSPDLIM" and a value of 150 to "UPSPDLIM".

DECINPUT (P31) (**DEC**eleration **INPUT**)

Indicates whether or not the spindle has a home switch to synchronize the spindle when working in M19.

NO = It has NO home switch.
YES = It has a home switch.

By default, the CNC assumes YES.

REFPULSE (P32) (**REF**erence **PULSE**)

Indicates the type of marker pulse Io to synchronize the spindle when working in M19.

+ = Positive pulse (5V).
- = Negative pulse (0V).

By default, the CNC assumes "+".

REFDIREC (P33) (**REF**erencing **DIREC**tion)

Indicates the rotating direction when synchronizing the spindle during M19.

+ = Positive direction.
- = Negative direction.

By default, the CNC assumes "+".

REFEED1 (P34) (**RE**ferencing **FEED**rate **1**)

Indicates the spindle's positioning speed when in M19 and the synchronizing speed until it finds the home switch.

Possible values: 0.00001 thru 199999.99999 degrees/min.

By default, the CNC assumes a value of 9000 degrees/min.

REFEED2 (P35) (**RE**ferencing **FEED**rate **2**)

Indicates the synchronizing speed of the spindle after hitting the home switch and until it finds the marker pulse.

Possible values: 0.00001 thru 99999.99999 degrees/min.

By default, the CNC assumes a value of 360 degrees/min.

REFVALUE (P36) (**REF**erence **VALUE**)

Indicates the position value assigned to the reference point of the spindle (home or marker pulse).

Possible values: ±99999.99999 degrees.

By default, the CNC assumes a value of 0.

MAXVOLT1 (P37) (**MAX**imum **VOLT**age gear **1**)

Indicates the analog voltage corresponding to the maximum speed of range 1.

It is given in millivolts and it admits any integer between 0 and 9999.

By default, the CNC assumes a value of 9500 (9.5 V).

MAXVOLT2 (P38) (**MAX**imum **VOLT**age gear **2**)

Indicates the analog voltage corresponding to the maximum speed of range 2.

It is given in millivolts and it admits any integer between 0 and 9999.

By default, the CNC assumes a value of 9500 (9.5 V).

MAXVOLT3 (P39) (**MAX**imum **VOLT**age gear **3**)

Indicates the analog voltage corresponding to the maximum speed of range 3.

It is given in millivolts and it admits any integer between 0 and 9999.

By default, the CNC assumes a value of 9500 (9.5 V).

MAXVOLT4 (P40) (**MAX**imum **VOLT**age gear **4**)

Indicates the analog voltage corresponding to the maximum speed of range 4.

It is given in millivolts and it admits any integer between 0 and 9999.

The default setting of this parameter is 9500 (9.5V).

GAIN UNIT (P41)

It indicates the units of the “PROGAIN” (P23) and “DERGAIN” (P24) parameters for the spindle.

0 = millivolts/degree
1 = millivolts/0.01 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. Therefore, the adjustment of machine parameters “PROGAIN” and “DERGAIN” will be more critical (sensitive).

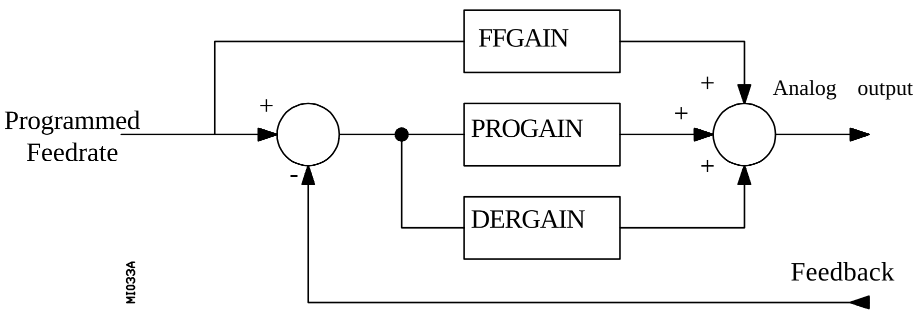
The default value for this parameter is 0 (mV/degree).

By default, the CNC assumes a value of 9500 (9.5 V).

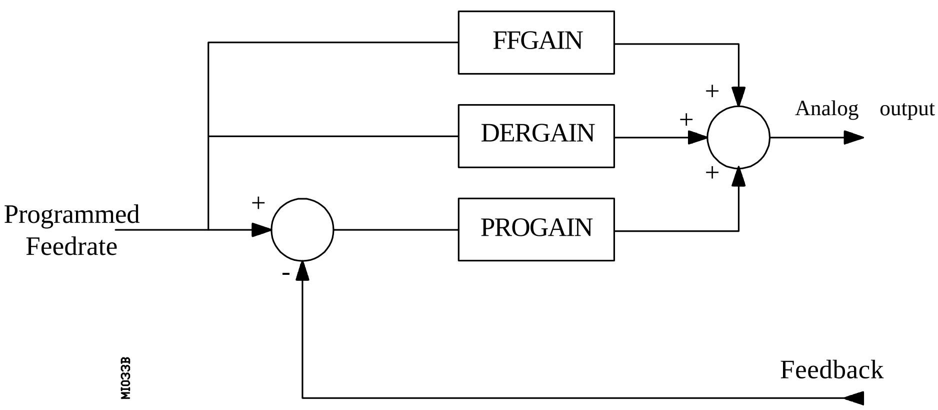
ACFGAIN (P42) (**AC-F**orward **GAIN**)

Indicates whether the "DERGAIN (P24)" value is to be applied onto the variations of following error(derivative gain) or onto the variations of programmed feedrate (AC-forward).

NO = It is applied onto the variations of following error (derivative gain).



YES = It is applied onto the variations of programmed feedrate due to ACC/DEC (AC-forward).



The default value of this parameter is "NO".

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.

0 = The spindle must be homed every time when switching from open to closed loop.
1 = It is enough to home the spindle once on power-up.

The default value for this parameter is "0".

3.3.3.3 MACHINE PARAMETERS FOR AUXILIARY SPINDLE

MAXSPEED (P0) (**Max**imum **SPEED**)

Indicates the maximum speed of the auxiliary spindle. It is given in rpms and it admits any integer value between 0 and 65535.

Its default value is 1000 rpms.

SPDLOVR (P1) (**SP**indle **OV**e**R**ide)

Indicates whether or not the spindle override buttons of the front panel affects the current speed of the auxiliary spindle when active.

NO = They do not alter the current spindle speed.
YES= They do alter it. The CNC will apply the values set for machine parameters for the main spindle "**MINSOVR**" (P10), "**MAXOVR**" (P11) and "**SOVRSTEP**" (P12).

By default, This parameter is set to **NO.**

MINANOUT (P2) (**MIN**imum **AN**alog **OUT**put)

Indicates the minimum value for the spindle analog output.

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.

MINANOUT	Minimum Analog
1	0.3 mV.
...	.....
3277	1 V.
...	.....
32767	10 V.

By default, the CNC assumes a value of 0 for this parameter.

SERVOFF (P3) (**SERV**o **OFF**set)

Indicates the analog offset value for the spindle drive.

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.

SERVOFF	Offset
-32767	-10 V.
...	.....
-3277	-1 V.
...	.....
1	0.3 mV.
...	.....
3277	1 V.
...	.....
32767	10 V.

By default, the CNC assumes a value of 0 (no offset being applied).

MAXVOLT (P4) (**MAX**imum **VOLT**age)

Indicates the analog voltage corresponding to the maximum speed defined by machine parameter **"MAXSPEED"**.

It is given in millivolts and it admits any integer value between 0 and 9999.

Its default value is 9500 (9.5V).

3.3.4 MACHINE PARAMETERS FOR THE SERIAL PORTS

BAUDRATE (P0)

Indicates the communication speed, in baud, between the CNC and the peripherals. The code is:

0 = 110 baud
1 = 150 baud
2 = 300 baud
3 = 600 baud
4 = 1200 baud
5 = 2400 baud
6 = 4800 baud
7 = 9600 baud
8 = 19200 baud
9 = 38400 baud (only when using CPU-Turbo)
10 = reserved
11 = reserved
12 = reserved

By default, the CNC assumes a value of 7 (9600 baud).

NBITSCHR (P1) (**N**umber of **BITS** per **CH**a**R**acter)

Indicates the number of data bits per transmitted character.

0 = Uses the 7 least significant bits of an 8-bit character. It is used when transmitting standard ASCII characters.

1 = Uses all 8 bits of the transmitting character. Used when transmitting special characters whose codes are greater than 127.

By default, the CNC assumes a value of 1 (8 bits).

PARITY (P2)

Indicates the type of parity check used.

0 = No parity.
1 = Odd parity.
2 = Even parity.

By default, the CNC assumes a value of 0 (no parity).

STOPBITS (P3)

Indicates the number of stop bits at the end of each transmitted word.

0 = 1 STOP bit.
1 = 2 STOP bits.

By default, the CNC assumes a value of 0 (1 STOP bit).

PROTOCOL (P4)

Indicates the type of communications protocol to be used.

0 = Communications protocol for general device.
1 = DNC protocol.
2 = Communications protocol for FAGOR floppy disc unit.

By default, the CNC assumes a value of 1. (DNC protocol).

PWONDNC (P5) (**P**o**W**er-**ON DNC**)

Indicates whether the DNC feature will be active on power-up or not.

NO = Not active on power-up.
YES = Active on power-up

By default, the DNC feature **WILL** be active on power-up (YES).

DNCDEBUG (P6)

Indicates whether the debugging feature for DNC communications is active or not. When this feature is NOT active, the CNC WILL abort DNC communications if they are not established within an internally set period of time.

It is advisable to use this safety feature in all DNC communications. It could be deactivated in the debugging process.

NO = Debug NOT active. CNC time-out active.
YES = Debug active. CNC time-out not active.

By default, the DEBUG will NOT be active.

ABORTCHR (P7) (**ABORT CH**a**R**acter)

Indicates the character used to abort communications with general peripheral device.

0 = CAN
1 = EOT

By default, the CNC assumes a value of 0 for this parameter.

EOLCHR (P8) (**E**nd **O**f **L**ine **CH**a**R**acter)

Indicates the character used to indicate "end of line" when communicating with general peripheral device.

0 = LF
1 = CR
2 = LF-CR
3 = CR-LF

By default, the CNC assumes a value of 0.

EOFCHR (P9) (**E**nd **O**f **F**ile **CH**a**R**acter)

Indicates the character used to indicate "end of text" (end of file) when communicating with a general peripheral device.

0 = EOT
1 = ESC
2 = SUB
3 = ETX

By default, the CNC assumes a value of 0 for this parameter.

XONXOFF (P10)

Indicates whether the XON-XOFF communications protocol is active or not when operating with a generic peripheral.

ON = It is active.
OFF = It is NOT active.

By default, XON-XOFF is **ON**.

3.3.5 MACHINE PARAMETERS FOR THE PLC

WDGPRG (P0) (**W**atch-**D**o**G PR**o**G**ram)

Indicates the Watch-dog time-out period for the main PLC program.

It is given in milliseconds and it admits integer values between 0 and 65535.

By default, the CNC assumes a value of 0 for this parameter.

WDGPER (P1) (**W**atch-**D**o**G PER**iodic)

Indicates the Watch-Dog time-out period for the periodic module of the PLC.

It is given in milliseconds and it admits integer values between 0 and 65535.

By default, the CNC assumes a value of 0 for this parameter.

USER0 (P2)
— —
— —
— —
USER23 (P25)

Parameters "USER0" thru "USER23" do not mean anything to the CNC. They could contain the type of information that the OEM may find necessary to customize this machine, such as:

Information about 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) thru USER7 (P9)	: Integers between 0 and 255.
USER8 (P10) thru USER15 (P17)	: Integers between 0 and 65535.
USER16 (P18) thru USER23 (P25)	: ±99999.9999 mm or ±3937.00787 inches.

By default, the CNC assigns a value of 0 to these parameters.

CPUTIME (P26)

This parameter is used when the PLC does not have its own CPU and it indicates the time the system CPU dedicates to the PLC.

Possible values:	0 = 1 millisecond every 8 samplings.
	1 = 1 millisecond every 4 samplings.
	2 = 1 millisecond every 2 samplings.
	3 = 1 millisecond every sampling.

The sampling period is determined by the general machine parameter "LOOPTIME".
Hence, for a sampling period of 4 msec. and a CPUTIME=0, the system CPU dedicates 1 millsecond every 8 samplings (thus, 32 milliseconds) to the PLC.

The default value for this parameter is "0".

PLCMEM (P27) (PLC MEMORY)

It indicates the percentage (%) of EEPROM memory used to store the PLC program.

This EEPROM memory stores the following information.

* The PLC program as mentioned above.

* The Pages (screens) and Symbols created with the Graphic Editor. The percentage of EEPROM memory space dedicated to this is established by the machine parameter "PAGESMEM".

* The part-programs of the CNC in the remaining free EEPROM memory space.

Possible integer values: between 0 and 100.

The default value for this parameter is 50.

Atention:

Changing this parameter can cause items previously stored in EEPROM memory to be lost.

To avoid this, proceed as follows:

1.- Transfer the data from the EEPROM to a peripheral or PC.

2.- Change the setting of "PAGESMEM" and "PLCMEM".

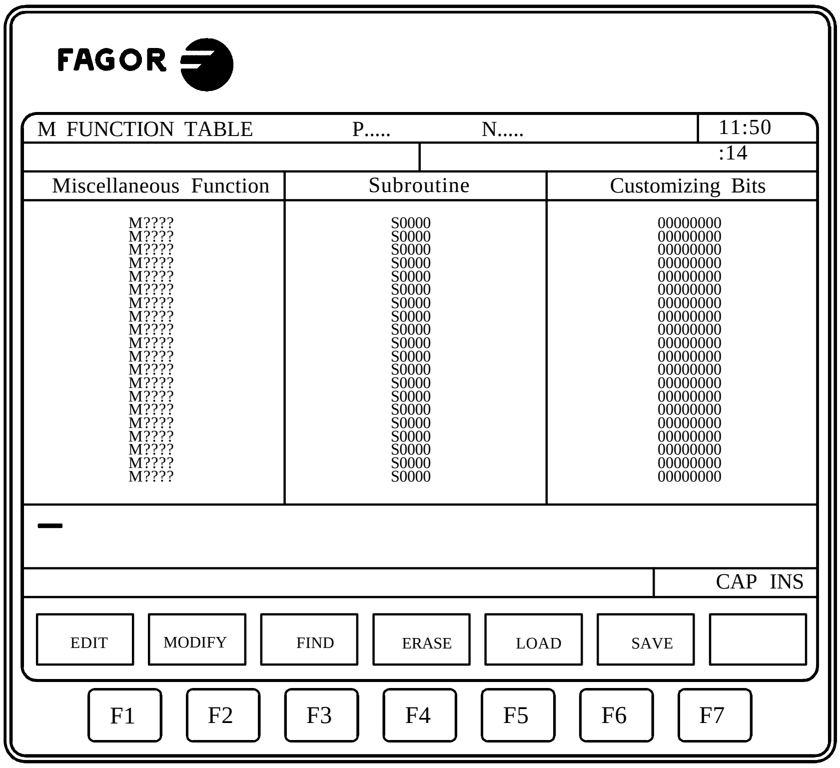
3.- Press [**SHIFT**] [**RESET**].

4.- Re-enter all the data back into the EEPROM.

3.3.6 MISCELLANEOUS (M) FUNCTION TABLE

The number of M functions in this table is determined by the general machine parameter “NMISCFUN”, being possible to define up to 255 M functions.

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



MI035

Each miscellaneous function will be called by its M number.

Possible values: Integers between 0 and 9999. The undefined table elements will be shown as M????.

A subroutine can be associated with each M function and it will be indicated by the letter S.

Possible values: Integers between 0 and 9999. If 0 is assigned to this field, it means that there is no associated subroutine.

The third field consists of 8 customizing bits called bit0 thru bit7:

* * * * * * * *
7) 0)

BIT 0: Indicates whether the CNC must wait or not for the AUX END signal (M done) to consider it executed and go on to the next program block

0 = It waits for the AUX END signal.
1 = It does NOT wait for AUX END.

BIT 1: Indicates whether the M function is executed before or after the movement block where it is programmed.

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 (the CNC reads 20 blocks ahead of their execution) or not.

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.

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).

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.

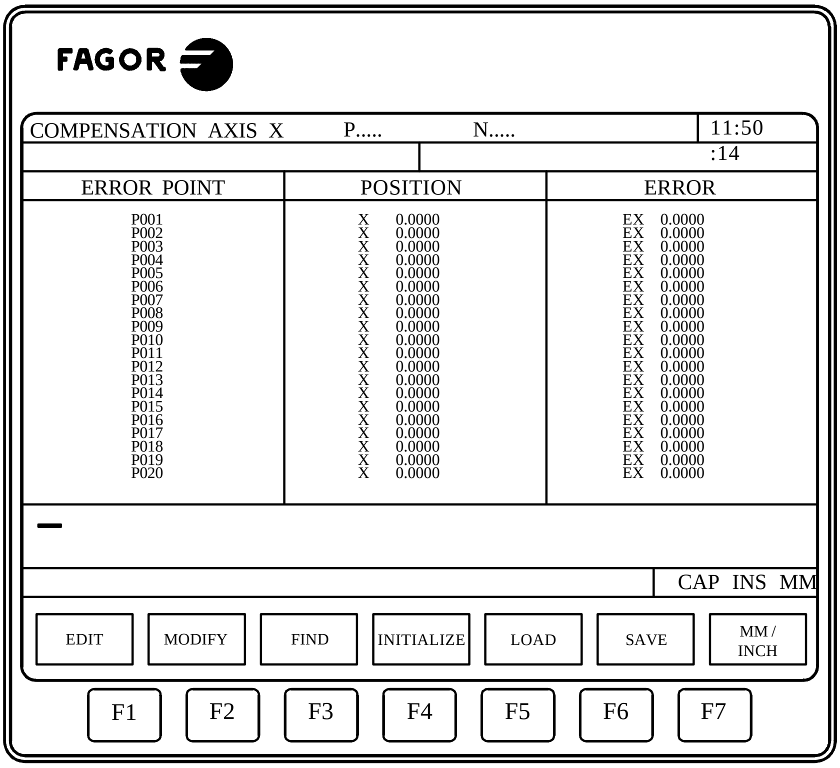
BITs 5 thru 7: Not being used at this time.

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.

3.3.7 LEADSCREW ERROR COMPENSATION TABLE

The CNC will provide a table for each one of the axes having leadscrew compensation. This type of compensation is activated by setting machine parameter “LSCRWCOM” for those axes.

The number of points (up to 255) affected by this compensation must be indicated by axis-machine-parameter “NPOINTS”.



MI036

Each table parameter represents one leadscrew point to be compensated. Each one defines:

The axis position for that Leadscrew point with respect to Machine Reference ZERO.

Possible values: ±99999.9999 millimeters.
±3937.00787 inches.

The leadscrew error in this point.

Possible values: ±99999.9999 millimeters.
±3937.00787 inches.

When defining the leadscrew compensation 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 Machine reference POINT (HOME or marker pulse location) must be assigned an error 0,

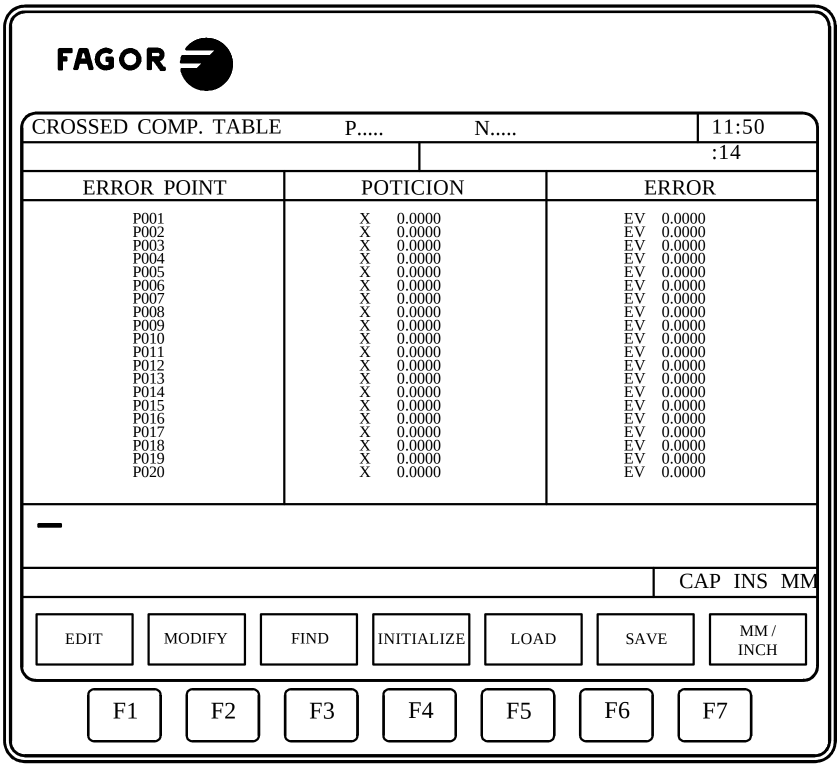
* The error difference between two consecutive points must not be greater than the distance between them (maximum slope= 100%).

3.3.8 CROSS COMPENSATION PARAMETER TABLE

This axis offers cross compensation for the axes. To apply it, define the axis causing the position variations by setting the general machine parameter “MOVAXIS” and define the axis suffering those position variations by setting the general machine parameter “COMPAXIS”.

Both parameters must be defined for this table to be active. The number of compensation points (up to 255) will be defined by setting axis-machine parameter “NPCROSS”.

Each parameter of this table defines:



MI037

The position of the point selected on the “guilty” axis (defined by “MOVAXIS” will be refered to the Machine Reference Zero.

Possible values: ±99999.9999 millimeters.
±3937.00787 inches.

The error generated on the axis defined by “COMPAXIS” when the “guilty” one is positioned in this point.

Possible values: ±99999.9999 millimeters.
±3937.00787 inches.

When defining the different table points, 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 Machine reference POINT (HOME or marker pulse location) must be assigned an error 0.

When both leadscrew and cross compensations are applied on the same axis, the CNC will apply the sum of the two.

4. CONCEPTS

Atention:

It is highly recommended to save the machine parameters in a peripheral device or computer in order to avoid losing them by replacing modules, checksum errors, operator errors, etc.

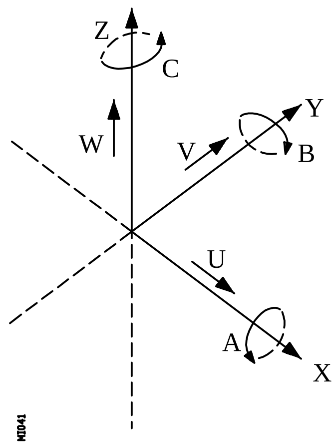
4.1. AXES AND COORDINATE SYSTEMS

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 8050 allows you to use absolute, relative or incremental coordinates throughout the same program.

4.1.1 NOMENCLATURE OF THE AXES

The axes are named according to DIN 66217.



Characteristics of the system of axes :

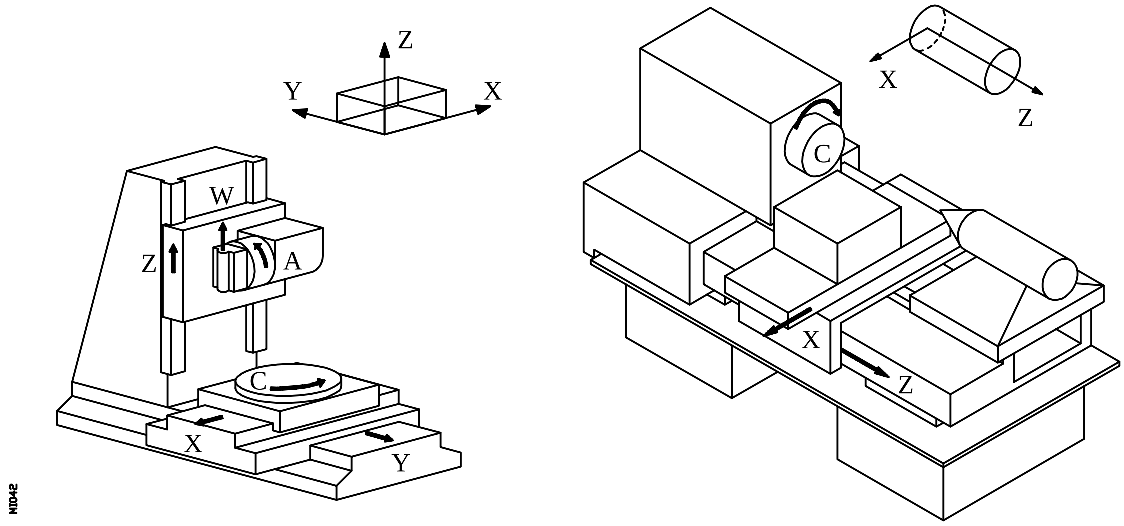
* **X & Y**: main movements on the main work plane of the machine.

* **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 of the X,Y, Z axes.

In the figure (below) an example of the nomenclature of the axes on a milling-profiling machine with a tilted table.



4.1.2 SELECTION OF THE AXES

Of the 9 possible axes which can exist, the CNC FAGOR 8050 allows the manufacturer to select up to 6 of them.

When selecting 6 axes, at least one of them must be either a GANTRY axis or commanded from the PLC.

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 5 axes.

<u>Mill example:</u>

The machine has three regular linear axes: X, Y and Z, one linear U axis controlled by the PLC, an analog Spindle (S) and an electronic handwheel.

"AXIS" general parameter setting:

AXIS1 (P0) = 1	X axis	with feedback connected to X1 and output O1
AXIS2 (P1) = 2	Y axis	with feedback connected to X2 and output O2
AXIS3 (P2) = 3	Z axis	with feedback connected to X3 and output O3
AXIS4 (P3) = 4	U axis	with feedback connected to X4 and output O4
AXIS5 (P4) = 10	Spindle (S)	with feedback connected to X5(1-6) & output O5
AXIS6 (P5) = 0		
AXIS7 (P6) = 11	Handwheel	associated with feedback connector 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).

Axis machine parameter "AXISTYPE" 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

Spindle machine parameter "SPDLTYPE" must be set as follows:

SPDLTYPE (P0) = 0	±10V analog output

Also, their corresponding "DFORMAT" parameter must be set accordingly indicating the way they will appear on the CRT.

Lathe example:

The machine has two regular linear axes: X and Z, a "C" axis, an analog spindle (S) and an auxiliary spindle (live tool).

"AXIS" general machine parameter setting:

AXIS1 (P0) = 1	X axis	feedback connected to X1 and output O1
AXIS2 (P1) = 3	Z axis	feedback connected to X2 and output O2
AXIS3 (P2) = 10	Spindle (S)	feedback connected to X3 and output O3
AXIS4 (P3) = 9	"C" axis	feedback connected to X4 and output O4
AXIS5 (P4) = 13	Aux. spindle	feedback connected to X5(1-6) & output O5
AXIS6 (P5) = 0		
AXIS7 (P6) = 0		
AXIS8 (P7) = 0		

The CNC activates a machine parameter table for each axis (X, Z, C), one for the main spindle (S) and another one for the auxiliary spindle.

Axis machine parameter "AXISTYPE" must be set as follows:

X axis	AXISTYPE (P0) = 0	Regular linear axis
Z axis	AXISTYPE (P0) = 0	Regular linear axis
C axis	AXISTYPE (P0) = 2	Regular rotary axis

Machine parameter "SPDLTYPE" for the main spindle must be set as follows:

SPDLTYPE (P0) = 0 ±10V analog output.

Also, their corresponding "DFORMAT" parameter must be set accordingly indicating the way they will appear on the CRT.

4.1.3 *GANTRY AXES, COUPLED AND SYNCHRONIZED AXES*

<u>GANTRY axes</u>

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".

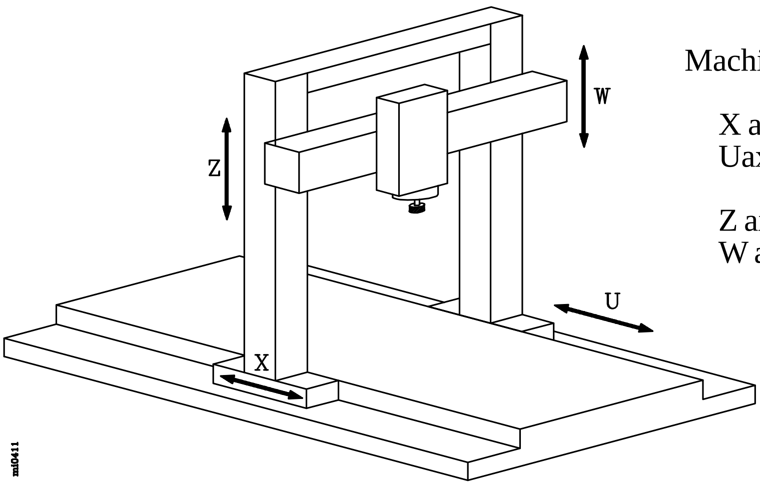
In order to operate this way, it is necessary to have the machine parameter "GANTRY" corresponding to both axes set as follows:

* 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, axis machine parameter "MAXCOUPE" of the slave axis must indicate the maximum allowed difference between the following errors of both axes.

It is possible to have more than one pair of gantry axes.

Example of a bridge type milling machine with two Gantry axes (X-U, Z-W).



Machine parameters:

X axis:	GANTRY=0
Uaxis:	GANTRY=1
Z axis:	GANTRY=0
W axis:	GANTRY=3

COUPLED 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.

Coupled 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 couple 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 whoud be "permanent" slaving).

Synchronized axes:

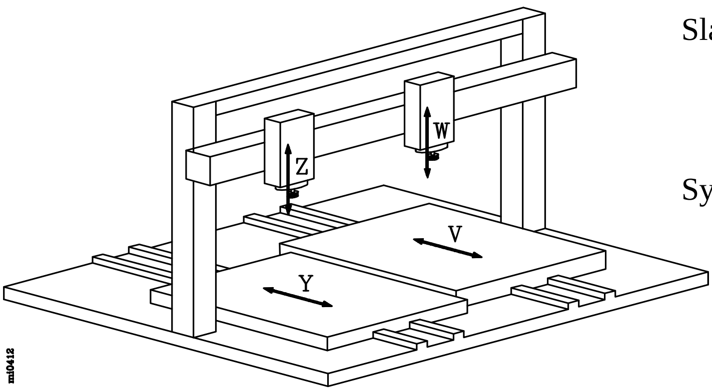
The axes are synchronized by the PLC, by activating the CNC input "SYNCHRO" of the axis to become the slave.

To be able to do this, machine parameter "SYNCHRO" of that axis must be set indicating which axis will be its master (or main axis).

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 machine parameters: "SYNCHRO".

To decouple (unslave) one of the slaved axes, the corresponding "SYNCRO" input of the CNC must be deactivated.

Example of a multi-spindle bridge type milling machine with two pairs of slaved axes (Y-V, Z-W). The two possible slaving methods are shown below:



Slaving (by program):

G77 Y V
G77 Z W

Synchronism (by external signal):

Y axis: SYNCHRO=0
V axis: SYNCHRO=2

Z axis: SYNCHRO=0
W axis: SYNCHRO=3

If the machine has the X, Y, Z, V, W axes, the following signals must be activated (logic state "1") at the PLC:

SYNCHRO4 to slave the V axis to the Y axis.
SYNCHRO5 to slave the W axis to the Z axis.

4.1.4 *RELATIONSHIP BETWEEN THE AXES AND THE JOG KEYS*

The Mill model CNC has 5 pairs of JOG keys and the Lathe model has 4 pairs of keys to jog the axes of the machine.

Mill Model   Lathe Model

The X, Y and Z axes slways use their own denomination; the "C" axis of a lathe model uses the [3+] and [3-] keys and the rest of the axes depend on the chosen name.

The logical order is: X Y Z U V W A B C.

Examples:

A milling machine has the X Y Z U B axes.

[X+] and [X-] keys for the X axis
[Y+] and [Y-] keys for the Y axis
[Z+] and [Z-] keys for the Z axis
[4+] and [4-] keys for the U axis
[5+] and [5-] keys for the B axis.

A laser machine has the X Y A B axes.

[X+] and [X-] keys for the X axis
[Y+] and [Y-] keys for the Y axis
[Z+] and [Z-] keys for the A axis
[4+] and [4-] keys for the B axis

A punch press has the X Y C axes

[X+] and [X-] keys for the X axis
[Y+] and [Y-] keys for the Y axis
[Z+] and [Z-] keys for the C axis

A lathe has the X Z U A axes

[X+] and [X-] keys for the X axis
[Z+] and [Z-] keys for the Z axis
[3+] and [3-] keys for the U axis
[4+] and [4-] keys for the A axis

A lathe has the X Z U C axes

[X+] and [X-] keys for the X axis
[Z+] and [Z-] keys for the Z axis
[4+] and [4-] keys for the U axis
[3+] and [3-] keys for the C axis.

4.2 FEEDBACK SYSTEMS

The various feedback inputs available at the FAGOR 8050 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)	Ballscrew pitch or that of the linear scale 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.

4.2.1 *COUNTING SPEED LIMITATIONS*

Sinewave signals

The maximum counting speed (frequency) for sinewave feedback is 50KHz.

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 scale, the signal pitch is 20 µm. Therefore, with a counting resolution of 1 µm, the maximum feedrate will be:

20 µm/pulse x 50000 pulses/sec = 1m/sec = 60 m/min.

Example 2:

When using a rotary table with a FAGOR sine-wave encoder of 3600 pulses/rev., the maximum feedrate for a 1µm resolution will be:

$$\frac{360 \text{ degrees/turn}}{3{,}600 \text{ pulses/turn}} \times 50{,}000 \text{ pulses/sec.} \times 60 \text{ sec/min.} = 300{,}000 \text{ degrees/min.}$$

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 90º ±20º.

The maximum feedrate for each axis will depend upon the selected resolution and the signal pitch (distance per pulse) in use.

When using FAGOR linear scales, their intrinsic speed limit is 60 m/min (2362 inch/min).

When using FAGOR rotary encoders, their intrinsic output frequency limit is (200Kz).

4.2.2 RESOLUTION

The FAGOR 8050 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 feedback device (scale) being used. When using a FAGOR scale, the signal pitch to be entered here will be either 20 µm or 100 µm.

When dealing with a rotary encoder, it must indicate the number of degrees per encoder turn. For example, if the encoder is mounted onto a motor with a 1/10 gear reduction, this parameter must be set to 360º/10 = 36.

NPULSES (P8) (**N**umber of **PULSES**)

Indicates the number or pulses/rev provided by the rotary encoder. Enter a value of 0 when using linear scales. When using gear ratios, the whole assembly must be taken into account when calculating this value.

SINMAGNI (P10) (**SIN**usoidal **MAGNI**fication)

Indicates the multiplying factor (x1, x4, x20, etc.) that the CNC must apply **only** to **sinewave** feedback signals.

Set this parameter to "**0**" when using **squarewave** feedback signals and the CNC will always apply a x4 multiplying factor.

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**
Squarewave encoder	Leadscrew pitch	# pulses	0
Sinewave encoder	Leadscrew pitch	# pulses	Multiplying factor
Squarewave scale	Scale pitch	0	0
Sinewave scale	Scale pitch	0	Multiplying factor

Example 1: Resolution in "mm" with square-wave encoder

We would like to obtain a 2µm resolution by using a square-wave encoder mounted on 5 mm pitch ballscrew.

Since the CNC applies a x4 multiplying factor to square-wave signals, we would require an encoder which provides the following number of pulses (lines) per turn.

$$\#\text{ of pulses} = \frac{\text{Ballscrew pitch}}{\text{Multiplying Factor x Resolution}} = \frac{5000\ \mu\text{m/rev}}{4 \times 2\ \mu\text{m/pulse}} = 625\ \text{pulses/rev.}$$

INCHES = 0 PITCH= 5 NPULSES = 625 SINMAGNI = 0

Although the CNC accepts a maximum square-wave frequency of 425 KHz, when using FAGOR square-wave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

$$F = \frac{200{,}000\ \text{pulses/sec x 60 sec/min. x 5 mm/rev.}}{625\ \text{pulses/rev.}} = 96\ \text{m/min.}$$

Example 2: Resolution in "mm" with sine-wave encoder

We would like to obtain a 2µm resolution by using a 250-line sine-wave 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.

$$\text{SINMAGNI} = \frac{\text{Ballscrew pitch}}{\text{Number of pulses x Resolution}} = \frac{5000\ \mu\text{m}}{250 \times 2\ \mu\text{m}} = 10$$

INCHES = 0 PITCH= 5 NPULSES = 250 SINMAGNI = 10

The maximum output frequency of FAGOR sine-wave rotary encoders is 200KHz; but the CNC accepts a maximum sine-wave frequency of 50 KHz. Thus, the maximum possible feedrate (F) will be:

$$F = \frac{50{,}000\ \text{pulses/sec x 60 sec/min. x 5 mm/rev.}}{250\ \text{pulses/rev.}} = 60\ \text{m/min.}$$

Example 3: Resolution in "mm" with square-wave linear scale

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

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

Consequently:

INCHES = 0 PITCH= 20 or 100 NPULSES = 0 SINMAGNI = 0

The CNC's maximum square-wave feedback input frequency is 425 KHz which means that the maximum feedrate obtainable with a 20 µm pitch scale is:

Maximum Feedrate = 425,000 pulses/sec x 20µm/pulse = 8500 mm/sec. 510 m/min.

However, when using FAGOR linear scales, the maximum feedrate is limited by their own characteristics to 60 m/min.

Example 4: Resolution in "mm" with sine-wave linear scale

We have a sine-wave linear scale and we would like to obtain 1 µm resolution.

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

$$\text{SINMAGNI} = \frac{\text{Scale pitch}}{\text{Resolution}} = \frac{20\ \mu\text{m}}{1\ \mu\text{m}} = 20$$

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

The CNC's maximum sine-wave feedback input frequency is 50 KHz which means that the maximum feedrate obtainable with a 20 µm pitch scale is:

Maximum Feedrate = 20 µm x 50,000 pulses/sec = 1000 mm/sec = 60 m/min

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

Example 5: Resolution in "inches" with square-wave encoder

Calculate the necessary square-wave 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.).

$$\# \text{ of pulses} = \frac{\text{Ballscrew pitch}}{\text{Multiplying Factor x Resolution}} = \frac{0.25 \text{ inch/rev.}}{4 \times 0.0001 \text{ inch/pulse}} = 625 \text{ pulses/rev.}$$

INCHES = 1 PITCH= 4 NPULSES = 625 SINMAGNI = 0

Although the CNC accepts a maximum square-wave frequency of 425 KHz, when using FAGOR square-wave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

$$F = \frac{100{,}000 \text{ pulses/sec x } 60 \text{ sec/min. x } 0.2 \text{ inch/rev.}}{500 \text{ pulses/rev.}} = 2{,}400 \text{ inches/min.}$$

Example 6: Resolution in "inches" with sine-wave encoder

Calculate the Multiplying Factor (SINMAGNI) required to obtain a **0.0001** inch counting resolution on a 5 pitch ballscrew (5 turns/inch = **0.2** inch/rev.) with a **sine-wave encoder** of **250** pulses/rev.

$$\text{SINMAGNI} = \frac{\text{Ballscrew pitch}}{\text{NPULSES x Resolution}} = \frac{0.2 \text{ inch/rev.}}{250 \text{ pulses/rev x } 0.0001 \text{ inch/pulse}} = 8$$

INCHES = 1 PITCH = 5 NPULSES = 250 SINMAGNI = 8

Although a FAGOR rotary encoder provides a counting frequency of up to 200KHz, the maximum frequency accepted by the CNC for sine-wave encoders is 50KHz; thus, the **maximum possible feedrate (F)** will be:

$$F = \frac{50{,}000 \text{ pulses/sec x } 60 \text{ sec/min. x } 0.2 \text{ inch/rev.}}{250 \text{ pulses/rev.}} = 2.400 \text{ inches/min.}$$

Example 7: Resolution in "degrees" with square-wave encoder

We would like to obtain a 0.0005º resolution by using a square-wave encoder mounted on a x10 reduction gear.

Since the CNC applies a x4 multiplying factor to square-wave signals, we would require an encoder which provides the following number of pulses (lines) per turn.

$$\#\ \text{of Lines} = \frac{\text{Degrees/turn}}{\text{Multiplying Factor x Reduction x Resolution}} = \frac{360}{4 \times 10 \times 0.0005} = 18000\ \text{lines/rev.}$$

INCHES = 0 PITCH= 36 NPULSES = 18000 SINMAGNI = 0

Although the CNC accepts a maximum square-wave frequency of 425 KHz, when using FAGOR square-wave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

$$F = \frac{200{,}000\ \text{pulses/sec}}{18000\ \text{pulses/turn}} = 11.111\ \text{turns/sec.} = 666.666\ \text{rpm}$$

Example 8: Resolution in "degrees" with sine-wave encoder

We would like to obtain a 0.001º resolution by using a 3600-line sine-wave 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.

$$\text{SINMAGNI} = \frac{\text{Degrees/turn}}{\text{Number of pulses x Resolution}} = \frac{360}{3600 \times 0.001} = 100$$

INCHES = 0 PITCH= 3600 NPULSES = 360 SINMAGNI = 100

The maximum output frequency of FAGOR sine-wave rotary encoders is 200KHz; but the CNC accepts a maximum sine-wave frequency of 50 KHz. Thus, the maximum possible feedrate (F) will be:

$$\text{Maximum Feedrate} = \frac{50{,}000\ \text{pulses/sec}}{3600\ \text{pulses/turn}} = 13.8888\ \text{turns/sec.} = 833.33\ \text{rpm}$$

4.3 AXIS SETTING

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.

Servo drive loop setting.

1.- Verify that the power output of the drives is OFF. Set all axis machine parameters "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.

CNC loop setting.

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 axis machine parameter "LOOPCHG" (P26) must be changed.

If the axis does not run away, but it does not move in the desired direction, Change both the "AXISCHG" (P13) and "LOOPCHG" (P26) parameters (for the axes).

4.3.1 SERVO DRIVE SETTING

<u>Offset adjustment</u>

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 **<u>at the drive</u>** (NOT AT THE CNC) until a "0" following error is obtained.

<u>Maximum feedrate adjustment</u>

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

Set each axis machine parameter "MAXVOLT" (P37) = 9500 so the CNC outputs a maximum analog voltage of 9.5 V.

The maximum axis feedrate, axis machine parameter "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 5mm/rev. = 15,000 mm/minute

In order to adjust the drive, machine parameter G00FEED" (P38) should be set to the same value as machine parameter "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
     (REP N10, N20)
```

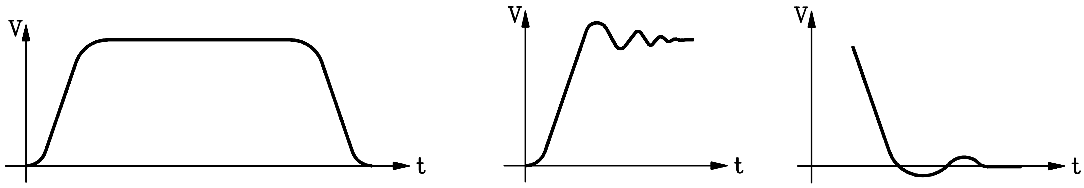
While the axis is moving back and forth, measure the analog voltage provided by the CNC to the drive and adjust the feed potentiometer **<u>at the drive</u>** (NOT AT THE CNC) until reaching 9.5 V.

It may happen that the axis does not have enough time to reach maximum speed in such short distance or it is too fast for your voltmeter. In that case, program a proportional F value (for example F7500) and, consequently, the analog voltage to adjust it for will also be proportional (in the example: 4.75 V)

4.3.2 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 mm (0.03937 inch) of following error.

It is set by axis machine parameter "PROGAIN" (P23)

Feed Forward Gain

It sets the percentage of analog output dependent of the programmed feedrate.

It must be set **only** when operating with acceleration / deceleration (axis machine parameter "ACCTIME" (P18)).

It is set by axis machine parameter "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).

It must be used only when operating with acc. /dec. (axis machine parameter "ACCTIME" (P18)).

It is set by axis machine parameters "DERGAIN" (P24) and "ACFGAIN" (P46).

With "ACFGAIN=No" it applies Derivative Gain.
With "ACFGAIN=Yes" it applies AC-Forward Gain.

4.3.3 *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

Axis machine parameter "PROGAIN" (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 1 in metric:

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 1m/min. (Gain of 1 in metric)

Axis machine parameter "G00FEED" (P38) must be set to 15,000 (15 m/min).

Axis machine parameter "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.

Axis machine parameter "MAXFEED" (P42) must be set to 3000 (3 m/min).

Analog output corresponding to F 1000 mm/min:

$$\text{Analog} = \frac{9.5\text{ V.}}{\text{"G00FEED"}} \times F$$

$$\text{Analog} = \frac{9.5\text{ V.}}{15{,}000\text{ mm/min.}} \times 1000\text{ mm/min.} = 0.633\text{V} = 633\text{mV}$$

Therefore, "PROGAIN" (P23) = 633

Example 1 in inches::

The maximum feedrate for a particular axis (rapid traverse G00) is 500 inch/min, but we would like to limit its maximum programmable machining feedrate (F) to 150 inch/min with a gain of 0.001 inch lag at a feedrate of 1 inch/min. (Gain of 1 in inches).

Axis machine parameter "G00FEED" (P38) must be set to 500 inches.

Axis machine parameter "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.

Axis machine parameter "MAXFEED" (P42) must be set to 150 inches/min. (although it is not being used in this calculation).

Analog output corresponding to 0.3937 inch. (1 mm) of following error:

$$\text{Analog} = \frac{9.5\text{ V.}}{500\text{ inch/min.}} \times 39.37\text{ inch/min.} = 748\text{mV}$$

Therefore, "PROGAIN" (P23) = 748

When setting the proportional gain, the following considerations must be taken into account:

* The maximum amount of following error allowed by the CNC for the axis is the value indicated by axis machine parameter "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).

Atention:

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.

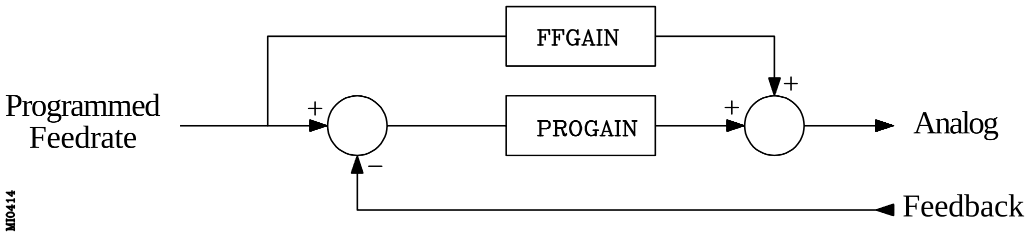
The more identical their following errors, the more "round" the programmed circles will turn out.

4.3.4 *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 "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 machine parameter "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 0,945V then: P37=9450.

Next, set parameter "FFGAIN" (P25) to the desired value.

As an example, the following values may be used:

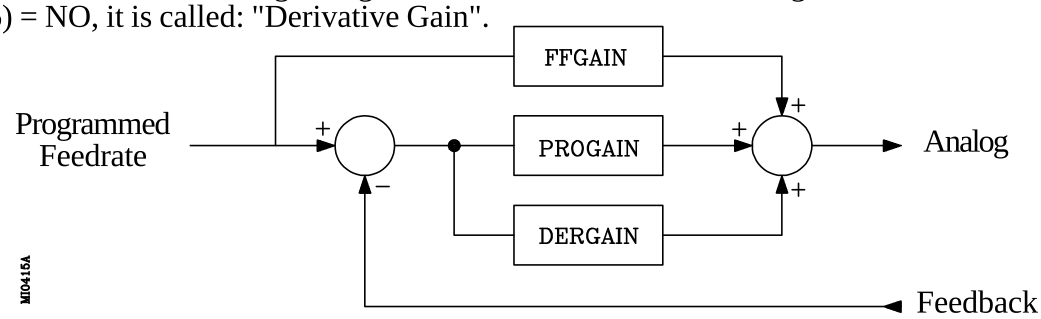
For slow machining.. between 40 and 60%
For regular feed machining .... between 60 and 80%
For fast machining (laser, plasma) .... between 80 and 100%

4.3.5 DERIVATIVE / AC-FORWARD GAIN SETTING

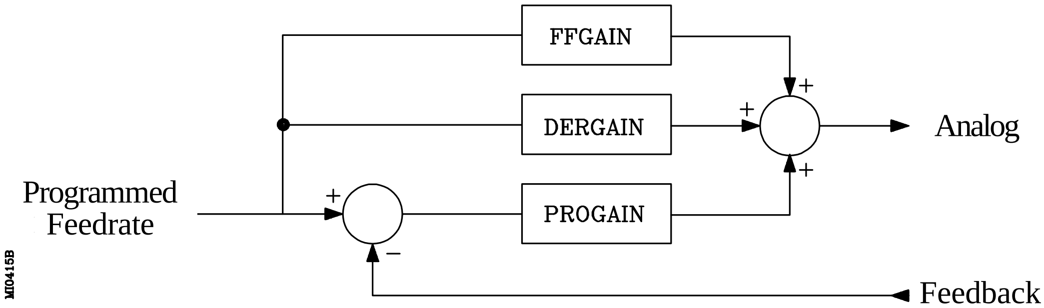
With the Derivative gain, it is possible to reduce the following error during the acc./dec. stages.

Its value is determined by axis machine parameter "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.

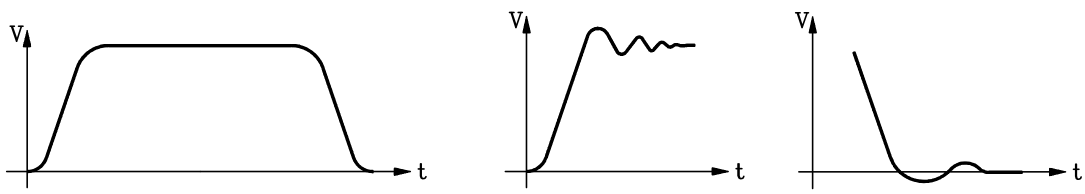
This gain is only to be used when operating with acceleration / deceleration.

A practical value between 2 to three 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).



4.3.6 LEADSCREW BACKLASH COMPENSATION

On this CNC, the leadscrew backlash may be compensated for when reversing the direction of movement.

The amount of backlash compensation to be applied is set by axis machine parameter "BACKLASH" (P14).

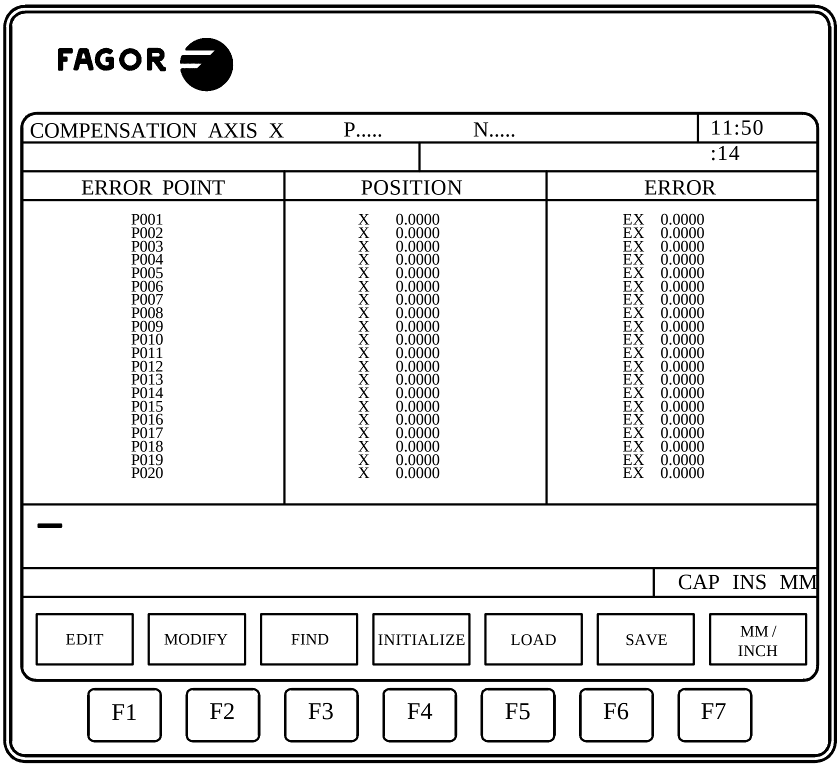
Sometimes, an additional analog pulse may also be needed to recover the possible backlash when reversing the axis movement.

Axis machine parameter "BAKANOUT" (P29) sets the value of this additional analog voltage pulse and "BACKTIME" (P30) sets its duration.

4.3.7 LEADSCREW ERROR COMPENSATION

The CNC provides a table for each one of the axes requiring leadscrew compensation. This type of compensation is activated by setting machine parameter “LSCRWCOM P15)=ON” for those axes.

The number of points (up to 255) affected by this compensation must be indicated by axis-machine-parameter “NPOINTS (P16)”.



MI036

Each table parameter represents one leadscrew point to be compensated. Each one defines:

The axis position for that Leadscrew point with respect to Machine Reference ZERO.

Possible values: ±99999.9999 millimeters.
±3937.00787 inches.

The leadscrew error in this point.

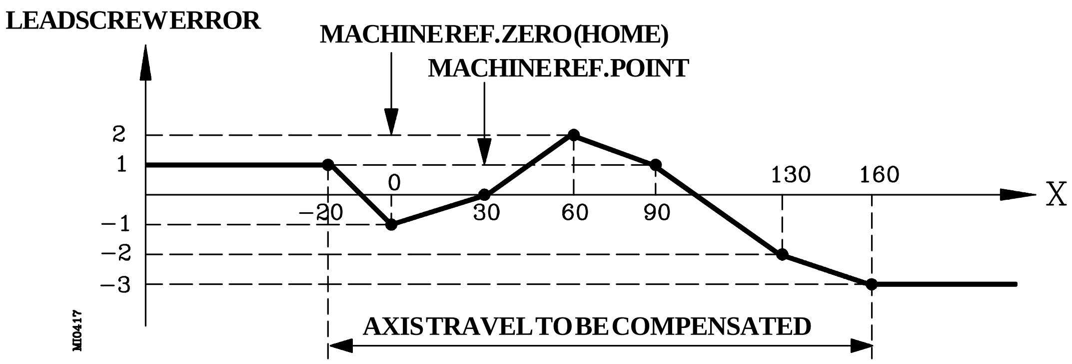
Possible values: ±99999.9999 millimeters.
±3937.00787 inches.

When defining the leadscrew compensation 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 Machine reference POINT (HOME or marker pulse location) must be assigned an error 0,
* The error difference between two consecutive points must not be greater than the distance between them (maximum slope= 100%).

<u>Programming example:</u>

The X axis ballscrew must be compensated for between X-20 and X160 according to the leadscrew error graph below:



Set axis machine parameters "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:

P001	X	-20,000	EX	0,001
P002	X	0,000	EX	-0,001
P003	X	30,000	EX	0,000
P004	X	60,000	EX	0,002
P005	X	90,000	EX	0,001
P006	X	130,000	EX	-0,002
P007	X	160,000	EX	-0,003

4.4 REFERENCE SYSTEMS

4.4.1 REFERENCE POINTS

A CNC machine needs the following origin and reference points defined :

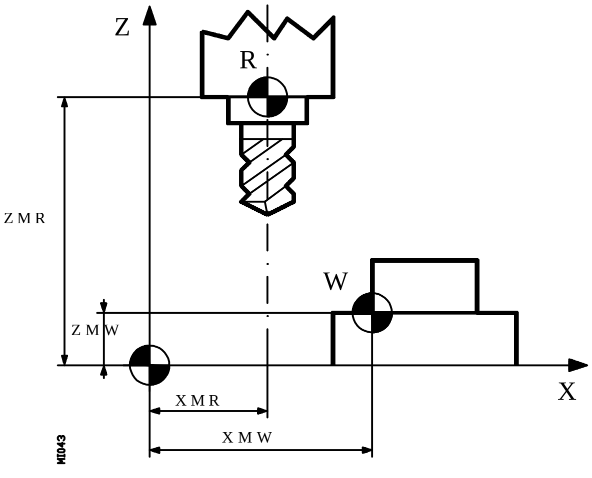
* **Machine zero** or point of origin of the machine. This is set by the manufacturer as the origin of the system of coordinates of the machine.

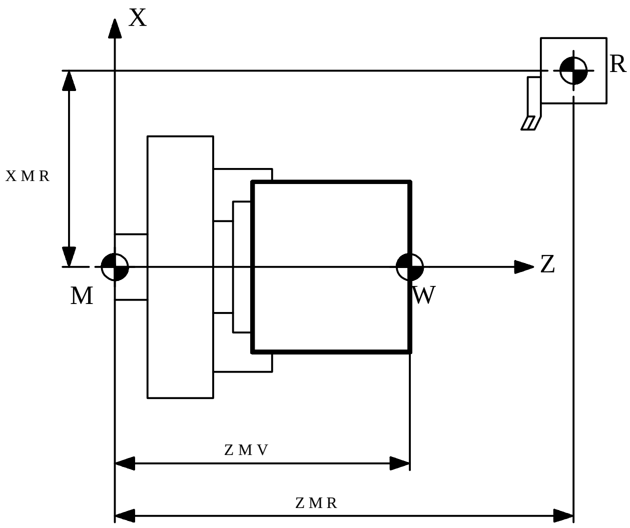
* **Part zero** or point of origin of the part. This is the point of origin which is set for programming the measurements of the part. it can be freely selected by the programmer, and its zero machine reference can be set by the zero offset.

* **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, Io), this point is only used when leadscrew error compensation must be applied onto the axis. The error amount at this reference point must be "0".

  - 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
ZMR,YMR,ZMR, etc.	Coordinates of machine reference point (“REFVALUE”)

4.4.2 *MACHINE REFERENCE SEARCH*

The FAGOR 8050 CNC allows to perform the machine reference search one axis at a time or several axes at the same time while in JOG mode or by program.

On axes **with no** semi-absolute (coded Io) feedback system.

The CNC will move all selected axes which have a home switch (machine parameter “DECINPUT (P31)” for each axis), and in the direction indicated by machine parameter “REFDIREC (P33)” for each axis.

This movement will be carried out at the feedrate established by machine parameter “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 machine parameter “REFEED2 (P35)” for each axis until the marker pulse is found; thus ending the home search.

On axes **with** semi-absolute (coded Io) feedback system.

Home switches are no longer necessary since the axes may be homed anywhere along its travel. However, axis machine parameter REFVALUE (P36) must be set when operating with leadscrew error compensation. The amount of leadscrew error to be assigned to this point is "0".

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 axis machine parameter "REFDIREC (P33" at the feedrate set by machine parameter “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.

When this search (with or without 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 machine parameter “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.

Atention:

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 machine parameter "REFSHIFT" 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 ("REFSHIFT (P47)" value) after finding the new marker pulse and it will position at the same physical home location as before. This additional movement will be carried out at the feedrate established by machine parameter "REFEED2 (P35)".

4.4.2.1 MACHINE REFERENCE SEARCH ON GANTRY AXES

The machine reference search on Gantry axes can be carried out in JOG mode or by program and it will be done as follows:

On axes **with no** semi-absolute (coded Io) feedback system.

The CNC starts the movements of both axes in the direction indicated by machine parameter “REFDIREC (P33)” corresponding to the main axis.

These movements will be performed at the feedrate indicated by machine parameter “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 machine parameter “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.

On axes **with** semi-absolute (coded Io) feedback system.

The CNC starts moving both axes in the direction indicated by machine parameter “REFDIREC (P33)” corresponding to the main axis at the feedrate indicated by machine parameter “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 machine parameters “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 machine parameter “**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.

Atention:

If the machine parameter "**REFDIREC (P33)**" for the main axis has been set for a positive direction, the machine parameter "**REFVALUE (P36)**" for the slaved axis must be set to a value lower than that assigned to the main axis.

Also, if the machine parameter "**REFDIREC (P33)**" for the main axis has been set for a negative direction, the machine parameter "**REFVALUE (P36)**" for the slaved axis must be set to a value greater than that assigned to the main axis. They cannot have the same value.

When encoders are used for feedback, the difference between the values assigned to the "**REFVALUE**" parameters 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.

4.4.3 SETTING ON SYSTEMS WITHOUT SEMI-ABSOLUTE FEEDBACK

4.4.3.1 MACHINE REFERENCE SETTING

The reference point must be adjusted on one axis at a time. The following procedure is recommended:

* Indicate in axis machine parameter "REFPULSE" (P32) the type of marker pulse (Io) used by the feedback device.

* Also, indicate in axis machine parameter "REFDIREC" (P33) the homing direction to look for that marker pulse.

* Set the axis machine parameters defining the home switch approaching feedrate "REFEED1" (P34), and the marker pulse searching feedrate after the home switch has been found "REFEED2" (P35).

* The machine reference point will be set to "0". Axis machine parameter "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 axis machine parameter "REFVALUE" (P36), which defines the coordinate corresponding to the Machine Reference Point (physical location of the marker pulse).

  "REFVALUE" P36 = Machine Coordinate of the 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" P36 = 230 - (-123.5) = 353.5 mm.

* After allocating this new value to the machine parameter, press SHIFT + RESET or turn the CNC off and back on in order for this value to be assumed by the CNC.

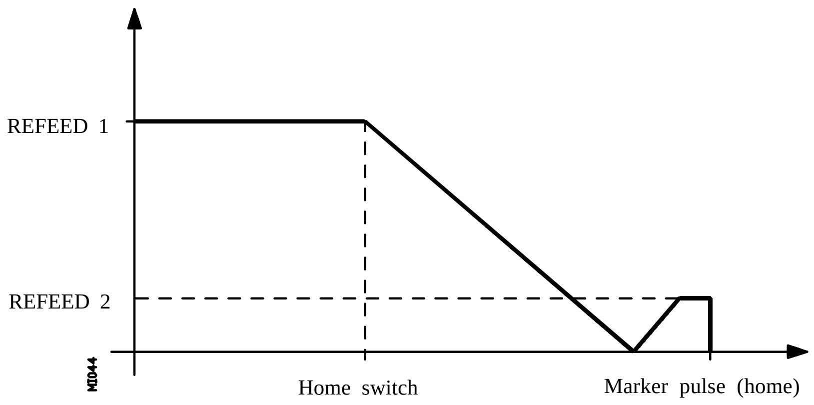
* The axis must be homed again in order for it to assume its right reference values.

4.4.3.2 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 outside the software travel limits (machine parameters "LIMIT+ (P5)" or "LIMIT- (P6)", it is necessary to jog the axis into the work zone so the home search is performed in the proper direction.

* Care must be taken when placing the home switch and when setting feedrates "REFEED1 (P34)" and "REFEED2 (P35)".



The home switch will be installed so the marker pulse 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 selected axis does not have a machine reference (home) switch (axis machine parameter "DECINPUT (P31)" = NO), the CNC will move the axis at the feedrate set by axis machine parameter "REFEED2 (P35)" until the first marker pulse from the current position is found, thus ending the home search.

* FAGOR linear transducers (scales) provide a negative marker (reference) pulse Io every 50mm (about 2 inches) and the FAGOR rotary encoders provide one positive reference pulse per revolution.

* Do not mistake the type of reference pulse being used (positive or negative) with the type of active flank (up or down) to be used when setting axis machine parameter "REFPULSE (P32)" since an up (positive) flank may be used with a negative type marker pulse.

4.4.4 SETTING ON SYSTEMS WITH SEMI-ABSOLUTE FEEDBACK

4.4.4.1 SCALE OFFSET SETTING

The offset of the scale must be adjusted on one axis at a time, preferably, following this procedure:

1.- Set axis machine parameters:

"REFDIREC" (P33) Homing direction.

"REFEED2" (P35) Homing feedrate.

2.- Verify that the value allocated to "REFPULSE" (P32) (type of marker pulse of the feedback system) is correct.

To do this, set "DECINPUT (P31) = NO" and "I0TYPE (P52) = 0". Then, home the axis.

If assumed immediately, change "REFPULSE" (P32) and check again.

3.- Set axis machine parameter "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 scale.

5.- Perform several consecutive home searches and observe the CNC display during the whole process.

The counting must be continuous. If it is not, if jerky, set axis machine parameter "I0TYPE (P52) = 1" and repeat steps 3 and 4.

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 scale.

7.- The value to be assigned to axis machine parameter "ABSOFF" (P53) must be calculated with the following formula:

"ABSOFF" (P53) = CNC reading at this point - Machine coordinate of this point

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 scale offset will be:

"ABSOFF" (P53) = -423,5 - 230 = -653.5 mm.

8.- After allocating this new value to the machine parameter, 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.

4.4.4.2 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 semi-absolute linear scales (with coded Io), 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.

* Semi-absolute FAGOR linear transducers have **negative** coded marker pulse (Io).

* Do not mistake the type of pulse provided by the feedback system with the value to be assigned to axis machine parameter "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.

4.4.5 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 position value (coordinate) of this point to the corresponding parameter for positive software limit, "LIMIT+" (P5).

* Repeat these steps in the negative direction assigning the resulting coordinate to axis machine parameter "LIMIT-" (P6).

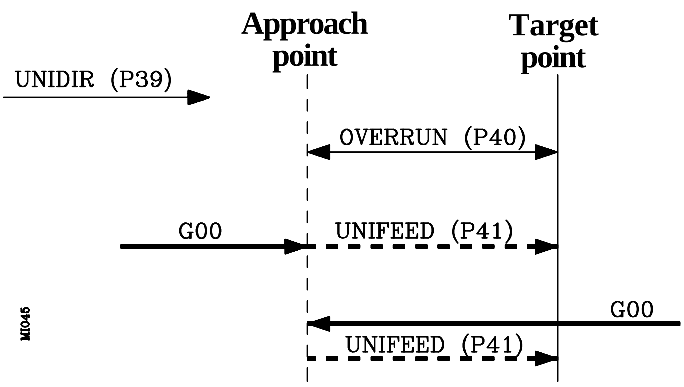
* Once both travel limits have been set for all the axes, press SHIFT + RESET or turn the CNC OFF and back ON in order for these new values to be assumed by the CNC.

4.5 *UNIDIRECTIONAL APPROACH*

The FAGOR 8050 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	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	Indicates the feedrate to be used from the approach point to the programmed point.

The CNC will calculate the approach point based on the programmed destination point (end point) and the machine parameters "UNIDIR" and "OVERRUN".



The positioning will be carried out in two stages:

* Rapid positioning (G00) up to the calculated approach point. If the axis is moving in the direction opposite to that indicated by "UNIDIR", it will overshoot the programmed point.

* Positioning at feedrate UNIFEED from this point to the programmed point.

4.6 *TRANSFERRING AUXILIARY M, S, T FUNCTIONS*

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 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 (see chapter 3 in this manual) 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.

Example 1:

Execution of a motion block containing 7 M functions 4 of which are executed before the axes move (M51, M52, M53, M54) and 3 afterwards (M61, M62, M63).

1.- It sends out to the PLC the 4 M functions programmed to be executed before the move

It sets logic outputs “MBCD1=51”, “MBCD2=52” “MBCD3=53” “MBCD4=54” and it activates the general logic output "MSTROBE to “tell” the PLC to go ahead with their execution.

Should any of them need the AUXEND activated, the CNC will “wait” for this signal to be activated before going on to executing the rest of the block.

If none of them need the AUXEND signal activated, the CNC will maintain the “MSTROBE” signal activated for a period of time set by the general machine parameter “MINAENDW (P30)”.

2.- The programmed axiss 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)”.

Example 2:

1.- It sends out to the PLC the 4 M functions programmed to be executed before the move

It sets logic outputs “MBCD1=51”, “MBCD2=52” “MBCD3=53” “MBCD4=54” and it activates the general logic output "MSTROBE to “tell” the PLC to go ahead with their execution.

Should any of them need the AUXEND activated, the CNC will “wait” for this signal to be activated before going on to executing the rest of the block.

If none of them need the AUXEND signal activated, the CNC will maintain the “MSTROBE” signal activated for a period of time set by the general machine parameter “MINAENDW (P30)”.

2.- It sends out to the PLC the 3 M functions programmed to be executed after the move.

It sets logic outputs “MBCD1=61”, “MBCD2=62”, “MBCD3=63” and it activates the general logic output "MSTROBE to “tell” the PLC to go ahead with their execution.

Should any of them need the AUXEND activated, the CNC will “wait” for this signal to be activated before going on to executing the rest of the block.

If none of them need the AUXEND signal activated, the CNC will maintain the “MSTROBE” signal activated for a period of time set by the general machine parameter “MINAENDW (P30)”.

S function:

The CNC transfers the "S function" out to the PLC only when using the BCD-coded "S" output. Spindle machine parameter "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 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 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 using a machining center with non-random tool magazine. General machine parameters "TOFFM06 (P28) = YES" and "RANDOMTC (P25) = NO".

* When using a random tool magazine, general machine parameter "RANDOMTC (P25) = YES" and a special tool change takes place. See chapter 6 of the Operating Manual: tool table, status.

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.

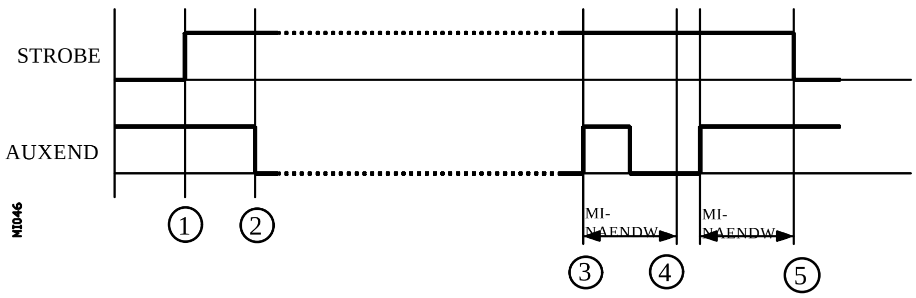
This indication will be made by means of the variable “T2BCD” (R559) and by activating the general logic output “T2STROBE” to tell the PLC that it must execute this. The CNC will wait for the general input AUXEND to be activated to consider the execution completed.

Atention:

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.

4.6.1 TRANSFERRING M, S, T USING THE AUXEND SIGNAL

1.- Once the block has been analyzed and after sending the corresponding values in the "MBCD1-7", "SBCD", "TBCD" and "T2BCD" variables, the CNC will tell the PLC by means of the general logic outputs "MSTROBE", "SSTROBE", "TSTROBE" and "T2STROBE" that the required auxiliary functions must be executed.



2.- When the PLC detects the activation of one of the STROBE signals, it must deactivate the general CNC logic input "AUXEND" to tell the CNC that the execution of the corresponding function or functions has begun.

3.- The PLC will execute all the auxiliary functions required, it being necessary to analyze the general CNC logic outputs:

"MBCD1 thru 7" 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.

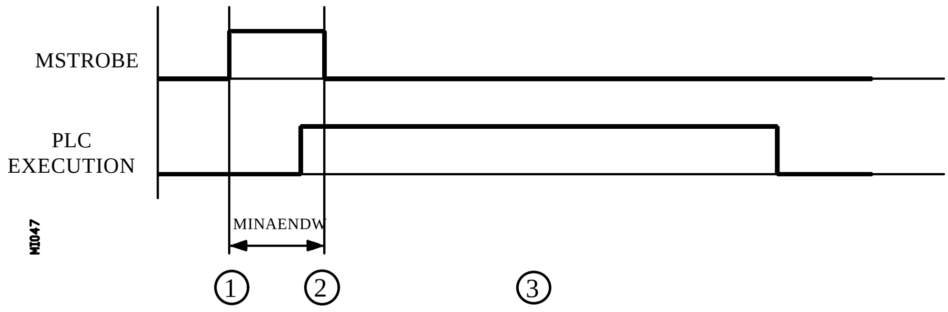
4.- Once the general input "AUXEND" is active, the CNC will require that this signal be kept active for a period of time greater than that defined by means of the general machine parameter "MINAENDW (P30)".

In this way erroneous interpretations of this signal by the CNC are avoided in the case of malfunctions caused by an incorrect logic in the PLC program.

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.

4.6.2 *TRANSFERRING THE MISCELLANEOUS (AUXILIARY) 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 general machine parameter “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”.

4.7 SPINDLE

The mill model 8050 M CNC has one single spindle (main spindle)

The lathe model may have 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 by means of functions G28 and G29 (see programming manual).

Next, the steps to be followed when using two spindles are described.

Parameter setting

Set general machine parameters "AXIS1" thru "AXIS8" to the desired values. 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

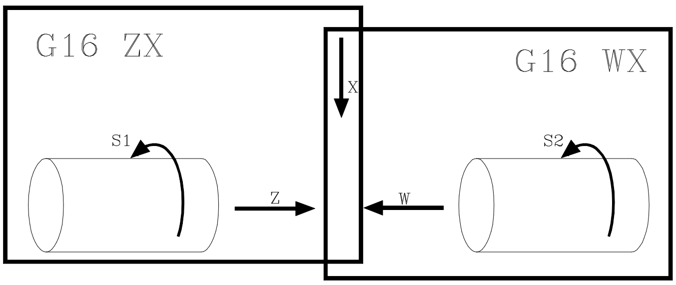
To select the second spindle, execute function G28.
From then on, All the keyboard actions and spindle related functions affect the second spindle.
The main spindle remains in its previous status.
Example: S1500 M4 Second spindle turns counter-clockwise at 1500 rpm.
The main spindle keeps turning at 1000 rpm

To select the main spindle again, execute function G29.
From then on, all the keyboard actions and spindle related functions affect the main spindle.
The second spindle stays in its previous status.
Example: S2000 The main spindle keeps turning clockwise but at 2000 rpm.
The second spindle keeps turning at 1500 rpm.

Work plane Selection

To select the work plane, use function G16 (see programming manual)

Example:



Machining canned cycles

When working in a plane other than 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 this, set general machine parameter "PLACOMP" (P78) to "1" (see chapter on "machine parameters" in this manual).

When setting general machine parameter "PLACOMP=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

4.7.1 SPINDLE TYPES

The setting of spindle machine parameter "SPDLTYPE" (P0) allows the following possibilities:

"SPDLTYPE" (P0) = 0	Analog spindle output..
"SPDLTYPE" (P0) = 1	2-digit BCD-coded spindle output.
"SPDLTYPE" (P0) = 2	8-digit BCD-coded spindle output.

When using BCD output (2 or 8 digits), the spindle will operate in open loop and it may 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 spindle machine parameter "NPULSES" (P13) must be set to a value other than "0".

* Controlled via PLC. With this feature, it is possible to have the PLC control the spindle for a while.

  A typical application of this feature is to control the oscillation of the spindle during a range (gear) change.

Regardless of the type of spindle output being used, the CNC admits up to 4 spindle speed ranges.

The spindle speed range change may be made either manually or automatically by the CNC.

To change spindle ranges, functions M41, M42, M43 and M44 are used to let the PLC know which one is to be selected.

4.7.2 *SPINDLE SPEED (S) CONTROL*

BCD output

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.

Machine parameter "**SPDLTYPE (P0)**" for the spindle must be set to indicate whether a 2-bit or a 8-bit BCD code will be used to indicate spindle speed.

"**SPDLTYPE (P0)**" = 1   2-digit BCD coded spindle output (S)
"**SPDLTYPE (P0)**" = 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:

Programmed S	S BCD	Programme S	S BCD	Programmed S	S BCD	Programmed S	S BCD
0	S 00	25-27	S 48	200-223	S 66	1600-1799	S 84
1	S 20	28-31	S 49	224-249	S 67	1800-1999	S 85
2	S 26	32-35	S 50	250-279	S 68	2000-2239	S 86
3	S 29	36-39	S 51	280-314	S 69	2240-2499	S 87
4	S 32	40-44	S 52	315-354	S 70	2500-2799	S 88
5	S 34	45-49	S 53	355-399	S 71	2800-3149	S 89
6	S 35	50-55	S 54	400-449	S 72	3150-3549	S 90
7	S 36	56-62	S 55	450-499	S 73	3550-3999	S 91
8	S 38	63-70	S 56	500-559	S 74	4000-4499	S 92
9	S 39	71-79	S 57	560-629	S 75	4500-4999	S 93
10-11	S 40	80-89	S 58	630-709	S 76	5000-5599	S 94
12	S 41	90-99	S 59	710-799	S 77	5600-6299	S 95
13	S 42	100-111	S 60	800-899	S 78	6300-7099	S 96
14-15	S 43	112-124	S 61	900-999	S 79	7100-7999	S 97
16-17	S 44	125-139	S 62	1000-1119	S 80	8000-8999	S 98
18-19	S 45	140-159	S 63	1120-1249	S 81	9000-9999	S 99
20-22	S 46	160-179	S 64	1250-1399	S 82		
23-24	S 47	180-199	S 65	1400-1599	S 83		

If a value of over 9999 is programmed, the CNC will "tell" the PLC the spindle speed corresponding to value 9999.

If an 8-digit BCD-coded output 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.

S12345.678	0001	0010	0011	0100	0101	0110	0111	1000

LSB

Analog voltage

In order for the CNC to provide an analog output to control the spindle speed, it is necessary to set machine parameter "SPDLTYPE (P0) = 0".

The CNC will generate the analog output (within $\pm$10V.) corresponding to the programmed rotation speed or a unipolar analog output voltage if the machine parameters for the spindle "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 this, follows 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 spindle analog voltage.

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.

A typical application of this feature is the control of the spindle oscillation during spindle speed range (gear) change.

4.7.3 *SPINDLE SPEED RANGE 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 speed ranges which are determined by machine parameters for the spindle: “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 range and the one assigned to “MAXGEAR4 (P5)” will be the one corresponding to the highest range.

When not using all 4 ranges, assign the values starting from “MAXGEAR1 (P2)” up and the highest speed value to the unused ranges.

The auxiliary functions M41, M42, M43 and M44 are used to “tell” the PLC that spindle range 1, 2, 3 or 4 must be selected.

In turn, the PLC must “tell” the CNC the speed range 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 range, before selecting a new "S" one must:

1.- Analyze whether the new "S" involves a range change.

2.- If it does, execute the M function corresponding to the new range (M41 thru M44) in order for the PLC to select it.

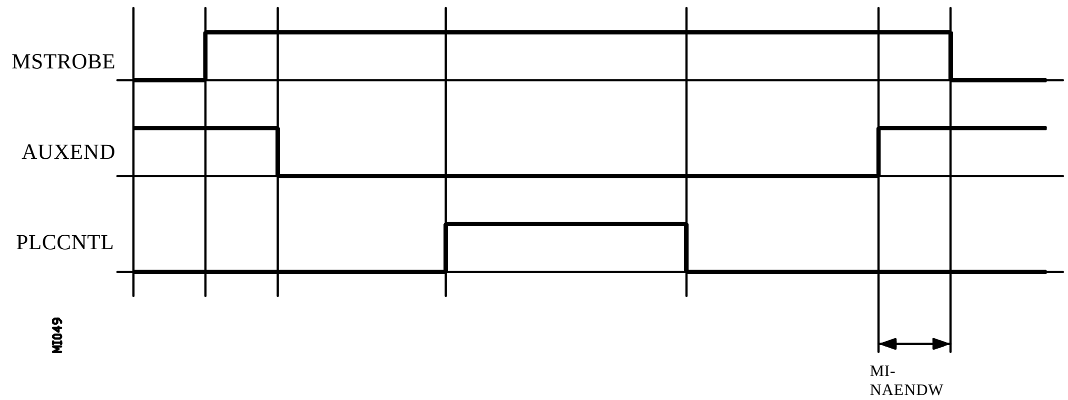
3.- Wait for the PLC to select the new range. Check spindle logic inputs "GEAR1" (M5458), "GEAR2" (M5459), "GEAR3" (M5460) and "GEAR4" (M5461).

4.- Select the new speed "S".

To have the CNC perform all these operations automatically, set spindle machine parameter “AUTOGEAR P6) =YES” to indicate that the range change is to be generated by the CNC.

When selecting an automatic range change, the CNC will inform the PLC of the new range (M41 thru M44; but it will not execute any subroutine associated with them.

4.7.3.1 AUTOMATIC SPINDLE RANGE CHANGE CONTROLLED BY PLC



When the CNC detects a range change, it sends out to the PLC the corresponding M code (M41 thru M44) via one of the logic outputs "MBCD1-7" (R550 thru R556).

It also activates general logic output "MSTROBE" (M5532) to "tell" the PLC to go ahead with the execution.

The PLC deactivates CNC general logic input "AUXEND" (M5016) to indicate to the CNC that it began processing the "M" function.

When requiring spindle oscillation control during a range 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 range 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.

4.7.3.2 AUTOMATIC SPINDLE RANGE CHANGE WHEN WORKING WITH M19

Every time M19 is programmed, it is recommended that the corresponding spindle range be selected.

If no range is already selected, the CNC proceeds as follows:

- It converts the speed indicated in degrees per minute at machine parameter "REFEED1" (P34) into rpm.

- It selects the speed range corresponding to those rpm.

The spindle range cannot be changed when operating in M19. It must be selected beforehand.

4.7.4 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). Spindle machine parameter "SPDLTYPE (P0) = 0".
* An encoder must be mounted onto the spindle. Spindle machine parameter "NPULSES" (P13) must indicated 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±5.5" must be executed.

The S±5.5 code indicates the spindle position, in degrees, from the spindle reference point (marker pulse).

When switching form open to closed loop, the CNC behaves as follows:

* If the spindle has a home switch, it performs a home-switch search at the turning speed set by spindle machine parameter "REFEED1" (P34).

  It then searches for actual marker pulse (Io) of the spindle encoder at the turning speed set by spindle machine parameter "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 spindle machine parameter "REFEED2" (P35).

  And, then, it positions the spindle at the programmed S±5.5 point.

4.7.4.1 CALCULATING SPINDLE RESOLUTION

The CNC assumes that one encoder revolution represents 360º. Therefore, the feedback (counting) resolution depends on the number of lines of the spindle encoder.

Resolution = 360° / (4 x number of encoder lines per revolution)

Hence, to obtain a resolution of 0.001º, a 90,000 line encoder is required and a 180,000 line encoder to obtain a resolution of 0.0005º.

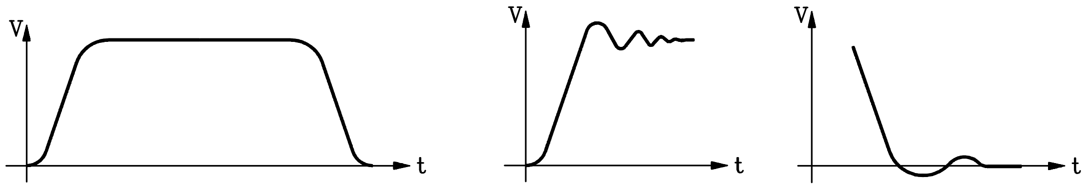
Spindle machine parameter "NPULSES" (P13) must indicate the number of square pulses supplied by the spindle encoder.

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".

4.7.4.2 *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:



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 1° of following error.

It is set by spindle machine parameter "PROGAIN" (P23)

Feed Forward Gain

It sets the percentage of analog output dependent of the programmed feedrate.

It must be set **only** when operating with acceleration / deceleration (spindle machine parameter "ACCTIME" (P18)).

It is set by spindle machine parameter "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).

It must be used only when operating with acc. /dec. (spindle machine parameter "ACCTIME" (P18)).

It is set by spindle machine parameters "DERGAIN" (P24) and "ACFGAIN" (P42).

With "ACFGAIN=No" it applies Derivative Gain.
With "ACFGAIN=Yes" it applies AC-Forward Gain.

4.7.4.3 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

Spindle machine parameter "PROGAIN" (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 1º of following error.

Example:

The maximum speed for the 1st range (rapid traverse G00) is 500 rpm and we would like to obtain 1º at a speed of 1000 º/min. (2.778 rpm)

Drive analog: 9.5V for 500 rpm

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

$$\text{Analog} = \frac{9.5\text{ V.}}{500\text{ rpm}} \times 2.778\text{ rpm} = 52.778\text{ mV}$$

$$\text{Analog} = \frac{9.5\text{ V.}}{15{,}000\text{ mm/min.}} \times 1000\text{ mm/min.} = 0.633\text{V} = 633\text{mV}$$

Therefore, "PROGAIN" (P23) = 53

When setting the proportional gain, the following considerations must be taken into account:

* The maximum amount of following error allowed by the CNC for the spindle is the value indicated by spindle machine parameter "MAXFLWE1" (P21). When exceeded, the CNC issues the corresponding following error message.

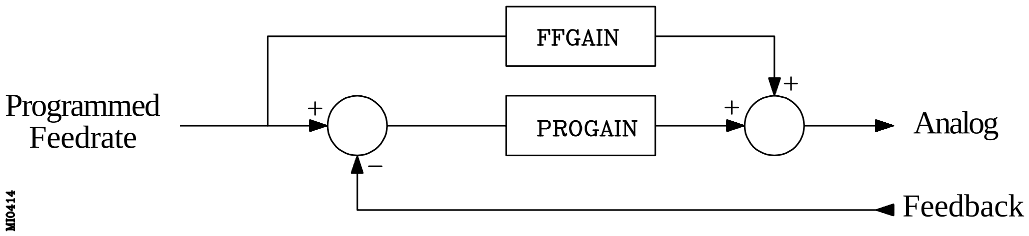
* The amount of following error decreases as the gain increases, but it tends to make the system unstable.

4.7.4.4 *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 "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 machine parameter "MAXVOLT" (P37).

1.- Move the spindle 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 0,945V then: P37=9450.

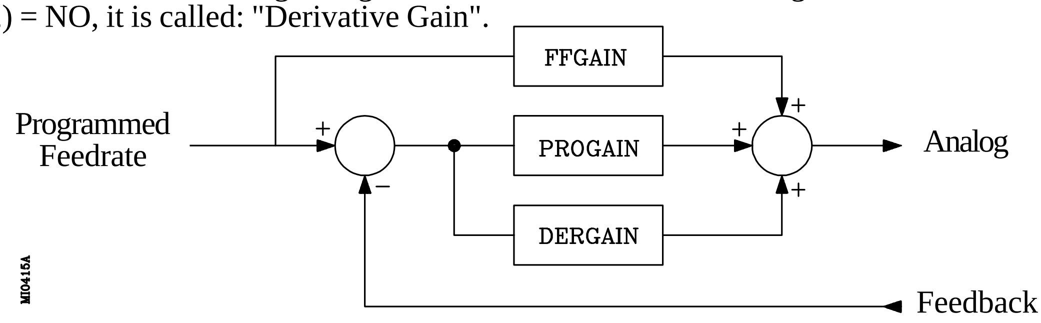
Next, set parameter "FFGAIN" (P25) to the desired value.

4.7.4.5 *DERIVATIVE / AC-FORWARD GAIN SETTING*

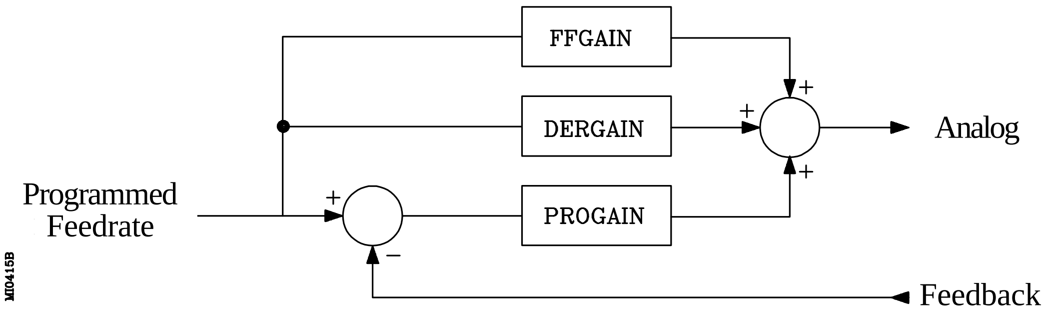
With the Derivative gain, it is possible to reduce the following error during the acc./dec. stages.

Its value is determined by spindle machine parameter "DERGAIN" (P24).

When this additional analog voltage is due to fluctuations of following error, "ACFGAIN" (P42) = NO, it is called: "Derivative Gain".



When it is due to variations of the programmed feedrate, "ACFGAIN" (P42) = YES, it is called AC-forward Gain" since it is due to acc./dec.



Best results are usually obtained when using it as AC-forward Gain, "ACFGAIN" (P42) = YES together with Feed-Forward Gain.

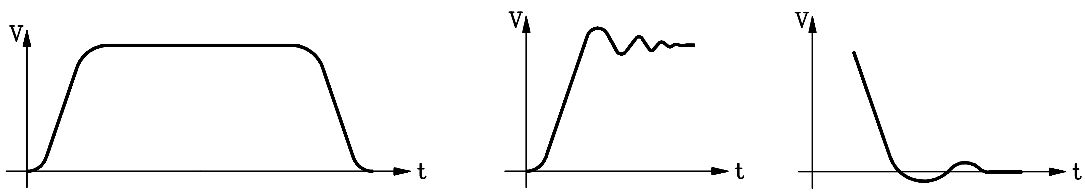
This gain is only to be used when operating with acceleration / deceleration.

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).



4.7.4.6 MACHINE REFERENCE SETTING

To set the machine reference point proceed as follows:

* Indicate in spindle machine parameter "REFPULSE" (P32) the type of marker pulse (Io) used by the feedback device.

* Also, indicate in spindle machine parameter "REFDIREC" (P33) the homing direction to look for that marker pulse.

* Set the spindle machine parameters defining the home switch approaching feedrate "REFEED1" (P34), and the marker pulse searching feedrate after the home switch has been found "REFEED2" (P35).

* The machine reference point will be set to "0". Spindle machine parameter "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.

* 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 spindle machine parameter "REFVALUE" (P36), which defines the coordinate corresponding to the Machine Reference Point (physical location of the marker pulse).

  "REFVALUE" P36 = Machine Coordinate of the 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 -123.5º as the coordinate value for this point, the coordinate of the Machine Reference Point with respect to Machine Reference Zero will be:

  "REFVALUE" P36 = 12 - (-123.5) = 135.5º

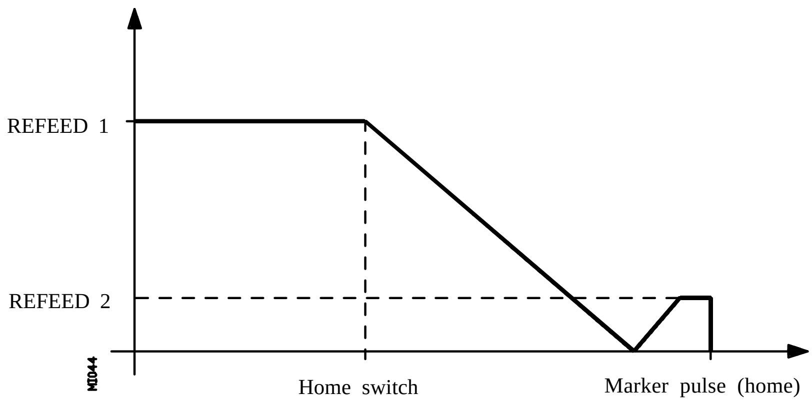
* After allocating this new value to the machine parameter, press SHIFT + RESET or turn the CNC off and back on in order for this value to be assumed by the CNC.

* The spindle must be homed again in order for it to assume its right reference values.

4.7.4.7 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 will be installed so the marker pulse 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 selected spindle does not have a machine reference (home) switch (spindle machine parameter "DECINPUT (P31)" = NO), the CNC will move the spindle at the feedrate set by spindle machine parameter "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 reference pulse being used (positive or negative) with the type of active flank (up or down) to be used when setting spindle machine parameter "REFPULSE (P32)".

4.8 TREATMENT OF EMERGENCY SIGNALS

4.8.1 EMERGENCY SIGNALS

The FAGOR 8050 CNC provides the following emergency signals:

/EMERGENCY STOP Physical emergency input.

It is generated from the outside and it corresponds to pin 2 of connector X9.

This signal is active low (0V).

/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).

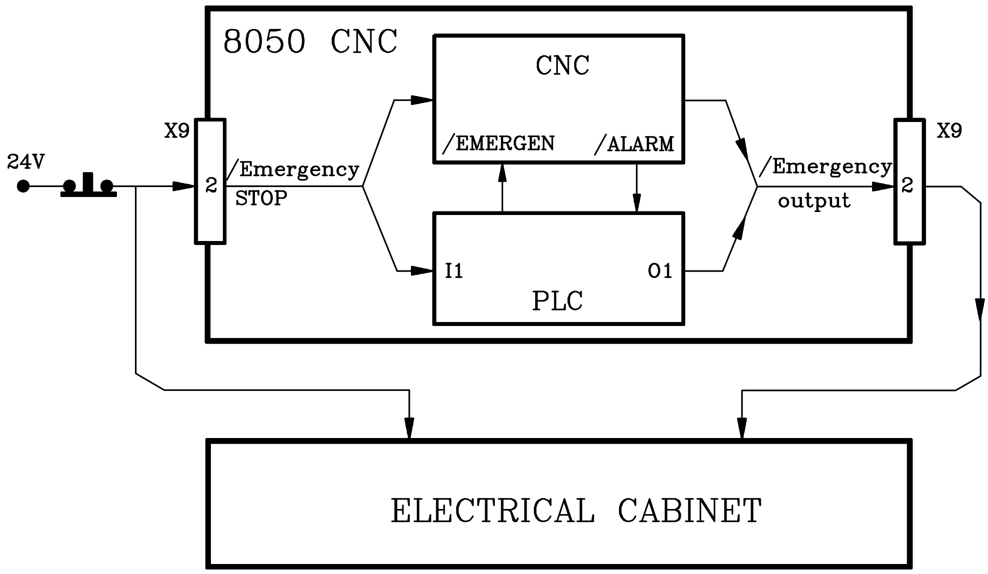
4.8.2 CNC TREATMENT OF EMERGENCY SIGNALS

The emergency inputs of the CNC are:

/EMERGENCY STOP Physical input coming from the outside.
/EMERGEN (M5000) Physical input coming from the PLC.

The emergency outputs of the CNC are:

/EMERGENCY OUTPUT Physical output to the outside.
/ALARM (M5507) Physical output to the PLC.



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.

4.8.3 *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.

5. *INTRODUCTION TO THE PLC*

It is highly recommended to save the machine parameters in a peripheral device or computer in order to avoid losing them by replacing modules, checksum errors, operator errors, etc.

The PLC program can be introduced from the front panel or from a computer or peripheral, using the RS232C and RS422 serial lines.

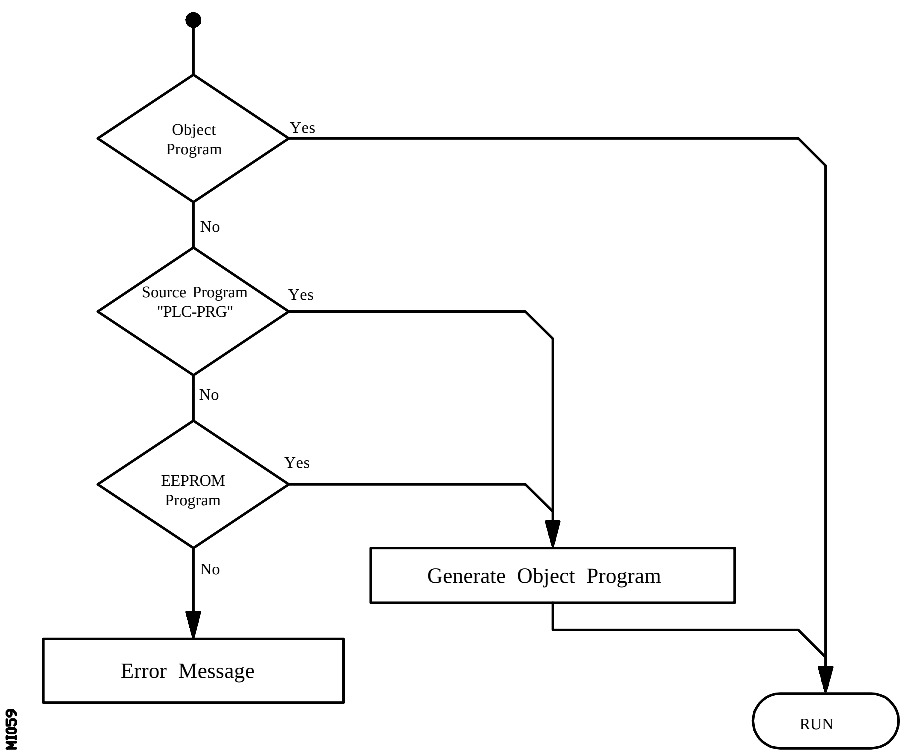
The PLC program will be stored in the internal memory of the CNC together with the part programs and are displayed in the program directory (utilities) together with the part programs.

Once the program has been compiled, the PLC will generate the executable code which will be stored in the PLC's internal memory.

The CNC has an EEPROM to store the PLC program and the User customized pages (screens).

On CNC power-up, the PLC will execute the Object Program. If this program does not exist, it will be generated by compiling the Source Program "PLC_PRG", if it exists, or the one saved in the EEPROM otherwise. Once the Object Program is generated, the PLC will execute it.

Should there be no source program, the CNC will issue the corresponding error message.



The PLC has 256 inputs and 256 outputs, some of which, depending on the configuration of the CNC acquired can communicate with the outside. The numbering of inputs and outputs for each of the modules will be as follows:

AXES module	I1 -I40	O1 -O24
INPUT/OUTPUT Module (1)	I65 -I128	O33-O64
INPUT/OUTPUT Module (2)	I129-I192	O65-O96
INPUT/OUTPUT Module (3)	I193-I256	O97-O128

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, T auxiliary functions.

* To display a screen previously defined by the user, as well as generating messages and errors in the CNC.

* The reading and modification of internal CNC variables from the PLC.

* Access to all PLC variables from any part program.

* Monitoring of PLC variables on the CNC screen.

* Access to all PLC variables from a computer, via DNC and by means of the RS 232 C and RS 422 serial ports.

5.1 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 256 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 256 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-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.

5.2 *PLC PROGRAM EXECUTION*

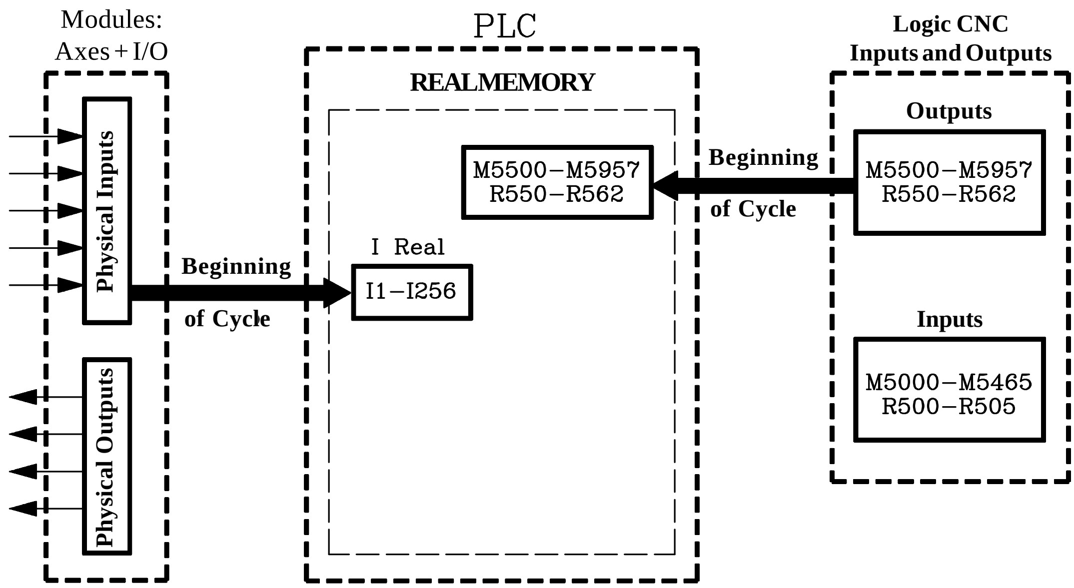
The PLC executes the user program cyclically. In other words, once it executes the complete program, it re-starts running this program from the first instruction.

This cyclic processing of the program is done as follows:

1. It allocates the current values of physical CNC inputs (AXES and I/O module connectors) to the "I" resources of the PLC.

   For instance, if physical input I10 (pin 25 of connector X9 of the Axes module) is receiving 24 V, the PLC sets resource "I10" to "1".

   Only the inputs corresponding to the modules being used are updated. This way, when having the axes module and one I/O module, the PLC will only update resources I1 thru I40 and I65 thru I128. The rest of the "I" resources will not be modified.

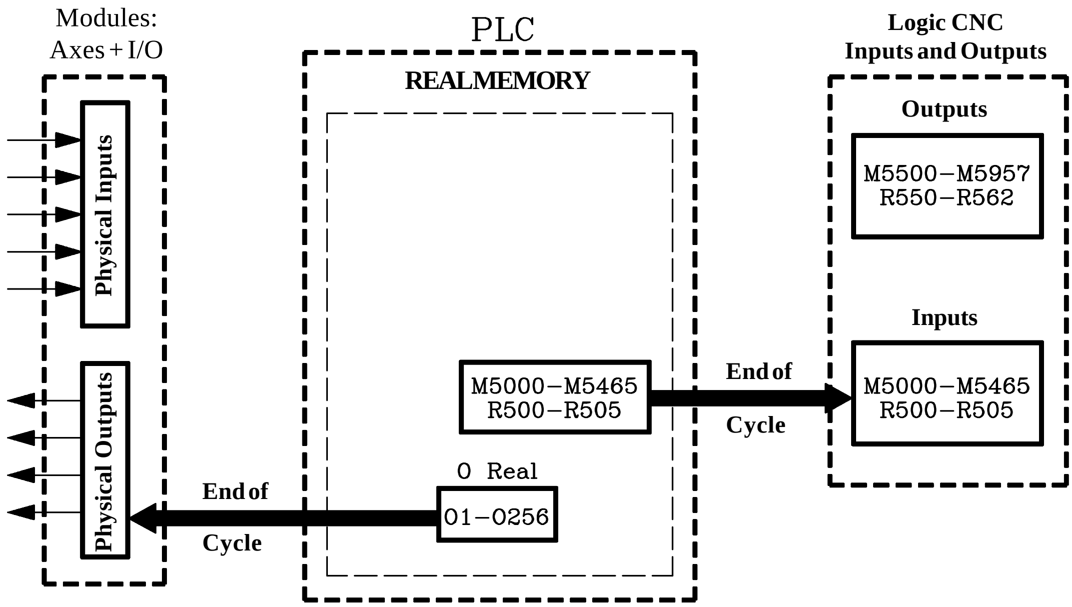


2. It allocates the current values of the logic CNC outputs (CNCREADY, START, FHOUT, ...) to PLC resources M5500 thru M5957 and R550 thru R562 .

3. It runs the program cycle.

   The next section of this chapter describes the structure of the PLC program and its execution modules.

4. It updates the Logic CNC inputs (/EMERGEN, /STOP, /FEEDHOL, ...) with the current values of PLC resources M5000 thru M5465 and R500 thru R505.



5. Allocates the current values of PLC "O" resources to the physical outputs (connectors of the Axes and I/O modules).

   For instance, if resource "O5" is set to "1", the PLC sets physical output "O5" (pin 4 of connector X10 of the Axes module) to 24V.

6. Bear in mind that all the actions of the program executed by the PLC alter the status of its resources.

   Example: I10 AND I20 = O5

   When this condition is met [resource I10 is "1" and I20 is also "1"], the PLC sets resource "O5" to "1". If this condition is not met, the PLC sets resource "O5" to "0".

   Therefore, the status of a resource may change during the execution of the PLC program.

   Example, assuming that the initial status of resource M100 is "0":

M100 AND I7 = O3	M100 = "0"
I10 = M100	M100 takes the value of resource I10
M100 AND I8 = M101	The value of M100 depends on the previous instruction.

   This type of problems may be prevented by careful programming or by using "Image" resource values (instead of "Real" values).

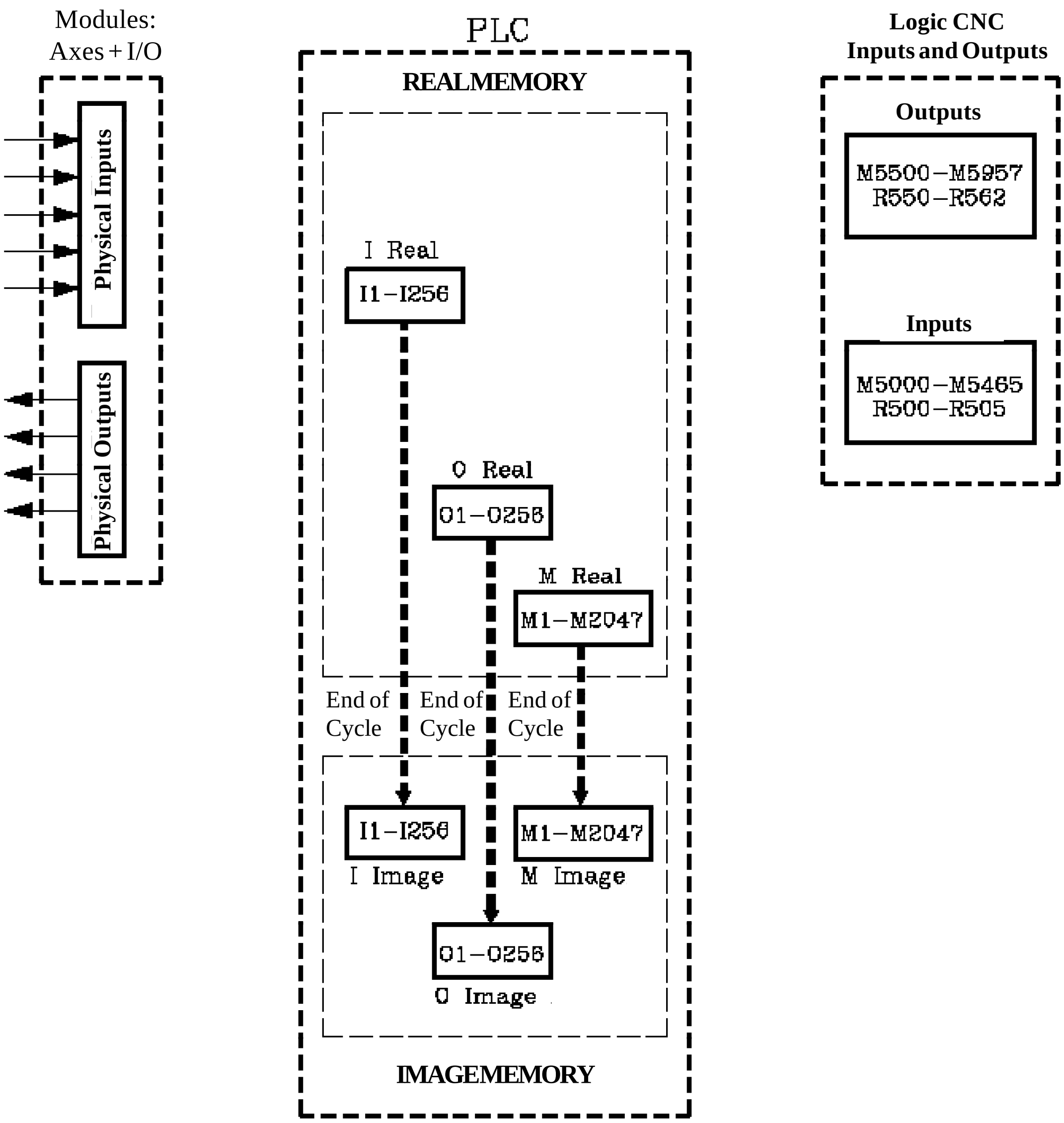
The PLC has 2 memories to store the status of the various registers: Real and Image.

All the steps described so far work with Real memory.
Saying "the value of such and such register" is the same as saying "the **Real** value of such and such register".

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 I256, O1 thru O256 and M1 thru M2047



The next example shows how the PLC acts when operating with real and image values.

PLC Program		Using Real values				Using Image values			
		M1	M2	M3	O5	M1	M2	M3	O5
() = M1	Initial status	0	0	0	0	0	0	0	0
M1 = M2	End of 1st Scan	1	1	1	1	1	0	0	0
M2 = M3	End of 2nd Scan	1	1	1	1	1	1	0	0
M3 = O1	End of 3rd Scan	1	1	1	1	1	1	1	0
	End of 4th Scan	1	1	1	1	1	1	1	1

The first program line indicates that resource M1 is set to "1".

Operating with real values:

M1 = M2 The real value of M1 is "1", it has been set by the previous line.
M2 = M3 The real value of M2 is "1", it has been set by the previous line.
M3 = O5 The real value of M3 is "1", it has been set by the previous line.

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".

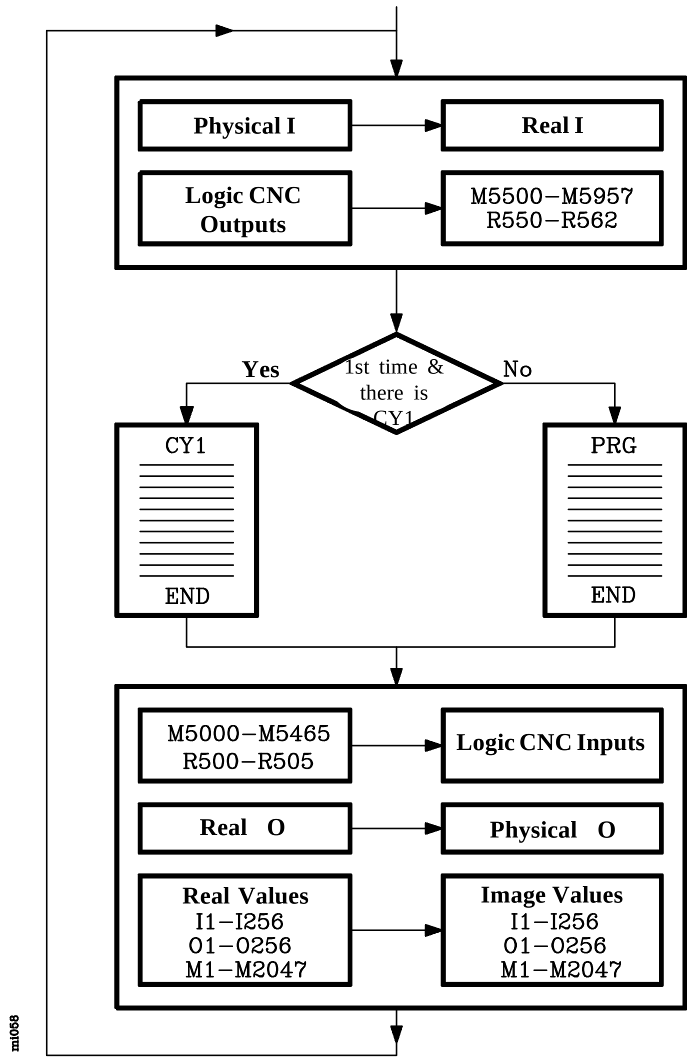
As can be observed, the system is faster when operating with real resource values.

Operating with image values permits analyzing the same resource along the whole program with the same value regardless of its current (instantaneous) real value.

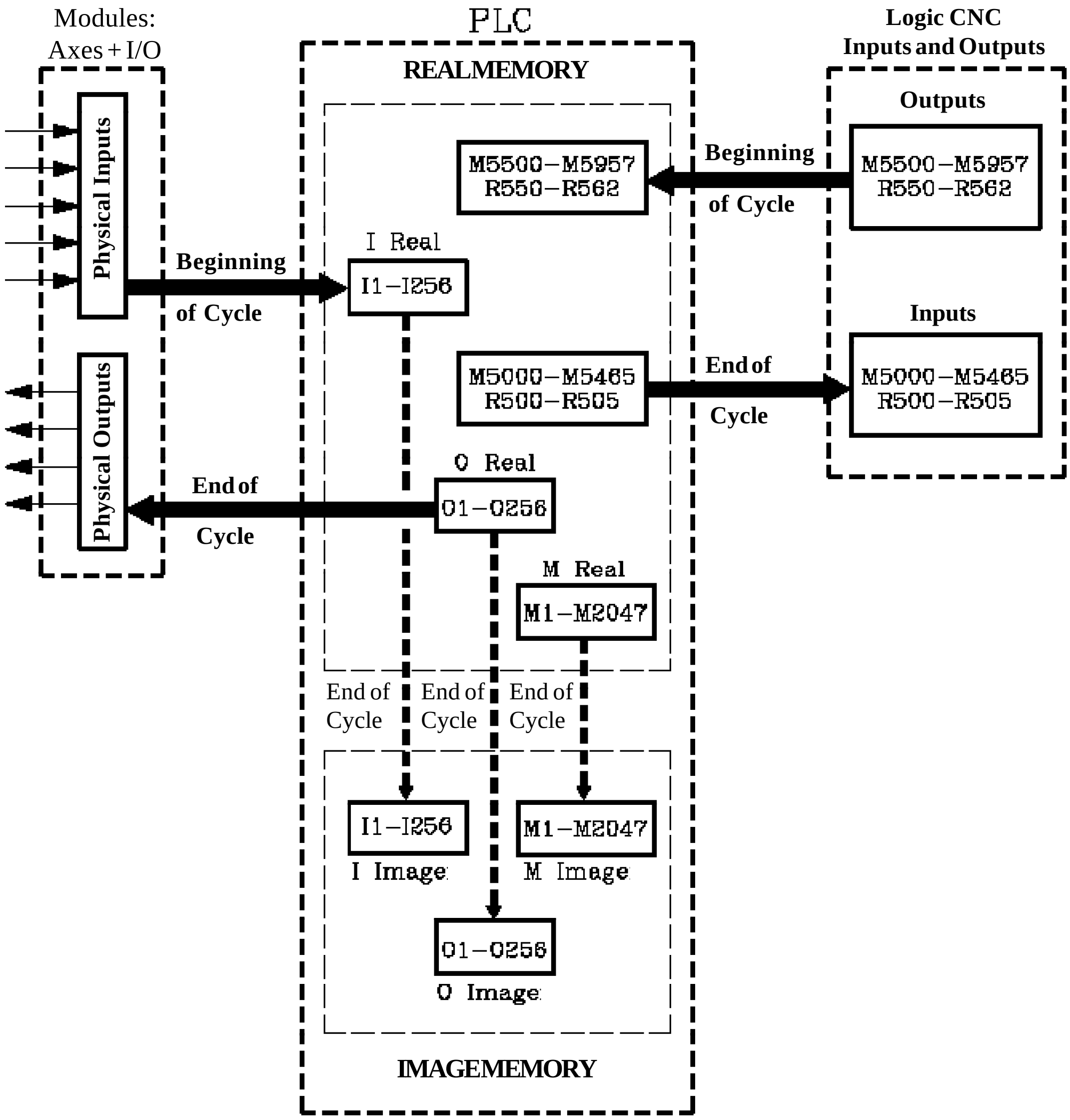
7. It concludes this cycle scan and it gets ready for the next one.

The flow-chart on the next page illustrates the cyclic program processing.

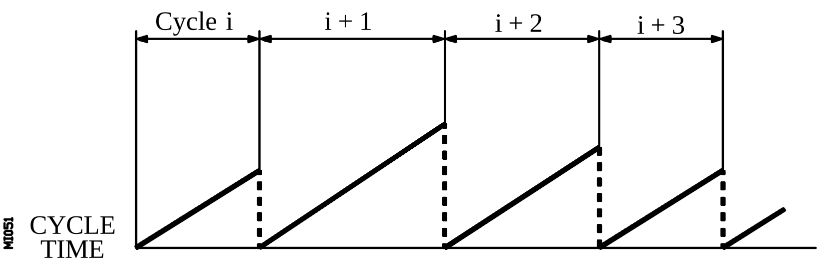
Cyclic program processing



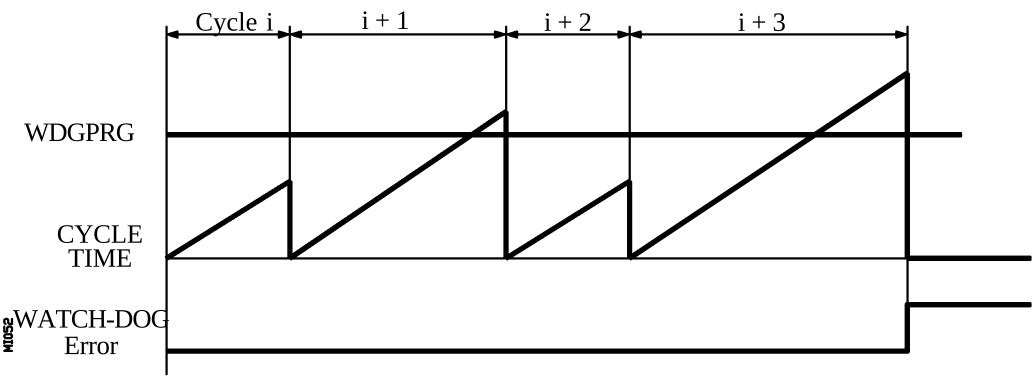
Information exchange:



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.



By means of the PLC machine parameter “WDGPRG” a maximum cycle execution time is established. 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.

5.3 MODULAR PROGRAM STRUCTURE

The program to be executed by the PLC consists of a series of MODULES which are appropriately defined by means of DIRECTING INSTRUCTIONS.

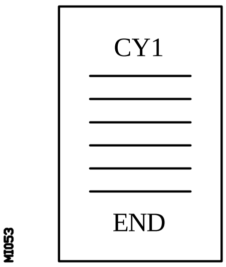
The modules which can make up the program are:

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.

5.3.1 FIRST CYCLE MODULE (CY1)

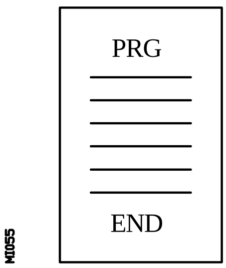


This module is optional and will only be executed when the PLC is turned on. It is used to initialize the different resources and variables with their initial values, before proceeding to execute the rest of the program.

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.

5.3.2 MAIN MODULE (PRG)

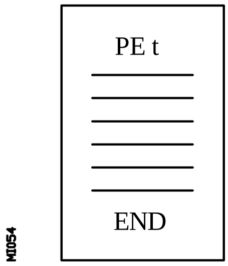


This module contains the user program. It will be executed cyclically and will be given the task of analyzing and modifying CNC inputs and outputs. Its time of execution will be limited by the value indicated in the PLC machine parameter “WDGPRG”.

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.

5.3.3 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 of between 1 and 65535 milliseconds can be programmed and the execution time of this module will be limited by the value indicated in the PLC machine parameter "WDGPER".

This module operates by default with the real values of resources I, O, M.

Example:

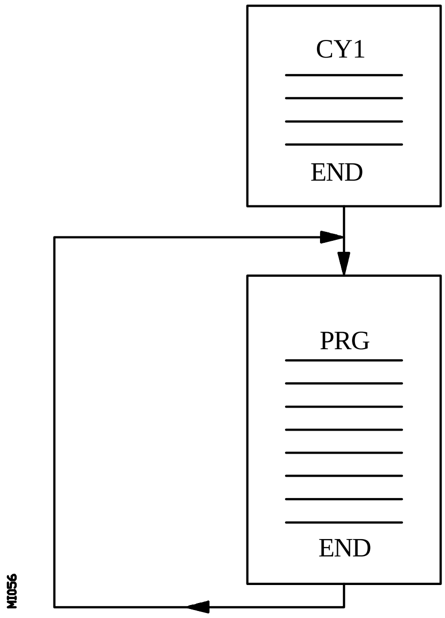
PE 10	Defines the beginning of the Periodic Module PE which will be executed every 10 milliseconds.

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.

5.3.4 PRIORITY IN THE EXECUTION OF PLC MODULES

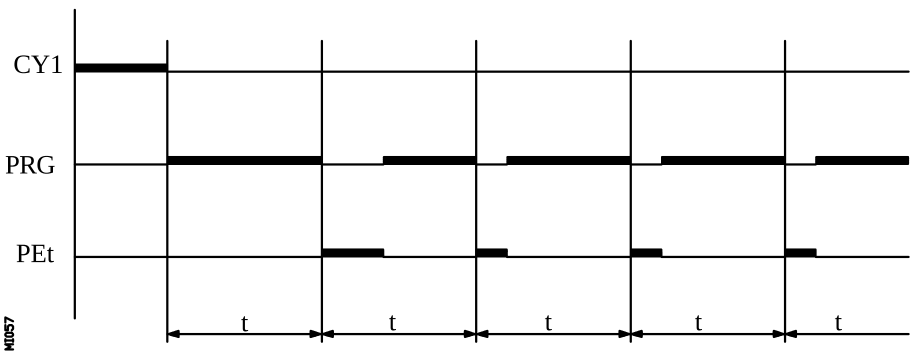
Every time the PLC program is started (command RUN) the first module to be executed is the first cycle Module (CY1). Once execution has been completed, it will continue with the main Module (PRG).

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.



6. PLC RESOURCES

6.1 INPUTS

These are elements which provide the PLC with information from the signals which are 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 256 inputs although when communicating with the outside world it can only access the ones indicated by each module.

The numbering of the inputs at each module is determined by the logic address (device select code) assigned to the module in such a way that the first group of inputs corresponds to the module with the lowest device select code and the last group of inputs corresponds to the module with the highest select code. For example:

Module	Device Select Code	Inputs
AXES	2	I1 -I40
I/O TRACING	3	I65 -I128
INPUT/OUTPUT (1)	4	I129-I192
INPUT/OUTPUT (2)	5	I193-I256

6.2 OUTPUTS

These are elements which allow the PLC to activate or deactivate the different devices or drives 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 256 outputs although when communicating with the outside world it can only access the ones indicated by each module.

The numbering of the outputs at each module is determined by the logic address (device select code) assigned to the module in such a way that the first group of outputs corresponds to the module with the lowest device select code and the last group of outputs corresponds to the module with the highest select code. For example:

Module	Device Select Code	Outputs
AXES	2	O1 -O24
I/O TRACING	3	O33 -O64
INPUT/OUTPUT (1)	4	O65-O96
INPUT/OUTPUT (2)	5	O97-O128

Output O1 coincides with the emergency output of the CNC (pin 2 of connector X10); thus, it must be kept high (logic level 1).

6.3 MARKS

These are elements capable of memorizing in one bit (as if they were an internal relay) information defined by the user, their value being inalterable even when the power supply to the system is turned off.

This will be programmed by the letter M followed by the number of the mark which it is wished to reference, for example, M1, M25, M102, etc.

The PLC controls the following marks:

User marks	M1 - M2000
Arithmetic flag marks	M2001
Clock marks	M2009 - M2024
Fixed status marks	M2046 & 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.

MARK	Half Period
M2009	100 msec.
M2010	200 msec.
M2011	400 msec.
M2012	800 msec.
M2013	1.6 sec.
M2014	3.2 sec.
M2015	6.4 sec.
M2016	12.8 sec.
M2017	1 sec.
M2018	2 sec.
M2019	4 sec.
M2020	8 sec.
M2021	16 sec.
M2022	32 sec.
M2023	64 sec.
M2024	128 sec.

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	MSG1
M4001	MSG2
.....	.....
.....	.....
M4126	MSG127
M4127	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. The activation of one of these marks does not activate the CNC external Emergency output.

They can be named by means of mark M4500-M4563 or by means of their associated mnemonic ERR1 - ERR64:

M4500	ERR1
M4501	ERR2
.....	.....
.....	.....
M4562	ERR63
M4563	ERR64

It is recommended to change the status 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.

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:

M4700	PIC0
M4701	PIC1
.....	.....
.....	.....
M4954	PIC254
M4955	PIC255

The PLC has marks M5000 to M5957 to exchange information with the CNC, all of which have associated mnemonics and are detailed in the chapter which deals with CNC - PLC communication.

6.4 REGISTERS

These are elements which store a numerical value in 32 bits, their value remaining unalterable even when the power supply to the system is cut off.

They do not have image values and are represented by the letter R, followed by the register number it is desired to reference, for example R1, R25, R102, etc.

The PLC has the following registers:

User registers	R1 - R256
Registers for communication with the CNC	R500 - R559

The PLC will consider each value stored in each register as an integer with a sign, and can be within ±2147483647.

It is also possible to make reference to a BIT of the REGISTER by putting the letter **B** and the bit number (0/31) in front of the selected register. For example:

B7 R155 Refers to Bit 7 of Register 155.

The PLC considers bit 0 as being the one with least significance and bit 31 as being the one with most significance.

The value stored iN a Register can be treated as being decimal, hexadecimal (preceded by "$"), binary (preceded by "B") or in BCD.

Example:

Decimal : 156
Hexadecimal : $9C
Binary : B0000 0000 0000 0000 0000 0000 1001 1100

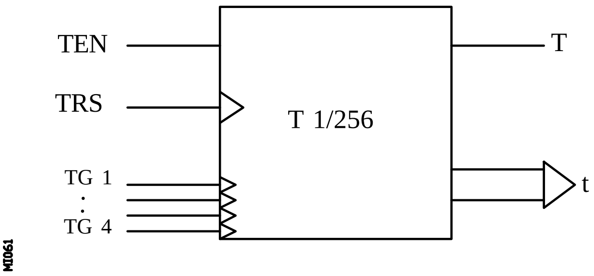
6.5 TIMERS

These are elements capable of maintaining their output at a determined logic level during a preset time (time constant), after which the output changes status.

They do not have image values and are represented by the letter T, followed by the number of the timer it is required to reference, for example, T1, T25, T102, etc.

The time constant is stored in a 32-bit variable, and so its value can be between 0 and 4294967295 milliseconds, which is equivalent to 1193 hours (almost 50 days).

The PLC has 256 timers, each of which has T status output and TEN, TRS, TG1, TG2, TG3 and TG4 inputs. It is also possible to consult at any moment the time which has elapsed from the moment it was activated.

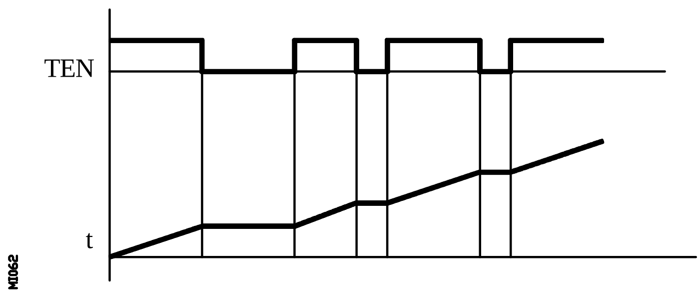


ENABLE INPUT (TEN)

This input allows the timing of the timer to be stopped. It is referred to by the letter TEN followed by the number of the timer which is wished to reference, for example TEN 1, TEN 25, TEN 102, etc.

So that the time elapses within the timer this input must be at level "1". By default every time a timer is activated the PLC will assign this input a logic level "1".

If, once the timer has been activated, TEN = 0 is selected, the PLC stops the timing, it being necessary to assign TEN = 1 for this timing to continue.



Example:

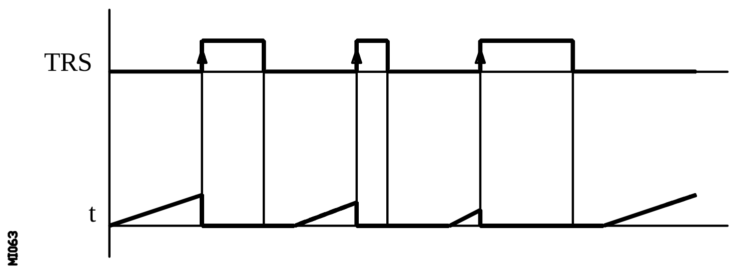
I2 = TEN 10; Input I2 controls the Enable input of timer T10.

RESET INPUT (TRS)

This input allows the timer to be initialized, by assigning the value 0 to its T status and by cancelling 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:

Input TG1 activates the timer in the MONOSTABLE mode.
Input TG2 activates the timer in the ACTIVATION DELAY mode.
Input TG3 activates the timer in the DEACTIVATION DELAY mode.
Input TG4 activates the timer in the SIGNAL LIMITER 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.

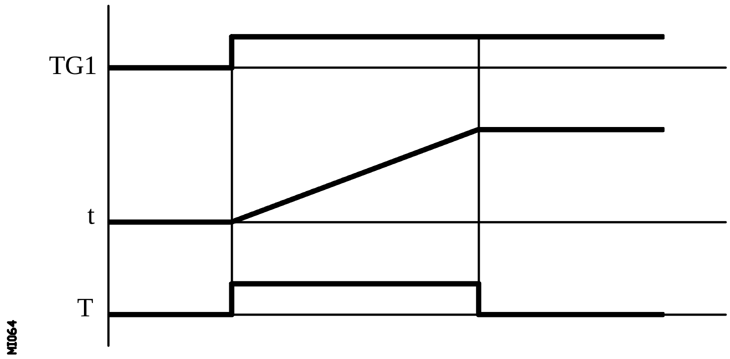
6.5.1 TIMER OPERATING MODES

The four operating modes which are available to each timer can be selected by means of the activation of one of the trigger inputs TG1, TG2, TG3, TG4.

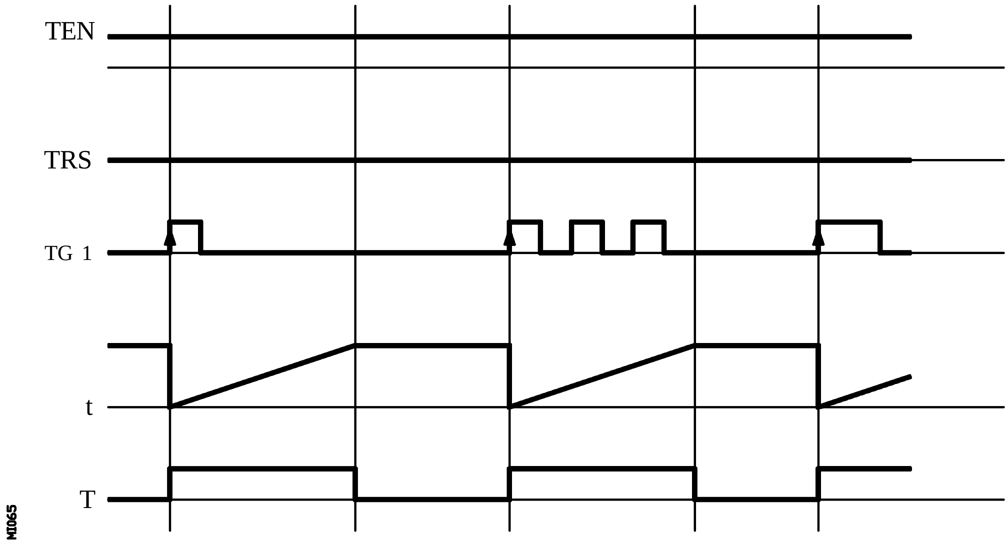
Input TG1 activates the timer in the MONOSTABLE mode
Input TG2 activates the timer in the ACTIVATION DELAY mode
Input TG3 activates the timer in the DEACTIVATION DELAY mode
Input TG4 activates the timer in the SIGNAL LIMITER mode

6.5.1.1 MONOSTABLE MODE. INPUT TG1

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 this moment the timer status output (T) changes status (T=1) and the timing t starts from 0.
Once the time specified by the time constant has elapsed the time will be considered as



having concluded. The timer status output (T) changes status (T=0) and the elapsed time will be maintained with the time value of the timer (T).

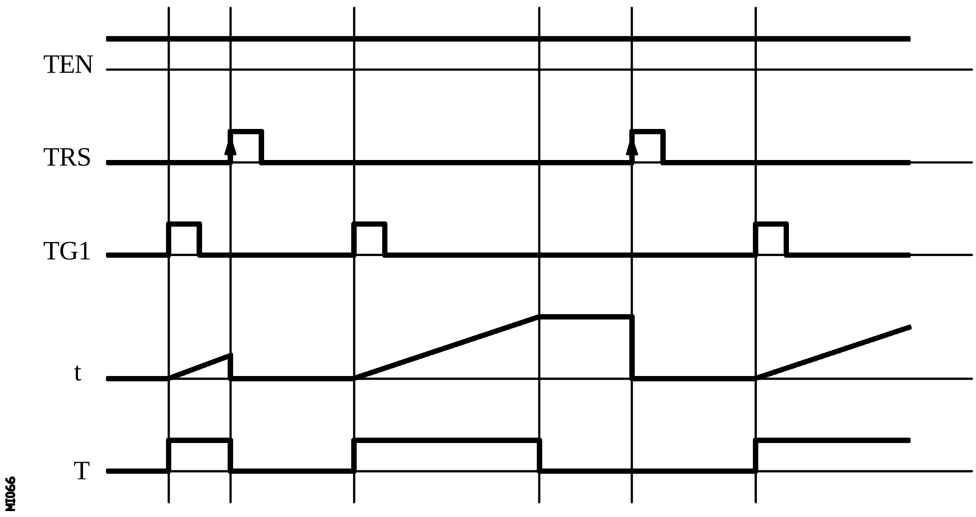
Any alteration which may be produced in input TG1 (leading or trailing edge) during the timing operation will have no effect whatsoever.

If, once the timing is complete it is required to activate the timer again, another leading edge must be produced at the TG1 input.

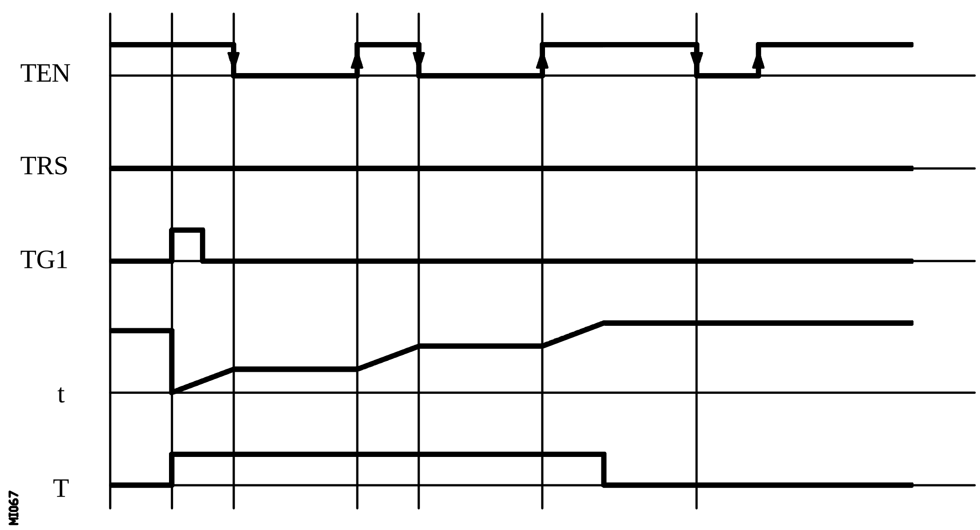
Operation of the TRS input in this mode

If a leading edge is produced at the TRS input at any moment during the timing or afterwards, the PLC initializes the timer, assigning the value 0 to its T status and cancelling its count (it initializes it 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



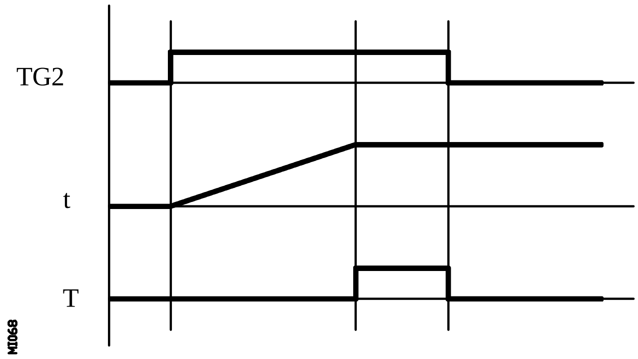
If, once the timer is activated, TEN = 0 is selected, the PLC stops timing, and it is necessary to assign TEN = 1 for this timing to continue.



6.5.1.2 ACTIVATION DELAY MODE. INPUT TG2

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.

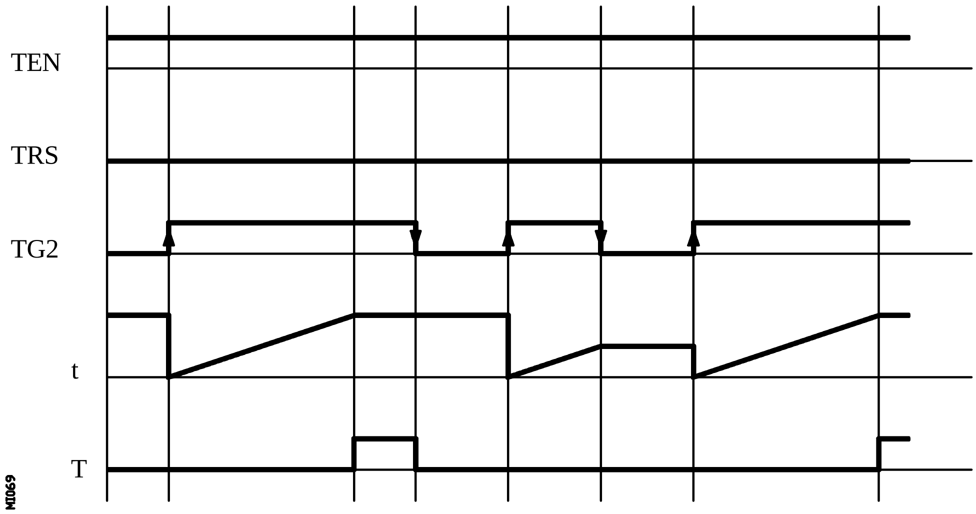
The duration of the delay is determined by the time constant.



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.

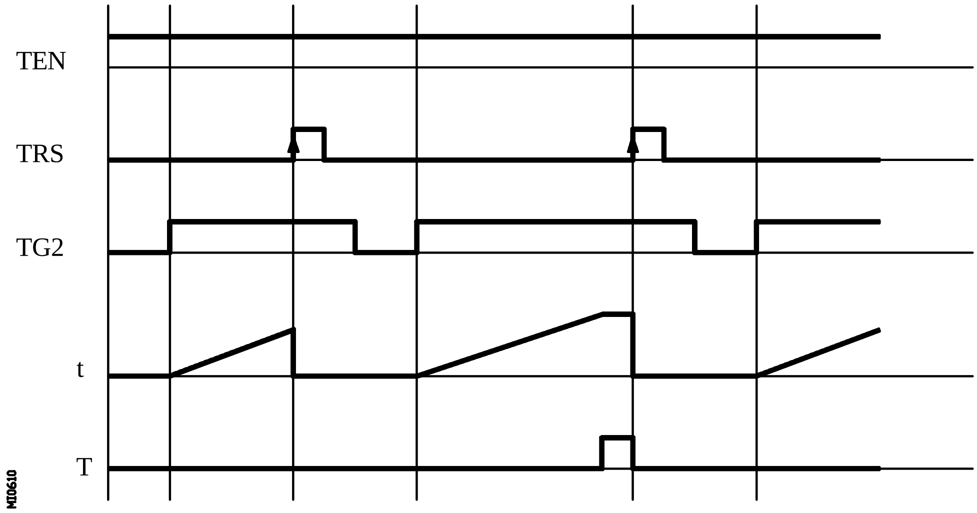
The elapsed time will remain as a timer time value (T) once timing has been completed. If, once the timing has finished, it is required to activate the timer again, another leading edge must be produced in the TG2 input.



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).

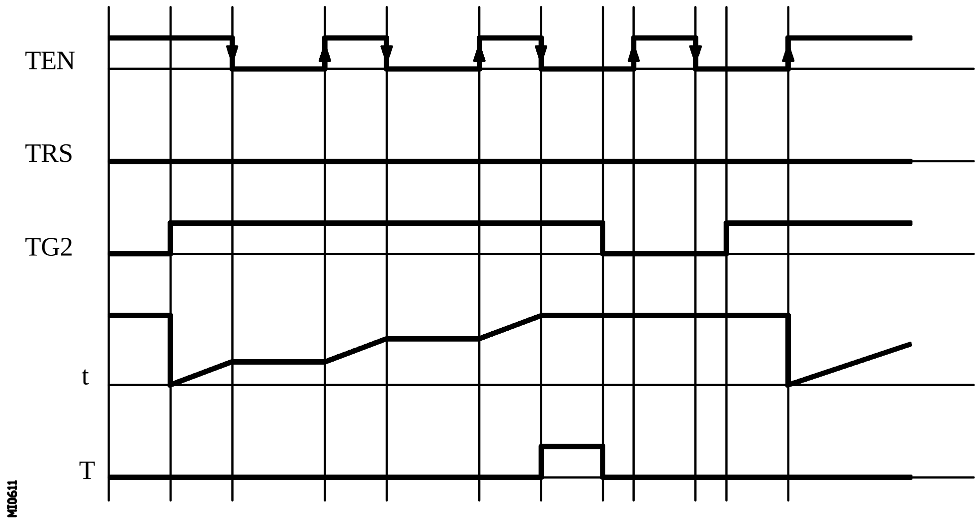
Operation of the TRS input in this mode

If a leading edge is produced in 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

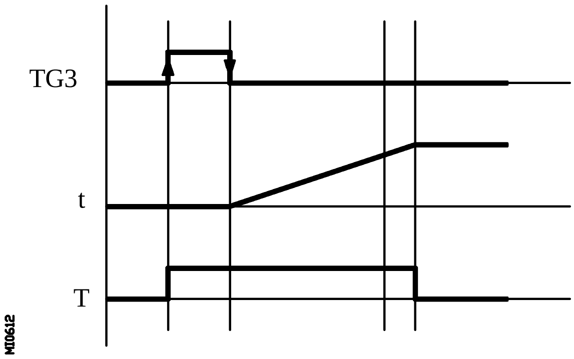
If, once the timer is activated, TEN = 0 is selected, the PLC stops timing, and it is necessary to assign TEN = 1 for this timing to continue.



6.5.1.3 DEACTIVATION DELAY MODE. INPUT TG3

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.



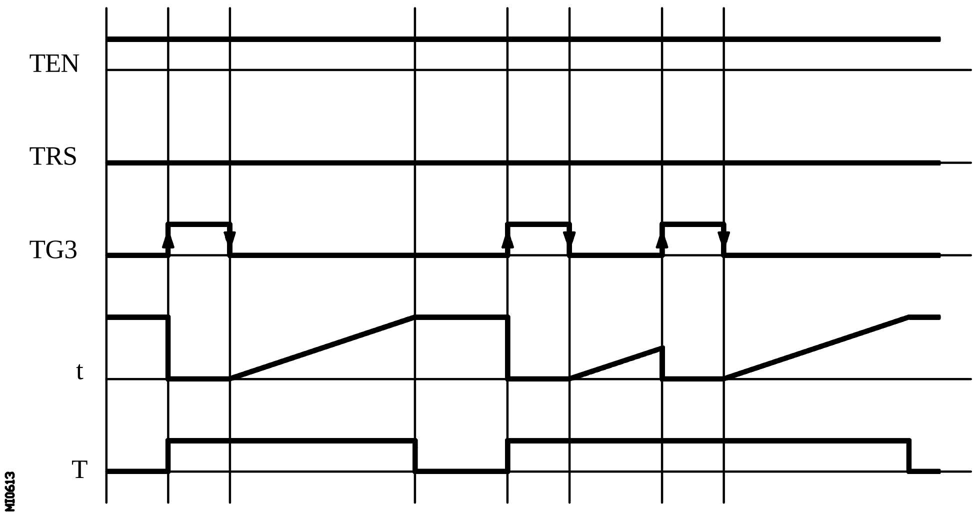
If the timer is initialized with values TEN=1 and TRS=0, the timer will be activated when a leading edge is produced at the TG3 input. At that moment, the timer status output will have a value of T=1.

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.

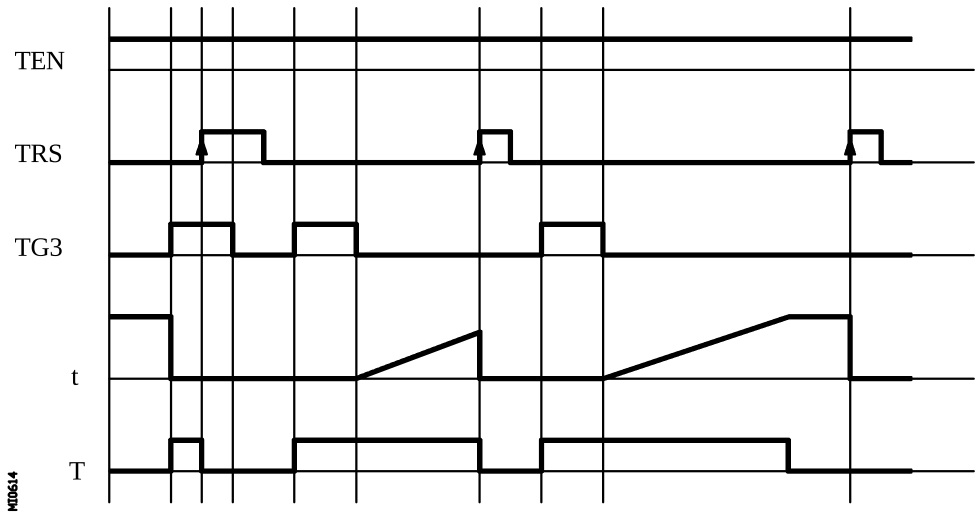
If, once the timing has finished, it is required to activate the timer again, another leading edge must be produced at the TG3 input.



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.

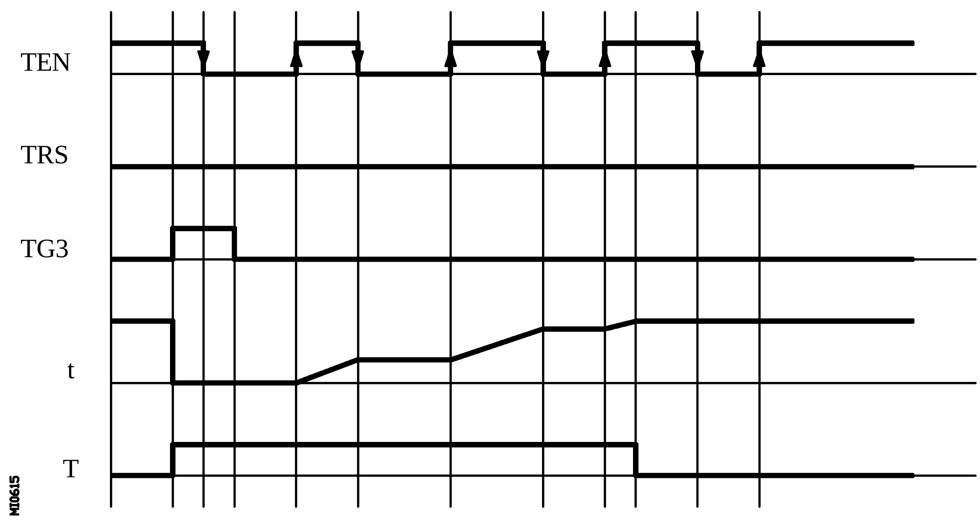
Operation of the TRS input in this mode

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.



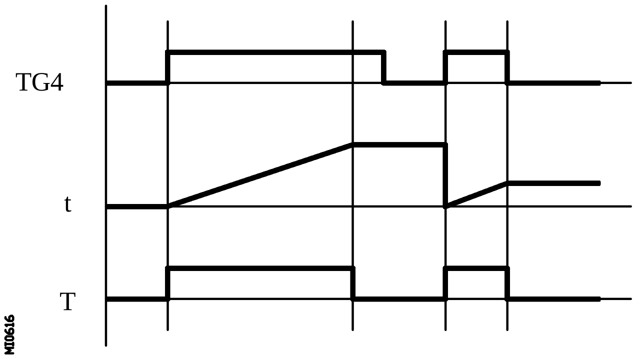
Operation of the TEN input in this mode

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.

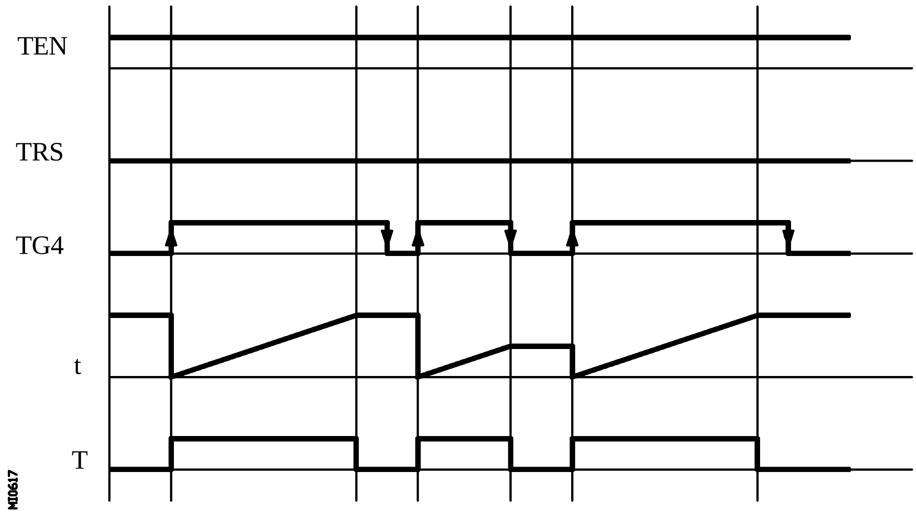


6.5.1.4 SIGNAL LIMITING MODE. INPUT TG4

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.



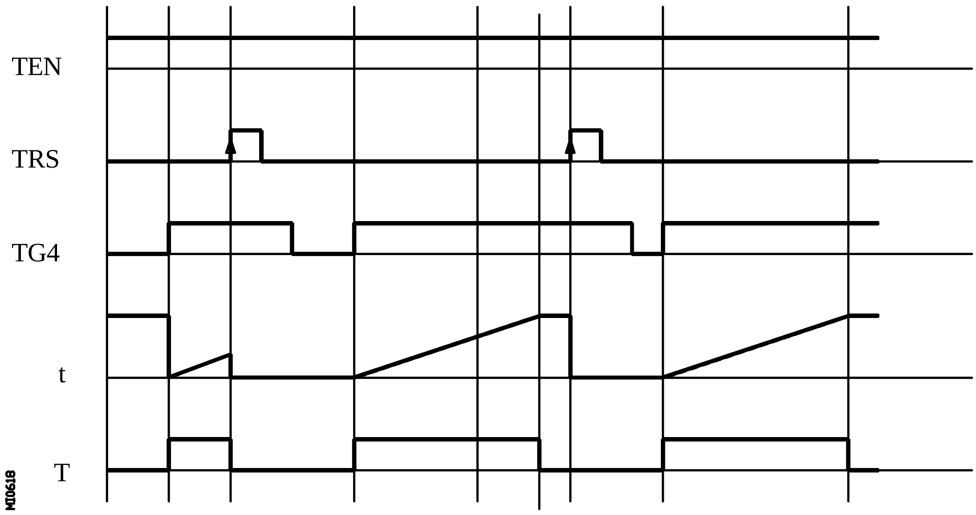
Once the time specified by the time constant has elapsed, timing will be considered as having finished. The time status output (T) changes status (T=0) and the elapsed time will be kept as a timer time value (T).

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.

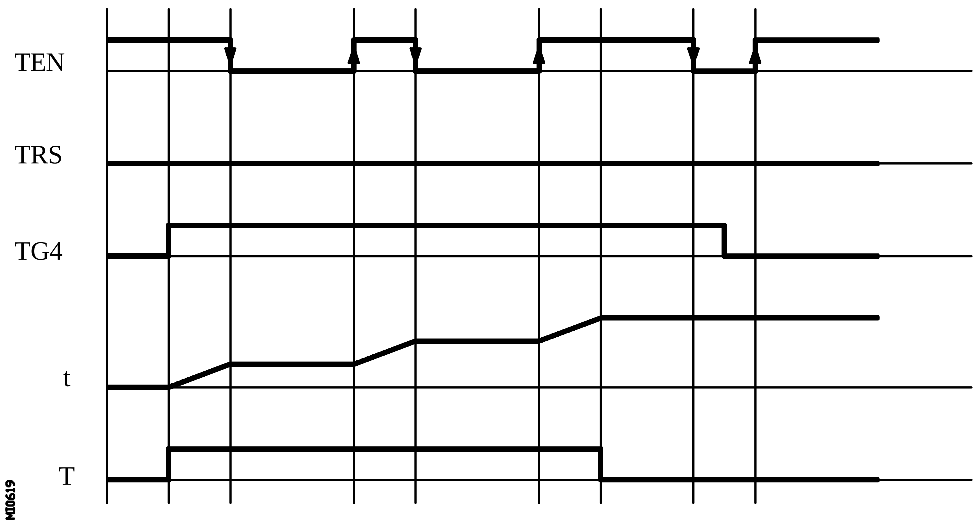
Operation of the TRS input in this mode

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.

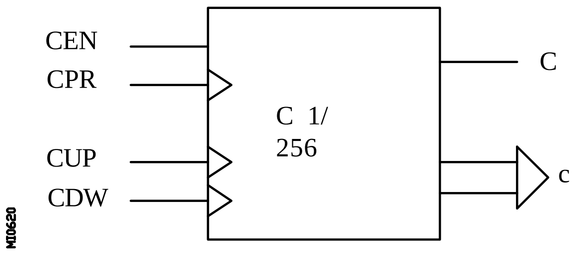


6.6 *COUNTERS*

These are elements capable of counting up or down a specific number 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 $\pm$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.



COUNT UP 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:

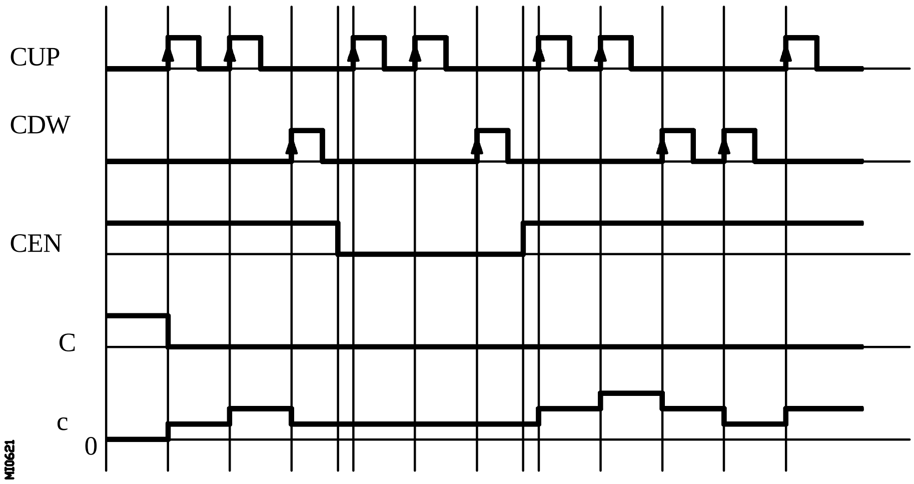
I3 = CDW 20 ; Every time a leading edge is produced at input I3 the counter count C20 will be decreased.

ENABLE INPUT (CEN)

This input allows the internal counter count to be stopped. It is referred to by the letters 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 logic level “1” to this input.

If CEN = 0 is selected the PLC stops the counter count, ignoring the inputs CUP and CDW until this input allows it (CEN = 1).



Example:

I10 = CEN 12 ; Input I10 controls the Enable input of counter C12

PRESET INPUT (CPR)

This input allows the counter to be preset with the desired value. It is referred to by the letters CPR followed by the number of the counter which is required to reference and the value to be assigned to the counter count.

For example CPR 1 100, CPR 25 224, CPR 102 0, CPR 200 500, etc.

The value of the count can be indicated by means of a numerical value or by assigning to it the internal value of an R register.

CPR 20 100	;	Presets the C20 counter with the value 100.
CPR 22 R200	;	Presets the C22 counter with the value of the Register 200 when the instruction is executed.

The counter is preset with the 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 it 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 (0/1) status of counter 123

In arithmetic and function comparison instructions C123 makes reference to the internal counter count.

I2 = MOV C123 R200	;	Transfers the count of C123 to register R200
CPS C123 GT 1000 = M100	;	Compares whether the count of C123 is greater than 1000, in which case it activates mark M100.

The PLC has a 32-bit variable to store the count of each counter.

6.6.1 *THE 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.

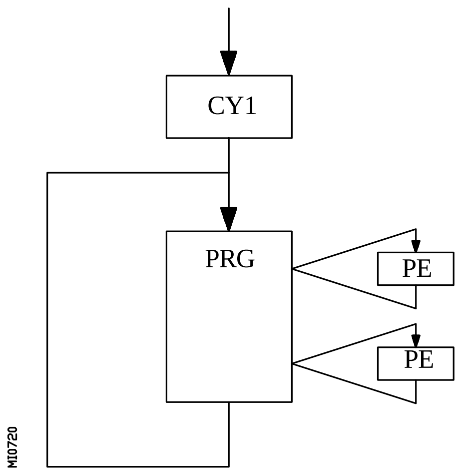
7. *PLC PROGRAMMING*

The PLC program is structured by modules and it could 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.

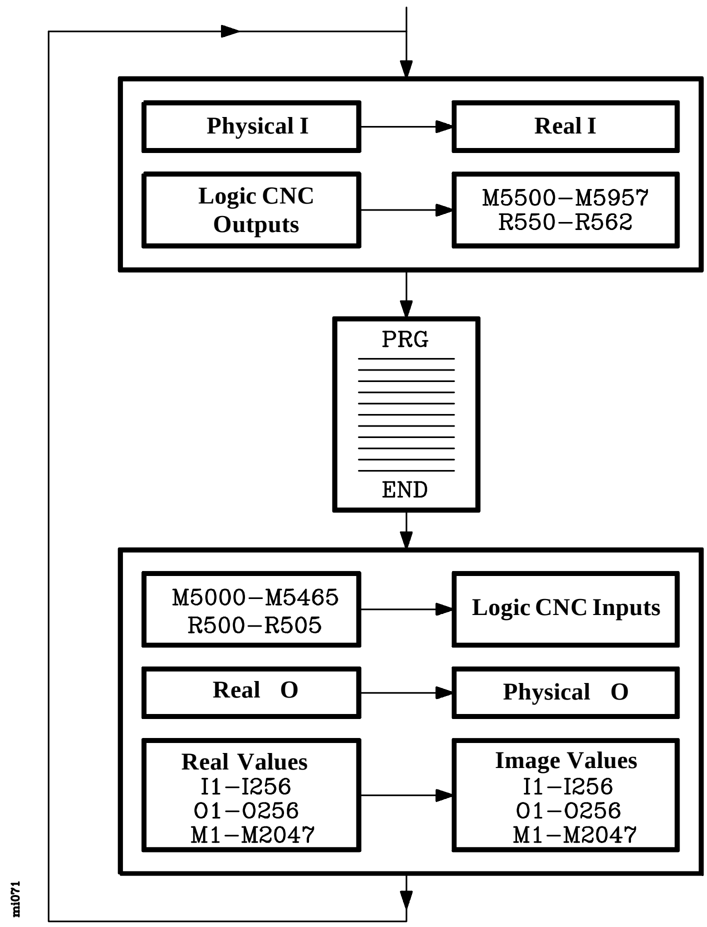
The periodic execution modules (PE) will be executed every so often with the frequency established for each of them. This time period starts counting from the time the CY1 cycle is ended. The execution of a periodic module temporarily interrupts the execution of the main module.



When defining the PLC program, both the processing of the main module (PRG) and the periodic modules (PE) must be taken into consideration.

Both are described next. Nevertheless, the chapter on "Introduction to the PLC" further elaborates on PLC program execution including the processing of the main module.

This cyclic processing of the main module is done as follows:



1. It allocates the current values of physical CNC inputs (AXES and I/O module connectors) to the "I" resources of the PLC.

2. It allocates the current values of the logic CNC outputs (CNCREADY, START, FHOUT, ...) to PLC resources M5500 thru M5957 and R550 thru R562 .

3. It runs the main module.

4. It updates the Logic CNC inputs (/EMERGEN, /STOP, /FEEDHOL, ...) with the current values of PLC resources M5000 thru M5465 and R500 thru R505.

5. Allocates the current values of PLC "O" resources to the physical outputs (connectors of the Axes and I/O modules).

6. It copies the real values of the I, O and M resources into their own image values.

   The resources having an image value are: I1 thru I256, O1 thru O256 and M1 thru M2047

7. It concludes this cycle scan and it gets ready for the next one.

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 (PE) is processed as follows:

1. The PLC takes into account the current values, as just before executing the PE module, of the physical inputs (connectors of the Axes and I/O modules).

2. It runs the Periodic Module.

3. It assigns to the physical outputs (connectors of the Axes and I/O modules) the current values of the "O" resources of the PLC.

4. It ends the execution of the Periodic Module and resumes the execution of the Main Module.

7.1 *STRUCTURE OF A MODULE*

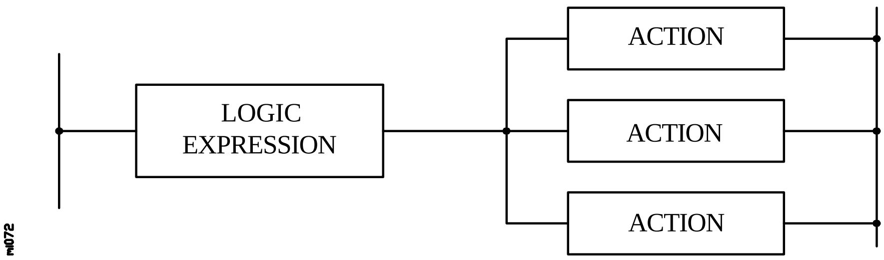
The modules which make up the PLC program (main module "PRG", periodic modules "PE" and first cycle module "CY1") consist of a series of Instructions which, depending on their functionality, can be divided into:

- Directing instructions.
- Executable instructions.

Directing Instructions provide the PLC with information on the type of module and on the way it must execute it.

Executable Instructions allow enquiries to be made on and/or alterations to the status of PLC resources (I,O,M,R,T,C) and consist of:

- A Logic Expression (Boolean 0/1).
- One or several Action instructions.



A Logic Expression consists of:

- One or several Enquiry Instructions on resource status
- One or several Operands.

Therefore, the structure of a module can be summed up as follows:

PLC Module
- Directing instruction (PRG)
- Executable instruction (I1 AND I2 =O2)
  - Logic expression (I1 AND I2)
    - Consulting instruction (I1)
    - Operand (AND)
  - Action instruction (=O2)

The PLC allows all the program lines to be associated with any type of information in the form of a comment. This comment will begin with the character "**;**" and if a line begins with this character the entire line will be considered as a comment and will not be executed.

Atention:

Empty lines are not allowed, they must contain at least one comment.

Programming example:

```
PRG              ; Directing Instruction
--------
I100 = M102      ; Executable Instruction
--------
I28 AND I30      ; Logic Expression
= O25            ; Action Instruction
--------
I32              ; Consulting instruction (I32)
AND I36          ; Operand (AND) and consulting instruction (I36)
= M300           ; Action instruction
--------
END              ; Directing Instruction
```

7.2 DIRECTING INSTRUCTIONS

These provide the PLC with information on the type of module and the way it must be executed.

Directing instructions available at the PLC for programming are PRG, PE, CY1, END, L, DEF, IMA, REA, IRD, OWR, MRD, MWR and TRACE.

PRG, PE, CY1: Define the module type.

PRG Main module
CY1 First cycle module
PE Periodic module. This will be executed periodically every time **t** (in milliseconds) indicated in the directing instruction itself. For example: PE1 100 ; This will be executed every 100 milliseconds.

END: Indicates the end of the module. If this is not defined, the PLC understands that this module ends in the last block of the program.

Example of programming using the directing instruction END:

```
CY1        ; Beginning of module CY1
-----
-----
END        ; End of module CY1
PRG        ; Beginning of module PRG
-----
-----
END        ; End of module PRG
PE1 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
-----
-----
PE1 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.

DEF: Definition of symbol. Allows a symbol to be associated with any PLC variable, it being possible to reference this variable throughout the program by means of the variable name or by means of the associated symbol.

Example:

DEF EMERG I1 ;	Assigns the EMERG symbol to input 11, so any reference throughout the program to EMERG will be interpreted by the PLC as a reference to I1.

It is also possible to associate a symbol to any number which can be given in decimal, with or without a sign, or hexadecimal format preceded with the "$" sign.

This option, among other applications, makes programming and later understanding of the PLC program much easier when trying to control the CNC by simulating its keyboard from the PLC program.

Example:

DEF HELP $FFF2	Assigns the symbol "HELP" to the HELP key on the front panel. The word "HELP" is easier to handle throughout the program than the "$FFF2" code.
()= MOVE HELP R101	Assigns the code corresponding to the "HELP" key to register R101. (R101 = $FFF2).
CNCWR (R101, KEY, M101)	Indicates to the CNC that the key whose code ($FFF2) is stored in register R101 (HELP key) has been pressed.

The PLC allows up to 200 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 ", ;".

It is not possible to define duplicated symbols, but it is possible to assign more than one symbol to the same variable.

Example: DEF EMRGOUT O1
DEF SALEMRG O1

The symbols associated to specialized marks and register (M>2047 y R 500) are pre-defined in the PLC and, therefore, it is not necessary to define them, nevertheless and if required, the PLC allows a different symbol to be assigned to them.

REA, IMA: Indicate to the PLC that the consultations defined below will be made on the real (REA) or image (IMA) values of I, O, M resources.

Counters. Timers and Registers do not have image values, so their real values will always be evaluated.

Action instructions (=O32) will always update the real values of PLC resources.

Example:

IMA I1 AND I2 = O1	; Consultations will evaluate Image values.
REA	; Consultations will evaluate Real values.
IMA I3 AND REA M4 = O2	; Evaluates the Image of I3 and the Real of M4
IMA I5 REA = O3	; Evaluates the Image of I5 and the next ones in Real

IRD: Updates the real values of the inputs after reading the physical inputs.

Care must be taken when using this instruction since the current real values of the inputs will be lost. After executing this instruction, the new values will match those of the physical inputs coming from the electrical cabinet.

MRD: Updates the values of resources M5000 thru M5957 and R500 thru R559 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).

OWR: Updates the physical outputs (electrical cabinet) with the current real values of the corresponding **O** resources.

MWR: Updates the logic inputs of the **CNC** (internal variables) with the current real values of resources M5000 thru M5957 and R500 thru R559.

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

PE5
    -----------
    TRACE          ; Data capture
    -----------
END
```

The data capture in the execution of the trace in this program takes place:

- At the beginning of each **PRG** cycle
- Every time the periodic cycle (**PE**) 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.

7.3 CONSULTING INSTRUCTIONS

Consulting instructions allow the PLC to evaluate the status of the different PLC resources (Input, Output, Mark, Timer, Counter) and are divided into:

Simple Consulting Instructions
Flank Detection Consulting Instructions
Comparative 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.

7.3.1 SIMPLE CONSULTING INSTRUCTIONS

These are instructions which test the status of the PLC, inputs, outputs, marks, timers, counters and register bits, returning their logic status.

Example:

I12 ; Will return a 1 if input 12 is active and a 0 if otherwise.

7.3.2 FLANK DETECTION CONSULTING INSTRUCTIONS

They are instructions which verify whether a status change has occurred at the specified input, output or mark.

This comparison may be carried out with the real or image values of the resources and they will be done between their current value and the one which the resources had when the instruction was executed last.

There are two types of Flank Detection Consulting Instructions:

DFU: Detects whether an Up Flank (leading edge), a change of status from 0 to 1 has been produced in the specified variable. It will return a "1" if that is the case.

DFD: Detects whether a Down Flank (trailing edge), a change of status from 1 to 0 has been produced in the specified variable. It will return a "1" if that is the case.

The programming format of the different combinations is:

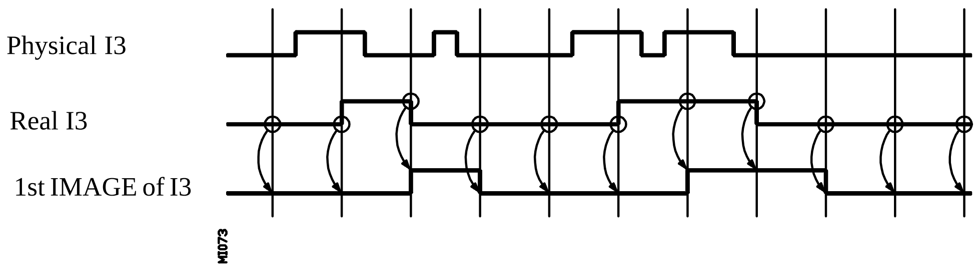
DFU	I 1/256
DFD	O 1/256
	M 1/5957

The consulting instructions to detect the flanks of marks M4000 thru M4127, M4500 thru M4563, M4700 thru M4955 and M5000 thru M5957 will be executed with their real values even when working with image values since these marks have no image values.

Considering that these instructions can evaluate real and image values, the following points must be taken into account:

* The PLC updates the real values of the inputs at the beginning of the cycle, taking the values of the physical inputs.

* The image values of the inputs, outputs and marks are updated after executing the program cycle.



7.3.3 COMPARATIVE CONSULTING INSTRUCTIONS

The PLC has a **CPS** instruction which makes comparisons between:

- The time elapsed in a timer (T).
- The internal count of a counter (C).
- The value of a register (R).
- An integer number within ±2147483647

The different types of comparison which can be made are:

GT (Greater than)	Checks whether the first operand is GREATER than the second one.
GE (Greater equal)	Checks whether the first operand is GREATER than or EQUAL to the second one.
EQ (Equal)	Checks whether the first operand is EQUAL to the second one.
NE (Not equal)	Checks whether the first operand is DIFFERENT from the second one.
LE (Less equal)	Checks whether the first operand is LESS than OR EQUAL to the second one.
LT (Less than)	Checks whether the first operand is LESS than the second one.

The programming format of the different combinations is:

CPS	T 1/256 C 1/256 R 1/559 #	GT GE EQ NE LE LT	T 1/256 C 1/256 R 1/559 #

Where the Registers can be R1/256 or R500/559 and the symbol # represents a defined number in one of the following formats:

Decimal	: Any integer within ±2147483647.
Hexadecimal	: Preceded by the sign $ and between 0 and FFFFFFFF
Binary	: Preceded by the letter B and made up of up to 32 bits (1 or 0).

If the required condition is met, the consulting instruction will return the logic value “1”; otherwise, value “0” is returned.

Programming examples:

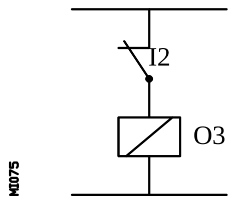
CPS C12 GT R14 = M100	; If the internal count of counter C12 is GREATER than the value of register R14, the PLC will make M100 = 1 and M100=0 in the opposite case.
CPS T2 EQ 100 = TG1 5 2000	; When the time elapsed on the counter T2 is EQUAL to the value of 100, timer T5 will be activated working as a monostable and with a time constant of 2 seconds.

7.4 *OPERATORS*

An operator is a symbol which indicates the logic manipulations which must be made within a Logic Expression, between the different Consulting Instructions. The PLC has the following operators:

NOT Inverts the result of the Consulting Instruction which it precedes.

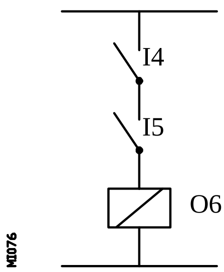
NOT I2 = O3 ; Output O3 will show the negated status of input I2.



I2	O3
0	1
1	0

AND Carries out the logic function “AND” between consulting instructions.

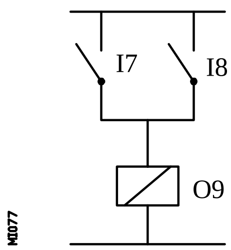
I4 AND I5 = O6 ; Output O6 will show the high logic level when input I4 and input I5 have a high logic level.



I	I5	O6
0	0	0
0	1	0
1	0	0
1	1	1

OR Carries out the logic function “OR” between consulting instructions.

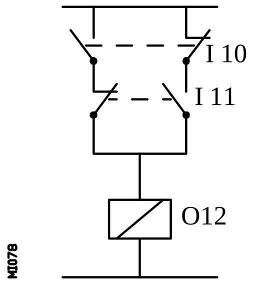
I7 OR I8 = O9; Output O9 will show the high logic level when input I7 or input I8 have a high logic level.



I7	I8	O9
0	0	0
0	1	1
1	0	1
1	1	1

XOR Carries out the logic function "EXCLUSIVE OR" between consulting instructions.

I10 XOR I11 = O12 ; Output O12 will show the high logic level when inputs I10 and I11 have different logic levels.



I10	I11	O12
0	0	0
0	1	1
1	0	1
1	1	0

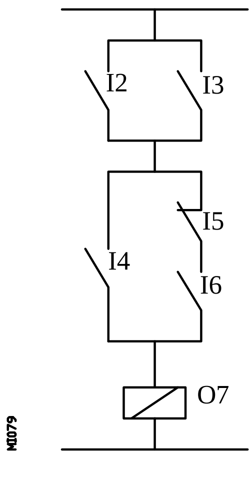
The associativity of all these Operators is left to right and the priorities set by the PLC for their use, classified from highest to lowest, are:

NOT
AND
XOR
OR

Besides, the PLC allows the operators "**(**" and "**)**" to be used for clarifying and selecting the order in which the evaluation of the logic expression is produced.

Example:

(I2 OR I3) AND (I4 OR (NOT I5 AND I6)) = O7



A consulting instruction formed exclusively by the operators "**(**" and "**)**" always has a value of "1", i.e.:

() = O2 ; Output O2 will always show the logic value "1".

7.5 ACTION INSTRUCTIONS

Action Instructions allow the status of PLC resources (I,O,M,R,T,C) to be altered in accordance with the result obtained in the logic Expression.

An executing Instruction is made up of a Logic Expression and one or several Action Instructions, and all Action Instructions must be preceded by the equal symbol (=).

Example:

I2 = O3 = M100 = TG1 2 100 = CPR 1 100

Output O3 and the mark M100 will show the status of input I2, while a leading edge at input I2 will activate the trigger input TG1 of timer T2 and counter C1 will be preset with value 100.

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 status of input I2.

A trailing edge (negated leading edge) at input I2 will activate the trigger input TG1 of timer T2.

A leading edge at input I2 will preset counter C1 with value 100.

Action instructions are divided into:

- Binary Action Instructions
- Sequence Breaking Action Instructions
- Arithmetic Action Instructions
- Logic Action Instructions
- Specific Action Instructions

7.5.1 BINARY ACTION INSTRUCTIONS

Binary Action Instructions are divided into:

Assignment Binary Action Instructions
Conditioned Binary Actions Instructions

7.5.1.1 ASSIGNMENT BINARY ACTION INSTRUCTIONS

This type of binary actions assigns the value obtained in the evaluation of the Logic Expression (0/1) to the specified PLC resource (inputs, outputs, marks, timers, counters and register bit).

Examples:

I3 = TG1 4 100

The PLC assigns the status of input I3 to the trigger input TG1 of timer T4, thereby the leading edge at input I3 will activate trigger input TG1 of timer T4.

(I2 OR I3) AND (I4 OR (NOT I5 AND I6)) = M111

The PLC assigns to Mark M111 the value obtained in the evaluation of the Logic Expression (I2 OR I3) AND (I4 OR (NOT I5 AND I6)) .

7.5.1.2 CONDITIONED BINARY ACTION INSTRUCTIONS

The PLC has 3 Conditioned Binary Action Instructions, **SET**, **RES** and **CPL**, which allow the status of the specified Input, Output or Register Bit to be modified.

The programming format for these is as follows:

SET RES CPL	I 1/256 O 1/256 M 1/5957 B 0/31 R 1/559

Marks can be M1/2047, M4000/4127, M4500/4563, M4700/4955 or M5000/5957 and Registers R1/256 or R500/559

=SET If the result obtained in the evaluation of the logic Expression is a "1" this action assigns a "1" to the Input, Output, Mark or Register Bit specified. If the result is a logic "0", this action will not modify the status of the specified resource.
Example:

CPS T2 EQ 100 = SET B0R100

When the time elapsed in the timer T2 equals 100, bit 0 of register R100 will be set to "1".

=RES If the result obtained in the evaluation of the logic Expression is a "1" this action assigns a "0" to the Input, Output, Mark or Register Bit specified. If the result is a logic "0", this action will not modify the status of the specified resource.
Example:

```
I12 OR NOT I22 = RES M55
               = NOT RES M65
```

When the result of the logic expression I12 OR NOT I22 is a "1", the PLC will set mark M55 to "0" and it will not modify mark M65. On the other hand, if the result is "0", the PLC will not modify the status of mark M55, but it will set mark M65 to "0".

=CPL If the result obtained in the evaluation of the logic Expression is a "1" this action complements the status of the Input, Output, Mark or Register Bit specified. If the result is a logic "0", this action will not modify the status of the specified 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.

7.5.2 *SEQUENCE BREAKING ACTION INSTRUCTIONS*

These actions interrupt the sequence of a program, and its execution continues from another executing instruction indicated by means of a label (L 1/256). This label may be situated before or after the executing instruction in which the action is indicated.

A subroutine is a section of a program which, once is properly labeled, can be called upoN from any executable instruction.

The first instruction of a subroutine will be its label (L1 thru L256) and its last will be the directing instruction END after the last executable instruction of the subroutine.

If END is not programmed as end of subroutine, the PLC will continue executing until it reaches the next END instruction in the program. At this point the PLC will consider the execution of the subroutine “ended”.

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.

= JMP L1/256 Unconditional Jump

If the result obtained in the evaluation of the logic Expression is a “1” this action causes a jump to the specified label, the execution of the program continuing in the executing instruction indicated by this label. If the result is a logic “0”, this action will be ignored by the PLC.

Example:

```
I8 = JMP L12                        ; If I8=1 the program continues in L12
NOT M14 AND NOT B7R120 = 08         ; If I8=1 it is not executed
CPS T2 EQ 2000 = O12                ; If I8=1 it is not executed

L12
(I12 AND I23) OR M54 = 06
```

CAL L 1/256 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 this action instruction has concluded, the PLC will continue with the execution of the next action instruction or executing instruction programmed after the CAL L1/256 command.

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.

Examples:

I2 = CAL L5 = O2

If input I2 has a value of 1 subroutine L5 will be executed and once this has concluded, the PLC will assign the value of input I2 (1) to output O2.

```
PRG
———————————
I9 = CAL L15                  ; If I9=1 it executes subroutine L15
———————————
END                           ; End of main program
L15                           ; Beginning of subroutine L15
———————————
(I12 AND I23) OR M54 = 06
NOT M14 AND NOT B7R120 = 08
CPS T2 EQ 2000 = O12
———————————
END                           ; End of subroutine L15
```

= 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 logic "0", this action will be ignored by the PLC.

If, during the execution of a subroutine, the PLC detects an active RET, it will be considered that the subroutine has finished as this instruction has a treatment similar to the directing instruction END.

If END is not programmed as the end of a subroutine and if no RET is executed, the PLC will continue executing until it reaches the next END instruction in the program. At this point the PLC will consider the execution of the subroutine "ended".

7.5.3 ARITHMETIC ACTION INSTRUCTIONS

The PLC has the following arithmetic Action Instructions: **MOV**, **NGU**, **NGS**, **ADS**, **SBS**, **MLS**, **DVS and MDS**, which allow working with the specified PLC resources.

=MOV Transfers the logic status of the origin indicated to the specified destination.

This transfer will be of 4, 8, 12, 16, 20, 24, 28 or 32 bits.

The Origin or source of information can be expressed in binary code or BCD and can be selected from among:

- I Input group after the one selected.
- O Output group after the one selected.
- M Mark group after the one selected.
- T Elapsed time of the selected timer.
- C Counter value of the selected counter.
- R Value of the selected register.
- # Number selected in decimal, hexadecimal or binary format.

The Destination or place where the transmitted information is placed can be expressed in binary code or in BCD and can be selected from among:

- I Input group after the one selected.
- O Output group after the one selected.
- M Mark group after the one selected.
- R Value of the selected register.

Their programming format is:

	Source	Destination	Source Code	Destination Code	No.bits to transmit
MOV	I 1/256 O 1/256 M 1/5957 T 1/256 C 1/256 R 1/559 #	I 1/256 O 1/256 M 1/5957 R 1/559	0(Bin) 1(BCD)	0(Bin) 1(BCD)	32 28 24 20 16 12 8

Marks can be M1/2047, M4000/4127, M4500/4563, M4700/4955 or M5000/5957 and Registers R1/256 or R500/559

The origin and destination codes as well as the number of bits to be transmitted must be defined always, except when it is required to transmit Bin to Bin and in 32 bits (0032) in which case it will not be required to program them.

Examples:

```
MOV I12 M100 0032      ; From Binary to Binary in 32 bits
MOV O21 R100 0012      ; From Binary to Binary in 12 bits
MOV C22 O23  0108      ; From Binary to BCD in  8 bits
MOV T10 M112 1020      ; From BCD to Binary in 20 bits
```

It should be borne in mind that when doing a conversion from binary (origin) to BCD (destination), the number of bits of the new calculated value may have more bits than those selected for transmission. If this happens, the PLC will truncate the value of the destination, ignoring the most significant digits.

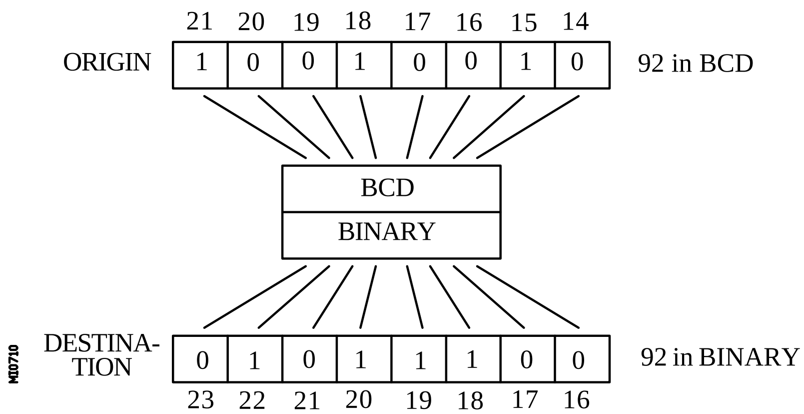
With 4 bits the maximum convertible value in BCD will be 9
With 8 bits the maximum convertible value in BCD will be 99
With 12 bits the maximum convertible value in BCD will be 999
With 16 bits the maximum convertible value in BCD will be 9999
With 20 bits the maximum convertible value in BCD will be 99999
With 24 bits the maximum convertible value in BCD will be 999999
With 28 bits the maximum convertible value in BCD will be 9999999
With 32 bits the maximum convertible value in BCD will be 99999999

In order to avoid the loss of these digits, it is suggested that the transfer be made increasing the number of bits, using intermediate registers or marks if necessary.

Example:

I11 = MOV I14 O16 108

If input I11 has a value of "1" the PLC transfers the logic states of input I14 and the next 7 inputs in BCD code, to the 8 outputs starting from O16, in binary code.



=NGU R 1/256 or **=NGU R 500/559** Negation Unsigned of all the bits in a Register.

If the result obtained in the evaluation of the logic Expression is a “1” this action negates the 32 bits of the specified register (changes the status of each bit).

Example:

I15 = NGU R152

If the input I15 has a value of “1” the PLC negates the 32 bits of register R152.

If register R152 is:

0	0	1	1	0	0	0	0	1	1	0	0	1	1	0	0	0	1	0	0	0	1	1	0	1	1	0	1	0	1	0	0

MI0711

after negating it, the result is:

1	1	0	0	1	1	1	1	0	0	1	1	0	0	1	1	1	0	1	1	1	0	0	1	0	0	1	0	1	0	1	1

MI0712

=NGS R 1/256 or **=NGS R 500/559** Changes the sign of the contents of a Register.

If the result obtained in the evaluation of the logic Expression is a “1” this action changes the sign to the contents of the specified register.

Example:

I16 = NGS R89

If the input I16 has a value of “1” the PLC changes the sign of the contents of register R89.

If register R89 is:

0	0	1	1	0	0	0	0	1	1	0	0	1	1	0	0	0	1	0	0	0	1	1	0	1	1	0	1	0	1	0	0

R 89=+818693844

MI0713

after negating it, it follows:

1	1	0	0	1	1	1	1	0	0	1	1	0	0	1	1	1	0	1	1	1	0	0	1	0	0	1	0	1	1	0	0

R 89=-818693844

MI0714

=ADS, =SBS, =MLS, =DVS, =MDS

If the result obtained in the evaluation of the logic Expression is a "1", these actions allow addition (ADS), subtraction (SBS), multiplication (MLS), division (DVS) and module (MDS) or remainder of a division, operations to be made between the contents of registers or between the contents of registers and a number. The result will always be placed in a specified register.

Their programming format is:

"Operation type" "1st operand" "2nd operand" "destination register".

The type of operation will be ADS, SBS, MLS, DVS or MDS.

Registers (R1/256 and R500/559) or numbers expressed in decimal, hexadecimal or binary formats can be defined as first and second operands.

The destination register indicates where the result of the operation will be deposited and will be defined by means of a register (R1/256 or R500/559).

Examples:

If registers R100 = 1234 and R101 = 100

M2047	; Is always met	
= 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

M2047	; Is always met	
= ADS 1563 R101 R112	; R112 = 1563 + 100	= 1663
= SBS R100 1010 R113	; R113 = 1234 - 1010	= 224
= MLS 1563 1000 R114	; R114 = 1563 x 1000	= 1563000
= DVS R100 1000 R115	; R115 = 1234 : 1000	= 1
= MDS 8765 1000 R116	; R116 = 8765 MOD 1000	= 765

Atention:

If a division by "0" is performed in the DVS operation, the CNC stops the execution of the PLC program and it displays the corrresponding error message.

7.5.4 *LOGIC ACTION INSTRUCTIONS*

The PLC has the following logic Action Instructions, **AND**, **OR**, **XOR**, **RR** and **RL**.

=AND, =OR, =XOR

If the result obtained in the evaluation of the logic Expression is a “1”, these actions allow the logic operations AND, OR and XOR to be made bit by bit between the contents of the register and the number. The result will always be placed in a specified register.

Their programming format is:

“Type of operation” “1st operand” “2nd operand” “destination register”.

The type of operation will be AND, OR or XOR.

Registers (R1/256 and R500/559) 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 (R1/256 or R500/559).

The mark M2003 is called Zero flag and indicates whether the result of an AND, OR, XOR, operation equals zero, in which case it follows that M2003=1.

Examples:

If registers R200 and R201 have the value of:

```
R200=B10010010
R201=B01000101
```

```
M2047                    ; Is always met
= AND R200 R201 R202     ; R202 = B0           M2003=1
= OR  R200 R201 R203     ; R203 = B11010111    M2003=0
= XOR R200 R201 R204     ; R204 = B11010111    M2003=0

M2047                    ; Is always met
= AND B1111 R201  R205   ; R205 = B00000101    M2003=0
= OR  R200  B1111 R206   ; R206 = B10011111    M2003=0
= XOR B1010 B1110 R207   ; R207 = B00000100    M2003=0
```

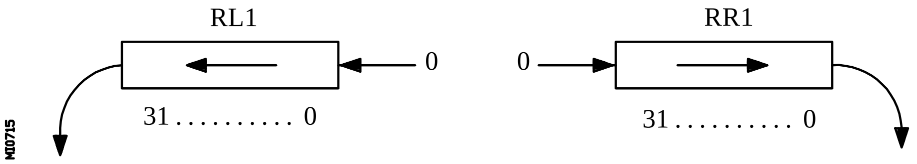
=RR, =RL Rotation of registers

If the result obtained in the evaluation of the logic Expression is a "1", these actions allow the register to be rotated.

It is possible to rotate to the right (RR) or to the left (RL) and there are two types of rotations: type 1 (RR1 or RL1) and type 2 (RR2 or RL2).

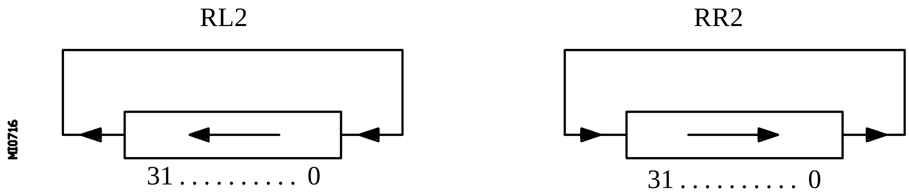
Type 1 rotation (RL1 or RR1):

This type of rotation enters a 0 in the least significant bit (RL1) or in the most significant bit (RR1), by shifting the remaining bits in the register. The value of the last bit disappears.



Type 2 rotation (RL2 or RR2):

A circular rotation of the register is made, i.e., the most significant bit becomes the new value of the least significant bit (RL2) or the least significant bit becomes the new value of the most significant bit (RR2).



Their programming format is:

"Type of operation" "origin" "No. repetitions" "destination"

The type of operation will be RR1, RR2, RL1 or RL2.

Both the origin and destination will be registers (R1/256 and R500/559). If the origin and destination registers coincide, it will be necessary to define both.

The number of repetitions will indicate the number of times the register will be rotated.

Examples:

RR1 R100 1 R200 ; 1 type 1 rotation to the right of the contents of R100 leaving the result in R200

RL2 R102 4 R101 ; 4 type 2 rotations to the left of the contents of R102 leaving the result in R101

If the contents of R17 is:

0	0	1	1	0	0	0	0	1	1	0	0	1	1	0	0	0	1	0	0	0	1	1	0	1	1	0	1	0	1	0	0	R 17

MI0717

and M2047 = RL2 R17 4 R20 is executed, the result is:

0	0	1	1	0	0	0	0	1	1	0	0	1	1	0	0	0	1	0	0	0	1	1	0	1	1	0	1	0	1	0	0	R 17

0	0	0	0	1	1	0	0	1	1	0	0	0	1	0	0	0	1	1	0	1	1	0	1	0	1	0	0	0	0	1	1	R 20

MI0718

7.5.5 SPECIFIC ACTION INSTRUCTIONS

=ERA group erase

If the result obtained in the evaluation of the logic Expression is a “1”, this action allows a group of inputs, outputs, marks or registers to be erased, or to initialize the status of a group of timers or counters.

If a group of inputs, outputs, marks or registers is erased; the PLC will assign the value 0 to the specified variables.

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.

Their programming format is:

ERA	I 1/256 O 1/256 M 1/5957 T 1/256 C 1/256 R 1/559	1/256 1/256 1/5957 1/256 1/256 1/559

The Marks can be M1/2047, M4000/4127, M4500/4563, M4700/4955 or M5000/5957 and Registers R1/256 or R500/559

This action is especially appropriate for execution in the first cycle module (CY1) with the aim of setting the required resources in the initial working conditions.

Examples:

I12 = ERA O5 12

If input I12 has a value of “1” the PLC will set to 0 outputs O5 thru O12.

I23 = ERA C15 18

If input I23 has a value of “1” the PLC will preset counters C15 thru C18 to 0.

=CNCRD, =CNCWR Access to internal CNC variables.

These Action Instructions allow reading (CNCRD) and writing (CNCWR) internal CNC variables, their programming format being:

CNCRD (Variable, Register, Mark)
CNCWR (Register, Variable, Mark)

If the result obtained in the logic expression is a “1”, the CNCRD action loads the contents of the indicated Variable into the selected Register, and the CNCWR action loads the contents of the indicated Register into the selected Variable.

The internal CNC variables which can be accessed by means of these Action Instructions are detailed in the chapter corresponding to CNC-PLC communications.

If information is requested about a non-existing variable (for example, the coordinates of an axis which is not used), these actions will not modify the contents of the indicated register and will assign a 1 to the selected mark, indicating in this way that the reading of a non-existing variable was requested.

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)

Presets the timer enabled by the PLC with the value contained in register R92.

= PAR Register Parity

If the result of the logic expression is a “1”, this action allows to check the type of parity of a register.

Its programming format is:

PAR Register Mark

If the register being checked has an EVEN parity, this instruction will set the indicated mark to “1” and if its parity is ODD, it will set it to “0”.

Example:

I15 = PAR R123 M222

If I15	= 1 and parity of R123	= EVEN, then M222	= 1
If I15	= 1 and parity of R123	= ODD, then M222	= 0

7.6 SUMMARY OF PLC PROGRAMMING INSTRUCTIONS

AVAILABLE PLC RESOURCES

Inputs:		**I** 1/256
Outputs:		**O** 1/256
Marks (user):		**M** 1/2000
	arithmetic flags:	**M** 2001
	clocks:	**M** 2009/2024
	set logic level:	**M** 2046/2047
	associated to messages:	**M** 4000/4127
	associated to errors:	**M** 4500/4563
	for screens or pages:	**M** 4700/4955
	to communicate with CNC:	**M** 5000/5957
Timers:		**T** 1/255
Counters:		
		C 1/255
User	registers	**R** 1/255
Registers	to communicate with CNC:	**R** 500/559

The value stored in each register will be considered by the PLC as signed integer which could be referred to in the following formats:

Decimal : Any integer number within ±2147483647.
Hexadecimal : Preceded by the symbol $ and between 0 and FFFFFFFF
Binary : Preceded by the letter B and consisting 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 outputs of the **CNC**.
OWR	Updates the physical outputs with the current real values of the corresponding **O** resources.
MWR	Updates the logic inputs of the CNC (internal variables) with the current real values of resources **M5000/5957** and **R500/559**.
TRACE	It captures data for the Logic Analyzer during the execution of a PLC cycle.

SIMPLE CONSULTING INSTRUCTIONS

I 1/256	Inputs
O 1/256	Outputs
M 1/5957	Marks
T 1/255	Timers
C 1/255	Counters
B 0/31 **R** 1/255	Register bit

FLANK DETECTING INSTRUCTIONS

DFU Up flank detection.
DFD Down flank detection.

DFU DFD	I 1/256 O 1/256 M 1/5957

COMPARATIVE INSTRUCTIONS

CPS allows comparisons.

CPS	T 1/256 C 1/256 R 1/559 #	GT GE EQ NE LE LT	T 1/256 C 1/256 R 1/559 #

OPERATORS

NOT Inverts the result of the consulting instruction it precedes.
AND Performs the logic function “AND” between consulting instructions.
OR Performs the logic function “OR” between consulting instructions.
XOR Performs the logic function “EXCLUSIVE OR” between consulting instructions.

ACTION INSTRUCTIONS FOR BINARY ASSIGNMENT

= **I** 1/256	Inputs
= **O** 1/256	Outputs
= **M** 1/5957	Marks
= **TEN** 1/256	Timers
= **TRS** 1/256	
= **TGn** 1/256 n/R	
= **CUP** 1/256	Counters
= **CDW** 1/256	
= **CEN** 1/256	
= **CPR** 1/256 n/R	
= **B** 0/31 **R** 1/256	Register Bits

CONDITIONED BINARY ACTION INSTRUCTIONS

= SET If the logic expression is “1”, this action assigns a “1” to the resource. If the logic expression is “0”, this action does not change the logic state of the resource.
= RES If the logic expression is “1”, this action assigns a “0” to the resource. If the logic expression is “0”, this action does not change the logic state of the resource.
= CPL If the logic expression is “1”, this action complements the logic state of the resource.

SET RES CPL	I 1/256 O 1/256 M 1/5957 B 0/31 R 1/55

JUMP ACTION INSTRUCTIONS

= JMP L 1/256 Unconditional Jump.
= RET Return or End of Subroutine.
= CAL L 1/256 Call a Subroutine.

ARITHMETIC ACTION INSTRUCTIONS

= MOV Transfers the logic states of the indicated source to the indicated destination.

	Source	Destination	Source Code	Destination Code	No.bits to transmit
MOV	I 1/256 O 1/256 M 1/5957 T 1/256 C 1/256 R 1/559 #	I 1/256 O 1/256 M 1/5957 R 1/559	0(Bin) 1(BCD)	0(Bin) 1(BCD)	32 28 24 20 16 12 8 4

= NGU R 1/559 Complements all register bits.
= NGS R 1/559 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.

ADS SBS MLS DVS MDS	R1/559 #	R1/599 #	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.

AND OR XOR	R1/559 #	R1/599 #	R1/559

= RR 1/2 Right-hand register rotation.
= RL 1/2 Left-hand register rotation.

RR1 RR2 RL1 RL2	R1/559	R1/559 0/31	R1/559

SPECIFIC ACTION INSTRUCTIONS

= ERA Group erase

ERA	I 1/25 O 1/256 M 1/5957 T 1/256 C 1/256 R 1/559	1/256 1/256 1/5957 1/256 1/256 1/559

= CNCRD Read internal CNC variables.

CNCRD (Variable, R1/559, M1/4955)

= CNCWR Write (modify) internal CNC variables.

CNCWR (R1/559, Variable, M1/4955)

= PAR Register Parity

PAR R1/559 M1/5957

8. *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.

8.1 AUXILIARY M, S, T FUNCTIONS

MBCD1 (R550)
MBCD2 (R551)
MBCD3 (R552)
MBCD4 (R553)
MBCD5 (R554)
MBCD6 (R555)
MBCD7 (R556)

The CNC tells the PLC by means of these 32 bit registers, the miscellaneous M functions programmed in the block being executed.

If there are less than 7 miscellaneous M functions in each block, the CNC will send the information in the lower-numbered registers, assigning the value $FFFFFFFF to those which are left free.

Each of these registers contains a coded miscellaneous M function in BCD format (8 digits).

M1234	0000	0000	0000	0000	**0001**	**0010**	**0011**	**0100**	
									LSB

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) = $FFFFFFFF
MBCD5 (R554) = $FFFFFFFF
MBCD6 (R555) = $FFFFFFFF
MBCD7 (R556) = $FFFFFFFF

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:

* Check all MBCD registers one by one until the specific "M" function is found or until one of them contains the $FFFFFFFF value.

* Use the "MBCD*" format which permits checking all MBCD registers at the same time.

  For example, to Detect M30:

  CPS MBCD* EQ $30 = ...   If detected, it will return a "1" 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. (spindle machine parameter "SPDLTYPE).

The auxiliary S function will always be executed at the beginning of the block and the CNC will wait for the general logic input AUXEND to be activated to consider the execution completed.

If S output in 2-digit BCD is used the CNC will tell the PLC, by means of this register the selected spindle speed according to the following conversion table:

Programmed S	"SBCD"	Programmed S	"SBCD"
0	S00	180-199	S65
1	S20	200-223	S66
2	S26	224-249	S67
3	S29	250-279	S68
4	S32	280-314	S69
5	S34	315-354	S70
6	S35	355-399	S71
7	S36	400-449	S72
8	S38	450-499	S73
9	S39	500-559	S74
10-11	S40	560-629	S75
12	S41	630-709	S76
13	S42	710-799	S77
14-15	S43	800-899	S78
16-17	S44	900-999	S79
18-19	S45	1000-1119	S80
20-22	S46	1120-1249	S81
23-24	S47	1250-1399	S82
25-27	S48	1400-1599	S83
28-31	S49	1600-1799	S84
32-35	S50	1800-1999	S85
36-39	S51	2000-2239	S86
40-44	S52	2240-2499	S87
45-49	S53	2500-2799	S88
50-55	S54	2800-3149	S89
56-62	S55	3150-3549	S90
63-70	S56	3550-3999	S91
71-79	S57	4000-4499	S92
80-89	S58	4500-4999	S93
90-99	S59	5000-5599	S94
100-111	S60	5600-6299	S95
112-124	S61	6300-7099	S96
125-139	S62	7100-7999	S97
140-159	S63	8000-8999	S98
160-179	S64	9000-9999	S99

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.

S12345.678	0001	0010	0011	0100	0101	0110	0111	1000	LSB

If no S has been programmed in the block, the CNC will assign a value of $FFFFFFFF to this register.

TBCD (R558)

The CNC tells the PLC by means of this 32-bit register, the pocket number in the magazine where the selected tool is. If the general machine parameter "RANDOMTC" 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).

T123	0000	0000	0000	0000	0000	0001	0010	0011	LSB

If no T has been programmed in the block, the CNC will assign a value of $FFFFFFFF to this register.

The T function will always be executed at the beginning of the block and the CNC will wait for the general logic input AUXEND to be activated to consider the execution completed.

T2BCD (R559)

This register is used when a special tool change has been made (family code $\geq$200) or with machining centers with a non-random tool magazine (general machine parameter “RANDOMTC”).

The CNC tells the PLC by means of the 32 bit register, the position of the magazine (empty pocket) in which the tool which was on the spindle must be deposited.

This will be coded in BCD code (8 digits). If a second T function is not required the CNC will assign a value $FFFFFFFF to the register.

The second T function will be sent together with M06 and the CNC will wait for the general logic input AUXEND to the activated to consider the execution completed.

8.1.1 TRANSFERRING AUXILIARY M, S, T FUNCTIONS

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:

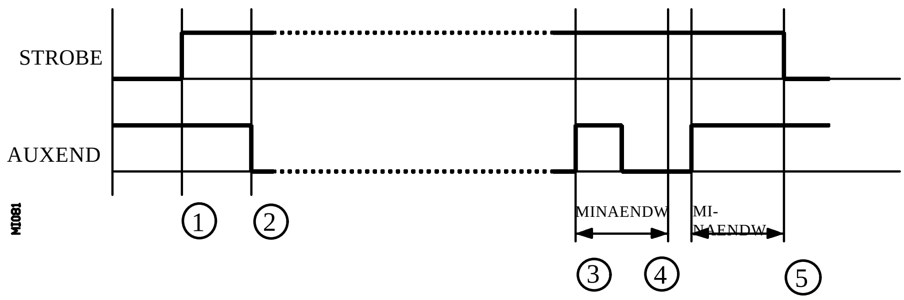
If this involves changing a special tool or a machining center with non-random tool magazine, the CNC will indicate, on executing the M06 function, the position of the magazine (empty pocket) in which the tool which was on the spindle must be deposited.

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.

8.1.1.1 TRANSFERRING M, S, T USING THE AUXEND SIGNAL

1.- Once the block has been analyzed and after sending the corresponding values in the "MBCD1-7", "SBCD", "TBCD" and "T2BCD" variables, the CNC will tell the PLC by means of the general logic outputs "MSTROBE", "SSTROBE", "TSTROBE" and "T2STROBE" that the required auxiliary functions must be executed.



2.- When the PLC detects the activation of one of the STROBE signals, it must deactivate the general logic output "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 logic outputs "MSTROBE", "SSTROBE", "TSTROBE", "T2STROBE" and the variables "MBCD1-7", "SBCD", "TBCD" and "T2BCD".

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 general machine parameter "MINAENDW".

In this way erroneous interpretations of this signal by the CNC are avoided in the case of malfunctions caused by an incorrect logic in the PLC program.

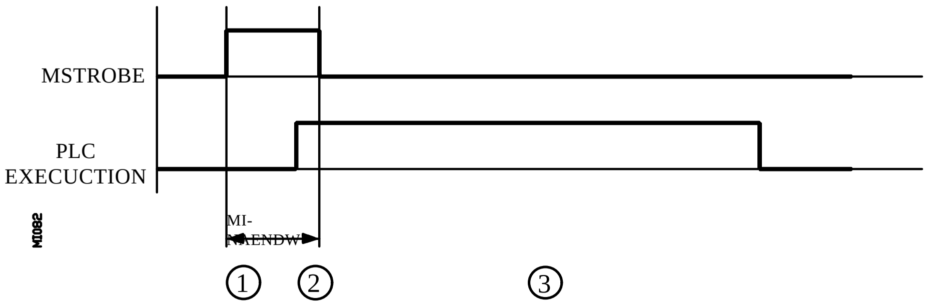
5.- Once the period of time "MINAENDW" has elapsed with the general input "AUXEND" at a high logic level, the CNC will deactivate the general logic outputs "MSTROBE", "SSTROBE", "TSTROBE", "T2STROBE" to tell the PLC that the execution of the required auxiliary function or functions has been completed.

Atention:

When the block being executed has several auxiliary functions (M, S, T), the CNC waits a time period set by general machine parameter "MINAENDW" between two consecutive transfers.

8.1.1.2 TRANSFERRING THE AUXILIARY (MISCELLANEOUS) M FUNCTION 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 general machine parameter "MINAENDW".

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.

8.2 DISPLAYING MESSAGES, ERRORS AND PAGES ON THE CNC

The PLC has a series of marks which allow messages and errors to be displayed in the CNC, as well as displaying screens which have been defined previously by the user.

Displaying messages

The PLC has 128 marks, with their corresponding mnemonic for displaying messages in the CNC.

M4000	MSG1
M4001	MSG2
M4002	MSG3
——	——
——	——
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: When changing the status of input I10 (from 0 to 1), messages MSG1 and MSG2 are activated. If, by maintaining I10=1 the user erases the messages from the keyboard, the following may occur in the next PLC cycle:

```
DFU  I10  =  MSG1        ;  The message will be erased.
     I10  =  MSG2        ;  The message will be reactivated.
```

Displaying errors

The PLC has 64 marks, with their corresponding mnemonic, for displaying errors in the CNC.

M4500	ERR1
M4501	ERR2
M4502	ERR3
———	———
———	———-
M4561	ERR62
M4562	ERR63
M4563	ERR64

If one of these marks is activated (high logic level), the CNC part program will be stopped, also displaying the selected error and associated text in the center 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 pages

The PLC has 256 marks with their corresponding mnemonic, for displaying pages on the CNC.

M4700	PIC0
M4701	PIC1
M4702	PIC2
———	———
———	———-
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) page 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.

8.3 ACCESS FROM THE CNC TO THE PLC PROGRAM AND RESOURCES

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.

8.4 ACCESS FROM A COMPUTER, VIA DNC, TO PLC RESOURCES

The FAGOR 8050 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.).
Etc.

The DNC manual can be applied for from the Commercial Department of FAGOR AUTOMATION S. COOP.

9. CNC LOGIC INPUTS AND OUTPUTS

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 FAGOR 8050 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 logic CNC inputs and outputs can be referred to by means of the corresponding PLC resource or by means of their associated Mnemonic. For example:

M5000	/EMERGEN
M5016	AUXEND
M5104	MIRROR1
M5507	/ALARM

Mnemonics which begin with "/" indicate that the signal is active low (0 V.).

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

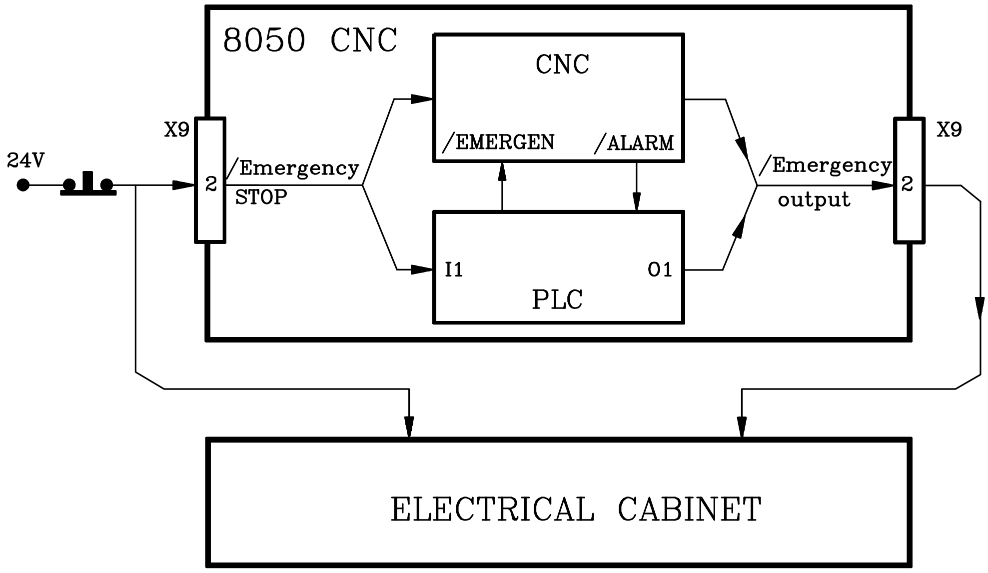
CNC logic inputs and outputs can be grouped in:

General logic inputs.
Key inhibiting logic inputs.
Axis logic inputs.
Spindle logic inputs.
General logic outputs.
Axis logic outputs.
Spindle logic outputs.

9.1 GENERAL LOGIC INPUTS

/EMERGEN (M5000) (**EMERGEN**cy stop)

There are two ways to cause an emergency at the CNC, by activating the physical input EMERGENCY STOP or by activating 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 (other conditions) = /EMERGEN
If the external emergency input is activated or any other emergency occurs, general CNC logic input "/EMERGEN" must be activated. When there is no emergency, this signal must be high (24V).

<u>Atention:</u>

This input must always be defined in the PLC program.

/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.

Atention:

This input must always be defined in the PLC program.

/FEEDHOL **(M5002)** (**FEED HOL**d)

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.

Atention:

This input must always be defined in the PLC program.

/XFERINH **(M5003)** (**XFER INH**ibit)

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.

Atention:

This input must always be defined in the PLC program.

CYSTART (M5007) (CYcle START)

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 (other conditions) = CYSTART

When pressing the Cycle Start key, the CNC activates general logic output "START". The PLC must check that the "other conditions" (hydraulic, safety, etc.) are also met before setting general logic input "CYSTART" high so the program starts running.

Atention:

This input must always be defined in the PLC program.

SBLOCK (M5008) (Single BLOCK)

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) (MANual RAPID)

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.

OVRCAN (M5010) (OVeRride CANcel)

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) (LATCH Manual)

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.

RESETIN (M5015) (RESET IN)

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.

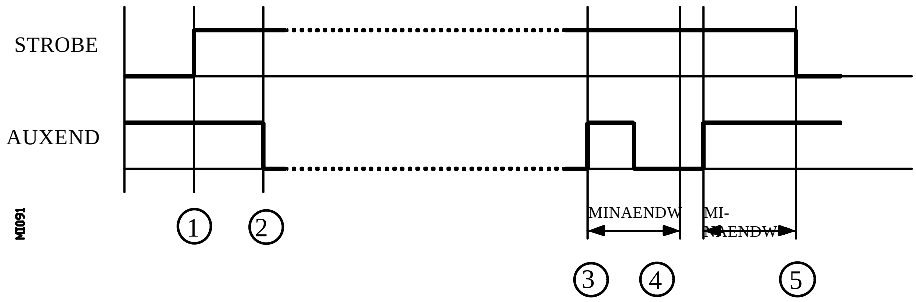
The treatment received by this signal is similar to that given to the RESET key on the Front Panel.

AUXEND (M5016) (AUXiliar **END)**

This signal is used in the execution of auxiliary functions M, S and T, to tell the CNC that the PLC is executing them.

It operates in the following way:

1.- Once the block has been analyzed and after sending the corresponding values in the variables “MBCD1-7”, “SBCD”, “TBCD” and “T2BCD”, the CNC will tell the PLC by means of the general logic outputs “MSTROBE”, “SSTROBE”, “TSTROBE” and “T2STROBE” that the required auxiliary functions must be executed.



2.- When the PLC detects that one of the STROBE signals is active, it must deactivate the general logic input “AUXEND” to tell the CNC that the execution of the corresponding function or functions is starting.

3.- The PLC will execute all the auxiliary functions required, it being necessary to analyze the “MSTROBE”, “SSTROBE”, “TSTROBE”, “T2STROBE” general logic outputs and the “MBCD1-7”, “SBCD”, “TBCD” and “T2BCD” variables in order to do this.

Once execution has been completed, the PLC must activate the general logic input “AUXEND” to tell the CNC that the treatment of the required functions has been completed.

4.- Once the general input “AUXEND” has been activated, the CNC will require that this signal be kept active for a period of time greater than that defined by means of the general machine parameter “MINAENDW”.

In this way, incorrect interpretations of this signal by the CNC are avoided when malfunctions are produced by an incorrect logic in the PLC program.

5.- Once the “MINAENDW” time has elapsed, with the general input “AUXEND” at a high logic level, the CNC will deactivate the general logic outputs “MSTROBE”, “SSTROBE”, “TSTROBE”, “T2STROBE” in order to tell the PLC that the execution of the required auxiliary function or functions has been completed.

TIMERON (M5017) (TIMER ON)

The CNC is provided with a timer which can be enabled and disabled. By means of this logic CNC input, it will be enabled (timing) when the PLC sets the signal TIMERON at a high logic level.

This general purpose timer can be accessed by means of the internal variable TIMER. An application of this timer is to monitor tool life.

TREJECT (M5018) (Tool REJECT)

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) (PANEL OFF)

The PLC sets this signal high in order to tell the CNC that the front panel keyboard (MONITOR/KEYBOARD) and the keyboard of the CONTROL 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 wil 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.

POINT (M5020) (POINT)

This signal is used by the CNC when digitizing point to point while tracing by hand.

Every time an up-flank (leading edge or transition from logic low to high) is detected, the CNC generates a new program point.

PLCABORT (M5022) (PLC ABORT)

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.

The PLC will order to move the "C" axis by 1 meter every time the "C+" button is pressed, but stopping it when this key is released.

```
DEF CPLUS I2                                 ; Define I2 as push-button "C+"
DFU CPLUS = CNCEX (G91 G1 C1000 F3000, M1)   ; Move 1000mm when button is pressed
DFD CPLUS = SET PLCABORT                     ; Stop when button is released.
```

On power-up, the CNC sets this mark to "0".

PLCREADY (M5023) (PLC READY)

This mark indicates the PLC status.

PLCREADY = 0 PLC stopped
PLCREADY = 1 PLC in execution

If this mark is set to 0. the PLC program will stop.

This mark MUST be set to 1 so the CNC allows the spindle and/or the axes to be moved. Otherwise, it will issue the corresponding error message.

INT1 (M5024)
INT2 (M5025)
INT3 (M5026)
INT4 (M5027)

The PLC sets one of these signals to logic state "1" to "tell" the CNC to interrupt the execution of the currently running program and jump to execute the interruption subroutine whose number is indicated in the general machine parameter "INT1SUB" (P35), "INT2SUB" (P36), "INT3SUB" (P37) or "INT4SUB" (P38) respectively.

All these inputs have the same priority and are active by level (not by flank or edge). Only the first one being detected high ("1") will be attended to.

The status of these signals 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) (BLocK SKIP1)

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) (BLocK SKIP2)

The PLC sets this signal at a high logic level to tell the CNC that the block skip condition “/2” is met, therefore, the blocks which have this block skip condition will not be executed.

BLKSKIP3 (M5030) (BLocK SKIP3)

The PLC sets this signal at a high logic level to tell the CNC that the block skip condition “/3 is met, therefore, the blocks which have this block skip condition will not be executed.

M01STOP (M5031) (M01 STOP)

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.

9.2 AXIS LOGIC INPUTS

There are 6 groups of logic inputs (LIMIT, DECEL, etc.) which refer to the 6 possible axes of the machine by means of digits 1 through 6 (LIMIT+2, DECEL1,etc.)

These numbers **have nothing to do** with the values assigned to the general machine parameters "AXIS1" through "AXIS8".

These variables are numbered according to the logic order of the axes.

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:

Variables: LIMIT+1, LIMIT-1, DECEL1, etc. correspond to the X axis.
Variables: LIMIT+2, LIMIT-2, DECEL2, etc. correspond to the Y axis.
Variables: LIMIT+3, LIMIT-3, DECEL3, etc. correspond to the Z axis.
Variables: LIMIT+4, LIMIT-4, DECEL4, etc. correspond to the U axis.
Variables: LIMIT+5, LIMIT-5, DECEL5, etc. correspond to the B axis.
Variables: LIMIT+6, LIMIT-6, DECEL6, etc. correspond to the C axis.

LIMIT+1 and **LIMIT-1**	**(M5100)** and **(M5101)**
LIMIT+2 and **LIMIT-2**	**(M5150)** and **(M5151)**
LIMIT+3 and **LIMIT-3**	**(M5200)** and **(M5201)**
LIMIT+4 and **LIMIT-4**	**(M5250)** and **(M5251)**
LIMIT+5 and **LIMIT-5**	**(M5300)** and **(M5301)**
LIMIT+6 and **LIMIT-6**	**(M5350)** and **(M5351)**

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 the screen.

In Manual (JOG) Operating Mode the axis which has overrun its range of travel can be moved in the correct direction in order to place it within the correct range of travel.

DECEL1	**(M5102)**
DECEL2	**(M5152)**
DECEL3	**(M5202)**
DECEL4	**(M5252)**
DECEL5	**(M5302)**
DECEL6	**(M5352)**

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 axis machine parameter "REFEED1", with the slow feedrate indicated by the axis machine parameter "REFEED2". After decelerating it accepts the following reference signal from the corresponding axis feedback system as being valid.

INHIBIT1	**(M5103)**
INHIBIT2	**(M5153)**
INHIBIT3	**(M5203)**
INHIBIT4	**(M5253)**
INHIBIT5	**(M5303)**
INHIBIT6	**(M5353)**

The PLC sets one of these signals at a high logic level in order to tell the CNC to prevent any movement of the corresponding axis. This movement will continue when the PLC sets this signal at the low logic level once more.

If the inhibited axis is moving together with other axes, all these stop moving until the signal returns to the low logic level.

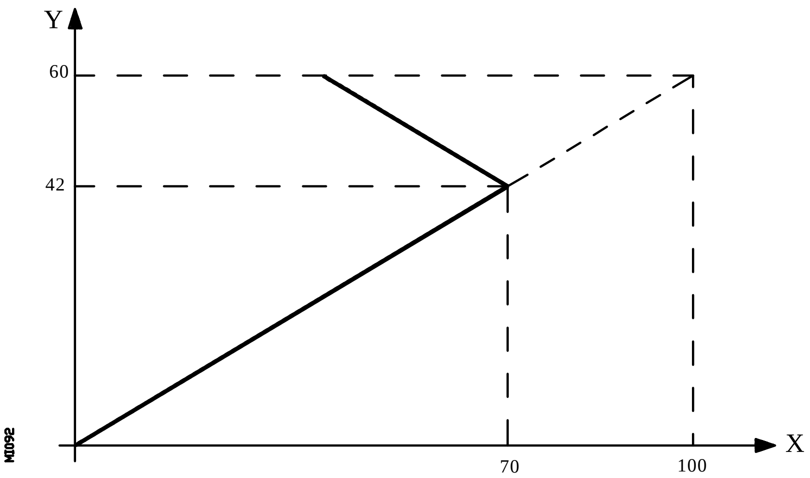
MIRROR1 (M5104)
MIRROR2 (M5154)
MIRROR3 (M5204)
MIRROR4 (M5254)
MIRROR5 (M5304)
MIRROR6 (M5354)

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.

Example:

```
N00   G01   X0     Y0    F1000
N10   G01   X70    Y42
N20   G01   X100   Y60
N30   M30
```



If, when executing the programmed movement in block N20 the signal corresponding to the X axis "MIRROR1" is active, the CNC will apply mirror image to the remaining movement in X.

This way, the new end of travel point will be X40 Y60.

By means of the activation of these signals, symmetrical parts can be executed by using a single program, for example, soles of shoes.

In order to obtain the same effect as functions G11, G12, G13 and G14, it is necessary for the corresponding axis or axes to be positioned at part zero when these signals are activated.

DRO1 M5106
DRO2 M5156
DRO3 M5206
DRO4 M5256
DRO5 M5306
DRO6 M5356

These inputs, together with the corresponding "SERVOON" inputs make it possible to operate with the axes as DRO.

In order for the axis to work in DRO mode, its **"DRO"** input must be **high** and its corresponding "**SERVOON**" input must be **low**.

When an axis works as a DRO, the positioning loop is open and its following error is ignored while in motion

If the DRO signal is brought back low, the axis will no longer behave as a DRO and the CNC will take as position value its current position assigning a 0 value to the following error.

SERVO1ON (M5107)
SERVO2ON (M5157)
SERVO3ON (M5207)
SERVO4ON (M5257)
SERVO5ON (M5307)
SERVO6ON (M5357)

When one of these signals is set high, the CNC closes the positioning loop of the corresponding axis.

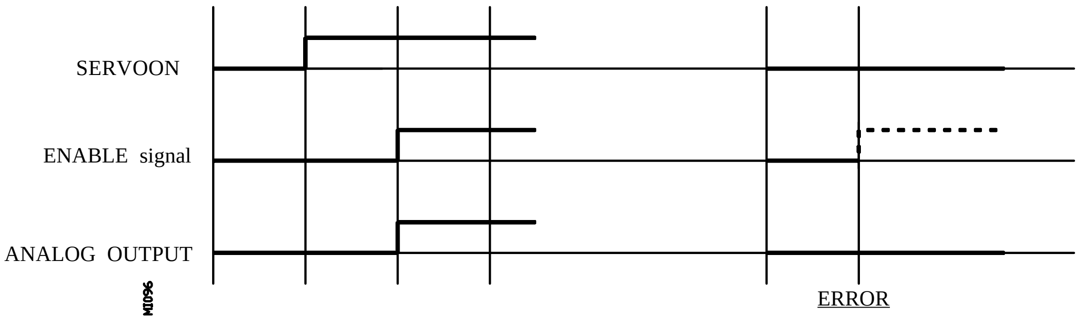
When the signal is set back low, the CNC opens the positioning loop for the axis, but it keeps track of the following error and when the signal comes back up, the CNC moves the axis to correct for the accumulated following error and gets back in position.

These signals are controlled by the PLC and when the positioning loop is tobe closed, they will be processed by the CNC according to the value given to machine parameter “DWELL” for the axes.

DWELL = 0

When this parameter 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.



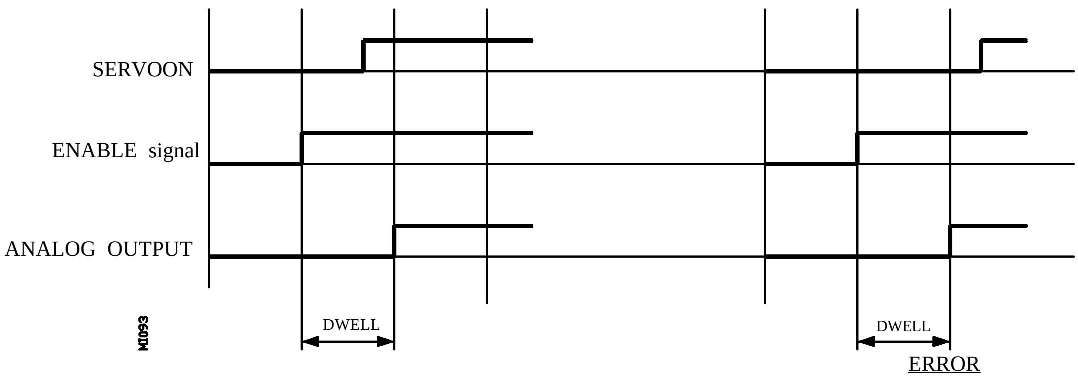
On the other hand, if the SERVOON signal is low or if it changes during the movement of the axes, the CNC stops the axes feed and the spindle rotation displaying the corresponding error message.

DWELL <> 0

If this machine parameter for the axis to be moved has been assigned a value other than 0, the CNC will check the status of the SERVOON signal at the time when that axis ENABLE signal 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, it will output the analog voltage for the servo drive, but if it is still low, it stops 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 and **AXIS-1**	**(M5108)** and **(M5109)**
AXIS+2 and **AXIS-2**	**(M5158)** and **(M5159)**
AXIS+3 and **AXIS-3**	**(M5208)** and **(M5209)**
AXIS+4 and **AXIS-4**	**(M5258)** and **(M5259)**
AXIS+5 and **AXIS-5**	**(M5308)** and **(M5309)**
AXIS+6 and **AXIS-6**	**(M5358)** and **(M5359)**

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 Control Panel.

SYNCHRO1	**(M5112)**
SYNCHRO2	**(M5162)**
SYNCHRO3	**(M5212)**
SYNCHRO4	**(M5262)**
SYNCHRO5	**(M5312)**
SYNCHRO6	**(M5362)**

The PLC sets one of these signals high to synchronize the corresponding axis to the axis defined by the axis machine parameter “SYNCHRO”.

LIM1OFF	**(M5115)**	(**LIM**its **1 OFF**)
LIM2OFF	**(M5165)**	(**LIM**its **2 OFF**)
LIM3OFF	**(M5215)**	(**LIM**its **3 OFF**)
LIM4OFF	**(M5265)**	(**LIM**its **4 OFF**)
LIM5OFF	**(M5315)**	(**LIM**its **5 OFF**)
LIM6OFF	**(M5365)**	(**LIM**its **6 OFF**)

The PLC sets one of these signals high so that the CNC ignores the software limits of the corresponding axis.

9.3 LOGIC SPINDLE INPUTS

The mill model 8050 M CNC has one single spindle (main spindle)

The lathe model may have 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 and **LIMIT-S** **(M5450)** and **(M5451)** .... **Main Spindle**
LIMIT+S2 and **LIMIT-S2** **(M5475)** and **(M5476)** .... **Second Spindle**

The CNC uses these signals when working with the spindle 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 spindle machine parameter “REFEED1”, with the slow feedrate indicated by the spindle machine parameter “REFEED2”. After decelerating, it accepts the following reference signal from the spindle feedback systems as being valid.

SPDLEINH **(M5453)** (**SP**in**DLE INH**ibit) .... **Main Spindle**
SPDLEIN2 **(M5478)** (**SP**in**DLE INH**ibit) .... **Second Spindle**

The CNC considers these two 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)** (**SP**in**DLE REV**erse) .... **Main Spindle**
SPDLERE2 **(M5479)** (**SP**in**DLE RE**verse) .... **Second Spindle**

The CNC considers these two 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.

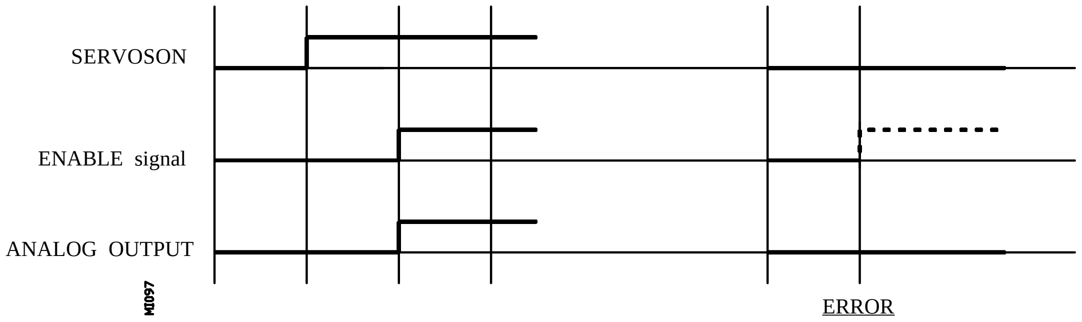
SERVOSON (M5457) .... Main Spindle
SERVOSO2 (M5482) .... Second Spindle

These signals are controlled by the PLC and the CNC will process them only when the spindle is working in closed loop (M19). Their treatment will depend on the value assigned to the spindle machine parameter “DWELL”.

DWELL = 0

In this case, 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.



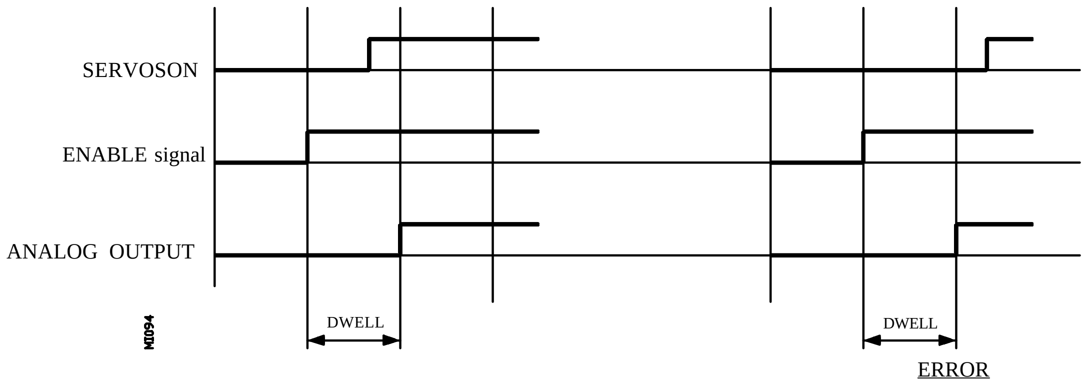
On the other hand, if the SERVOSON signal is low or if it changes to low during the rotation of the spindle, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.

DWELL <> 0

In this case, 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.

GEAR1, GEAR2, GEAR3, GEAR4 (M5458), (M5459), (M5460), (M5461)
Main Spindle
GEAR12, GEAR22, GEAR32, GEAR42 (M5483), (M5484), (M5485), (M5486)
Second Spindle

The PLC uses these signals to indicate to the CNC which spindle speed range 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 range even if it is already selected.

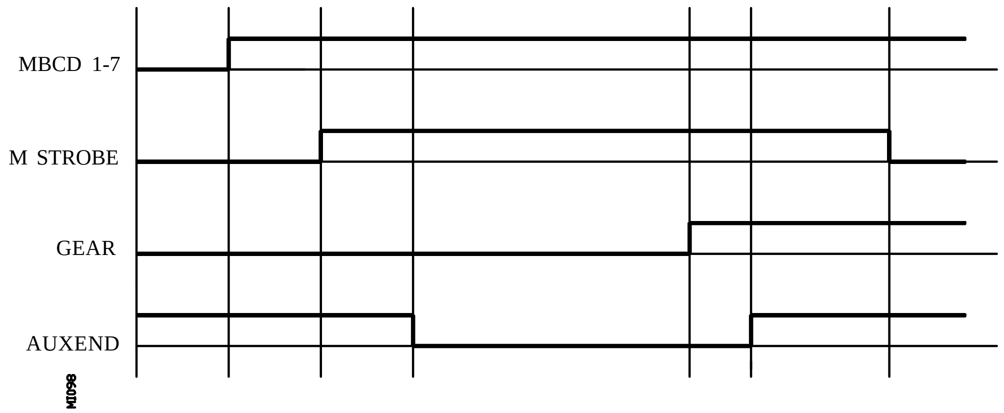
When working with an automatic gear changer, the CNC will check the currently selected gear (GEAR1...GEAR4) and if this does not correspond to the selected speed, the CNC will indicate it to the PLC by means of the miscellaneous function M41, M42, M43 or M44 so it can select it.

Once the PLC selects the proper gear, it indicates it to the CNC by means of the logic input corresponding to the spindle (GEAR1 thru GEAR4).

The spindle gear change depends on the setting of functions M41 thru M44 in the M function table:

If these functions use the "AUXEND" signal:

> The CNC indicates to the PLC the selected range (M41 thru M44) in one of the registers "MBCD1" thru "MBCD7" and it activates the general logic output "MSTROBE" to "tell" the PLC 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 function 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 thru 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 activated, the CNC will require that this signal be kept active for a time period greater than the value given to the general machine parameter “MINAENDW”.

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 gear range has been selected by verifying that the corresponding input among GEAR1 thru GEAR4 is set high.

If that is the case, it will deactivate the “MSTROBE” signal to “tell” the PLC that the gear change has been completed and if the corresponding GEAR input is not set high, the CNC stops the axes feed and spindle rotation displaying the corresponding error message.

If these functions do not use the “AUXEND” signal:

The CNC indicates to the PLC the selected gear range M41, M42, M43 or M44 in one of the registers “MBCD1” thru “MBCD7” and it activates the “MSTROBE” signal to let the PLC “know” that it must execute it.

The CNC will keep the MSTROBE output active for the time period indicated by the general machine parameter “MINAENDW”.

After this time, the CNC will check whether the new gear range has been physically selected by verifying that the corresponding GEAR input (GEAR1 thru 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.

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 synchrnoized speed value to assume when operating in closed loop (M19).

When this **input is low**, the CNC assumes the value set by spindle machine parameter "**REFEED1" (P34)**

When this **input is high**, the CNC assumes the value set by the spindle input register **"M19FEED" (R505)**.

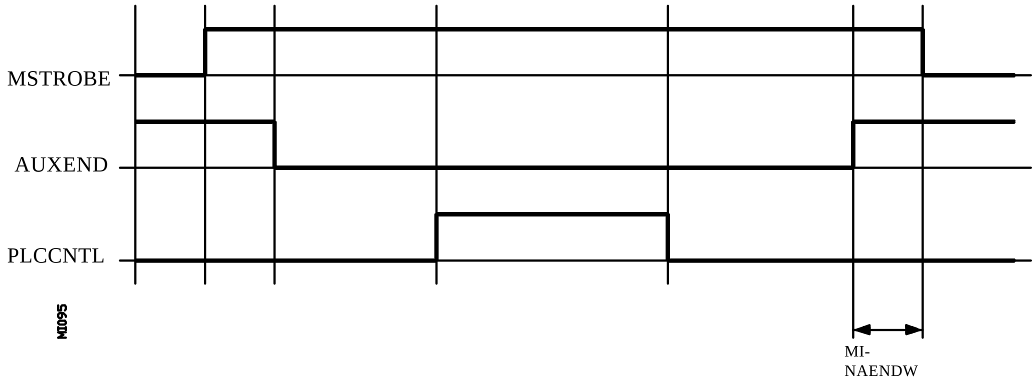
The "**M19FEED**" value is given in 0,0001º/min.

PLCCNTL (M5465) (PLC CoNTroL) .... Main Spindle
PLCCNTL2 (M5490) (PLC CoNTroL) .... 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 gear changes or for changing tools.



The following example shows how a new spindle speed is selected involving a range change.

After analyzing the block and detecting the speed 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 range 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 speed 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.

In this way, if an analog output of 4V is required, the following will be programmed:

SANALOG = (4 x 32767)/10 = 13107

And if an analog output of -4V is required, the following will be programmed:

SANALOG = (-4 x 32767)/10 = -13107

9.4 KEY INHIBITING LOGIC INPUTS

KEYDIS1 (R500)
KEYDIS2 (R501)
KEYDIS3 (R502)
KEYDIS4 (R503)

The PLC can individually inhibit the operation of the panel keys, setting the corresponding bit of one of these 4 32-bit registers high.

Register	Bit	Key Inhibited
KEYDIS1 (R500)	0	F
KEYDIS1 (R500)	1	L
KEYDIS1 (R500)	2	Q
KEYDIS1 (R500)	3	W
KEYDIS1 (R500)	4	SHIFT
KEYDIS1 (R500)	5	9
KEYDIS1 (R500)	6	6
KEYDIS1 (R500)	7	3
KEYDIS1 (R500)	8	E
KEYDIS1 (R500)	9	K
KEYDIS1 (R500)	10	P
KEYDIS1 (R500)	11	V
KEYDIS1 (R500)	12	CAPS
KEYDIS1 (R500)	13	8
KEYDIS1 (R500)	14	5
KEYDIS1 (R500)	15	2
KEYDIS1 (R500)	16	D
KEYDIS1 (R500)	17	J
KEYDIS1 (R500)	18	O
KEYDIS1 (R500)	19	U
KEYDIS1 (R500)	20	SP
KEYDIS1 (R500)	21	7
KEYDIS1 (R500)	22	4
KEYDIS1 (R500)	23	1
KEYDIS1 (R500)	24	C
KEYDIS1 (R500)	25	I
KEYDIS1 (R500)	26	Ñ
KEYDIS1 (R500)	27	T
KEYDIS1 (R500)	28	Z
KEYDIS1 (R500)	29	=
KEYDIS1 (R500)	30	/
KEYDIS1 (R500)	31	*

Register	Bit	Key Inhibited
KEYDIS2 (R501)	0	B
KEYDIS2 (R501)	1	H
KEYDIS2 (R501)	2	N
KEYDIS2 (R501)	3	S
KEYDIS2 (R501)	4	Y
KEYDIS2 (R501)	5	RESET
KEYDIS2 (R501)	6	ESC
KEYDIS2 (R501)	7	MAIN MENU
KEYDIS2 (R501)	8	A
KEYDIS2 (R501)	9	G
KEYDIS2 (R501)	10	M
KEYDIS2 (R501)	11	R
KEYDIS2 (R501)	12	X
KEYDIS2 (R501)	13	ENTER
KEYDIS2 (R501)	14	HELP
KEYDIS2 (R501)	15	
KEYDIS2 (R501)	16	.
KEYDIS2 (R501)	17	0
KEYDIS2 (R501)	18	-
KEYDIS2 (R501)	19	+
KEYDIS2 (R501)	20	
KEYDIS2 (R501)	21	
KEYDIS2 (R501)	22	
KEYDIS2 (R501)	23	
KEYDIS2 (R501)	24	Next page
KEYDIS2 (R501)	25	Previous page
KEYDIS2 (R501)	26	Arrow up
KEYDIS2 (R501)	27	Arrow down
KEYDIS2 (R501)	28	Arrow right
KEYDIS2 (R501)	29	Arrow left
KEYDIS2 (R501)	30	CL
KEYDIS2 (R501)	31	INS

Register	Bit	Key Inhibited	
		Milling Model	Lathe Model
KEYDIS3 (R502)	0	F1	F1
KEYDIS3 (R502)	1	F2	F2
KEYDIS3 (R502)	2	F3	F3
KEYDIS3 (R502)	3	F4	F4
KEYDIS3 (R502)	4	F5	F5
KEYDIS3 (R502)	5	F6	F6
KEYDIS3 (R502)	6	F7	F7
KEYDIS3 (R502)	7		
KEYDIS3 (R502)	8	X +	3rd axis +
KEYDIS3 (R502)	9	Y +	
KEYDIS3 (R502)	10	Z +	X +
KEYDIS3 (R502)	11	4 +	
KEYDIS3 (R502)	12	5 +	4th axis +
KEYDIS3 (R502)	13	Speed Override +	Speed Override +
KEYDIS3 (R502)	14	Spindle clockwise	Spindle clockwise
KEYDIS3 (R502)	15	Cycle Start	Cycle Start
KEYDIS3 (R502)	16		
KEYDIS3 (R502)	17		Z -
KEYDIS3 (R502)	18	Rapid	Rapid
KEYDIS3 (R502)	19		Z +
KEYDIS3 (R502)	20		
KEYDIS3 (R502)	21		
KEYDIS3 (R502)	22	Spindle stop	Spindle stop
KEYDIS3 (R502)	23		
KEYDIS3 (R502)	24	X -	3rd axis -
KEYDIS3 (R502)	25	Y -	
KEYDIS3 (R502)	26	Z -	X -
KEYDIS3 (R502)	27	4 -	
KEYDIS3 (R502)	28	5 -	4th axis -
KEYDIS3 (R502)	29	Speed Override -	Speed Override -
KEYDIS3 (R502)	30	Spindle c.clockwise	Spindle c.clockwise
KEYDIS3 (R502)	31	Cycle Stop	Cycle Stop

Register	Bit	Key Inhibited
KEYDIS4 (R503)	0	Handwheel x100
KEYDIS4 (R503)	1	Handwheel x 10
KEYDIS4 (R503)	2	Handwheel x 1
KEYDIS4 (R503)	3	JOG 10000
KEYDIS4 (R503)	4	JOG 1000
KEYDIS4 (R503)	5	JOG 100
KEYDIS4 (R503)	6	JOG 10
KEYDIS4 (R503)	7	JOG 1
KEYDIS4 (R503)	8	Feedrate Override 0%
KEYDIS4 (R503)	9	Feedrate Override 2%
KEYDIS4 (R503)	10	Feedrate Override 4%
KEYDIS4 (R503)	11	Feedrate Override 10%
KEYDIS4 (R503)	12	Feedrate Override 20%
KEYDIS4 (R503)	13	Feedrate Override 30%
KEYDIS4 (R503)	14	Feedrate Override 40%
KEYDIS4 (R503)	15	Feedrate Override 50%
KEYDIS4 (R503)	16	Feedrate Override 60%
KEYDIS4 (R503)	17	Feedrate Override 70%
KEYDIS4 (R503)	18	Feedrate Override 80%
KEYDIS4 (R503)	19	Feedrate Override 90%
KEYDIS4 (R503)	20	Feedrate Override 100%
KEYDIS4 (R503)	21	Feedrate Override 110%
KEYDIS4 (R503)	22	Feedrate Override 120%
KEYDIS4 (R503)	23	
KEYDIS4 (R503)	24	
KEYDIS4 (R503)	25	
KEYDIS4 (R503)	26	
KEYDIS4 (R503)	27	
KEYDIS4 (R503)	28	
KEYDIS4 (R503)	29	
KEYDIS4 (R503)	30	
KEYDIS4 (R503)	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%.

9.5 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, EPROM, 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 (other conditions) = CYSTART
When pressing the Cycle Start key, the CNC activates general logic output "START".
The PLC must check that the "other conditions" are also met (safety and so forth) before setting general logic input "CYSTART" high so the program starts running.

FHOUT (M5502)

The CNC sets this signal high in order to tell the PLC that the execution of the program is stopped due to one of the following causes:

- Because the CONTROL PANEL STOP key has been pressed.
- Because the general logic input /STOP has been set low, even though later it has returned high.
- Because the general logic input /FEEDHOL is low.

RESETOUT (M5503)

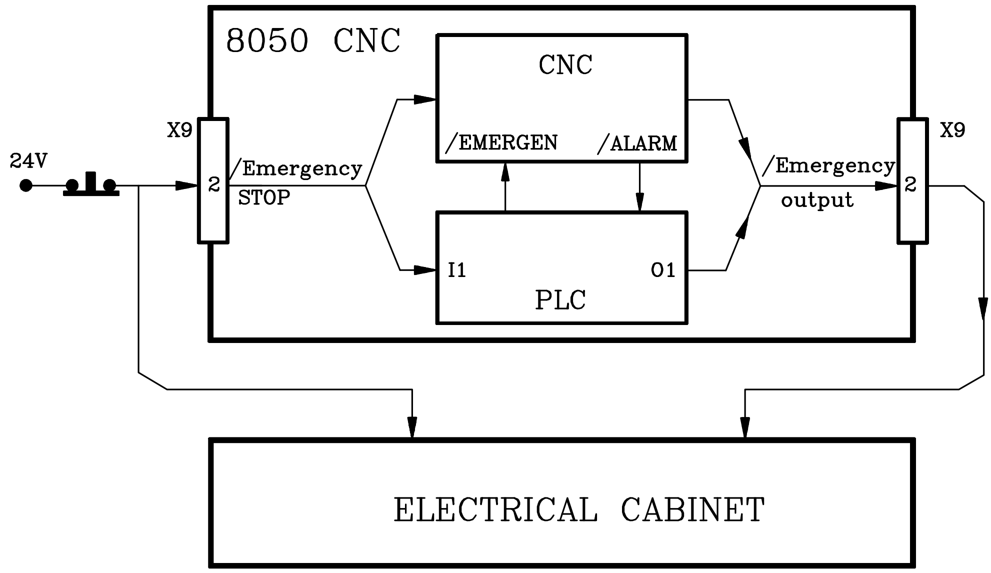
The CNC sets this signal high for 100 milliseconds, in order to tell the PLC that it is under initial conditions because the Reset key on the Front Panel has been pressed or because the general logic input RESETIN has been activated.

LOPEN (M5506)

The CNC sets this signal high in order to tell the PLC that the positioning loop of the axes is open since an error has occurred.

/ALARM (M5507)

The CNC sets this signal low in order to tell the PLC that an alarm or emergency condition has been detected. This signal will be set high once again, once the message from the CNC has been eliminated and the cause of the alarm has disappeared.



During the period of time when this signal is low, the CNC will activate (low logic level) the Emergency output (pin 2 of the X10 connector of the Axis module).

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) (**AUTOMAT**ic)

The CNC sets this signal high to tell the PLC that the Automatic Operating Mode is selected.

MDI (M5510)

The CNC sets this signal high to tell the PLC that the MDI Mode (manual data input) is selected in one of the operating modes (JOG, Automatic, etc).

SBOUT (M5511)

The CNC sets this signal high to tell the PLC that the Single Block Execution Mode is selected.

INCYCLE (M5515)

The CNC sets this signal high while executing a block or moving an axis.

Once the execution of the program has been requested by the PLC to the CNC by means of the logic input CYSTART, the latter will indicate that it is being executed by setting the INCYCLE signal high.

This signal is maintained high until the CNC finishes the part program or when this is stopped by means of the STOP key on the CONTROL PANEL or the general logic input /STOP.

If the CNC is in the Single Block Execution Mode, the INCYCLE signal is set low as soon as the block execution is concluded.

If the CNC is in the 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)

This output is only available on the Mill model. 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.

CSS (M5523)

The CNC sets this signal high to tell the PLC that the constant cutting speed function is selected (G96). Lathe Model.

SELECT0 (M5524)
SELECT1 (M5525)
SELECT2 (M5526)
SELECT3 (M5527)
SELECT4 (M5528)

The CNC indicates to the PLC by means of this group of signals the Front Panel switch position selected.

Its value will be coded according to the following table:

SELECT4	SELECT3	SELECT2	SELECT1	SELECT0	Switch position
0	0	0	0	0	Handwheel x 100
0	0	0	0	1	Handwheel x 10
0	0	0	1	0	Handwheel x 1
0	0	0	1	1	JOG 10000
0	0	1	0	0	JOG 1000
0	0	1	0	1	JOG 100
0	0	1	1	0	JOG 10
0	0	1	1	1	JOG 1
0	1	0	0	0	Feed Override 0%
0	1	0	0	1	Feed Override 2%
0	1	0	1	0	Feed Override 4%
0	1	0	1	1	Feed Override 10%
0	1	1	0	0	Feed Override 20%
0	1	1	0	1	Feed Override 30%
0	1	1	1	0	Feed Override 40%
0	1	1	1	1	Feed Override 50%
1	0	0	0	0	Feed Override 60%
1	0	0	0	1	Feed Override 70%
1	0	0	1	0	Feed Override 80%
1	0	0	1	1	Feed Override 90%
1	0	1	0	0	Feed Override 100%
1	0	1	0	1	Feed Override 110%
1	0	1	1	0	Feed Override 120%

MSTROBE (M5532)

The CNC sets this signal high to tell the PLC that it must execute the auxiliary M function or functions which are indicated in Registers “MBCD1” to “MBCD7” (R550 to R556).

SSTROBE (M5533)

This signal is used when the type of spindle used has S input in BCD (spindle machine parameter “SPDLTYPE”).

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 T 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 general machine parameter “RANDOMTC” is set for “not random”, the tool pocket in the magazine coincides with the tool number.

T2STROBE (M5535)

This signal is used when making a special tool change, family code ≥200 or in the case of a machining center with a non random tool magazine (general machine parameter “RANDOMTC”).

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 available on the Lathe model.
It indicates which spindle is controlled by the CNC. This selection is made via part-program using 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) (**ADV**anced **IN POS**ition)

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 period is set by general machine parameter "**ANTIME**".

This reduces the idle time, thus resulting in more punches per minute.

INTEREND (M5538) (**INTER**polation **END**)
INPOS (M5539) (**IN POS**ition)

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) for a time period indicated in the axis machine parameter INPOTIME, it will consider that all of them are in position and it will inform the PLC by setting the logic output "INPOS" high.

The logic output "INTEREND" can be used when it is required to activate mechanisms before the axes reach their position.

DM00 (M5547)

The CNC sets this signal high to tell the PLC that the auxiliary function M00 (program stop) is programmed in the block being executed.

DM01 (M5546)

The CNC sets this signal high to tell the PLC that the auxiliary function M01 (conditional stop) is programmed in the block being executed.

DM02 (M5545)

The CNC sets this signal high to tell the PLC that the auxiliary function M02 (program end) is programmed in the block being executed.

DM03 (M5544)

The CNC sets this signal high to tell the PLC that the spindle is turning clockwise or that the auxiliary function M03 is programmed in the block being executed.

DM04 (M5543)

The CNC sets this signal high to tell the PLC that the spindle is turning counter-clockwise or that the auxiliary function M04 is programmed in the block being executed.

DM05 (M5542)

The CNC sets this signal high to tell the PLC that the spindle is stopped or that the auxiliary function M05 is programmed in the block being executed.

DM06 (M5541)

The CNC sets this signal high to tell the PLC that the spindle is stopped or that the auxiliary function M06 is programmed in the block being executed (tool change).

DM08 (M5540)

The CNC sets this signal high to tell the PLC that the coolant output is activated or that the auxiliary function M08 is programmed in the block being executed.

DM09 (M5555)

The CNC sets this signal high to tell the PLC that the coolant output is deactivated or that the auxiliary function M09 is programmed in the block being executed.

DM19 (M5554)

The CNC sets this signal high to tell the PLC that it is working with spindle orientation or that the auxiliary function M19 is programmed in the block being executed.

DM30 (M5553)

The CNC sets this signal high to tell the PLC that the auxiliary function M30 (program end) is programmed in the block being executed.

DM41 (M5552)

The CNC sets this signal high to tell the PLC that the first spindle speed range 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 speed range 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 speed range is selected or that the auxiliary function M43 is programmed in the block being executed.

DM44 (M5549)

The CNC sets this signal high to tell the PLC that the fourth spindle speed range 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.

9.6 AXIS LOGIC OUTPUTS

There are 6 groups of logic outputs (ENABLE, DIR, etc.) which refer to the 6 possible axes of the machine by means of digits 1 through 6 (ENABLE2, DIR1,etc.)

These numbers **have nothing to do** with the values assigned to the general machine parameters "AXIS1" through "AXIS8".

These variables are numbered according to the logic order of the axes.

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:

Variables: ENABLE1, DIR1, REFPOINT1, INPOS1 correspond to the X axis.
Variables: ENABLE2, DIR2, REFPOINT2, INPOS2 correspond to the Y axis.
Variables: ENABLE3, DIR3, REFPOINT3, INPOS3 correspond to the Z axis.
Variables: ENABLE4, DIR4, REFPOINT4, INPOS4 correspond to the U axis.
Variables: ENABLE5, DIR5, REFPOINT5, INPOS5 correspond to the B axis.
Variables: ENABLE6, DIR6, REFPOINT6, INPOS6 correspond to the C axis.

ENABLE1 (M5600)
ENABLE2 (M5650)
ENABLE3 (M5700)
ENABLE4 (M5750)
ENABLE5 (M5800)
ENABLE6 (M5850)

The CNC sets these signals at a high logic level to tell the PLC to allow the corresponding axis to move.

DIR1 (M5601)
DIR2 (M5651)
DIR3 (M5701)
DIR4 (M5751)
DIR5 (M5801)
DIR6 (M5851)

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)** (**R**eference **POINT 1**)
REFPOIN2 **(M5652)** (**R**eference **POINT 2**)
REFPOIN3 **(M5702)** (**R**eference **POINT 3**)
REFPOIN4 **(M5752)** (**R**eference **POINT 4**)
REFPOIN5 **(M5802)** (**R**eference **POINT 5**)
REFPOIN6 **(M5852)** (**R**eference **POINT 6**)

The CNC sets these signals high to tell the PLC that the machine reference search has been made already.

It 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 will be put high after carrying out the machine reference search.

ANT1 **(M5606**) **(ANT**icipation **1**)
ANT2 **(M5656**) **(ANT**icipation **2**)
ANT3 **(M5706**) **(ANT**icipation **3**)
ANT4 **(M5756**) **(ANT**icipation **4**)
ANT5 **(M5806**) **(ANT**icipation **5**)
ANT6 **(M5856**) **(ANT**icipation **6**)

These signals are related to axis machine parameter "MINMOVE (P54)".

If the programmed axis movement is smaller than the value indicated by its corresponding axis machine machine parameter "MINMOVE (P54)", the corresponding logic axis output "ANT1 thru ANT6" goes high.

INPOS1 (M5607) (**IN POS**ition **1**)
INPOS2 (M5657) (**IN POS**ition **2**)
INPOS3 (M5707) (**IN POS**ition **3**)
INPOS4 (M5757) (**IN POS**ition **4**)
INPOS5 (M5807) (**IN POS**ition **5**)
INPOS6 (M5857) (**IN POS**ition **6**)

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.

9.7 SPINDLE LOGIC OUTPUTS

The mill model 8050 M CNC has one single spindle (main spindle)

The lathe model may have 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.

REPOINS (M5952) (**REF**erence **POIN**t**S**) .... **Main Spindle**
REPOIS2 (M5977) (**REF**erence **POI**nt**S**) .... **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.

CAXIS (M5955) (**C AXIS**) .... **Main Spindle**
CAXIS2 (M5980) (**C AXIS**) .... **Second Spindle**

This signal is used when working with the spindle as C axis (G15). The CNC only considers the signals for the currently selected spindle.

The CNC sets this signal high to tell the PLC that the C axis is active.

REVOK (M5956) .... Main Spindle
REVOK2 (M5981) .... Second Spindle

The CNC only considers the signals for the currently selected spindle.

When working with M03 and M04 the CNC sets this signal high to tell the PLC that the real spindle revolutions correspond to those programmed.

The CNC will activate this signal every time the real revolutions are within the range defined by spindle machine parameters “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) (**IN POS**isition **S**) .... **Main Spindle**
INPOSS2 (M5982) (**IN POS**isition **S**) .... **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.

9.8 LOGIC OUTPUTS OF KEY STATUS

KEYBD1 (R560)
KEYBD2 (R561)
KEYBD3 (R562)

These registers indicate whether or not one of the keys on the keyboard or on the operator panel is pressed.

When one of these keys is pressed, the corresponding bit will be set high and it will return low when the key is released.

Register	Bit	Key pressed
KEYBD1 (R560)	0	F
KEYBD1 (R560)	1	L
KEYBD1 (R560)	2	Q
KEYBD1 (R560)	3	W
KEYBD1 (R560)	4	SHIFT
KEYBD1 (R560)	5	9
KEYBD1 (R560)	6	6
KEYBD1 (R560)	7	3
KEYBD1 (R560)	8	E
KEYBD1 (R560)	9	K
KEYBD1 (R560)	10	P
KEYBD1 (R560)	11	V
KEYBD1 (R560)	12	CAPS
KEYBD1 (R560)	13	8
KEYBD1 (R560)	14	5
KEYBD1 (R560)	15	2
KEYBD1 (R560)	16	D
KEYBD1 (R560)	17	J
KEYBD1 (R560)	18	O
KEYBD1 (R560)	19	U
KEYBD1 (R560)	20	SP
KEYBD1 (R560)	21	7
KEYBD1 (R560)	22	4
KEYBD1 (R560)	23	1
KEYBD1 (R560)	24	C
KEYBD1 (R560)	25	I
KEYBD1 (R560)	26	Ñ
KEYBD1 (R560)	27	T
KEYBD1 (R560)	28	Z
KEYBD1 (R560)	29	=
KEYBD1 (R560)	30	/
KEYBD1 (R560)	31	*

Register	Bit	Key pressed
KEYBD2 (R561)	0	B
KEYBD2 (R561)	1	H
KEYBD2 (R561)	2	N
KEYBD2 (R561)	3	S
KEYBD2 (R561)	4	Y
KEYBD2 (R561)	5	RESET
KEYBD2 (R561)	6	ESC
KEYBD2 (R561)	7	MAIN MENU
KEYBD2 (R561)	8	A
KEYBD2 (R561)	9	G
KEYBD2 (R561)	10	M
KEYBD2 (R561)	11	R
KEYBD2 (R561)	12	X
KEYBD2 (R561)	13	ENTER
KEYBD2 (R561)	14	HELP
KEYBD2 (R561)	15	
KEYBD2 (R561)	16	.
KEYBD2 (R561)	17	0
KEYBD2 (R561)	18	-
KEYBD2 (R561)	19	+
KEYBD2 (R561)	20	
KEYBD2 (R561)	21	
KEYBD2 (R561)	22	
KEYBD2 (R561)	23	
KEYBD2 (R561)	24	Next page
KEYBD2 (R561)	25	Previous page
KEYBD2 (R561)	26	Arrow up
KEYBD2 (R561)	27	Arrow down
KEYBD2 (R561)	28	Arrow right
KEYBD2 (R561)	29	Arrow left
KEYBD2 (R561)	30	CL
KEYBD2 (R561)	31	INS

Register	Bit	Key pressed	
		Mill Model	Lathe Model
KEYBD3 (R562)	0	F1	F1
KEYBD3 (R562)	1	F2	F2
KEYBD3 (R562)	2	F3	F3
KEYBD3 (R562)	3	F4	F4
KEYBD3 (R562)	4	F5	F5
KEYBD3 (R562)	5	F6	F6
KEYBD3 (R562)	6	F7	F7
KEYBD3 (R562)	7		
KEYBD3 (R562)	8	X +	3rd axis +
KEYBD3 (R562)	9	Y +	
KEYBD3 (R562)	10	Z +	X +
KEYBD3 (R562)	11	4 +	
KEYBD3 (R562)	12	5 +	4th axis +
KEYBD3 (R562)	13	Speed Override +	Speed Override +
KEYBD3 (R562)	14	Spindle clockwise	Spindle clockwise
KEYBD3 (R562)	15	Cycle Start	Cycle Start
KEYBD3 (R562)	16		
KEYBD3 (R562)	17		Z -
KEYBD3 (R562)	18	Rapid	Rapid
KEYBD3 (R562)	19		Z +
KEYBD3 (R562)	20		
KEYBD3 (R562)	21		
KEYBD3 (R562)	22	Spindle stop	Spindle stop
KEYBD3 (R562)	23		
KEYBD3 (R562)	24	X -	3rd axis -
KEYBD3 (R562)	25	Y -	
KEYBD3 (R562)	26	Z -	X -
KEYBD3 (R562)	27	4 -	
KEYBD3 (R562)	28	5 -	4th axis -
KEYBD3 (R562)	29	Speed Override -	Speed Override -
KEYBD3 (R562)	30	Spindle c.clockwise	Spindle c.clockwise
KEYBD3 (R562)	31	Cycle Stop	Cycle Stop

10. *ACCESS TO INTERNAL CNC VARIABLES*

The PLC provides two instructions (actions) which permit to read or modify the various internal variables of the CNC.

CNCRD: Allows reading access to the CNC internal variables. It is programmed as follows:

CNCRD (Variable, Register, Mark)

This instruction loads the selected PLC register with the contents of the indicated CNC 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.

Example: 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 non-existing variable (i.e. the position value of a non-existing axis), this instruction will not alter the contents of R150 and it will set error mark M200 to “1” indicating that the variable does not exist.

CNCWR: Allows writing access to internal CNC variables. It is programmed as follows:

CNCWR (Register, Variable, Mark)

This instruction loads the contents of the indicated PLC register into the selected CNC 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.

Example: CNCWR (R92, TIMER, M200)

Presets the timer enabled by the PLC with the value contained in Register R92.

When trying to modify the contents of a non-existing 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.

The internal CNC variables which can be accessed by the PLC can be read-only or read-and-write variables.

Every internal CNC table (tools, tool offset, Zero offsets, etc.) has a mnemonic to identify the fields. Use one of the following formats to access a specific variable:

* The corresponding mnemonic followed by the element number of that table. Example (TOR3):

  CNCRD (TOR3, R100, M102); Assigns the R value of tool offset 3 to register R100.

* The corresponding mnemonic and a register containing the element number of that table. Example (TOR R222):

  CNCRD (TOR R222, R100, M102); assigns to register R100 the R value of the tool offset indicated by register R222.

The variables available at the CNC 8050 can be classified in the following way:

- Variables associated with tools.
- Variables associated with zero offsets.
- Variables associated with machine parameters
- Variables associated with work zones
- Variables associated with feedrates
- Variables associated with coordinates
- Variables associated with the spindle
- Variables associated with local and global parameters
- Other variables

10.1 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 offset table of the MILL model FAGOR 8050 CNC:

R,L,I,K	In the units set by machine parameter "INCHES"
	If "INCHES"=0, in 0.0001mm. Max.: ±999999999
	If "INCHES"=1, in 0,00001 inch. Max or ±393700787
	If rotary axis, in 0.0001°. Max.:± 999999999

Tool offset table of the LATHE model FAGOR 8050 CNC:

X,Z,R,I,K	In the units set by machine parameter "INCHES"
	If "INCHES"=0, in 0.0001mm. Max.: ±999999999
	If "INCHES"=1, in 0,00001 inch. Max or ±393700787
	If rotary axis, in 0.0001°. Max.:± 999999999
F	Integer value between 0 and 99.

Tool table for Mill model CNC:

Tool offset number	0...NT OFFSET (maximum 255)
Family code	If normal tool, 0 < n < 200
	If special tool, 200 < n < 255
Nominal life	0...65535 minutes or operations.
Real life	0...99,999,99 hundredths of a minute or 99,999 operations

Tool table for lathe model CNC:

Tool offset number	0...NT OFFSET (maximum 255)
Family code	If normal tool, 0 < n < 200
	If special tool, 200 < n < 255
Nominal life	0...65535 minutes or operations.
Real life	0...99,999,99 hundredths of a minute or 99,999 operations.
Cutter angle	In 0.0001^0 units up to 359.9999^0 degrees.
Cutter width	In the units set by machine parameter "INCHES"
	If "INCHES"=0, in 0.0001mm. Max.: ±999999999
	If "INCHES"=1, in 0,00001 inch. Max or ±393700787
Cutting angle	In 0.0001^0 units, up to 359.9999^0.

Tool magazine table:

Contents of each magazine position

Tool number 1 ...NTOOL (maximum 255)
0 Empty
-1 Cancelled

Tool position in magazine

Position number 1 ..NPOCKET (maximum 255)
0 On spindle
-1 Not found
-2 In change position

Read-only variables

TOOL: Returns the active tool number

CNCRD(TOOL,R100,M100); Loads register R100 with the number of the active tool

TOD: Returns the active tool offset number

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.

CNCRD(TFLD3,R100,M102); Loads register R100 with the tool offset number of tool 3.

CNCWR(R101,TFLD3,M101); Assigns the tool offset number indicated in register R101 to tool number 3.

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.

Read-and-write variables of the FAGOR 8050 CNC MILL model

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);	Loads register R100 with the R value of tool offset 3.
CNCWR(R101,TOR3,M101);	Assigns the value indicated in R101 to the R 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 FAGOR 8050 CNC 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 the tool offset 3.

TOZn: This variable allows reading or modifying the length value along the Z axis assigned to the indicated tool offset (n).

TOFn: This variable allows reading or modifying the location code (F) of the indicated tool offset (n).

TORn: This variable allows reading or modifying the radius R value of the indicated tool offset (n).

TOIn: This variable allows reading or modifying the length wear value (I) along the X axis of the indicated tool offset (n).

TOKn: This variable allows reading or modifying the length wear value (K) along the Z axis of the indicated tool offset (n).

NOSEAn: This variable allows reading or modifying the cutter angle assigned to the indicated tool (n) in the tool table.

NOSEWn: This variable allows reading or modifying the cutter width assigned to the indicated tool (n) in the tool table.

CUTAn: This variable allows reading or modifying the cutting angle assigned to the indicated tool (n) in the tool table.

10.2. VARIABLES ASSOCIATED WITH ZERO OFFSETS

These variables are associated with the zero offset table, due to which the values that will be assigned to or read from these fields will comply with the formats established for this table.

The zero offsets, in addition to the additive offset indicated by the PLC, are G54, G55, G56, G57, G58 and G59 and

The values of each axis are given in the units set by machine parameter "INCHES".

If "INCHES"=0, in 0.0001mm. Max.: ±999999999
If "INCHES"=1, in 0,00001 inch. Max or ±393700787
If rotary axis, in 0.0001º. Max.:± 999999999

Although there are variables which refer to each axis, the CNC only allows those referring to the selected axes in 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 is not included in this value.

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,ORGY 54,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 additive zero offset table indicated by the PLC.

10.3 VARIABLES ASSOCIATED WITH MACHINE PARAMETERS

Variables associated with machine parameters are read-only variables.

In order to become familiar with the values returned it is advisable to consult the chapter dealing with those parameters taking into account the following indications:

Values 1/0 correspond to the parameters which are defined with YES/NO, +/- and ON/OFF.

Values regarding position and feedrate values will be given in the units set by machine parameter "INCHES".

If "INCHES"=0, in 0.0001mm. Max.: ±999999999
If "INCHES"=1, in 0,00001 inch. Max or ±393700787

Values regarding the spindle (when working in M19) and rotary axes will be given in 0.0001 degree units. Max.:± 999999999.

Read-only variables

MPGn: Returns the value assigned to the indicated general machine parameter (n).

CNCRD (MPG 8,R100,M102); Loads register R100 with the value of general machine parameter P8 (INCHES). 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.

CNCRD (MPY 1,R100,M102); Loads register R100 with the value of machine parameter P1 (DFORMAT) for the Y axis which indicates the display format for this axis.

MPSn: Returns the value assigned to the indicated machine parameter (n) of the spindle.

MPASn: Returns the value of the indicated (n) machine parameter for the auxiliary spindle.

MPLCn: Returns the value assigned to the indicated machine parameter (n) of the PLC.

10.4 VARIABLES ASSOCIATED WITH WORK ZONES

The values of the limits are given in the units set by machine parameter "INCHES".

If "INCHES"=0, in 0.0001mm. Max.: ±999999999
If "INCHES"=1, in 0,00001 inch. Max or ±393700787
If rotary axis, in 0.0001º. Max.:± 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: This variable permits reading or writing the status of work zone 1.

FZLO(X-C): This variable permits reading or writing the value of the lower limit of Zone 1 according to the selected axis (X-C).

FZUP(X-C): This variable permits reading or writing the value of the upper limit of Zone 1 according to the selected axis (X-C).

SZONE: This variable permits reading or writing the status of work zone 2.

SZLO(X-C): This variable permits reading or writing the value of the lower limit of Zone 2 according to the selected axis (X-C).

SZUP(X-C): This variable permits reading or writing the value of the upper limit of Zone 2 according to the selected axis (X-C).

TZONE: This variable permits reading or writing the status of work zone 3.

TZLO(X-C): This variable permits reading or writing the value of the lower limit of Zone 3 according to the selected axis (X-C).

TZUP(X-C): This variable permits reading or writing the value of the upper limit of Zone 3 according to the selected axis (X-C).

The following example shows how it is possible to define as forbidden zone for the X axis the area between 0 and 100mm (1000000 tenths of microns).

```
(condition)   =  MOV 0 R1          = CNCWR (R1,FZLOX,M1)
              =  MOV 1000000 R1    = CNCWR (R1,FZUPX,M1)
              =  MOV 1 R1          = CNCWR (R1,FZONE,M1)
```

10.5 VARIABLES ASSOCIATED WITH FEEDRATES

Read-only variables

FREAL: Returns the real feedrate (taking the feedrate override % into account). Its value is given in 0.0001 mm/min. or 0.00001 inch/min. units.

FEED: Returns the active feedrate (ignoring the override) selected at the CNC when working in G94. This will be given in 0.0001mm/minute or 0.00001 inch/minute units.

This feedrate can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCF: Returns the feedrate selected by DNC, in 0.0001mm/minute or 0.00001 inches/minute. If this has a value of 0 it means that it is not selected.

PRGF: Returns the feedrate selected by program, in 0.0001 mm/minute or 0.00001 inches/minute units.

FPREV: Returns the active (ignoring the override) feedrate selected at the CNC when working in G95. This will be given in 0.0001 mm/rev. or 0.00001 inches/rev units.

This feedrate can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCFPR: Returns the feedrate selected by DNC, in 0.0001 mm/rev. or 0.00001 inches/rev units. If this has a value of 0 it means that it is not selected.

PRGFPR: Returns the feedrate selected by program in 0.0001 mm/rev. or 0.00001 inches/rev units.

FRO: Returns the Feedrate Override (%) selected at the CNC. This will be given by an integer between 0 and "MAXFOVR" (maximum 255).

This feedrate percentage may be indicated by the PLC, by DNC or from the front panel, and the CNC will select one of them, the order of priority (from highest to lowest) being: by program, by DNC, by PLC and from the front panel switch.

PRGFRO: Returns the % of the feedrate selected by program. It is given in integer values between 0 and "MAXFOVR" (maximum 255). A value of 0 means that it is not selected.

DNCFRO: Returns the feedrate, in mm/minute or inches/minute, which is selected by DNC. If this has a value of 0 it means that it is not selected.

CNCFRO: Returns the % of feedrate selected from the front panel knob.

Read-and-write variables

PLCF: This variable allows reading or modifying the feedrate selected by PLC. It is given in 0.0001 mm/min. or 0.00001 inch/min units. A value of 0 means that it is not selected.

PLCFPR: This variable allows reading or modifying the feedrate selected by PLC. It is given in 0.0001 mm/rev. or 0.00001 inch/rev units. A value of 0 means that it is not selected.

PLCFRO: This variable allows reading or modifying the % of feedrate selected by PLC. A value of 0 means that it is not selected.

10.6 VARIABLES ASSOCIATED WITH POSITION COORDINATES

The coordinate values of each axis are given in the units set by machine parameter "INCHES".

If "INCHES" = 0, in 0.0001 mm. Max. ±999999999
If "INCHES" = 1, in 0.00001 inch. Max. ±39370078
If rotary axis, in 0.0001º. Max. ±999999999

Read-only variables

POS(X-C): Returns the real position value of the selected axis referred to machine reference zero (home).

On the **Lathe** model **FAGOR 8050 CNC**, the coordinates of each axis are shown in either radius or diameter depending on the setting of machine parameter "**DFORMAT**".

TPOS(X-C): Returns the theoretical position value (real + following error) of the selected axis referred to machine reference zero (home).

On the **Lathe** model **FAGOR 8050 CNC**, the coordinates of each axis are shown in either radius or diameter depending on the setting of machine parameter "**DFORMAT**".

FLWE(X-C): Returns the following error of the selected axis, in 0.0001mm or 0.00001 inch units .

DEFLEX
DEFLEY
DEFLEZ: These variables can only be used on the **Mill** model **FAGOR 8050 CNC**. They return the amount of deflection obtained at the time by the Renishaw probe SP2 on each axis X, Y, Z.

Read-and-write variables

DIST(X-C): These variables allow the distance travelled by the selected axis to be read or modified. This value is accumulative and is very useful when it is required to perform an operation which depends on the distance travelled by the axes, for example: in their lubrication.

The CNC will set this value to 0 when changing the software version or when a checksum error occurs.

10.7 VARIABLES ASSOCIATED WITH THE SPINDLE

Read-only variables

SREAL: Returns the real spindle turning speed. This will be in 0.0001 rev./min. units.

SPEED: Returns the spindle turning speed which is selected at the CNC. This will be given in 0.0001 rev./min. units.

This turning speed can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

DNCS: Returns the turning speed in 0.0001 rev./min. units selected by DNC. If this has a value of 0 it means that it is not selected.

PRGS: Returns the turning speed in 0.0001 rev./min. units selected by program.

CSS This variable can only be used on FAGOR 8050 CNC LATHE models. It returns the constant surface speed selected at the CNC. its value will be in meters/min. or feet/min. depending on the setting of machine parameter "INCHES" (=0 or =1 respectively).

This constant surface speed can be indicated by program, PLC or via DNC. The CNC will select one of them. The DNC has the highest priority and the program the lowest.

DNCCSS This variable can only be used on FAGOR 8050 CNC LATHE models. It returns the constant surface speed selected via DNC. Its value is given in meters/min. or feet/min. A value of “0” means that it is not selected.

PRGCSS This variable can only be used on FAGOR 8050 CNC LATHE models. It returns the constant surface speed selected by program. Its value is given in meters/min. or feet/min.

SSO: Returns the Override (%) of the spindle turning speed which is selected at the CNC. This will be given by an integer between 0 and “MAXSOVR” (maximum 255).

This spindle turning speed percentage may be indicated by the PLC, by DNC or from the front panel, and the CNC will select one of them, the order of priority (from highest to lowest) being: by program, by DNC, by PLC and from the front panel.

PRGSSO: It returns the percentage of the spindle turning speed selected by program. This will be given by an integer between 0 and “MAXSOVR” (maximum 255). If this has a value of 0 it means that it is not selected.

DNCSSO: Returns the spindle turning speed percentage which is selected by DNC. If this has a value of 0 it means that it is not selected.

CNCSSO: Returns spindle turning speed percentage which is selected from the front panel.

SLIMIT: Returns the value established for the spindle turning speed limit which is selected at the CNC. This will be in 0.0001 rev./min. units.

This limit can be indicated by program, by the PLC or DNC, and the CNC selects one of these, the one with the highest priority being that indicated by DNC and the one with the lowest priority that indicated by program.

If this limit has not been defined, this variable will return a value of 0.

DNCSL: Returns the spindle turning speed limit selected by DNC in 0.0001 rev./min. units. If this has a value of 0 it means that it is not selected.

PRGSL: Returns the spindle turning speed limit which is selected by program in 0.0001 rev./min. units.

POSS: Returns the spindle real position value, when it is in closed loop (M19). Its value will be given in 0.0001 degree units between ±999999999.

If the spindle is not in closed loop (M19), this variable will return a value of 0.

RPOSS: Returns the spindle real position value when operating in closed loop (M19) and as rollover. Its value will be given in 0.0001 degree units between 0 and 360º,

TPOSS: Returns the spindle theoretical position value (real + following error) when it is in closed loop (M19). Its value will be given in 0.0001 degree units between ±999999999.

When not in M19, this variable returns a "0" value.

RTPOSS: Returns the spindle theoretical position value (real + following error) when operating in closed loop (M19) and as rollover. Its value will be given in 0.0001 degree units between 0 and 360º.

FLWES: Returns the spindle following error when it is operating in closed loop (M19).

When the spindle is not in closed loop (M19), this variable will return a value of "0".

Read-and-write variables

PLCS: This variable allows reading or modifying the spindle speed in 0.0001 rev./min. units. A value of “0” means that it is not selected.

PLCCSS: This variable can only be used on FAGOR 8050 CNC LATHE models. It allows reading or modifying the constant surface speed selected by PLC. Its value is given in meters/min. or feet/min.

PLCSSO: It allows reading or modifying the spindle turning speed percentage which is selected by PLC. If this has a value of 0 it means that it is not selected.

PLCSL: It allows reading or modifying the spindle turning speed limit selected by PLC in 0.0001 rev./min units. If this has a value of 0 it means that it is not selected.

10.8 VARIABLES ASSOCIATED TO GLOBAL AND LOCAL ARITHMETIC PARAMETERS

The FAGOR 8050 CNC offers two types of general purpose variables, local parameters P0 thru P25 and global parameters P100 thru 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.

When reading the value of one of these parameters by means of functions GUP and LUP the obtained value will always be an integer (dropping the decimals if it has them). If the parameter value is greater than ±2147483647 the variable will return this maximum value.

Read-and-write variables

GUP n: This variable 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, M101); Assigns the value in register R101 to global parameter P155.

LUP a b: This variable allows reading or modifying the indicated local parameter (b) (P0-P25) corresponding to a nesting level (a).

CNCRD (LUP 3 15, R100, M102); Loads register R100 with the value of local parameter P15 corresponding to nesting level 3.

CNCWR (R101, LUP 2 15, M101); Assigns the value in R101 to local parameter P15 corresponding to nesting level 2.

10.9 *OTHER VARIABLES*

Read-only variables

OPMODE: 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

20 = Theoretical path movement simulation
21 = G functions simulation
22 = G, M, S and T functions simulation
23 = Simulation with movement on main plane
24 = Simulation with rapid movement

30 = Normal editing
31 = User editing
32 = TEACH-IN editing
33 = Interactive edior
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 = JOG user operation

50 = Zero offset table
51 = Tool Offset table
52 = Tool table
53 = Tool magazine table
54 = Global parameter table
55 = Local parameter table

60 = Utilities

70 = DNC

80 = Editing PLC files
81 = Compiling PLC program
82 = PLC monitoring
83 = Active PLC messages
84 = Active PLC pages (screens)
85 = Save PLC program
86 = Restore PLC program
87 = PLC usage maps
88 = PLC statistics

90 = Graphic Editor

100 = General machine parameter table
101 = Axis machine parameter tables
102 = Spindle machine parameter tables
103 = Serial port machine parameter tables
104 = PLC machine parameter table
105 = M function table
106 = Leadscrew and cross compensation table

110 = Diagnosis: configuration
111 = Diagnosis: hardware test
112 = Diagnosis: RAM memory test
113 = Diagnosis: EPROM memory test (checksums)
114 = User diagnosis

PRGN: Returns the program number which is being executed. Should none be selected, a value of -1 is returned.

BLKN: Returns the label number of the block being executed or that of the last block executed. If none, it returns -1.

GGSA: It returns the status of functions G00 thru G24. The status of each one of these 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.

00	00	00	00	00	00	00	G 24	G 23	G 22	G 21	G 20	—	G 02	G 01	G 00

LSB

CNCRD (GGSA, R100, M102); Loads register R100 with the status of functions G00 thru G24.

GGSB: It returns the status of functions G25 thru G49. The status of each one of these 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.

00	00	00	00	00	00	00	G 49	G 48	G 47	G 46	G 45	—	G 27	G 26	G 25

LSB

GGSC: It returns the status of functions G50 thru G74. The status of each one of these 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.

00	00	00	00	00	00	00	G 74	G 73	G 72	G 71	G 70		G 52	G 51	G 50

LSB

GGSD: It returns the status of functions G75 thru G99. The status of each one of these 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.

00	00	00	00	00	00	00	G 99	G 98	G 97	G 96	G 95		G 77	G 76	G 75

LSB

PLANE: Returns data on the abscissa axis (bits 4 to 7) and the ordinate axis (bits 0 to 3) of the active plane in 32 bits and in binary.

....	....	....	....	....	....	7654	3210

LSB

Ordinate axis

Abscissa axis

The axes are coded in 4 bits and indicate the axis number (from 1 to 6) according to the programming order.

Example: If the CNC controls the X,Y,Z,U,B,C axes and the ZX plane (G18) is selected.

CNCRD (PLANE, R100, M102); Loads register R100 with Hexadecimal value $31.

0000	0000	0000	0000	0000	0000	0011	0001

LSB

Abscissa axis = 3 (0011) --> Z axis

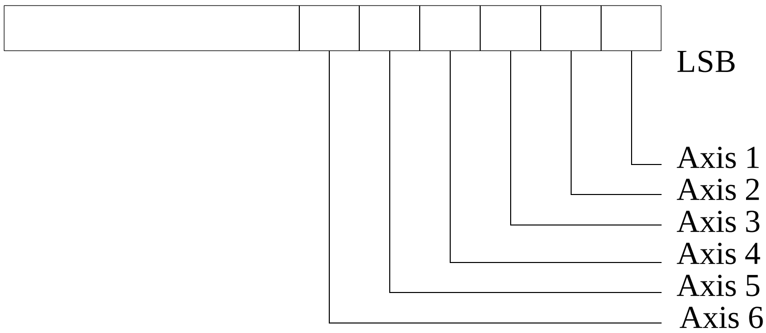
Ordinate axis = 1 (0001) --> X axis

LONGAX: This variable can only be used on the **Mill** model **FAGOR8050 CNC.** It returns the number (1 to 6) 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, R100, M102); Loads register R100 with a value of 4 (4th axis = U).

MIRROR: Returns in the 6 least significant bits of the 32-bit word, the status of the mirror image of each axis, 1 in the case of being active and 0 if not.



The axis name corresponds to the number (1 to 6) 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: Returns the active general scaling factor multiplied by 10000.

SCALE(X-C): Returns the specific scaling factor of the indicated axis (X-C) multiplied by 10000.

ORGROT: Returns, in 0.0001 degree units, the rotation angle of the coordinate system selected with the G73 function.

PRBST: Returns the status of the probe.

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);	Loads register R100 hh-mm-ss. For example if the time is: 18h 22m 34sec. R100 = 182234.

DATE: Returns the date in year-month-day format.

CNCRD (DATE, R100, M102);	Loads register R100 with year-month-day. For example: if the date is April 25th 1992, R100 = 920425.

CYTIME: Returns in hundredths of a second the time elapsed in making the part not counting the time period when the program execution might have been stopped. Possible values 0...4294967295

The CNC will consider the execution of the program finished after executing the last block of the program 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 the “Shift-Reset” keys.
Every time a new program is selected.

ANAIn: Returns the value (+/-5 Volts) in 0.0001 mV units,of the indicated analog input (n), it being possible to select one among the eight (1...8) analog inputs.

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”.

Read-and-write variables

TIMER: This variable allows reading or modifying the time, in seconds, indicated by the clock enabled by the PLC. Possible values 0...4294967295

The CNC will set this value to 0 when changing the software version or when a checksum error occurs.

PARTC: The CNC has a part counter whose count increases every time M30 or M02 is executed and this variable allows its value to be read or modified. This value will be between 0 and 4294967295

The CNC will set this value to 0 when changing the software version or when a checksum error occurs.

KEY: This variable allows reading the last accepted keystroke or simulating the CNC keyboard assigning the desired key code to it.

CNCRD (KEY,R100,M102);	Loads register R100 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,KEYSRC,M101);	Indicates to the CNC that only keystrokes coming from the PLC must be processed (CNC keyboard inhibited).
CNCWR (R101, KEY, M101);	Indicates to the CNC that the key corresponding to the code contained in R101 has been pressed.
CNCWR (R110,KEYSRC,M101);	Process only keystrokes coming from the CNC (CNC keyboard enabled).

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 (by the PLC) 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 Volts.

The analog outputs which are free among the eight (1..8) available at the CNC may be modified, the corresponding error being displayed if an attempt is made to write in one which is occupied.

11. *AXES CONTROLLED FROM THE PLC*

The PLC offers the action: **CNCEX** which allows to send commands to the CNC in order for it to execute movements on one or several axes.

The programming format for this action is:

CNCEX (ASCII Block , Mark)

By means of this action, the PLC will send to the CNC the command indicated in the "ASCII Block" for it to execute it.

If the "ASCII Block" has been accepted by the CNC, the PLC will set the indicated Mark to "0" and to "1" if otherwise.

The CNC only indicates that the "ASCII block" has been accepted. It is up to the operator to check whether the command has been executed by the CNC or not.

Example: CNCEX (G1 U125 V300 F500, M200)

Sends to the CNC the command: G1 U125 V300 F500 so it carries out a linear interpolation between the U and V axes at a feedrate of F500 and being U125 V300 the end point.

11.1 CONSIDERATIONS

Set-up

Each axis machine parameter "**AXISTYPE**" must be set properly indicating whether that axis is controlled by the CNC or from the PLC.

An axis cannot be controlled both from the CNC and from the PLC at the same time.

PLC Execution channel

The CNC offers a parallel execution channel to execute commands received from the PLC. This channel will have its own history and it permits the execution of blocks programmed from the PLC regardless of the operating mode being selected at the CNC at the time.

When the CNC receives a command from the PLC and it is executing another command received previously, it will store it in an internal buffer and it will execute it as soon as the previous one is done.

The internal buffer can store up to three blocks received from the PLC besides the one currently in execution.

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.

11.2 BLOCKS WHICH CAN BE EXECUTED FROM THE PLC

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.

An ASCII block **may** contain preparatory functions, axis position and feedrate values. On the other hand, it **may not** contain any auxiliary functions M, S, or T; nor tool offset D as well as any high level language blocks.

The information which an ASCII block **may** contain is the following:

Preparatory functions

The preparatory functions which can be used in the PLC execution channel are the following:

G00: Rapid travers
G01: Linear interpolation
G02: Clockwise circular (helical) interpolation
G03: Counter-clockwise circular (helical) interpolation
G04: Dwell
G05: Round corner
G06: Arc center in absolute coordinates
G07: Square corner
G08: Arc tangent to previous path.
G09: Arc defined by three points
G16: Main plane selection by two addresses
G50: Controlled corner rounding
G52: Movement until making contact
G53: Programming with respect to machine reference zero (home)
G70: Inch programming
G71: Metric programming
G74: Home search
G75: Probing move until touching
G76: Probing move while touching
G90: Absolute programming
G91: Incremental programming
G92: Preset
G94: Feedrate in millimeters (inches) per minute.
G95: Feedrate in millimeters (inces) per revolution.

All these functions must be programmed as indicated in the programming manual and all their related notes in the manual are valid.

Axes position values (coordinates)

Only those axes set by means of machine parameter "**AXISTYPE**" 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.

Example:

...... =	MOV 150 R1	Assigns a 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.

Axes feedrate

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 move corresponds to a rotary axis, the CNC will assume the feedrate to be programmed in degrees/min.

Blocks programmed in high-level language

The programming of high-level language blocks from the PLC has the following restrictions:

* The programmed blocks can only operate with global arithmetic parameters.
* The "PCALL" and "MCALL" instructions cannot be used because they work with local arithmetic parameters.
* Up to 5 nesting levels are permitted for standard subroutines. Neither parametric nor global subroutines may be used.

Example in mm: Move the W axis to the position indicated by register R101.

Since the PLC works with 32-bit integers, the value contained in R2 is given in ten-thousandths of a millimeters (tenths of a micron).

CNCWR (R101, GUP 155, M101)	Assigns the value of R101 to global parameter P155
CNCEX (P155=P155/10000)	Converts the value of P155 to mm.
CNCEX (G1 WP155 F2000)	Moves the W axis.

11.3 CONTROL OF THE PLC PROGRAM FROM THE CNC

The section of the PLC program regarding the "axes controlled from the PLC" can be controlled from the CNC itself.

To do this, the inputs, outputs, marks, registers, timers or counters of the PLC itself are used.

The CNC has the following PLC related variables to read or change the status of the selected resource.

PLCI	To read or modify up to 32 PLC inputs.
PLCO	To read or modify up to 32 PLC outputs.
PLCM	To read or modify up to 32 PLC marks (internal relays).
PLCR	To read or modify the status of a register.
PLCT	To read or modify the count of a timer.
PLCC	To read or modify the count of a counter.

With these variables, the desired values will be assigned, in the part-program of the CNC, to the PLC resources used in the communication. The setting of these values will be carried out whenever an axis or axes are to be controlled from the PLC.

In turn, the PLC program must check the status of such resources and when detecting that one of them is activated, it must execute the corresponding section of the PLC program.

It is also possible to transfer data from the CNC to the PLC via global and local arithmetic parameters. The PLC has the following variables related to those CNC parameters:

GUP	To read or modify a global parameter of the CNC.
LUP	To read or modify a local parameter of the CNC.

Example:

The "U" axis is controlled by the PLC and we want to command it from any part-program of the CNC in such way that we could select the type of move (G00 or G01), the positioning coordinate and the feedrate for that move.

In order to command it from any part-program, it is convenient to have in a subroutine the section of the CNC program allowing the data transfer with the PLC.

This example uses subroutine SUB1 and, for data exchange, it uses global CNC parameters.

P100 Type of move. If P100 = 0 ==> G00; If P100 = 1 ==> G01.

P101 "U" axis positioning coordinate.

P102 Feedrate. It only makes sense when moving in G01.

To indicate to the PLC that it must execute this move, it activates the following PLC resource:

M1000 Command to begin movement.

Any part-program of the CNC may contain a block of the type:

(PCALL 1, G1, U100, F1000)

This block calls subroutine SUB1 and it transfers the local parameters G, U and F with the following information:

G Type of movement
U "U" axis positioning coordinate
F Feedrate for the movement

Subroutine SUB1 can be programmed as follows:

```
( SUB 1)
( P100 = G, P101 = U, P102 = F )      ;Data transfer to global parameters
( PLCM1000 = PLCM1000 OR 1 )          ;Execution command for the PLC.
( RET )
```

The PLC program, in turn, will have to contain the following instructions:

```
M1000 = CNCEX ( G90 GP100 UP101 FP102, M111)   ;When mark M1000 is active, it sends the indicated block to the CNC

NOT M111 = RES M1000                           ;If the CNC accepts this block, it resets mark M1000.
```

12. *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

Module	Device Select	Inputs	Outputs
CPU	1		
AXES	2	I1 - I40	O1 - O24
INPUTS-OUTPUTS (1)	3	I65 - I128	O33 - O64
INPUTS-OUTPUTS (2)	4	I129 - I192	O65 - O96
INPUTS-OUTPUTS (3)	5	I193 - I256	O97 - O128

Atention:

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.

Inputs I41 through I64 and outputs O25 through O32 are not physically connected to the outside world.

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.

----- Symbols used in : Basic and necessary programming -----

DEF	**I-EMERG**	**I1**	External emergency input
DEF	**I-CONDI**	**I10**	Conditional mode. The CNC interrupts part-program execution when executing auxiliary function M01
DEF	**SERVO-OK**	**I11**	The servo drives are O.K.
DEF	**O-EMERG**	**O1**	Emergency output. It must be normally high

----- Symbols used in: Treatment of the axis overtravel limit switches -----

DEF	**I-LIMTX1**	**I12**	X axis positive overtravel limit switch
DEF	**I-LIMTX2**	**I13**	X axis negative overtravel limit switch
DEF	**I-LIMTY1**	**I14**	Y axis positive overtravel limit switch
DEF	**I-LIMTY2**	**I15**	Y axis negative overtravel limit switch
DEF	**I-LIMTZ1**	**I16**	Z axis positive overtravel limit switch
DEF	**I-LIMTZ2**	**I17**	Z axis negative overtravel limit switch

----- Symbols used in: Treatment of the machine reference (home) switches -----

DEF	**I-REF0X**	**I18**	X axis home switch
DEF	**I-REF0Y**	**I19**	Y axis home switch
DEF	**I-REF0Z**	**I20**	Z axis home switch

----- Symbols 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

----- Symbols used in: Lubrication treatment -----

DEF	**I-LUBING**	**I21**	Operator request to lubricate the ways of the machine
DEF	**O-LUBING**	**O10**	Ways lubrication output

----- Symbols used in: Coolant treatment -----

DEF	**I-COOLMA**	**I22**	The coolant is controlled by the operator. Manual mode
DEF	**I-COOLAU**	**I23**	The CNC controls the the coolant. Automatic mode
DEF	**O-COOL**	**O11**	Coolant output

----- Symbols used in: Spindle turning control -----

DEF	**O-S-ENAB**	**O12**	Spindle enable output

----- Symbols used in: Treatment of the spindle speed range change -----

DEF	**O-RANGE1**	**O13**	Move gears to select range 1
DEF	**O-RANGE2**	**O14**	Move gears to select range 2
DEF	**I-RANGE1**	**I24**	Indicates that Range 1 is selected
DEF	**I-RANGE2**	**I25**	Indicates that Range 2 is selected

----- Symbols used in: Keyboard simulation -----

DEF	**I-SIMULA**	**I26**	The operator requests the simulation of program P12
DEF	**SENDKEY**	**M1100**	Indicates that the code of a key is to be sent out to the CNC
DEF	**KEYCODE**	**R55**	Indicates the code of the key to be simulated
DEF	**LASTKEY**	**R56**	Indicates which is the last key accepted by the CNC
DEF	**SENTOK**	**M1101**	Indicates that the key code has been sent correctly
DEF	**KEYBOARD**	**R57**	Used to indicate to the CNC the source of the keys
DEF	**CNCKEY**	**0**	Used to indicate that the keys come from the CNC keyboard
DEF	**PLCKEY**	**1**	Used to indicate that the keys come from the PLC

DEF	**MAINMENU**	**$FFF4**	Code of the "MAIN MENU" key
DEF	**SIMULATE**	**$FC01**	Code of the "SIMULATE" key (F2)
DEF	**KEY1**	**$31**	Code of the "1" key
DEF	**KEY2**	**$32**	Code of the "2" key
DEF	**ENTER**	**$0D**	Code of the "ENTER" key
DEF	**THEOPATH**	**$FC00**	Code of the "THEORETICAL PATH" key (F1)
DEF	**START**	**$FFF1**	Code of the "START" key

FIRST CYCLE MODULE

CY1

( ) = ERA O1 256 = ERA C1 256 = ERA T1 256 = ERA R1 256 = ERA M1 2000
= ERA M4000 4127 = ERA M4500 4563 = ERA M4700 4955
Initializes all PLC resources to low logic level "0"

( ) = TG1 2 120000 Initializes the timer which controls the lubrication of the machine ways on power-up. This lubrication lasts 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

MAIN MODULE

PRG
REA

----- Basic and necessary programming -----

( ) = /STOP Permission to execute the part-program
= /FEEDHOL Permission to move the axes
= /XFERINH Permission to execute the next block

I-EMERG AND (rest of conditions) = /EMERGEN

If the external emergency input is activated or any other emergency occurs, the general logic input /EMERGEN of the CNC. When there is no emergency, this signal must remain high.

/ALARM AND CNCREADY = O-EMERG

The emergency output, O1, of the PLC (O-EMERG) must be normally high
If an alarm or emergency is detected at the CNC (/ALARM) or a problem was detected when powering the CNC up (CNCREADY), the emergency output O-EMERG must be brought low

I-CONDI = M01STOP

When the operator selects the conditional mode (I-CONDI), the CNC general logic input M01STOP must be activated. It interrupts the program when executing M01

START AND (rest of conditions) = CYSTART

When the cycle START key is pressed, the CNC activates the general logic output START. The PLC must check that the rest of the conditions (hydraulic, safety devices, etc.) are met before setting the general input CYSTART high in order to start executing the program

SERVO-OK AND NOT LOPEN = SERVO1ON = SERVO2ON = SERVO3ON

If the servo drives are OK and the CNC does not detect any errors in the positioning loop of the axes (LOPEN), the positioning loop must be closed on all axes. Axis logic inputs of the CNC: SERVO1ON, SERVO2ON, SERVO3ON.

----- Treatment of the axis overtravel limit switches -----

I-LIMTX1 = LIMIT+1
I-LIMTX2 = LIMIT-1
I-LIMTY1 = LIMIT+2
I-LIMTY2 = LIMIT-2
I-LIMTZ1 = LIMIT+3
I-LIMTZ2 = LIMIT-3

----- Treatment of the machine reference (home) switches -----

I-REF0X = DECEL1
I-REF0Y = DECEL2
I-REF0Z = DECEL3

----- Message treatment -----

The PLC allows displaying the corresponding PLC message at the CNC screen by activating marks MSG1 through 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 -----

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 table.

The next example shows how to generate the X axis overtravel limit overrun error when one of the overtravel limit switches is pressed.

NOT I-LIMTX1 OR NOT I-LIMTX2 = ERR10

----- Treatment of M, S, T functions -----

The CNC activates the general logic output MSTROBE to "tell" the PLC to execute the M functions indicated at the variables MBCD1 through MBCD7.

It also activates: the SSTROBE output when the S function indicated at variable SBCD must be executed, the TSTROBE output when the T function indicated at variable TBCD must be executed and the T2STROBE output when the T function indicated at variable T2BCD must be executed.

Whenever the CNC activates one of these signals, it is convenient to deactivate the general CNC input AUXEND in order to interrupt the execution of the CNC. When the PLC concludes the processing of the required function, this AUXEND signal must be activated back so that the CNC resumes the execution of the interrupted program.

This example deactivates the AUXEND signal for 100 milliseconds using the timer T1.

MSTROBE OR SSTROBE OR TSTROBE OR T2STROBE = TG1 1 100

The activation of the STROBE signals activates timer T1 in the mono-stable mode for 100 milliseconds.

Whenever timer T1 is active, the PLC must set the AUXEND signal low as described in: "Treatment of the general CNC input AUXEND".

When the CNC activates the MSTROBE signal, the contents of variables MBCD1 through MBCD7 must be analyzed in order to know which auxiliary functions are to be executed. All MBCD variables may be analyzed at the same time by using "MBCD*".

This example **SET**s the auxiliary marks so they can be analyzed later. Once analyzed, they must be **RES**et so that the PLC does not analyze them again on the next cycle (scan)

DFU MSTROBE AND CPS MBCD* EQ $0 = RES M-08
DFU MSTROBE AND CPS MBCD* EQ $2 = RES M-08

Functions M00 and M02 cancel the coolant (M08)

DFU MSTROBE AND CPS MBCD* EQ $3 = SET M-03 = RES M-04
DFU MSTROBE AND CPS MBCD* EQ $4 = SET M-04 = RES M-03
DFU MSTROBE AND CPS MBCD* EQ $5 = RES M-03 = RES M-04

Functions M03 and M04 are incompatible with each other and M05 cancels both.

DFU MSTROBE AND CPS MBCD* EQ $8 = SET M-08
DFU MSTROBE AND CPS MBCD* EQ $9 = RES M-08
DFU MSTROBE AND CPS MBCD* EQ $30 = RES M-08

Functions M09 and M30 cancel the coolant (M08)

DFU MSTROBE AND CPS MBCD* EQ $41 = SET M-41 = RES M-42
DFU MSTROBE AND CPS MBCD* EQ $42 = SET M-42 = RES M-41

Functions M41 and M42 are incompatible with each other.

----- Spindle turning control -----

The spindle enable output O-S-ENAB will be activated when selecting function M03 or M04.

M-03 OR M-04 = O-S-ENAB

----- Treatment of spindle speed range change -----

The spindle in this example has two ranges (high and low). To perform a range change, the following steps must be taken:

* Deactivate the general CNC input AUXEND
* Remove the control of the spindle loop from the CNC and give it to the PLC
* Output an oscillating analog signal to change gears
* Move the gears
* Verify that the range change has been completed
* Remove the oscillating analog signal
* Return the control of the spindle loop to the CNC
* Activate the general CNC input AUXEND

Deactivate the general CNC input AUXEND

While changing gears, it is recommended to keep the general CNC input AUXEND deactivated so that the CNC interrupts the execution of the program as described in "Treatment of the general CNC input AUXEND".

Remove the control of the spindle loop from the CNC and give it to the PLC.
Output an oscillating analog signal to change gears.

DFU M-41 OR DFU M-42	When a range 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 range output (O-RANGE) is kept active until the range selection is completed (I-RANGE).

M-41 AND NOT I-RANGE1 = O-RANGE1
M-42 AND NOT I-RANGE2 = O-RANGE2

Verify that the gear change has been completed
Remove the oscillating analog signal
Return the control of the spindle back to the CNC

M-41 AND I-RANGE1 = RES M-41	Once the gear change has concluded, the following must be done:
M-42 AND I-RANGE2 = RES M-42	... remove the request for range 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-RANGE1 = GEAR1	Also, the corresponding logic CNC input (GEAR1, GEAR2) must
I-RANGE2 = GEAR2	be activated to confirm the range change.

----- Lubrication of the machine ways -----

In this example, the machine axes are lubricated in the following instances:
* On machine power-up. For 2 minutes.
* When requesting a manual lubrication. For 5 minutes.
* After the axes have been moving for 1 hour. For 5 minutes.
* After an axis has travelled a specific distance since last lubricated. For 4 minutes.

Lubrication on machine power-up.

This operation will be performed for 2 minutes.

Whenever the machine is powered up, the PLC program starts running. Therefore, the first cycle module CY1 must activate timer T2 in the mono-stable mode for 2 minutes (120000 milliseconds).

( ) = TG1 2 120000

Manual lubrication

This operation will last 5 minutes and it will be performed at operator's request.

DFU I-LUBING = TG1 3 300000

Whenever the operator requests the lubricating (lubing) operation, T3 must be activated in the mono-stable mode for 5 minutes (300000 milliseconds).

Lubrication every hour of axis motion

This operation takes place when the axes of the machine have been moving for an accumulated time period of 1 hour. They will be lubricated for 5 minutes.

Timer T4 is used to keep track of the axis accumulated moving time and T5 to time the 5 minute lubrication period.

The first cycle module CY1 must activate timer T4 in the delayed activation mode with a time constant of 1 hour (3600 000 milliseconds).

( ) = TG2 4 3600000

ENABLE1 OR ENABLE2 OR ENABLE3 = TEN 4

T4 only times when any of the axis is moving.

T4 = TG1 5 300000 After having timed 1 hour, T5 must be activated in the mono-stable mode for 5 minutes. (300000 milliseconds)

T5 = TRS 4 = TG2 4 3600000 Resets the axis-motion timer T4 to zero.

Lubrication when an axis has travelled a specific distance since the last time it was lubricated

PLC machine parameters "USER12", "USER13" and "USER14" 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", "USER13" and "USER14"

( ) = 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 travelled 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 travelled by each axis.

Activate the lubricating (lubing) operation

T2 OR T3 OR T5 OR T6 = O-LUBING
If any of these conditions is met, the lubing output will be activated.

DFD O-LUBING = TRS2 = TRS3 = TRS4 = TRS5 = TRS6
Once the lubricating operation has concluded, All timers must be reset to "0".

----- Coolant treatment -----

The CNC executes function M08 to turn the coolant on and function M09 to turn it off.

Also, in this case, the operator has a switch to select whether the coolant is activated manually by the operator or automatically by the CNC.

I-COOLMA	The operator control the coolant. Manual mode.
I-COOLAU	The CNC controls the coolant. Automatic mode.
O-COOL	Coolant on/off output.

I-COOLMA OR (I-COOLAU AND M-08) = O-COOL Coolant ON.

RESETOUT = NOT O-COOL = RES M-08
The coolant will be turned off when the CNC is reset to initial conditions (RESETOUT) or when executing functions M00, M02, M09 and M30.

This instruction does not contemplate functions M00, M02, M09 and M30 since the treatment of M, S, T functions turns mark M-08 off when activating any of them.

----- Treatment of the general CNC input AUXEND -----

It is advisable to have one single instruction to control each one of the logic CNC inputs, thus preventing undesired functioning.

When having several instructions which can activate or deactivate an input, the PLC will always assign the result of analyzing the last one of those instructions.

This example shows how to group in a single instruction all the conditions that activate or deactivate one logic CNC input.

NOT T1 AND NOT M-41 AND NOT M-42 = AUXEND

Input AUXEND will remain low while:

* The "Treatment of the MSTROBE, TSTROBE, STROBE signals" is in progress (timer T1 active)

* A spindle range change is being performed (M-41, M-42)

----- Keyboard simulation -----

With this example it is possible to simulate the theoretical path of part-program P12 whenever the operator requests it.

To do this, follow these steps:

* Indicate to the CNC that from now on the keys will come from the PLC.
* Simulate all the necessary steps sending the code of each one of the keys.
* Indicate to the CNC that from now on the keys will be coming from the CNC keyboard, not from the PLC.

In order to make sending the keys easier, a subroutine is used which utilizes the following parameters:

SENDKEY	(Send Key) Calling parameter that must be activated whenever a key is to be sent.
KEYCODE	(Code of the key) Calling parameter that must contain the code corresponding to the key being simulated.
SENTOK	(Sent OK) Outgoing parameter indicating that the key code has been sent successfully.

DFU I-SIMULA = SET M120 = ERA M121 126

Whenever the operator requests the simulation (I-SIMULA), marks M120 through M126 must be activated....

= MOV PLCKEY KEYBOARD = CNCWR (KEYBOARD, KEYSRC, M100)

.. indicate to the CNC that, from now on, the keys will be coming from the PLC (PLCKEY)

= MOV MAINMENU KEYCODE = SET SENDKEY

... and send the code for the "MAIN MENU" key.

M120 AND SENTOK = RES M120 = RES SENTOK = SET M121

If the previous key was sent out successfully (SENTOK), flags M120 and SENTOK will be turned off, the flag for the next stage (M121) is activated ....

= MOV SIMULATE KEYCODE = SET SENDKEY

... and the code for the SIMULATE key (F2) is sent out.

M121 AND SENTOK = RES M121 = RES SENTOK = SET M122

If the previous key was sent out successfully (SENTOK), flags M121 and SENTOK will be turned off, the flag for the next stage (M122) is activated ....

= MOV KEY1 KEYCODE = SET SENDKEY

... and the code for the "1" key is sent out.

M122 AND SENTOK = RES M122 = RES SENTOK = SET M123

If the previous key was sent out successfully (SENTOK), flags M122 and SENTOK will be turned off, the flag for the next stage (M123) is activated ....

= MOV KEY2 KEYCODE = SET SENDKEY

...and the code for the "2" key is sent out.

M123 AND SENTOK = RES M123 = RES SENTOK = SET M124

If the previous key was sent out successfully (SENTOK), flags M123 and SENTOK will be turned off, the flag for the next stage (M124) is activated ....

= MOV ENTER KEYCODE = SET SENDKEY

...and the code for the "ENTER" key is sent out.

M124 AND SENTOK = RES M124 = RES SENTOK = SET M125

If the previous key was sent out successfully (SENTOK), flags M124 and SENTOK will be turned off, the flag for the next stage (M125) is activated ....

= MOV THEOPATH KEYCODE = SET SENDKEY

... and the code for the "THEORETICAL PATH" (F1) is sent out.

M125 AND SENTOK = RES M125 = RES SENTOK = SET M126

If the previous key was sent out successfully (SENTOK), flags M125 and SENTOK will be turned off, the flag for the next stage (M126) is activated ....

= MOV START KEYCODE = SET SENDKEY

... and the code for the START key is sent out.

M126 AND SENTOK = RES M126 = RES SENTOK

If the last key was sent out successfully (SENTOK), flags M126 and SENTOK will be turned off....

= MOV CNCKEY KEYBOARD = CNCWR (KEYBOARD, KEYSRC, M100)

.. and the CNC is "told" that from now on the keys will be coming from CNC keyboard (CNCKEY), not from the PLC.

Subroutine used to send a key

SENDKEY =SET M100 =SET M101 =SET M102 =RES SENDKEY

To send a key (SENDKEY), set to "1" internal marks M100 through M102 and reset the SENDKEY flag to "0".

M100 = CNCWR (KEYCODE, KEY, M100)

Sends to the CNC the code of the key to be simulated (KEYCODE). If this command is not executed correctly (M100=1), the PLC will try again on the next cycle scan.

M101 AND NOT M100 = CNCRD (KEY, LASTKEY, M101)

If the previous command was executed correctly, (M100=0), it reads the last key accepted by the CNC (LASTKEY).

M102 AND NOT M101 AND CPS LASTKEY EQ KEYCODE

If the previous command was executed correctly (M101=0) and the CNC accepted the key sent to it (LASTKEY = KEYCODE), ...

= RES M102 = SET SENTOK

.... the flag is turned off (M102=0) and the key is considered to be sent out successfully (SENTOK=1)...

= NOT M101

... But if the CNC did not accept the key sent to it, it waits until it does (M101=1).

End of subroutine

END *End of program*

APPENDIX A

TECHNICAL CHARACTERISTICS OF THE 8050 CNC

CENTRAL UNIT

GENERAL CHARACTERISTICS

8 feedback inputs up to 5 axes + spindle encoder + electronic handwheel.
8 analog inputs to monitor external devices: ±5V. Resolution : 46.8 mV
8 analog outputs: ±10 V. (one per axis + spindle)
Approximate weight: 12 Kg
Maximum consumption in normal operation : 80W

PACKAGING

Meets the "EN 60068-2-32" standard

FAN MODULE

Supplied with a separate power supply independent from the rest of the equipment.
DC: 24V DC, +15% -25%

POWER SUPPLY MODULE

High performance Switching power supply.
Universal power supply with any input between 100 V AC and 240 V AC (±10% and -15%).
AC frequency: 50 - 60 Hz ±1% and ±2% during very short periods.
Power outages. Meets the EN 61000-4-11 standard. It is capable of withstanding micro outages of up to 10 milliseconds and 50 Hz starting from 0º and 180º (dual polarity, positive and negative)
Harmonic distortion: Less than 10% of the rms voltage between low voltage conductors (sum of the 2nd through the 5th harmonic)

CPU MODULE

32 bit processor
Math coprocessor
Graphics coprocessor
128 Kb CNC program memory expandible to 640 Kb.
10 ms block processing time with PLC CPU
15 ms block processing time without PLC CPU
4 ms sampling and analog output update time.
2 RS232C and RS422 communication lines
System software in removable cartridge.

AXES MODULE

8 feedback inputs up to 5 axes + spindle encoder + electronic handwheel
0.0001 or 0.00001 inch resolution
Multiplying factor up to x 25 with sinewave input.
Feedrates from 0.0001 mm/min up to 99999.9999 mm/min
(0.00001 inches/min. up to 3937 inches/min.
Maximum travel ±99999.9999 mm (±3937 inches)
Input for digital probe (TTL or 24 Vdc)
40 optocoupled digital inputs
24 optocoupled digital outputs
8 analog inputs for supervision and control of external devices: ±5V. Resolution 48.6 mV.
8 analog outputs: ±10V (one for each axis + spindle)
PLC with 16/32 bit processor.

INPUT/OUTPUT MODULE

64 optocoupled digital inputs
32 optocoupled digital outputs

PLC

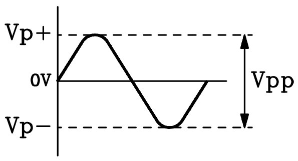
CPU with 64 Kb RAM, expandible to 256 Kb.
256 inputs
256 outputs
2047 user marks (internal relays)
256 32-bit registers
256 32-bit counters
256 32-bit timers
32 K EEPROM memory to store PLC and graphic editor programs (10,000 instructions if used only for PLC programs), expandible up to 128 K.
Programming in mnemonic and contact diagram
Time unit 1 millisecond
Cycle time 3 milliseconds/1000 instructions with PLC CPU
Cycle time 15 milliseconds/1000 instructions without PLC CPU

FEEDBACK INPUT ELECTRICAL CHARACTERISTICS

+5V power consumption. 2A (250 mA each axis)
-5V power consumption. 0.4A (100 mA each axis)
Operating levels for differential squarewave signals (A, /A, B, /B, Io, /Io)
- Maximum frequency: 425kHz
- Maximum separation between flanks: 460ns.
- Phase shift: 90° ± 20°
- Vmax. in common mode: ±7V.
- Vmax. in differential mode: ±6V
- Hysteresis: 0.2V
- Maximum differential input current: 3mA.

Operating levels for non-differential squareware signals (A, B, Io)
- Maximum frequency: 425kHz
- Maximum separation between flanks: 460ns.
- Phase shift: 90° ± 20°
- High threshold (logic level 1): $1.25V < V_{IH} < 7V$
- Low threshold (logic level 0): $-7V < V_{IL} < 1V$
- Vmax. in common mode: ±7V
- Hysteresis: 0.2 V
- Maximum input current: 3mA.

Operating levels for sinewave signals
- Maximum frequency: 50kHz
- Peak-to-peak voltage: 2V < Vpp < 6V.
- Vp+ = Vp- ±10%
- $V_{APP} = V_{BPP}$ ±10%
- Maximum input current: I_H = 1mA



INPUT ELECTRICAL CHARACTERISTICS

Nominal voltage + 24 V.D.C.
Maximum nominal value + 30 V.D.C.
Minimum voltage value + 18 V.D.C.
High threshold (logic level “1”) V_{IH} : from +18 V.D.C. up.
low threshold (logic level “0”) V_{IL} < +5 V.D.C. 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 V.D.C.

OUTPUT ELECTRICAL CHARACTERISTICS

Nominal supply voltage + 24 V.D.C.
Maximum nominal voltage + 30 V.D.C.
Minimum nominal voltage + 18 V.D.C.
Output voltage Vout = Supply voltage (VDC) -3 V
Maximum output amps 100 mA
Protection by means of galvanic isolation by optocouplers.
Protection by external 3 amp fuse against reverse connection of -30 V D.C. and against overvoltage of external power supply greater than 33V D.C.

ANALOG INPUT ELECTRICAL CHARACTERISTICS

Voltage within ±5 V
Resolution 46,8 mV
Shielded cable should be used.

ANALOG OUTPUT ELECTRICAL CHARACTERISTICS

Voltage within ±10 V
Minimum impedance of the connected connector: 10 KW
Shielded cable should be used.

ELECTRICAL CHARACTERISICS OF THE 5V PROBE INPUT

Typical value 0.25 mA. @ Vin = 5V.
High threshold (logic level “1”) $V_{IH} \geq 1.7$ V.
Low threshold (logic level “0”) $V_{IL} \leq 0.9$ V.
Maximum nominal voltage Vimax = +15 VDC.

ELECTRICAL CHARACTERISTICS OF THE 24V PROBE INPUT.

Typical value 0.30 mA. @ Vin = 24V.
High threshold (logic level “1”) $V_{IH} \geq 12.5$ V.
Low threshold (logic level “0”) $V_{IL} \leq 8.5$ V.
Maximum nominal voltage Vimax = +35 Vcc.

AMBIENT CONDITIONS

Relative humidity: 30-95% non condensing
Operating temperature: 5ºC - 40ºC (41º F - 104ºF) with an average lower than 35ºC (95º F)
Storage temperature : between 25º C (77ºF and 70º C (158º F).
Maximum operating altitude : Meets the “IEC 1131-2” standard.

VIBRATION

Under working conditions 10-50 Hz amplitude 0.2 mm
Under transport conditions 10-50 Hz amplitude 1 mm
Free fall of packaged equipment 1 m

ELECTROMAGNETIC COMPATIBILITY

See Declaration of Conformity in the introduction of this manual.

SAFETY

See Declaration of Conformity in the introduction of this manual

DEGREE OF PROTECTION

Central Unit : IP 2X

Accessible parts inside the enclosure: IP 1X

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

BATTERY

3.5 V lithium battery

Estimated life: 10 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.

Precaution, due to the risk of explosion or combustion.

- Do not attempt to recharge the battery.
- Do not expose to temperatures > 100°C (232° F)
- Do not shortcircuit the leads

MONITOR (Color 14")

CRT

Monitor: 14" Trinitron

Screen: Anti-glare

Resolution: 640 points x 480 lines

Deflection: 90 degrees

Phosphorous: P22

Screen surface: 255 x 193 mm

SWEEP FREQUENCY

Vertical synchronism: 60 Hz negative

Horizontal synchronism: 31.25 KHz negative

VIDEO INPUT SIGNALS

Separate video and synchronism signals

Impedance: 120 Ohms.

Differential RS-422 A (TTL level)

POWER SUPPLY

Alternating **C**urrent 220-240 V +10%, -15%

Consumption: 100 W maximum

Mains frequency 50 - 60 Hz ±1.

Fuse: 2 of 2AT 220V (2Amps. Slow)

CONTROLS

Brightness

Contrast

CONNECTORS

Monitor supply: Bipolar connection base + ground connection, according to IEC-320 and EEC-22 standards

Video signals: 25 pin SUB-D connector (male)

Keyboard connection: 25 pin SUB-D connector (female)

Control panel connection: 15 pin SUB-D connector (female)

PACKAGING

Meets the “EN 60068-2-32” standard

AMBIENT CONDITIONS

Relative humidity: 30-95% non condensing

Operating temperature: 5ºC - 40ºC (41º F - 104ºF) with an average lower than 35ºC (95º F)

Storage temperature : between 25º C (77ºF and 70º C (158º F).

Maximum operating altitude : Meets the “IEC 1131-2” standard.

ELECTROMAGNETIC COMPATIBILITY

See Declaration of Conformity in the introduction of this manual.

SAFETY

See Declaration of Conformity in the introduction of this manual.

DEGREE OF PROTECTION

Front panel:IP54

Rear panel: IP2X

Accessible parts inside the enclosure: IP 2X

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

Access to the inside of the unit is absolutely forbidden to non authorized personnel.

Atention:

To avoid excessive heating of internal circuits, the several ventilation slits must not be obstructed, it also being necessary to install a ventilation system which extracts hot air from the housing or desk which supports the MONITOR/KEYBOARD.

MONITOR (Color 10")

CRT

Monitor:	10" color	Deflection: :	90 degrees
Screen:	Anti-glare	Phosphorous:	P22
Resolution:	640 points x 480 lines	Display screen:	168 x 131 mm

SWEEP FREQUENCY

Vertical synchronism: 60 Hz negative   Horizontal synchronism: 31.25 KHz negative

VIDEO INPUT SIGNALS

Separate video and synchronism signals:	Differential RS-422 A (TTL level)
Impedance:	120 Ohms.

POWER SUPPLY

Universal **A**lternating **C**urrent 110V thru 240 V +10%, -15%

Consumption:	60 W maximum	Mains frequency 50 - 60 Hz ±1.
Fuse:	2 of 2AT 220V (2Amps. Slow)	

CONTROLS

Brightness   Contrast

CONNECTORS

Monitor supply: Bipolar connection base + ground connection, according to IEC-320 and EEC-22 standards
Video signals: 25 pin SUB-D connector (male)
Keyboard connection: 25 pin SUB-D connector (female)
Control panel connection: 15 pin SUB-D connector (female)

PACKAGING

Meets the “EN 60068-2-32” standard

AMBIENT CONDITIONS

Relative humidity: 30-95% non condensing
Operating temperature: 5ºC - 40ºC (41º F - 104ºF) with an average lower than 35ºC (95º F)
Storage temperature : between 25º C (77ºF and 70º C (158º F).
Maximum operating altitude : Meets the “IEC 1131-2” standard.

ELECTROMAGNETIC COMPATIBILITY

See Declaration of Conformity in the introduction of this manual.

SAFETY

See Declaration of Conformity in the introduction of this manual.

DEGREE OF PROTECTION

Front panel: IP54
Rear panel: IP2X
Accessible parts inside the enclosure: IP 2X

The machine manufacturer must comply with the “EN 60204-1 (IEC-204-1)”, standard regarding protection against electrical shock due to I/O contact failures with external power supply when not hooking up this connector before turning the power supply on. Access to the inside of the unit is absolutely forbidden to non authorized personnel.

Atention:

To avoid excessive heating of internal circuits, the several ventilation slits must not be obstructed, it also being necessary to install a ventilation system which extracts hot air from the housing or desk which supports the MONITOR/ KEYBOARD.

MONITOR (Monochrome 9")

CRT

Monitor:	9" Monochrome	Deflection:	90 degrees
Screen:	Anti-glare	Phosphorous:	H17 or Paper white
Resolution:	640 points x 480 lines	Display surface:	168 x 131 mm

SWEEP FREQUENCY

Vertical synchronism: 60 Hz negative   Horizontal synchronism: 31.25 KHz negative

VIDEO INPUT SIGNALS

Separate video and synchronism signals	Differential RS-422 A (TTL level)
Impedance:	120 Ohms.

POWER SUPPLY

Alternating **C**urrent. Selectable for 110V or 220V. +10%, -15%

Consumption:	30 W maximum	Mains frequency 50 - 60 Hz ±1.
Fuse:	2 of 2AF 220V (2Amps. Fast)	

CONTROLS

Brightness   Contrast

CONNECTORS

Monitor supply: Bipolar connection base + ground connection, according to IEC-320 and EEC-22 standards
Video signals: 25 pin SUB-D connector (male)
Keyboard connection: 25 pin SUB-D connector (female)
Control panel connection: 15 pin SUB-D connector (female)

PACKAGING

Meets the “EN 60068-2-32” standard

AMBIENT CONDITIONS

Relative humidity: 30-95% non condensing
Operating temperature: 5ºC - 40ºC (41º F - 104ºF) with an average lower than 35ºC (95º F)
Storage temperature : between 25º C (77ºF and 70º C (158º F).
Maximum operating altitude : Meets the “IEC 1131-2” standard.

ELECTROMAGNETIC COMPATIBILITY

See Declaration of Conformity in the introduction of this manual.

SAFETY

See Declaration of Conformity in the introduction of this manual.

DEGREE OF PROTECTION

Front panel: IP54
Rear panel: IP2X
Accessible parts inside the enclosure: IP 2X

The machine manufacturer must comply with the “EN 60204-1 (IEC-204-1)”, standard regarding protection against electrical shock due to I/O contact failures with external power supply when not hooking up this connector before turning the power supply on.
Access to the inside of the unit is absolutely forbidden to non authorized personnel.

Atention:

To avoid excessive heating of internal circuits, the several ventilation slits must not be obstructed, it also being necessary to install a ventilation system which extracts hot air from the housing or desk which supports the MONITOR/KEYBOARD.

APPENDIX B

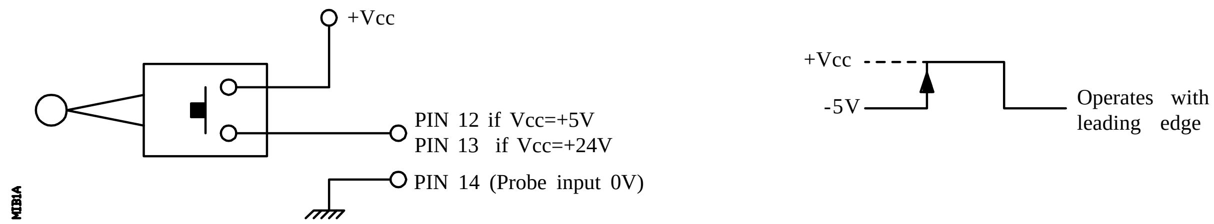
RECOMMENDED PROBE CONNECTION CIRCUITS

The CNC has two probe inputs situated in the X7 connector of the AXES module, one for 5 V inputs, the other for 24 V.

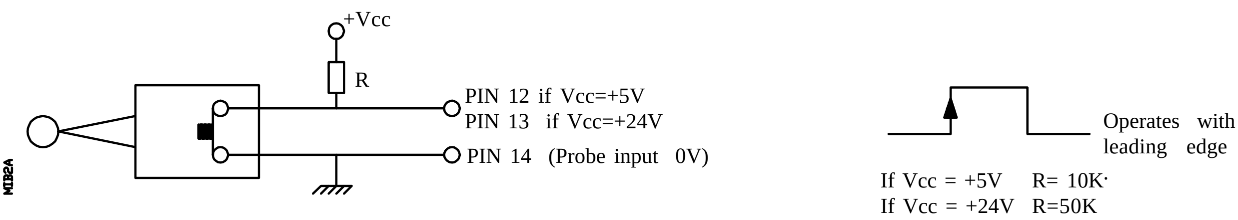
Depending upon the type of connection applied the general machine parameter "PRBPULSE" must be set, indicating whether it operates with the leading edge or trailing edge of the signal which the probe provides.

DIRECT CONNECTION

- Probe with "normally open contact" output



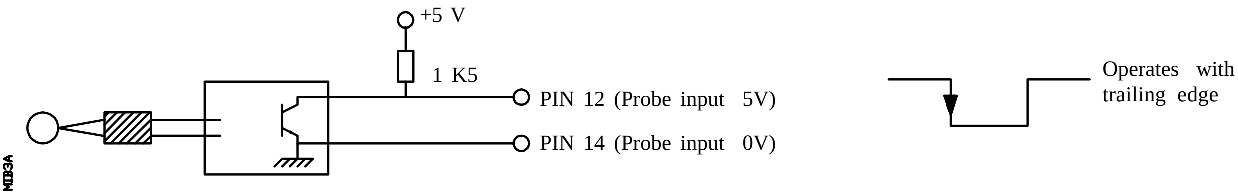
- Probe with "normally closed contact" output



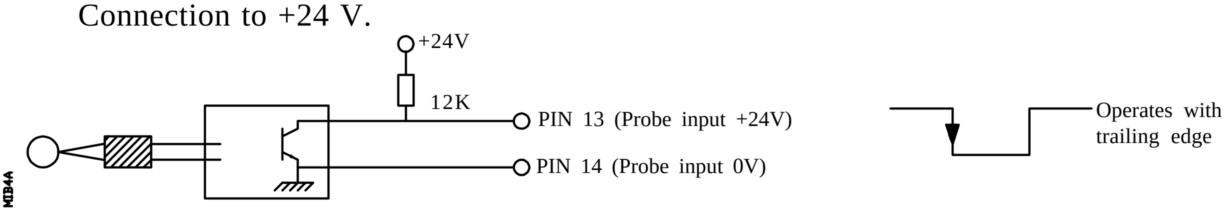
CONNECTION BY MEANS OF INTERFACE

- Interface with output in open collector

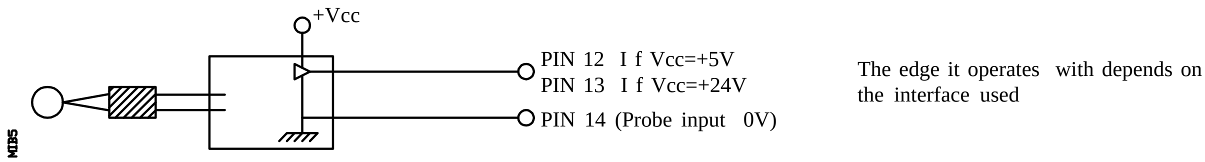
Connection to +5 V.



Connection to +24 V.



- Interface with output in PUSH-PULL



APPENDIX C

PLC PROGRAMMING INSTRUCTIONS

PLC RESOURCES AVAILABLE *(Chapter 6)*

Inputs:		**I** 1/256
Outputs:		**O** 1/256
Marks	user:	**M** 1/2000
	arithmetic flags:	**M** 2001
	clocks:	**M** 2009/2024
	set logic level:	**M** 2046/2047
	associated to messages:	**M** 4000/4127
	associated to errors:	**M** 4500/4563
	for screens or pages:	**M** 4700/4955
	to communicate with CNC:	**M** 5000/5957
Timers:		**T** 1/256
Counters:		**C** 1/256
User registers		**R** 1/256
Registers to communicate with 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	:	Any integer number within ±2147483647.
Hexadecimal	:	Preceded by the symbol $ and between 0 and FFFFFFFF
Binary	:	Preceded by the letter B and consisting of up to 32 bits (1 or 0).

DIRECTING INSTRUCTIONS *(Section 7.2)*

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.

SIMPLE CONSULTING INSTRUCTIONS *(Section 7.3.1)*

I 1/256	Inputs
O 1/256	Outputs
M 1/5957	Marks
T 1/256	Timers
C 1/256	Counters
B 0/31 **R** 1/256	Register bit

FLANK DETECTING INSTRUCTIONS *(Chapter 7.3.2)*

DFU Up flank detection.
DFD Down flank detection.

DFU DFD	I 1/256 O 1/256 M 1/5957

COMPARATIVE INSTRUCTIONS *(Section 7.3.3)*

CPS allows comparisons.

CPS	T 1/256 C 1/256 R 1/559 #	GT GE EQ NE LE LT	T 1/256 C 1/256 R 1/559 #

OPERATORS *(Section 7.4)*

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.

ACTION INSTRUCTIONS FOR BINARY ASSIGNMENT *(Section 7.5.1.1)*

= **I** 1/256 Inputs
= **O** 1/256 Outputs
= **M** 1/5957 Marks
= **TEN** 1/256 Timers
= **TRS** 1/256
= **TGn** 1/256 n/R
= **CUP** 1/256 Counters
= **CDW** 1/256
= **CEN** 1/256
= **CPR** 1/256 n/R
= **B** 0/31 **R** 1/256 Register Bits

CONDITIONED BINARY ACTION INSTRUCTIONS *(Section 7.5.1.2)*

= SET If the logic expression is “1”, this action assigns a “1” to the variable. If the logic expression is “0”, this action does not change the logic state of the variable.

= RES If the logic expression is “1”, this action assigns a “0” to the variable. If the logic expression is “0”, this action does not change the logic state of the variable.

= CPL If the logic expression is “1”, this action complements the logic state of the variable.

SET RES CPL	I 1/256 O 1/256 M 1/5957 B 0/31 R 1/559

JUMP ACTION INSTRUCTIONS *(Section 7.5.2)*

= JMP L 1/256 Unconditional Jump.
= RET Return or End of Subroutine.
= CAL L 1/256 Call a Subroutine.

ARITHMETIC ACTION INSTRUCTIONS *(Section 7.5.3)*

= MOV Transfers the logic states of the indicated source to the indicated destination.

	Source	Destination	Source Code	Destination Code	# bits to transmit
MOV	I 1/256 O 1/256 M 1/5957 T 1/256 C 1/256 R 1/559 #	I 1/256 O 1/256 M 1/5957 R 1/559	0(Bin) 1(BCD)	0(Bin) 1(BCD)	32 28 24 20 16 12 8 4

= NGU R 1/559 Complements all register bits.
= NGS R 1/559 Changes the sign of the Register contents.

= ADS Adds the contents of 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 (remainder of a division).

ADS SBS MLS DVS MDS	R1/559 #	R1/599 #	R1/559

LOGIC ACTION INSTRUCTIONS *(Section 7.5.4)*

= 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.

AND OR XOR	R1/559 #	R1/599 #	R1/559

= RR 1/2 Right-hand register rotation.
= RL 1/2 Left-hand register rotation.

RR1 RR2 RL1 RL2	R1/559	R1/559 0/31	R1/559

SPECIAL ACTION INSTRUCTIONS *(Section 7.5.5)*

= ERA Group erase

ERA	I 1/256 O 1/256 M 1/5957 T 1/256 C 1/256 R 1/559	1/256 1/256 1/5957 1/256 1/256 1/559

= CNCRD Read internal CNC variables.

CNCRD (Variable, R1/559, M1/5957)

= CNCWR Write (modify) internal CNC variables.

CNCWR (R1/559, Variable, M1/5957)

= PAR Parity of register.

PAR R1/559 M1/5957

LENGTH AND EXECUTION TIME OF THE INSTRUCTIONS

INSTRUCTION	EEPROM	OCCUPIED RAM (bytes)	EXECUTION TIME (µs)
PRG	2	0	0
PE	4	0	0
CY1	2	0	0
END	2	0	0
REA	2	0	0
IMA	2	0	0
Ln	3	0	0
DEF	0	0	0
In	3	6	2
On	3	6	2
Mn	4	6	2
Tn	3	6	2
Cn	3	6	2
BnRn	4	6	2
DFU I/O/M	4	20	5.75
DFD I/O/M	4	20	5.75
CPS	5/6	14	6.25
(	2	0	0
)	2	0	00
NOT	(*)	4	2
AND	(*)	4	2
OR	(*)	4	2
XOR	(*)	12	3
= In	3	18	5
= On	3	18	5
= Mn	4	18	5
= TGn	6/8	36	50/300
= TEN	4	22	50/250
= TRS	4	22	50/200
= CPR	6/8	30	38
= CUP	4	22	48
= CDW	4	22	48
= CEN	4	18	44
= BnRn	4	18	3
= SET	4	8	3.25/5
= RES	4	8	3.25/5
= CPL	4	8	3.25/5
= JMP Ln	3	16	3/6.75
= CAL Ln	3	16	3/11
= RET	2	8	3

INSTRUCTION	EEPROM	OCCUPIED RAM (bytes)	EXECUTION TIME (µs)
= MOV	7/9	18/74	12/590
= NGU	4	14	25
= NGD	4	14	25
= ADS	8/12	24	28
= SBS	8/12	24	28
= MLS	8/12	30	74
= DVS	8/12	30	100
= MDS	8/12	30	100
= AND	8/12	30	30
= OR	8/12	30	30
= XOR	8/12	26	28
= RR1	8/10	22	23
= RR2	8/10	22	23
= RL1	8/10	22	23
= RL2	8/10	22	23
= ERA	8/6	44	50/700
= CNCRD	10/20	50	300/500
= CNCWR	10/20	50	300/500

(*) Included in the operand.

If the result of an expression is known before finishing the whole calculation, the PLC ignores the rest of the calculations.

Example:

In the expression: I6 OR (M3 AND NOT I7) = O12

Whenever I6=1, the PLC will ignore the calculation between () since the result of the expression will be “1” regardless of the status of ().

APPENDIX D

INTERNAL CNC VARIABLES

R indicates that the variable can be read.
W indicates that the variable can be modified.

VARIABLES ASSOCIATED WITH TOOLS

(Section 10.1)

Variable	CNC	PLC	DNC	
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).
TORn	R/W	R/W	-	Tool radius (R) value of offset (n). Mill model.
TOLn	R/W	R/W	-	Tool length (L) value of offset (n). Mill model.
TOIn	R/W	R/W	-	Tool radius wear (I) of offset (n). Mill model.
TOKn	R/W	R/W	-	Tool length wear (K) of offset (n). Mill model.
TOXn	R/W	R/W	-	Tool length offset (n) along X axis. Lathe model.
TOZn	R/W	R/W	-	Tool length offset (n) along Z axis. Lathe model.
TOFn	R/W	R/W	-	Location code (F) of offset (n). Lathe model.
TORn	R/W	R/W	-	Tool radius (R) value of offset (n). Lathe model.
TOIn	R/W	R/W	-	Tool length wear (I) of offset (n) along X axis. Lathe model
TOKn	R/W	R/W	-	Tool length wear (K) of offset (n) along Z axis. Lathe model
NOSEAn	R/W	R/W	-	Cutter angle of indicated tool (n). Lathe model.
NOSEWn	R/W	R/W	-	Cutter width of indicated tool (n). Lathe model.
CUTAn	R/W	R/W	-	Cutting angle of indicated tool (n). Lathe model.

VARIABLES ASSOCIATED WITH ZERO OFFSETS

(Section 10.2)

Variable	CNC	PLC	DNC	
ORG(X-C)	R	R	-	Zero offset active on the selected axis without including the additive Zero offset activated via PLC.
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.

VARIABLES ASSOCIATED WITH MACHINE PARAMETERS

(Section 10.3)

Variable	CNC	PLC	DNC	
MPGn	R	R	-	Value assigned to general machine parameter (n).
MP(X-C)n	R	R	-	Value assigned to machine parameter (n) of an axis.
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. Lathe model.
MPLCn	R	R	-	Value assigned to machine parameter (n) of the PLC.

VARIABLES ASSOCIATED WITH THE WORK ZONES

(Section 10.4)

Variable	CNC	PLC	DNC	
FZONE	R	R/W	R	Status of work zone 1.
FZLO(X-C)	R	R/W	R	Lower limit of work zone 1 along the selected axis (X/C).
FZUP(X-C) R	R/W	R/W	R	Upper limit of work zone 1 along the selected axis (X/C).
SZONE	R	R/W	R	Status of work zone 2.
SZLO(X-C)	R	R/W	R	Lower limit of work zone 2 along the selected axis (X/C).
SZUP(X-C) R	R/W	R/W	R	Upper limit of work zone 2 along the selected axis (X/C).
TZONE	R	R/W	R	Status of work zone 3.
TZLO(X-C)	R	R/W	R	Lower limit of work zone 3 along the selected axis (X/C).
TZUP(X-C)	R	R/W	R	Upper limit of work zone 3 along the selected axis (X/C).

VARIABLES ASSOCIATED WITH FEEDRATES

(Section 10.5)

Variable	CNC	PLC	DNC	
FREAL	R	R	R	Real feedrate of the CNC in mm/min or inch/min.
FEED	R	R	R	Active feedrate at the CNC (G94) 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.
FPREV	R	R	R	Active feedrate at CNC (G95), 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.
FRO	R	R	R	Feedrate Override (%) active at the CNC.
PRGFRO	R/W	R	R	Feedrate Override (%) selected by program.
DNCFRO	R	R	R/W	Feedrate Override (%) selected by DNC.
PLCFRO	R	R/W	R	Feedrate Override (%) selected by PLC.
CNCFRO	R	R	R	Feedrate Override (%) selected from the front panel knob.

VARIABLES ASSOCIATED WITH POSITION VALUES

(Section 10.6)

Variable	CNC	PLC	DNC	
PPOS(X-C)	R	-	-	Theoretical programmed position value (coordinate).
POS(X-C)	R	R	R	Real position value of the indicated axis.
TPOS(X-C)	R	R	R	Theoretical position value (real + lag) of the indicated axis.
FLWE(X-C)	R	R	R	Following error of the indicated axis.
DEFLEX	R	R	R	Probe deflection along X axis. Mill model.
DEFLEY	R	R	R	Probe deflection along Y axis. Mill model.
DEFLEZ	R	R	R	Probe deflection along Z axis. Mill model.
DIST(X-C)	R/W	R/W	R	Distance travelled by the indicated axis.

VARIABLES ASSOCIATED WITH THE MAIN SPINDLE *(Section 10.7)*

Variable	CNC	PLC	DNC	
SREAL	R	R	R	Real spindle speed in r.p.m.
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.
CSS	R	R	R	Constant surface feed active at the CNC in meters/min or feet/min. Lathe model.
DNCCSS	R	R	R/W	Constant surface feed selected via DNC. Lathe model.
PLCCSS	R	R/W	R	Constant surface feed selected via PLC. lathe model.
PRGCSS	R	R	R	Constant surface feed selected by program.
SSO	R	R	R	Spindle Speed Override (%) active at the CNC.
PRGSSO	R/W	R	R	Spindle Speed Override (%) selected by program.
DNCSSO	R	R	R/W	Spindle Speed Override (%) selected via DNC.
PLCSSO	R	R/W	R	Spindle Speed Override (%) selected via PLC.
CNCSSO	R	R	R	Spindle Speed Override (%) selected from front panel.
SLIMIT	R	R	R	Spindle speed limit, in rpm, 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.
POSS	R	R	R	Real Spindle position. Between ±999999999 ten-thousandths º
RPOSS	R	R	R	Real Spindle position. Between 0 and 360º (in ten-thousandths º)
TPOSS	R	R	R	Theoretical Spindle position (real + lag) Between ±999999999 ten-thousandths of a degree.
RTPOSS	R	R	R	Theoretical Spindle position (real + lag). Between 0 and 360º (in ten-thousandths of a degree.
FLWES	R	R	R	spindle following error in Closed Loop (M19) in degrees.

VARIABLES ASSOCIATED WITH THE SECOND SPINDLE *(Section 10.8)*

Variable	CNC	PLC	DNC	
SSREAL	R	R	R	Real spindle speed in r.p.m.
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.
SCSS	R	R	R	Constant surface feed active at the CNC in meters/min or feet/min. Lathe model.
SDNCCS	R	R	R/W	Constant surface feed selected via DNC. Lathe model.
SPLCCS	R	R/W	R	Constant surface feed selected via PLC. lathe model.
SPRGCS	R	R	R	Constant surface feed selected by program.
SSSO	R	R	R	Spindle Speed Override (%) active at the CNC.
SPRGSO	R/W	R	R	Spindle Speed Override (%) selected by program.
SDNCSO	R	R	R/W	Spindle Speed Override (%) selected via DNC.
SPLCSO	R	R/W	R	Spindle Speed Override (%) selected via PLC.
SCNCSO	R	R	R	Spindle Speed Override (%) selected from front panel.
SSLIMI	R	R	R	Spindle speed limit, in rpm, 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.
SPOSS	R	R	R	Real Spindle position. Between ±999999999 ten-thousandths º
SRPOSS	R	R	R	Real Spindle position. Between 0 and 360º (in ten-thousandths º)
STPOSS	R	R	R	Theoretical Spindle position (real + lag) Between ±999999999 ten-thousandths of a degree.
SRTPOS	R	R	R	Theoretical Spindle position (real + lag). Between 0 and 360º (in ten-thousandths of a degree.
SFLWES	R	R	R	spindle following error in Closed Loop (M19) in degrees.

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	-	-	Indicated (n) Register.
PLCTn	R/W	-	-	Indicated (n) Timer's count.
PLCCn	R/W	-	-	Indicated (n) Counter's count.

VARIABLES ASSOCIATED WITH GLOBAL AND LOCAL ARITHMETIC PARAMETERS

(Section 10.9)

Variable	CNC	PLC	DNC	
GUP n	-	R/W	-	Global parameter (n) (100-P299).
LUP (a,b)	-	R/W	-	Local parameter (b) and its nesting level (a). (P0-P25).
CALLP	R	-	-	Indicates which local parameters have been defined by means of a PCALL or MCALL instruction (calling a subroutine).

OTHER VARIABLES

(Section 10.10)

Variable	CNC	PLC	DNC	
OPMODE	R	R	R	Operate mode.
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 (0..6, 8, 9, 19, 30, 41..44)
PLANE	R	R	R	Axes which form the active main plane.
LONGAX	R	R	R	Axis affected by the tool length compensation (G15). Mill model.
MIRROR	R	R	R	Active mirror images.
SCALE	R	R	R	Active general Scaling factor.
SCALE(X-C)	R	R	R	Scaling Factor applied only to the indicated axis.
ORGROT	R	R	R	Rotation angle (G73) of the coordinate system in degrees. Mill model.
ROTPF	R	-	-	Abscissa of rotation center. Mill model.
ROTPS	R	-	-	Ordinate of rotation center. Mill model.
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	Part counter of the CNC.
FIRST	R	R	R	Flag to indicate first time of program execution.
KEY	R/W*	R/W	R/W	keystroke code.
KEYSRC	R/W	R/W	R/W	Keystroke source, 0=keyboard, 1=PLC, 2=DNC
ANAIn	R	R	R	Voltage (in volts) of the indicated analog input (n).
ANAOn	W	W	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.

Atention: The "KEY" variable can be "written" at the CNC only via the user channel.

APPENDIX E

LOGIC CNC INPUTS AND OUTPUTS

GENERAL LOGIC INPUTS

(Section.9.1)

/EMERGEN	M5000	Stops axis feed and spindle rotation, displaying the corresponding error on screen.
/STOP	M5001	Stops execution of the part program, maintaining spindle rotation.
/FEEDHOL	M5002	Stops axis feed momentarily, maintaining spindle rotation.
/XFERINH	M5003	Prevents the following block from being executed, but finishes the block which is being executed.
CYSTART	M5007	Starts program execution.
SBLOCK	M5008	The CNC changes to the Single Block execution mode.
MANRAPID	M5009	Selects rapid travel for all the movements which are executed in Manual 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.
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: TIMER.
TREJECT	M5018	Rejection of tool in use.
PANELOFF	M5019	Deactivation of keyboard.
POINT	M5020	Takes a new digitized point.
PLCABORT	M5022	Possibility to abort the PLC channel
PLCREADY	M5023	PLC without errors.
INT1 INT2 INT3 INT4	M5024 M5025 M5026 M5027	Executes the interruption subroutine whose number is assigned to general machine parameter P35, P36, P37, P38 respectively.
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.

LIMIT+1 LIMIT-1 LIMIT+2 LIMIT-2 LIMIT+3 LIMIT-3 LIMIT+4 LIMIT-4 LIMIT+5 LIMIT-5 LIMIT+6 LIMIT-6	M5100 M5101 M5150 M5151 M5200 M5201 M5250 M5251 M5300 M5301 M5350 M5351	Indicates that the axis has overrun the travel limit. The CNC stops axis feed and spindle rotation, and displays the corresponding error on screen.
DECEL1 DECEL2 DECEL3 DECEL4 DECEL5 DECEL6	M5102 M5152 M5202 M5252 M5302 M5352	Machine reference search switch is pressed.
INHIBIT1 INHIBIT2 INHIBIT3 INHIBIT4 INHIBIT5 INHIBIT6	M5103 M5153 M5203 M5253 M5303 M5353	Momentarily prevents any axis movement.
MIRROR1 MIRROR2 MIRROR3 MIRROR4 MIRROR5 MIRROR6	M5104 M5154 M5204 M5254 M5304 M5354	Applies mirror image to the movements of the corresponding axis.
DRO1 DRO2 DRO3 DRO4 DRO5 DRO6	M5106 M5156 M5206 M5256 M5306 M5356	DRO axis This signal must be set high and SERVOON must be set low.
SERVO1ON SERVO2ON SERVO3ON SERVO4ON SERVO5ON SERVO6ON	M5107 M5157 M5207 M5257 M5307 M5357	Servo signal, must be set high in order to close the positioning loop.
AXIS+1 AXIS-1 AXIS+2 AXIS-2 AXIS+3 AXIS-3 AXIS+4 AXIS-4 AXIS+5 AXIS-5 AXIS+6 AXIS-6	M5108 M5109 M5158 M5159 M5208 M5209 M5258 M5259 M5308 M5309 M5358 M5359	Moves the axis (in the JOG Mode), in the indicated direction, positive (+) or negative (-). Similar to JOG keys.
SYNCHRO1 SYNCHRO2 SYNCHRO3 SYNCHRO4 SYNCHRO5 SYNCHRO6	M5112 M5162 M5212 M5262 M5312 M5362	Couples the axis electronically to the one defined by the axis machine parameter “SYNCHRO”.
LIM1OFF LIM2OFF LIM3OFF LIM4OFF LIM5OFF LIM6OFF	M5115 M5165 M5215 M5265 M5315 M5365	It ignores the software limits of the corresponding axis

LOGIC MAIN SPINDLE INPUTS

(Section 9.3)

LIMIT+S LIMIT-S	M5450 M5451	Indicates that the spindle has overrun the travel limit. The CNC stops axis feed and spindle rotation, and displays the corresponding error on screen.
DECELS	M5452	Machine reference search switch pressed.
SPDLEINH	M5453	Sends out zero analog output for spindle.
SPDLEREV	M5454	Reverses spindle rotation direction.
SERVOSON	M5457	Servo signal must be at high logic level to be able to move the spindle in Closed Loop (M19).
GEAR1 GEAR2 GEAR3 GEAR4	M5458 M5459 M5460 M5461	Indicates which spindle speed range is selected.
PLCM19 M19FEED	M5464 R505	Indicates positioning and fast synchronized speed in M19
PLCCNTL	M5465	The spindle is controlled directly by the PLC.
SANALOG	R504	Indicates spindle analog output. Only for the spindle controlled by the PLC.

LOGIC SECOND SPINDLE INPUTS

(Section 9.3)

LIMIT+S2 LIMIT-S2	M5475 M5476	Indicates that the spindle has overrun the travel limit. The CNC stops axis feed and spindle rotation, and displays the corresponding error on screen.
DECELS	M5477	Machine reference search switch pressed.
SPDLEINH	M5478	Sends out zero analog output for spindle.
SPDLEREV	M5479	Reverses spindle rotation direction.
SERVOSON	M5482	Servo signal must be at high logic level to be able to move the spindle in Closed Loop (M19).
GEAR1 GEAR2 GEAR3 GEAR4	M5483 M5484 M5485 M5486	Indicates which spindle speed range is selected.
PLCM19 M19FEED	M5489 R507	Indicates positioning and fast synchronized speed in M19
PLCCNTL	M5490	The spindle is controlled directly by the PLC.
SANALOG	R506	Indicates spindle analog output. Only for the spindle controlled by the PLC.

LOGIC KEY INHIBITION INPUTS

(Section 9.4)

KEYDIS1 KEYDIS2 KEYDIS3 KEYDIS4	R500 R501 R502 R503	Inhibit the operation of the panel keys.

CNCREADY	M5500	CNC without errors.
START	M5501	Indicates that the START key on the Front Panel has been pressed.
FHOUT	M5502	Indicates that program execution has been stopped.
RESETOUT	M5503	Indicates that the CNC is set at 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.
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). Mill model.
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.
CSS	M5523	The G96 function is selected.
SELECT0 SELECT1 SELECT2 SELECT3 SELECT4	M5524 M5525 M5526 M5527 M5528	Indicates in code the position selected on 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. Indicates that the punching may begin.
INTEREND	M5538	Indicates that interpolation has concluded.
INPOS	M5539	Axes are in position.
DM00	M5547	The execution of the program has stopped after executing the auxiliary M00 function.
DM01	M5546	The execution of the program has stopped after executing the auxiliary M01 function.
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 counter-clockwise (M04).
DM05	M5542	The spindle is stopped (M05).
DM06	M5541	The auxiliary M06 function has been executed.
DM08	M5540	The coolant output has been activated (M08).
DM09	M5555	The coolant output has been deactivated (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	The first spindle speed range has been selected (M41).
DM42	M5551	The second spindle speed range has been selected (M42).
DM43	M5550	The third spindle speed range has been selected (M43).
DM44	M5549	The fourth spindle speed range has been selected (M44).
DM45	M5548	Auxiliary spindle or live tool activated (M45).

ENABLE1 ENABLE2 ENABLE3 ENABLE4 ENABLE5 ENABLE6	M5600 M5650 M5700 M5750 M5800 M5850	Allow movement of axis.
DIR1 DIR2 DIR3 DIR4 DIR5 DIR6	M5601 M5651 M5701 M5751 M5801 M5851	Indicate axis movement direction.
ANT1 ANT2 ANT3 ANT4 ANT5 ANT6	M5606 M5656 M5706 M5756 M5806 M5856	If the programmed movement is smaller than the value of axis parameter "MINMOVE (P54)", the ANT* output goes high.
REFPOIN1 REFPOIN2 REFPOIN3 REFPOIN4 REFPOIN5 REFPOIN6	M5602 M5652 M5702 M5752 M5802 M5852	Machine reference search has already been done.
INPOS1 INPOS2 INPOS3 INPOS4 INPOS5 INPOS6	M5607 M5657 M5707 M5757 M5807 M5857	The corresponding axis is in position.

LOGIC MAIN SPINDLE OUTPUTS *(Section 9.7)*

ENABLES	M5950	Allows movement of spindle.
DIRS	M5951	Indicates in which direction the spindle is moving.
REFPOINS	M5952	The spindle reference point search has already been done.
CAXIS	M5955	The C axis is active
REVOK	M5956	The real revolutions of the spindle correspond to programmed ones.
INPOSS	M5957	The spindle is in position.

LOGIC SECOND SPINDLE OUTPUTS *(Section 9.7)*

ENABLES2	M5975	Allows movement of spindle.
DIRS2	M5976	Indicates in which direction the spindle is moving.
REFPOIS2	M5977	The spindle reference point search has already been done.
CAXIS2	M5980	The C axis is active
REVOK2	M5981	The real revolutions of the spindle correspond to programmed ones.
INPOSS2	M5982	The spindle is in position.

LOGIC AUXILIARY M, S, T FUNCTION OUTPUTS *(Section 8.1)*

MBCD1 MBCD2 MBCD3 MBCD4 MBCD5 MBCD6 MBCD7	R550 R551 R552 R553 R554 R555 R556	Indicate the auxiliary M functions which must be executed.
SBCD	R557	Indicates 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	Indicates the magazine position (pocket) for the tool which was in the spindle.

LOGIC OUTPUTS OF KEY STATUS *(Section 9.8)*

KEYBD1 KEYBD2 KEYBD3	R560 R561 R562	Indicate whether a key of the operator panel is pressed.

APPENDIX F

CONVERSION TABLE FOR S OUTPUT IN 2-DIGIT BCD CODE

Programmed S	"SBCD"	Programmed S	"SBCD"
0	S00	180-199	S65
1	S20	200-223	S66
2	S26	224-249	S67
3	S29	250-279	S68
4	S32	280-314	S69
5	S34	315-354	S70
6	S35	355-399	S71
7	S36	400-449	S72
8	S38	450-499	S73
9	S39	500-559	S74
10-11	S40	560-629	S75
12	S41	630-709	S76
13	S42	710-799	S77
14-15	S43	800-899	S78
16-17	S44	900-999	S79
18-19	S45	1000-1119	S80
20-22	S46	1120-1249	S81
23-24	S47	1250-1399	S82
25-27	S48	1400-1599	S83
28-31	S49	1600-1799	S84
32-35	S50	1800-1999	S85
36-39	S51	2000-2239	S86
40-44	S52	2240-2499	S87
45-49	S53	2500-2799	S88
50-55	S54	2800-3149	S89
56-62	S55	3150-3549	S90
63-70	S56	3550-3999	S91
71-79	S57	4000-4499	S92
80-89	S58	4500-4999	S93
90-99	S59	5000-5599	S94
100-111	S60	5600-6299	S95
112-124	S61	6300-7099	S96
125-139	S62	7100-7999	S97
140-159	S63	8000-8999	S98
160-179	S64	9000-9999	S99

APPENDIX G

KEY CODES

Each key can generate up to four different codes when pressed, depending on the status of the SHIFT and CAPS functions.

Therefore, when key A is pressed the following codes are obtained:

Hexad.	Decimal	
$61	097	If, when key A is pressed, no function is selected.
$41	065	If, when key A is pressed, the SHIFT function is selected.
$41	065	If, when key A is pressed, the CAPS function is selected.
$61	097	If, when key A is pressed, both functions are selected.

Key	Hexadecimal	Decimal
A	$61	097
A+SHIFT	$41	065
A+CAPS	$41	065
A+SHIFT+CAPS	$61	097
B	$62	098
B+SHIFT	$42	066
B+CAPS	$42	066
B+SHIFT+CAPS	$62	098
C	$63	099
C+SHIFT	$43	067
C+CAPS	$43	067
C+SHIFT+CAPS	$63	099
D	$64	100
D+SHIFT	$44	068
D+CAPS	$44	068
D+SHIFT+CAPS	$64	100
E	$65	101
E+SHIFT	$45	069
E+CAPS	$45	069
E+SHIFT+CAPS	$65	101
F	$66	102
F+SHIFT	$46	070
F+CAPS	$46	070
F+SHIFT+CAPS	$66	102
G	$67	103
G+SHIFT	$47	071
G+CAPS	$47	071
G+SHIFT+CAPS	$67	103

Key	Hexadecimal	Decimal
H	$68	104
H+SHIFT	$48	072
H+CAPS	$48	072
H+SHIFT+CAPS	$68	104
I	$69	105
I+SHIFT	$49	073
I+CAPS	$49	073
I+SHIFT+CAPS	$69	105
J	$6A	106
J+SHIFT	$4A	074
J+CAPS	$4A	074
J+SHIFT+CAPS	$6A	106
K	$6B	107
K+SHIFT	$4B	075
K+CAPS	$4B	075
K+SHIFT+CAPS	$6B	107
L	$6C	108
L+SHIFT	$4C	076
L+CAPS	$4C	076
L+SHIFT+CAPS	$6C	108
M	$6D	109
M+SHIFT	$4D	077
M+CAPS	$4D	077
M+SHIFT+CAPS	$6D	109
N	$6E	110
N+SHIFT	$4E	078
N+CAPS	$4E	078
N+SHIFT+CAPS	$6E	110

Key	Hexadecimal	Decimal
Ñ	$A5	164
Ñ+SHIFT	$A4	165
Ñ+CAPS	$A4	165
Ñ+SHIFT+CAPS	$A5	164
O	$6F	111
O+SHIFT	$4F	079
O+CAPS	$4F	079
O+SHIFT+CAPS	$6F	111
P	$70	112
P+SHIFT	$50	080
P+CAPS	$50	080
P+SHIFT+CAPS	$70	112
Q	$71	113
Q+SHIFT	$51	081
Q+CAPS	$51	081
Q+SHIFT+CAPS	$71	113
R	$72	114
R+SHIFT	$52	082
R+CAPS	$52	082
R+SHIFT+CAPS	$72	114
S	$73	115
S+SHIFT	$53	083
S+CAPS	$53	083
S+SHIFT+CAPS	$73	115
T	$74	116
T+SHIFT	$54	084
T+CAPS	$54	084
T+SHIFT+CAPS	$74	116

Key	Hexadecimal	Decimal
U	$75	117
U+SHIFT	$55	085
U+CAPS	$55	085
U+SHIFT+CAPS	$75	117
V	$76	118
V+SHIFT	$56	086
V+CAPS	$56	086
V+SHIFT+CAPS	$76	118
W	$77	119
W+SHIFT	$57	087
W+CAPS	$57	087
W+SHIFT+CAPS	$77	119
X	$78	120
X+SHIFT	$58	088
X+CAPS	$58	088
X+SHIFT+CAPS	$78	120
Y	$79	121
Y+SHIFT	$59	089
Y+CAPS	$59	089
Y+SHIFT+CAPS	$79	121
Z	$7A	122
Z+SHIFT	$5A	090
Z+CAPS	$5A	090
Z+SHIFT+CAPS	$7A	122
SP	$20	032
SP+SHIFT	$20	032
SP+CAPS	$20	032
SP+SHIFT+CAPS	$20	032

Key	Hexadecimal	Decimal
; **0** +SHIFT +CAPS +SHIFT+CAPS	 $30 $3B $30 $3B	 048 059 048 059
! **1** +SHIFT +CAPS +SHIFT+CAPS	 $31 $21 $31 $21	 049 033 049 033
“ **2** +SHIFT +CAPS +SHIFT+CAPS	 $32 $22 $32 $22	 050 034 050 034
, **3** +SHIFT +CAPS +SHIFT+CAPS	 $33 $27 $33 $27	 051 039 051 039
[**4** +SHIFT +CAPS +SHIFT+CAPS	 $34 $5B $34 $5B	 052 091 052 091
] **5** +SHIFT +CAPS +SHIFT+CAPS	 $35 $5D $35 $5D	 053 093 053 093
& **6** +SHIFT +CAPS +SHIFT+CAPS	 $36 $26 $36 $26	 054 038 054 038
(**7** +SHIFT +CAPS +SHIFT+CAPS	 $37 $28 $37 $28	 055 040 055 040

Key	Hexadecimal	Decimal
) **8** +SHIFT +CAPS +SHIFT+CAPS	 $38 $29 $38 $29	 056 041 056 041
$ **9** +SHIFT +CAPS +SHIFT+CAPS	 $39 $24 $39 $24	 057 036 057 036
: **.** +SHIFT +CAPS +SHIFT+CAPS	 $2E $3A $2E $3A	 046 058 046 058
> **+** +SHIFT +CAPS +SHIFT+CAPS	 $2B $3E $2B $3E	 043 062 043 062
< **-** +SHIFT +CAPS +SHIFT+CAPS	 $2D $3C $2D $3C	 045 060 045 060
? ***** +SHIFT +CAPS +SHIFT+CAPS	 $2A $3F $2A $3F	 042 063 042 063
% **/** +SHIFT +CAPS +SHIFT+CAPS	 $2F $25 $2F $25	 047 037 047 037
# **=** +SHIFT +CAPS +SHIFT+CAPS	 $3D $23 $3D $23	 061 035 061 035

Key	Hexadecimal	Decimal
ENTER	$0D	013
+SHIFT	$0D	013
+CAPS	$0D	013
+SHIFT+CAPS	$0D	013
HELP	$FFF2	65522
+SHIFT	$FFF2	65522
+CAPS	$FFF2	65522
+SHIFT+CAPS	$FFF2	65522
RESET	$FFF3	65523
+SHIFT		
+CAPS	$FFF3	65523
+SHIFT+CAPS		
ESC	$1B	027
ESC+SHIFT	$1B	027
ESC+CAPS	$1B	027
ESC+SHIFT+CAPS	$1B	027
MAIN MENU	$FFF4	65524
+SHIFT	$FFF4	65524
+CAPS	$FFF4	65524
+SHIFT+CAPS	$FFF4	65524
CL	$FFAD	65453
CL+SHIFT		
CL+CAPS	$FFAD	65453
CL+SHIFT+CAPS		
INS	$FFAE	65454
INS+SHIFT	$FFAE	65454
INS+CAPS	$FFAE	65454
INS+SHIFT+CAPS	$FFAE	65454

Key	Hexadecimal	Decimal
Previous Page	$FFA5	65445
+SHIFT	$FFA5	65445
+CAPS	$FFA5	65445
+SHIFT+CAPS	$FFA5	65445
Next Page	$FFAF	65455
+SHIFT		
+CAPS	$FFA5	65455
+SHIFT+CAPS		
Up arrow	$FFB0	65456
+SHIFT	$FFB1	65457
+CAPS	$FFB0	65456
+SHIFT+CAPS	$FFB1	65457
Down arrow	$FFB2	65458
+SHIFT	$FFB3	65459
+CAPS	$FFB2	65458
+SHIFT+CAPS	$FFB3	65459
Left arrow	$FFB4	65460
+SHIFT	$FFB5	65461
+CAPS	$FFB4	65460
+SHIFT+CAPS	$FFB5	65461
Right arrow	$FFB6	65462
+SHIFT	$FFB7	65463
+CAPS	$FFB6	65462
+SHIFT+CAPS	$FFB7	65463

Key	Hexadecimal	Decimal
F1	$FC00	64512
F1+SHIFT	$FC00	64512
F1+CAPS	$FC00	64512
F1+SHIFT+CAPS	$FC00	64512
F2	$FC01	64513
F2+SHIFT	$FC01	64513
F2+CAPS	$FC01	64513
F2+SHIFT+CAPS	$FC01	64513
F3	$FC02	64514
F3+SHIFT	$FC02	64514
F3+CAPS	$FC02	64514
F3+SHIFT+CAPS	$FC02	64514
F4	$FC03	64515
F4+SHIFT	$FC03	64515
F4+CAPS	$FC03	64515
F4+SHIFT+CAPS	$FC03	64515
F5	$FC04	64516
F5+SHIFT	$FC04	64516
F5+CAPS	$FC04	64516
F5+SHIFT+CAPS	$FC04	64516
F6	$FC05	64517
F6+SHIFT	$FC05	64517
F6+CAPS	$FC05	64517
F6+SHIFT+CAPS	$FC05	64517
F7	$FC06	64518
F7+SHIFT	$FC06	64518
F7+CAPS	$FC06	64518
F7+SHIFT+CAPS	$FC06	64518

Key	Hexadecimal	Decimal
Cycle Start	$FFF1	65521
+SHIFT	$FFF1	65521
+CAPS	$FFF1	65521
+SHIFT+CAPS	$FFF1	65521
Cycle Stop	$FFF0	65520
+SHIFT	$FFF0	65520
+CAPS	$FFF0	65520
+SHIFT+CAPS	$FFF0	65520

APPENDIX H

MACHINE PARAMETER SETTING

GENERAL PARAMETERS

MACHINE PARAMETER	VALUE	MACHINE PARAMETER	VALUE	MACHINE PARAMETER	VALUE
P 0		P 24		P 48	
P 1		P 25		P 49	
P 2		P 26		P 50	
P 3		P 27		P 51	
P 4		P 28		P 52	
P 5		P 29		P 53	
P 6		P 30		P 54	
P 7		P 31		P 55	
P 8		P 32		P 56	
P 9		P 33		P 57	
P 10		P 34		P 58	
P 11		P 35		P 59	
P 12		P 36		P 60	
P 13		P 37		P 61	
P 14		P 38		P 62	
P 15		P 39		P 63	
P 16		P 40		P 64	
P 17		P 41		P 65	
P 18		P 42		P 66	
P 19		P 43		P 67	
P 20		P 44		P 68	
P 21		P 45		P 69	
P 22		P 46		P 70	
P 23		P 47		P 71	

______ AXIS PARAMETERS

MACHINE PARAMETER	VALUE	MACHINE PARAMETER	VALUE	MACHINE PARAMETER	VALUE
P 0		P 24		P 48	
P 1		P 25		P 49	
P 2		P 26		P 50	
P 3		P 27		P 51	
P 4		P 28		P 52	
P 5		P 29		P 53	
P 6		P 30		P 54	
P 7		P 31		P 55	
P 8		P 32		P 56	
P 9		P 33		P 57	
P 10		P 34		P 58	
P 11		P 35		P 59	
P 12		P 36		P 60	
P 13		P 37		P 61	
P 14		P 38		P 62	
P 15		P 39		P 63	
P 16		P 40		P 64	
P 17		P 41		P 65	
P 18		P 42		P 66	
P 19		P 43		P 67	
P 20		P 44		P 68	
P 21		P 45		P 69	
P 22		P 46		P 70	
P 23		P 47		P 71	

_____ AXIS PARAMETERS

MACHINE PARAMETER	VALUE
P 0	
P 1	
P 2	
P 3	
P 4	
P 5	
P 6	
P 7	
P 8	
P 9	
P 10	
P 11	
P 12	
P 13	
P 14	
P 15	
P 16	
P 17	
P 18	
P 19	
P 20	
P 21	
P 22	
P 23	

MACHINE PARAMETER	VALUE
P 24	
P 25	
P 26	
P 27	
P 28	
P 29	
P 30	
P 31	
P 32	
P 33	
P 34	
P 35	
P 36	
P 37	
P 38	
P 39	
P 40	
P 41	
P 42	
P 43	
P 44	
P 45	
P 46	
P 47	

MACHINE PARAMETER	VALUE
P 48	
P 49	
P 50	
P 51	
P 52	
P 53	
P 54	
P 55	
P 56	
P 57	
P 58	
P 59	
P 60	
P 61	
P 62	
P 63	
P 64	
P 65	
P 66	
P 67	
P 68	
P 69	
P 70	
P 71	

______ AXIS PARAMETERS

MACHINE PARAMETER	VALUE
P 0	
P 1	
P 2	
P 3	
P 4	
P 5	
P 6	
P 7	
P 8	
P 9	
P 10	
P 11	
P 12	
P 13	
P 14	
P 15	
P 16	
P 17	
P 18	
P 19	
P 20	
P 21	
P 22	
P 23	

MACHINE PARAMETER	VALUE
P 24	
P 25	
P 26	
P 27	
P 28	
P 29	
P 30	
P 31	
P 32	
P 33	
P 34	
P 35	
P 36	
P 37	
P 38	
P 39	
P 40	
P 41	
P 42	
P 43	
P 44	
P 45	
P 46	
P 47	

MACHINE PARAMETER	VALUE
P 48	
P 49	
P 50	
P 51	
P 52	
P 53	
P 54	
P 55	
P 56	
P 57	
P 58	
P 59	
P 60	
P 61	
P 62	
P 63	
P 64	
P 65	
P 66	
P 67	
P 68	
P 69	
P 70	
P 71	

_____ AXIS PARAMETERS

MACHINE PARAMETER	VALUE
P 0	
P 1	
P 2	
P 3	
P 4	
P 5	
P 6	
P 7	
P 8	
P 9	
P 10	
P 11	
P 12	
P 13	
P 14	
P 15	
P 16	
P 17	
P 18	
P 19	
P 20	
P 21	
P 22	
P 23	

MACHINE PARAMETER	VALUE
P 24	
P 25	
P 26	
P 27	
P 28	
P 29	
P 30	
P 31	
P 32	
P 33	
P 34	
P 35	
P 36	
P 37	
P 38	
P 39	
P 40	
P 41	
P 42	
P 43	
P 44	
P 45	
P 46	
P 47	

MACHINE PARAMETER	VALUE
P 48	
P 49	
P 50	
P 51	
P 52	
P 53	
P 54	
P 55	
P 56	
P 57	
P 58	
P 59	
P 60	
P 61	
P 62	
P 63	
P 64	
P 65	
P 66	
P 67	
P 68	
P 69	
P 70	
P 71	

AXIS PARAMETERS

MACHINE PARAMETER	VALUE
P 0	
P 1	
P 2	
P 3	
P 4	
P 5	
P 6	
P 7	
P 8	
P 9	
P 10	
P 11	
P 12	
P 13	
P 14	
P 15	
P 16	
P 17	
P 18	
P 19	
P 20	
P 21	
P 22	
P 23	

MACHINE PARAMETER	VALUE
P 24	
P 25	
P 26	
P 27	
P 28	
P 29	
P 30	
P 31	
P 32	
P 33	
P 34	
P 35	
P 36	
P 37	
P 38	
P 39	
P 40	
P 41	
P 42	
P 43	
P 44	
P 45	
P 46	
P 47	

MACHINE PARAMETER	VALUE
P 48	
P 49	
P 50	
P 51	
P 52	
P 53	
P 54	
P 55	
P 56	
P 57	
P 58	
P 59	
P 60	
P 61	
P 62	
P 63	
P 64	
P 65	
P 66	
P 67	
P 68	
P 69	
P 70	
P 71	

SPINDLE PARAMETERS

MACHINE PARAMETER	VALUE	MACHINE PARAMETER	VALUE	MACHINE PARAMETER	VALUE
P 0		P 24		P 48	
P 1		P 25		P 49	
P 2		P 26		P 50	
P 3		P 27		P 51	
P 4		P 28		P 52	
P 5		P 29		P 53	
P 6		P 30		P 54	
P 7		P 31		P 55	
P 8		P 32		P 56	
P 9		P 33		P 57	
P 10		P 34		P 58	
P 11		P 35		P 59	
P 12		P 36		P 60	
P 13		P 37		P 61	
P 14		P 38		P 62	
P 15		P 39		P 63	
P 16		P 40		P 64	
P 17		P 41		P 65	
P 18		P 42		P 66	
P 19		P 43		P 67	
P 20		P 44		P 68	
P 21		P 45		P 69	
P 22		P 46		P 70	
P 23		P 47		P 71	

RS-232C MACHINE PARAMETERS

MACHINE PARAMETER	VALUE
P 0	
P 1	
P 2	
P 3	
P 4	
P 5	
P 6	
P 7	
P 8	
P 9	
P 10	
P 11	
P 12	
P 13	
P 14	
P 15	
P 16	
P 17	
P 18	
P 19	
P 20	
P 21	
P 22	
P 23	

MACHINE PARAMETER	VALUE
P 24	
P 25	
P 26	
P 27	
P 28	
P 29	
P 30	
P 31	
P 32	
P 33	
P 34	
P 35	
P 36	
P 37	
P 38	
P 39	
P 40	
P 41	
P 42	
P 43	
P 44	
P 45	
P 46	
P 47	

MACHINE PARAMETER	VALUE
P 48	
P 49	
P 50	
P 51	
P 52	
P 53	
P 54	
P 55	
P 56	
P 57	
P 58	
P 59	
P 60	
P 61	
P 62	
P 63	
P 64	
P 65	
P 66	
P 67	
P 68	
P 69	
P 70	
P 71	

RS-422 MACHINE PARAMETERS

MACHINE PARAMETER	VALUE	MACHINE PARAMETER	VALUE	MACHINE PARAMETER	VALUE
P 0		P 24		P 48	
P 1		P 25		P 49	
P 2		P 26		P 50	
P 3		P 27		P 51	
P 4		P 28		P 52	
P 5		P 29		P 53	
P 6		P 30		P 54	
P 7		P 31		P 55	
P 8		P 32		P 56	
P 9		P 33		P 57	
P 10		P 34		P 58	
P 11		P 35		P 59	
P 12		P 36		P 60	
P 13		P 37		P 61	
P 14		P 38		P 62	
P 15		P 39		P 63	
P 16		P 40		P 64	
P 17		P 41		P 65	
P 18		P 42		P 66	
P 19		P 43		P 67	
P 20		P 44		P 68	
P 21		P 45		P 69	
P 22		P 46		P 70	
P 23		P 47		P 71	

PLC MACHINE PARAMETERS

MACHINE PARAMETER	VALUE
P 0	
P 1	
P 2	
P 3	
P 4	
P 5	
P 6	
P 7	
P 8	
P 9	
P 10	
P 11	
P 12	
P 13	
P 14	
P 15	
P 16	
P 17	
P 18	
P 19	
P 20	
P 21	
P 22	
P 23	

MACHINE PARAMETER	VALUE
P 24	
P 25	
P 26	
P 27	
P 28	
P 29	
P 30	
P 31	
P 32	
P 33	
P 34	
P 35	
P 36	
P 37	
P 38	
P 39	
P 40	
P 41	
P 42	
P 43	
P 44	
P 45	
P 46	
P 47	

MACHINE PARAMETER	VALUE
P 48	
P 49	
P 50	
P 51	
P 52	
P 53	
P 54	
P 55	
P 56	
P 57	
P 58	
P 59	
P 60	
P 61	
P 62	
P 63	
P 64	
P 65	
P 66	
P 67	
P 68	
P 69	
P 70	
P 71	

M FUNCTIONS

M FUNCTION	ASSOCIATED SUBROUTINE	Bits to be set 7	6	5	4	3	2	1	0
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								

M FUNCTIONS

M FUNCTION	ASSOCIATED SUBROUTINE	Bits to be set							
		7	6	5	4	3	2	1	0
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								

LEADSCREW ERROR COMPENSATION FOR____ AXIS

POINT NUMBER	POSITION	ERROR
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

POINT NUMBER	POSITION	ERROR
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

LEADSCREW ERROR COMPENSATION FOR____ AXIS

POINT NUMBER	POSITION	ERROR
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

POINT NUMBER	POSITION	ERROR
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

LEADSCREW ERROR COMPENSATION FOR ____ AXIS

POINT NUMBER	POSITION	ERROR
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

POINT NUMBER	POSITION	ERROR
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

LEADSCREW ERROR COMPENSATION FOR____ AXIS

POINT NUMBER	POSITION	ERROR
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

POINT NUMBER	POSITION	ERROR
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

LEADSCREW ERROR COMPENSATION FOR____ AXIS

POINT NUMBER	POSITION	ERROR
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

POINT NUMBER	POSITION	ERROR
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

FIRST CROSS COMPENSATION TABLE

Moving axis "MOVAXIS" : ____________
Axis to compensate "COMPAXIS" : ____________

POINT NUMBER	POSITION	ERROR
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

POINT NUMBER	POSITION	ERROR
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

SECOND CROSS COMPENSATION TABLE

Moving axis "MOVAXIS" : ____________
Axis to be compensated "COMAXIS2" : ____________

POINT NUMBER	POSITION	ERROR
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

POINT NUMBER	POSITION	ERROR
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

THIRD CROSS COMPENSATION TABLE

Moving axis "MOVAXIS" :____________
Axis to be compensated "COMAXIS3" : ____________

POINT NUMBER	POSITION	ERROR
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

POINT NUMBER	POSITION	ERROR
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

APPENDIX I

MAINTENANCE

Cleaning:

The accumulated dirt inside the unit may act as a screen preventing the proper dissipation of the heat generated by the internal circuitry which could result in a harmful overheating of the CNC and, consequently, possible malfunctions.

On the other hand, accumulated dirt can sometimes act as an electrical conductor and shortcircuit the internal circuitry, especially under high humidity conditions.

To clean the operator panel and the monitor, a smooth cloth should be used which has been dipped into de-ionized water and /or non abrasive dish-washer soap (liquid, never powder) or 75º alcohol.

Do not use highly compressed air to clean the unit because it could generate electrostatic discharges.

The plastics used on the front panel are resistant to :

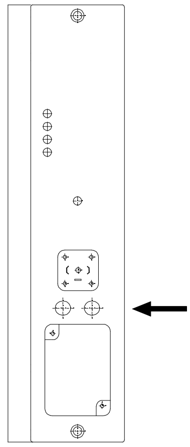
1.- Grease and mineral oils
2.- Bases and bleach
3.- Dissolved detergents
4.- Alcohol

> Avoid the action of solvents such as Chlorine hydrocarbons , Benzole , Esters and Ether which can damage the plastics used to make the unit's front panel.

Preventive Inspection:

If the CNC does not turn on when actuating the start-up switch, verify that the monitor fuse as well as that of the power supply module of the Central Unit are in good condition and that they are the right ones.

The Power Supply unit of the Central Unit has two 3.15Amp./250V fast fuses (F), one per line.



To check the fuses, first disconnect the power to the CNC.

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

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

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

List of materials, parts that could be replaced

Part Description		*Code*	*Manufacturer*	*Reference*
Rear Panel	w/ 4 modules w/ 6 modules	80500023 80500024	Fagor Automation	
Power Supply Module		80500001		
CPU Module	w/ 128K w/ 640K	80500075 80500076	Fagor Automation	
Eprom cartridge	w/ 32K w/ 128K	80500081 80500082	Fagor Automation	
Axes Module		80500004	Fagor Automation	
Input/Output Module		80500006	Fagor Automation	
Input/Output and Tracing Module		83210000	Fagor Automation	
Fan Module	w/ 4 modules w/ 6 modules	80500025 80500026	Fagor Automation	
9" Monitor	for 8050M for 8050T	83390000 83390001	Fagor Automation	
10" Monitor	for 8050M for 8050T	83420001 83420003	Fagor Automation	
14" Monitor		83420004	Fagor Automation	
Operator Panel 8050M	w/o Handwheel w/ Handwheel	80300010 80300014	Fagor Automation	
Operator Panel 8050T	w/o Handwheel w/ Handwheel	80300011 80300015	Fagor Automation	
Set of cables	5 m 10 m 15 m	80500018 80500019 80500020	Fagor Automation	
Mains AC Cable 3x0,75		11313000	Fagor Automation	
3.15A/250V Fuse		12130015	Schurter Wickmann	FST-034-1521 Ref. 19115
Manual	OEM USER (M) USER (T)	83750004 83750002 83750000	Fagor Automation	