FAGOR AUTOMATION

CNC 8025 GP, M, MS

New Features (Ref. 0107 in)

FAGOR

ERRORS FOUND IN THE INSTALLATION MANUAL (REF. 9707)

Appendix "F" page 10. P621(7)

It is wrong, it should say:

P621(7) The M06 function executes the M19 function (0=Yes, 1=No)

Appendix "G" page 20. P621(7)

It is wrong, it should say:

P621(7) The M06 function executes the M19 function (0=Yes, 1=No)

MODIFICATIONS TO THE INSTALLATION MANUAL (REF. 9707)

Comparison table xii. Technical description. Inputs/Outputs.

Feedback inputs for rotary axes, it should say: W (GP), W (M), W (MG), W V (MS)

Comparison table xii. Technical description. Axis control.

	GP	M	MG	MS
Electronic threading. It should say		x	x	x

Comparison table xii. Technical description. Others. Add fields

	GP	M	MG	MS
Open loop motors without servodrives.....	x			
Laser machines.....		x	x	x
JIG Grinders.....		x	x	x

Section 3.3.3 (chapter 3 page 15). P612(6). Another example:

Having a Fagor electronic handwheel (25 lines/turn) set as follows:

P612(3) = 0 Millimeters P612(4) = 0 and P612(5) = 0 Resolution: 0.001 mm.

P612(6) = 0 Multiplication factor x4

Depending on the position of the MFO switch (Manual Feedrate Override) the selected axis will move:

Position	1	1 x 25 x 4 =	0.100 mm per turn
Position	10	10 x 25 x 4 =	1.000 mm per turn
Position	100	100 x 25 x 4 =	10.000 mm per turn

MODIFICATIONS TO THE PROGRAMMING MANUAL (REF. 9701)

Section 6.30.4 (page 128). G76 Automatic block generation

If the new program to be created is to be sent to a PC (G76 N), the DNC communication must be enabled and, at the PC, the "program management" "Digitizing Reception" option must be selected.

If it is not, the CNC will issue error 56.

1. EXPANSION OF THE INTEGRATED PLC RESOURCES

1.1 INPUTS

1.1.1 AXIS BEING HOMED (REFERENCED)

Input I88 indicates whether a home search is taking place and inputs I100, I101, I102, I103 and I104 indicates which axis is being homed.

I88	Indicates whether any axis is being homed (0=No / 1=Yes)
I100	Indicates whether the X axis is being homed (0=No / 1=Yes)
I101	Indicates whether the Y axis is being homed (0=No / 1=Yes)
I102	Indicates whether the Z axis is being homed (0=No / 1=Yes)
I103	Indicates whether the W axis is being homed (0=No / 1=Yes)
I104	Indicates whether the V axis is being homed (0=No / 1=Yes)

1.1.2 AXIS MOVING DIRECTION

Inputs I42, I43, I44, I45 and I46 will show, at all times, the moving direction of each axis.

I42	Indicates the moving direction of the X axis (0=Positive / 1=negative)
I43	Indicates the moving direction of the Y axis (0=Positive / 1=negative)
I44	Indicates the moving direction of the Z axis (0=Positive / 1=negative)
I45	Indicates the moving direction of the W axis (0=Positive / 1=negative)
I46	Indicates the moving direction of the V axis (0=Positive / 1=negative)

1.2 OUTPUTS

1.2.1 ENABLING THE CYCLE-START KEY VIA PLCI

With this feature it is possible to set the treatment of the [CYCLE START] of the CNC via PLCI. Machine parameter "P627(7)" indicates whether this feature is available or not.

P627(7) = 0 This feature **is not** available.
P627(7) = 1 This feature **is** available.

When using this feature, the way the CNC handles the [CYCLE START] key depends on the status of PLCI output O25 (CYCLE START ENABLE).

O25 = 0	The CNC **ignores** both the [CYCLE-START] key and the external [CYCLE-START] signal.
O25 = 1	The CNC takes into account both the [CYCLE-START] key and the external [CYCLE-START] signal.

1.2.2 TRAVEL LIMITS SET VIA PLCI

With this feature, the travel limits of the axes may be set via PLCI. Machine parameter "P627(7)" indicates whether this feature is available or not.

P627(7) = 0 This feature **is not** available.
P627(7) = 1 This feature **is** available.

To set the travel limits for each axis, use the following outputs:

O52 / O53	Positive / negative X axis limits
O54 / O55	Positive / negative Y axis limits
O56 / O57	Positive / negative Z axis limits
O58 / O59	Positive / negative W axis limits
O60 / O61	Positive / negative V axis limits

When the PLCI activates one of this outputs while the axis is moving in the same direction, the CNC stops the axes and the spindle and it displays an axis-travel-limit-overrun error.

1.2.3 DENYING ACCESS TO THE EDITOR MODE VIA PLCI

Machine parameter"P627(7)" indicates whether this feature is available or not.

P627(7) = 0 This feature **is not** available.
P627(7) = 1 This feature **is** available.

When using this feature, access to the editor mode at the CNC depends on the status of PLCI output O26, as well as on the current conditions (protected memory, number of the program to be locked).

O26 = 0 Free access to the editor mode (it is protected by current conditions).
O26 = 1 Denied access to the editor mode.

1.2.4 SPINDLE CONTROLLED VIA CNC OR VIA PLCI

From this version on, the spindle analog output may be set either by the CNC or by the PLCI. Machine parameter "P627(7)" indicates whether this feature is available or not.

P627(7) = 0 This feature **is not** available
P627(7) = 1 This feature **is** available

Setting the spindle analog output via PLCI

To do this, use the combination: M1956 - R156.

Register R156 sets the spindle analog output in units of 2.442 mV. (10 / 4095)

R156 = 0000 1111 1111 1111	(R1256=4095)	= 10V.
R156 = 0001 1111 1111 1111		= -10V.
R156 = 0000 0000 0000 0001	(R1256=1)	= 2.5 mV.
R156 = 0001 0000 0000 0001		= -2.5 mV.

In order for the CNC to assume the value allocated to register R156, one must activate mark M1956 as described in the PLCI Manual (section 5.5.2. Writing internal CNC variables).

Spindle controlled either by the CNC or by the PLCI

The CNC may have two internal spindle analog outputs, that of the CNC itself and the one set by the PLCI.

Use PLCI output O27 to "tell" the CNC which one of them to output.

O27 = 0 Spindle analog output set by the CNC itself.
O27 = 1 Spindle analog output set by the PLCI (combination: M1956-R156).

1.3 READING INTERNAL CNC VARIABLES

From this version on, the PLCI and the PLC64 have access to more internal CNC information.

With the PLCI, there is no need to activate a mark to access this information. The CNC itself updates this information at the beginning of each PLCI cycle scan.

With the PLC 64, the corresponding mark must be consulted every time a CNC variable is to be checked.

The CNC information now accessible is:

Real S in rpm (REG119 at the PLCI, M1919 at the PLC64)

Not to be mistaken with R112 which indicates the programmed Spindle speed (S).

It is given in rpm and in hexadecimal format. Example: S 2487 R119= 967

Number of the block in execution (REG120 at the PLCI, M1920 at the PLC64)

It is given in hexadecimal format. Example: N120 R120= 78

Code of the last key pressed (B0-7 REG121 at the PLCI, Not available at the PLC64)

Not to be mistaken with register R118 which also indicates the code corresponding to the last key pressed**but-**

When pressing a key, both registers have the same value; but **the data in R121 is only kept there for one cycle scan** whereas **R118 keeps its value until another key is pressed**

When pressing the same key several times, (for example: 1111):
R121 will show code "1" four times (once per cycle scan).
R118 will always show the same value, thus not being able to tell whether the "1" key has been pressed once or more times.

The key codes are listed in the appendix of the PLCI manual.

Operating mode selected at the CNC (B8-11 REG121 at the PLCI, Not available at the PLC64)

B8	B9	B10	B11	
0	0	0	0	Automatic
0	0	0	1	Single block
0	0	1	0	Play-Back
0	0	1	1	Teach-in
0	1	0	0	Dry-Run
0	1	0	1	JOG
0	1	1	0	Editor
0	1	1	1	Peripherals
1	0	0	0	Tool Table and G functions
1	0	0	1	Special modes

Status of the miscellaneous "M" functions (REG122 at the PLCI, Not available at the PLC64)

The status of each one of these functions is given by a bit and will appear as a "1" when active and "0" when inactive.

B15	B14	B13	B12	B11	B10	B9	B8	B7	B6	B5	B4	B3	B2	B1	B0
							M19	M1	M30	M6	M5	M4	M3	M2	M0

2. RETRACE FUNCTION.

This feature is available on the following models:

CNC-8025M CNC-8025MG CNC-8025MS
CNC-8025MI CNC-8025MGI CNC-8025MSI

Machine parameter "P627(6)" indicates whether this feature is available or not.

P627(6) = 0 This feature **is not** available
P627(6) = 1 This feature **is** available

This function may be selected by the operator. To do this, activate:

On models without PLCI: pin 7 of connector A5.
On models **with** PLCI: PLCI output O47

Operation:

As the CNC executes motion blocks, it always stores the last 10 blocks already executed

Whenever it executes a block containing an M,S,T type function, the machining conditions change and the CNC deletes those previously stored motion blocks.

When the retrace function is activated, the block currently in execution is interrupted and the retrace process begins.
First to the starting point of the current block and, then, to that of the previously stored program blocks.

If all the stored blocks are executed, the CNC stops the machine until the retrace function is canceled.

When this function is canceled, the CNC interrupts the current movement (if any) and it executes all the retraced blocks again. Once the interruption point is reached, the CNC resumes the execution of the program.

3. OPERATION WITH TWO MOTORS AND 3 AXES.

Machine parameter "P627(8)" indicates whether this feature is available or not.

P627(8) = 0 This feature **is not** available.
P627(8) = 1 This feature **is** available.

Operation:

The CNC permits using 2 motors to move 3 axes with the following conditions:

One of the axes shared by a motor must be the Z axis and the other one must be either the X or the Y axis.

Only interpolations between the X and Y axes are possible. The Z axis cannot be interpolated with any other axis. It must be moved alone.

Example: To move the tool from "X0 Y0 Z0" to "X20 Y20 Z20", The CNC will make this move in two steps. First, it will move the X and Y axes to X20 Y20 and, then, the Z axis to Z20.

4. SPINDLE FOLLOWING ERROR DISPLAY WHILE IN M19

From this version on, when operating with spindle orient (M19), the CNC also shows the spindle following error on the screen corresponding to the following error in Automatic and Single block modes.

The Following Error screen shows, in large characters, the amount of axis lag and, under it, the following information line.

F 00000.0000 & 100 S 0000 % 100 T 00.00 **S 0000.000**

The last value of this line**"S 0000.000"** shows the amount of following error (lag) of the spindle when it operates in spindle orient mode (M19).

5. GANTRY AXES NOT MECHANICALLY SLAVED

From this version on, depending on the setting of machine parameter "P629(8)", it is possible to work with two different types of Gantry axes.

"P629(8)=0" Mechanically slaved Gantry Axes. As until now.
When being homed, both axes behave as a single axis. The CNC takes into account only the parameter settings and feedback pulses of the main axis, the slaved one being just a follower of the main axis,

"P629(8)=1" Not-mechanically slaved Gantry axes.
When being homed, the two axes behave as separate independent axes. First the main axis is homed and, then, the slaved one.

6. SHEETMETAL FORMING MACHINES

This feature is available on GP models.

To enable it, set machine parameter "P626(7)=1". The CNC enables functions M98 and M99 to control the X axis positioning loop.

Function M98 opens the X axis loop and M99 closes it.
When the CNC executes an M30, it also closes the X axis positioning loop.

When operating in jog mode, the CNC enables he following keys to control the X axis positioning loop:

 Executes an M98 opening the X axis loop.

 Executes an M98 opening the X axis loop.

Executes an M98 closing the X axis loop.

Version 7.2 (April 1997)

1. SCREEN SAVER

The screen saver function works as follows:

After 5 minutes without pressing a key or without the CNC refreshing the screen, the screen goes blank. Press any key to restore the display.

Machine parameter "P626(5)" indicates whether this feature is to be used or not.

P626(5) = 0 This feature is not being used.
P626(5) = 1 This feature is being used.

2. JOGGING FEEDRATE

If while in JOG mode, the conditional input (block skip), pin 18 of connector I/O1, the CNC does not allow entering a new F value. Only the feedrate override (%) may be varied by means of the MFO switch.

3. PARAMETRIC PROGRAMMING NEW FUNCTION: F34

Function F34 returns the number of the tool being dealt with.

P27=F34 Parameter P27 takes the value of the new tool being dealt with.

This function must be used when working with a subroutine associated with the tool change.
When using it outside that subroutine, function F34 returns the value of "100".

Version 7.3 (March 1998)

1. PLCI. Input I87

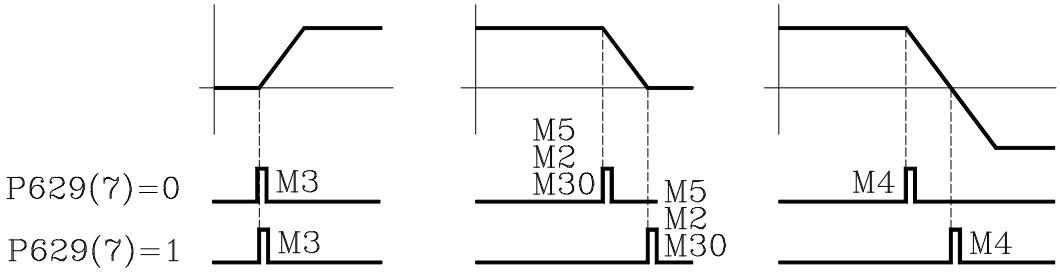
While the CNC is threading (G84), PLCI input I87 is set to “1”.

Note: Input I97 indicates rigid tapping.

Version 7.4 (May 1999)

1. NEW MACHINE PARAMETER ASSOCIATED WITH THE M FUNCTIONS

Machine parameter "P629(7)" indicates when the M3, M4, M5 functions are sent out while accelerating or decelerating the spindle.



2. CANCEL TOOL OFFSET DURING A TOOL CHANGE

From this version on, it is possible to execute a "T.0" type block inside the subroutine associated with the tool to cancel the tool offset. This lets move to a particular position without the need for cumbersome calculations.

Only the tool offset may be canceled (T.0) or modified (T.xx). The tool cannot be changed (Txx.xx) inside the subroutine associated with the tool.

3. DIVIDING FACTOR FOR FEEDBACK SIGNALS

Parameters P631(8), P631(7), P631(6), P631(5) and P631(4) are used together with P604(8), P604(7), P604(6), P604(5) and P616(8) which indicate the multiplying factor to be applied to the feedback signals of the X, Y, Z, W, V axes respectively.

X axis	Y axis	Z axis	W axis	V axis
P604(8)	P604(7)	P604(6)	P604(5)	P616(8)
P631(8)	P631(7)	P631(6)	P631(5)	P631(4)

Indicate whether the feedback signals are divided (=1) or not (=0).

P631(8)=0, P631(7)=0, P631(6)=0, P631(5)=0 and P631(4)=0	They are not divided
P631(8)=1, P631(7)=1, P631(6)=1, P631(5)=1 and P631(4)=1	They are divided by two.

Example:
We wish to obtain a resolution of 0.01 mm with a squarewave encoders mounted on the X axis with 5mm pitch ballscrew

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

With P604(8)=0 & P631(8)=0	x4 multiplying factor	Nr of pulses = 125
With P604(8)=1 & P631(8)=0	x2 multiplying factor	Nr of pulses = 250
With P604(8)=0 & P631(8)=1	x2 multiplying factor	Nr of pulses = 250
With P604(8)=1 & P631(8)=1	x1 multiplying factor	Nr of pulses = 500

Version 7.6 (July 2001)

1. G75 AFFECTED BY FEEDRATE OVERRIDE

From this version on, there is a new machine parameter indicating whether G75 is affected by the feedrate override or not.

P631(1) = 0 Not affected. It is always at 100%, like in previous versions.
P631(1) = 1 It is affected by the Feedrate override.

2. FEEDBACK FACTOR.

From this version on, there is a new machine parameter to set the resolution of an axis having an encoder and a leadscrew.

P819 Feedback factor for the X axis P820 Feedback factor for the Y axis P821 Feedback factor for the Z axis
P822 Feedback factor for the W axis P823 Feedback factor for the V axis

Values between 0 and 65534. The "0" value indicates that this feature is not being used.

Use the following formula to calculate the "Feedback Factor" :

Feedback factor = (Gear Ratio x Leadscrew pitch / Number of Encoder pulses) x 8.192

Examples:	Gear Ratio	1	1	2	1	
	Leadscrew pitch	5000	6000	6000	8000	(microns)
	Encoder	2500	2500	2500	2500	(pulses/turn)
	Feedback factor	16384	19660.8	39321.6	26214.4	

The machine parameters only admit integer values and sometimes the "Feedback Factor" has decimals. In those cases, assign the integer part to the machine parameter and use the leadscrew compensation table to make up for the decimal part.

The values to be entered in the table are calculated with the following formula:

Leadscrew position = Leadscrew Error (microns) x Integer of feedback factor / decimals of the feedback factor

For example: Gear ratio = 1 Leadscrew pitch = 6000 Encoder = 2500
Feedback factor = 19660.8 Machine parameter = 19660
For a leadscrew error of 20 microns Leadscrew position = 20 x 19660 / 0.8 = 491520
Going on with the calculation, we come up with the following table.

Leadscrew position		Leadscrew error	
P0 =	-1966.000	P1 =	-0.080
P2 =	-1474.500	P3 =	-0.060
P4 =	-983.000	P5 =	-0.040
P6 =	-491.500	P7 =	-0.020
P8 =	0	P9 =	0
P10 =	491.500	P11 =	0.020
P12 =	983.000	P13 =	0.040
P14 =	1472.500	P15 =	0.060
P16 =	1966.000	P17 =	0.080

3. NEW MODEL

From this version on, the new model MLI is now available.
It offers the same features as the MGI model and it is sold together with the motors and ACS drives..

Headquarters (SPAIN): **Fagor Automation S. Coop.**
Bº San Andrés s/n, Apdo. 144
20500 Arrasate - Mondragón
Tel: +34-943-719200
Fax: +34- 943-791712
+34-943-771118 (Service Dept.)
www.fagorautomation.com
E-mail: info@fagorautomation.es

FAGOR 8025/8030 CNC

Models: M, MG, MS, GP

INSTALLATION MANUAL

Ref. 9707 (in)

ABOUT THE INFORMATION IN THIS MANUAL

This manual is addressed to the machine manufacturer.

It includes the necessary information for new users as well as advanced subjects for those who are already familiar with the 8025 CNC.

It may not be necessary to read this whole manual. Consult the list of "New Features and Modifications" and the appendix related to the machine parameters. Practically all of them are cross referenced indicating the chapter and section of the manual where they are described.

This manual explains all the functions of the 8025 CNC family. Consult the Comparison Table for the models in order to find the specific ones offered by your CNC.

To install the CNC onto your machine, we suggest that you consult the appendix regarding the enclosures required to mount the CNC as well as chapter 1 (CNC configuration) which indicates the CNC dimensions and details the pin-out of its connectors.

If your CNC has an integrated PLC (PLCI), the I/O pin-out is different. Therefore, the PLCI manual must also be consulted.

Chapter 2 (Power and Machine Interface) shows how to connect the CNC to A.C. power (Mains) and to the electrical cabinet.

To adapt the CNC to the machine, set the CNC machine parameters. Consult chapters 3, 4 and 5 as well as the appendix concerning machine parameters.

There are 2 appendices; one where the parameters are ordered by subject and the other one where the parameters are in numerical order.

Both appendices offer cross references indicating the section of the manual describing each parameter.

When explaining each parameter in detail, chapters 3, 4 and 5, they sometimes refer to chapter 6 (Concepts) where some of them are dealt with in further detail indicating how to perform various adjustments of the CNC-machine interface.

Once all machine parameters are set, we suggest that you write their settings down on the charts provided for this purpose in the appendix on "Machine Parameter Setting Chart".

There is also an appendix on error codes which indicates some of the probable reasons which could cause each one of them.

Also, if you wish this CNC to communicate with other FAGOR products, you must use the Fagor Local Area Network (LAN). To do that, refer to the manual on FAGOR LAN.

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

FAGOR AUTOMATION, S. Coop. Ltda. reserves the right to modify the contents of this manual without prior notice.

INDEX

Section Page

Chapter 2 POWER AND MACHINE INTERFACE

Chapter 3 MACHINE PARAMETERS

Chapter 4 MACHINE PARAMETERS FOR THE AXES

Section Page

Chapter 5 SPINDLE MACHINE PARAMETERS

Chapter 6 CONCEPTS

Section Page

APPENDICES

ERROR CODES

COMPARISON TABLE
FOR MILL MODEL
FAGOR 8025/8030 CNCs

8025/8030 MILL MODEL CNCS

Fagor offers the 8025 and 8030 mill type CNCs.

Both types operate the same way and offer similar characteristics. Their basic difference is that the former is compact and the latter is modular.

Both CNC types offer basic models. Although the differences between the basic models are detailed later on, each model may be defined as follows:

8025/8030 GP	Oriented to General Purpose machines
8025/8030 M	Oriented to Milling machines of up to 4 axes.
8025/8030 MG	Same as the M model, but with dynamic graphics.
8025/8030 MS	Oriented to Machining Centers (up to 5 axes).

When the CNC has an Integrated Programmable Logic Controller (PLCI), the letter "I" is added to the CNC model denomination: GPI, MI, MGI, MSI.

Also, When the CNC has 512Kb of part-program memory, the letter "K" is added to the CNC model denomination: GPK, MK, MGK, MSK, GPIK, MIK, MGIK, MSIK.

	Basic	**With PLCI**	**Basic With 512Kb**	**With PLCI and 512Kb**
General Purpose	**GP**	**GPI**	**GPK**	**GPKI**
Mills up to 4 axes	**M**	**MI**	**MK**	**MIK**
Up to 4 axes with graphics	**MG**	**MGI**	**MGK**	**MGIK**
Machining Centers	**MS**	**MSI**	**MSK**	**MSIK**

TECHNICAL DESCRIPTION

	GP	M	MG	MS
INPUTS/OUTPUTS				
Feedback inputs.	**6**	**6**	**6**	**6**
Linear axes	**4**	**4**	**4**	**5**
Rotary axes	**2**	**2**	**2**	**2**
Spindle encoder	**1**	**1**	**1**	**1**
Electronic handwheels	**1**	**1**	**1**	**1**
Probe input	**x**	**x**	**x**	**x**
Square-wave feedback signal multiplying factor, x2/x4	**x**	**x**	**x**	**x**
Sine-wave feedback signal multiplying factor, x2/x4/10/x20	**x**	**x**	**x**	**x**
Maximum counting resolution 0.001mm/0.001°/0.0001inch	**x**	**x**	**x**	**x**
Analog outputs (±10V) for axis servo drives	**4**	**4**	**4**	**5**
Spindle analog output (±10V)	**1**	**1**	**1**	**1**
AXIS CONTROL				
Axes involved in linear interpolations	**3**	**3**	**3**	**3**
Axes involved in circular interpolations	**2**	**2**	**2**	**2**
Helical interpolation	**x**	**x**	**x**	**x**
Electronic threading	**x**	**x**	**x**	
Spindle control	**x**	**x**	**x**	**x**
Software travel limits	**x**	**x**	**x**	**x**
Spindle orientation	**x**	**x**	**x**	**x**
Management of non-servo-controlled Open-Loop motor	x			
PROGRAMMING				
Part Zero preset by user	**x**	**x**	**x**	**x**
Absolute/incremental programming	**x**	**x**	**x**	**x**
Programming in cartesian coordinates	**x**	**x**	**x**	**x**
Programming in polar coordinates	**x**	**x**	**x**	**x**
Programming in cylindrical coordinates (radius, angle, axis)	**x**	**x**	**x**	**x**
Programming by angle and cartesian coordinate	**x**	**x**	**x**	**x**
COMPENSATION				
Tool radius compensation		**x**	**x**	**x**
Tool length compensation	**x**	**x**	**x**	**x**
Leadscrew backlash compensation	**x**	**x**	**x**	**x**
Leadscrew error compensation	**x**	**x**	**x**	**x**
Cross compensation (beam sag)	**x**	**x**	**x**	**x**
DISPLAY				
CNC text in Spanish, English, French, German and Italian	**x**	**x**	**x**	**x**
Display of execution time	**x**	**x**	**x**	**x**
Piece counter	**x**	**x**	**x**	**x**
Graphic movement display and part simulation			**x**	**x**
Tool base position display	**x**	**x**	**x**	**x**
Tool tip position display	**x**	**x**	**x**	**x**
Geometric programming aide	**x**	**x**	**x**	**x**
COMMUNICATION WITH OTHER DEVICES				
Communication vía RS232C	**x**	**x**	**x**	**x**
Communication via DNC	**x**	**x**	**x**	**x**
Communication via RS485 (FAGOR LAN)	**x**	**x**	**x**	**x**
ISO program loading from peripherals	**x**	**x**	**x**	**x**
OTHERS				
Parametric programming	**x**	**x**	**x**	**x**
Model digitizing	**x**	**x**	**x**	**x**
Possibility of an integrated PLC	**x**	**x**	**x**	**x**
Sheetmetal tracing on LASER machines				**x**
Jig Grinder				**x**

PREPARATORY FUNCTIONS

	GP	M	MG	MS
AXES AND COORDINATE SYSTEMS				
XY (G17) plane selection	x	x	x	x
XZ and YZ plane selection (G18,G19)	x	x	x	x
Part measuring units. Millimeters or inches (G70,G71)	x	x	x	x
Absolute/incremental programming (G90,G91)	x	x	x	x
Independent axis (G65)	x	x	x	x
REFERENCE SYSTEMS				
Machine reference (home) search (G74)	x	x	x	x
Coordinate preset (G92)	x	x	x	x
Zero offsets (G53...G59)	x	x	x	x
Polar origin preset (G93)	x	x	x	x
Store current part zero (G31)	x	x	x	x
Recover stored part zero (G32)	x	x	x	x
PREPARATORY FUNCTIONS				
Feedrate F	x	x	x	x
Feedrate in mm/min. or inches/minute (G94)	x	x	x	x
Feedrate in mm/revolution or inches/revolution (G95)	x	x	x	x
Constant surface speed (G96)	x	x	x	x
Constant tool center speed (G97)	x	x	x	x
Programmable feedrate override (G49)	x	x	x	x
Spindle speed (S)	x	x	x	x
S value limit (G92)	x	x	x	x
Tool and tool offset selection (T)	x	x	x	x
AUXILIARY FUNCTIONS				
Program stop (M00)	x	x	x	x
Conditional program stop (M01)	x	x	x	x
End of program (M02)	x	x	x	x
End of program with return to first block (M30)	x	x	x	x
Clockwise spindle start (M03)	x	x	x	x
Counter-clockwise spindle start (M04)	x	x	x	x
Spindle stop (M05)	x	x	x	x
Tool change in machining centers (M06)	x	x	x	x
Spindle orientation (M19)	x	x	x	x
Spindle speed range change (M41, M42, M43, M44)	x	x	x	x
Functions associated with pallets (M22, M23, M24, M25)	x	x	x	
PATH CONTROL				
Rapid traverse (G00)	x	x	x	x
Linear interpolation (G01)	x	x	x	x
Circular interpolation (G02,G03)	x	x	x	x
Circular interpolation with absolute center coordinates (G06)	x	x	x	x
Circular path tangent to previous path (G08)	x	x	x	x
Arc defined by three points (G09)	x	x	x	x
Tangential entry at beginning of a machining operation (G37)	x	x	x	x
Tangential exit at the end of a machining operation (G38)	x	x	x	x
Controlled radius blend (G36)	x	x	x	x
Chamfer (G39)	x	x	x	x
Electronic threading (G33)		x	x	x
ADDITIONAL PREPARATORY FUNCTIONS				
Dwell (G04 K)	x	x	x	x
Round and square corner (G05, G07)	x	x	x	x
Mirror image (G10,G11,G12)	x	x	x	x
Mirror image along the Z axis (G13)	x	x	x	x
Scaling factor (G72)	x	x	x	x
Pattern rotation (G73)	x	x	x	x
Slaving/unslaving of axes (G77, G78)	x	x	x	x
Single block treatment (G47, G48)	x	x	x	x
User error display (G30)	x	x	x	x
Automatic block generation (G76)				x
Communication with FAGOR Local Area Network (G52)	x	x	x	x

	GP	M	MG	MS
COMPENSATION				
Tool radius compensation (G40,G41,G42)		x	x	x
Tool length compensation (G43,G44)	x	x	x	x
Loading of tool dimensions into internal tool table (G50)	x	x	x	x
CANNED CYCLES				
Multiple arc-pattern machining (G64)		x	x	x
User defined canned cycle (G79)	x	x	x	x
Drilling cycle (G81)		x	x	x
Drilling cycle with dwell (G82)		x	x	x
Deep hole drilling cycle (G83)		x	x	x
Tapping cycle (G84)		x	x	x
Rigid tapping cycle (G84R)		x	x	x
Reaming cycle (G85)		x	x	x
Boring cycle with withdrawal in G00 (G86)		x	x	x
Rectangular pocket milling cycle (G87)		x	x	x
Circular pocket milling cycle (G88)		x	x	x
Boring cycle with withdrawal in G01 (G89)		x	x	x
Canned cycle cancellation (G80)	x	x	x	x
Return to starting point (G98)		x	x	x
Return to reference plane (G99)		x	x	x
PROBING				
Probing (G75)	x	x	x	x
Tool length calibration canned cycle (G75N0)				x
Probe calibration canned cycle (G75N1)				x
Surface measuring canned cycle (G75N2)				x
Surface measuring canned cycle with tool offset (G75N3)				x
Outside edge measuring canned cycle (G75N4)				x
Inside edge measuring canned cycle (G75N5)				x
Angle measuring canned cycle (G75N6)				x
Outside edge and angle measuring canned cycle (G75N7)				x
Hole centering canned cycle (G75N8)				x
Boss centering canned cycle (G75N9)				x
Hole measuring canned cycle (G75N10)				x
Boss measuring canned cycle (G75N11)				x
SUBROUTINES				
Number of standard subroutines	99	99	99	99
Definition of standard subroutine (G22)	x	x	x	x
Call to a standard subroutine (G20)	x	x	x	x
Number of parametric subroutines	99	99	99	99
Definition of parametric subroutine (G23)	x	x	x	x
Call to a parametric subroutine (G21)	x	x	x	x
End of standard or parametric subroutine (G24)	x	x	x	x
JUMP OR CALL FUNCTIONS				
Unconditional jump/call (G25)	x	x	x	x
Jump or call if zero (G26)	x	x	x	x
Jump or call if not zero (G27)	x	x	x	x
Jump or call if smaller (G28)	x	x	x	x
Jump or call if equal or greater (G29)	x	x	x	x

NEW FEATURES AND MODIFICATIONS

Date: **February 1991** **Software version:** **2.1 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Error 65 is not issued while probing (G75)	Installation Manual	Section 3.3.4
It is possible to select the home searching direction for each axis	Installation Manual	Section 4.6
New 1, 2, 5, 10 resolution values for sine-wave feedback signals of each axis	Installation Manual	Section 4.1
PLCI register access from the CNC	Programming Manual	G52
Sheetmetal tracing on laser machines	Applications Manual	
Jig Grinder	Applications Manual	

Date: **June 1991** **Software version:** **3.1 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Repetitive emergency subroutine	Installation Manual	Section 3.3.8
New function F29. It takes the value of the selected tool	Programming Manual	Chapter 13
Function M06 does not execute M19	Installation Manual	Section 3.3.5
Greater speed when executing several parametric blocks in a row.		

Date: **March 1992** **Software version:** **4.1 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Bell-shape acceleration/deceleration control	Installation Manual	Section 4.7
Expansion of cross compensation	Installation Manual	Section 4.10
Rigid Tapping G84 R	Programming Manual	G84
Possibility to enter the sign of the leadscrew backlash for each axis	Installation Manual	Section 4.9
Independent execution of an axis	Programming Manual	G65

Date: **July 1993** **Software version:** **5.1 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Double cross compensation	Installation Manual	Section 4.10
Linear and bell-shaped acc./dec. ramp combination for the axes	Installation Manual	Section 4.7
Acceleration/deceleration control for the the spindle	Installation Manual	Section 5.
Multiple arc pattern machining	Programming Manual	G64
Tool tip position display	Installation Manual	Section 3.3.5
The associated subroutine is executed before the T function	Installation Manual	Section 3.3.5
The additional circular sections of a compensated path are executed in G05 or G07	Installation Manual	Section 3.3.8
VGA monitor 8030 CNC.	Installation Manual	Chapter 1

Date: **March 1995** **Software version:** **5.3 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Management of feedback with coded Io	Installation Manual	Section 4.6 & 6.5
Spindle inhibit by PLC	Installation Manual	Section 3.3.9
Handwheel management by PLC	Installation Manual	Section 3.3.3
Rapid (JOG) key simulation via PLC	PLCI Manual	
Non-servo-controlled open-loop motors	Applications Manual	
Function G64, multiple machining in an arc. To be selected by machine parameter.	Installation Manual	Section 3.3.9
Initialization of machine parameters after memory loss.		

Date: **September 1995** **Software version:** **6.0 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
512 Kb of part-program memory	Operating Manual	Section 3.6
When conditional input (block skip) active while in JOG mode, the  key is ignored	Installation Manual	Section 1.3.6

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 responsibility that the product:

Fagor 8025 M 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 61000-3-2 Current Harmonics
- EN 61000-3-3 Voltage fluctuations and flickers

EN 50082-2 Immunity
- EN 61000-4-2 Electrostatic Discharges.
- EN 61000-4-3 Radiofrequency Radiated Electromagnetic Fields.
- EN 61000-4-4 Bursts and fast transients.
- EN 61000-4-5 Conducted high voltage pulses in mains (Surges)
- EN 61000-4-6 Conducted disturbance induced by radio frequency fields.
- EN 61000-4-8 Magnetic fields at mains frequency
- EN 61000-4-11 Voltage fluctuations and Outages.
- ENV 50204 Fields generated by digital radio-telephones

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

In Mondragón, on October 1st, 2001

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

Module interconnection

Use the cables supplied 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 not 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

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

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 over voltage and reverse connection of the power supply.

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

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 CNC, 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.

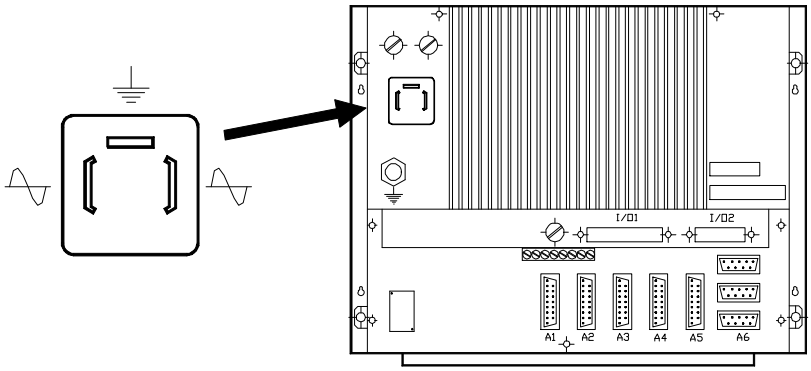
5.- Seal the cardboard box with packing tape or industrial staples.

ADDITIONAL REMARKS

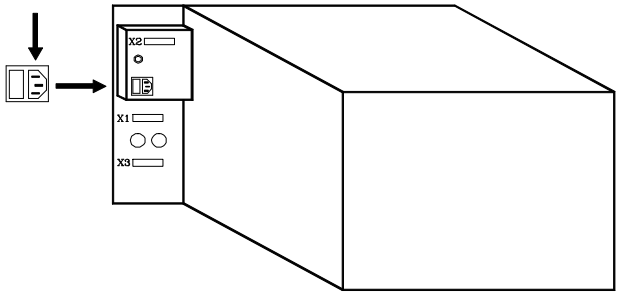
* Mount the CNC away from coolants, chemical products, blows, etc. which could damage it.

* 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)



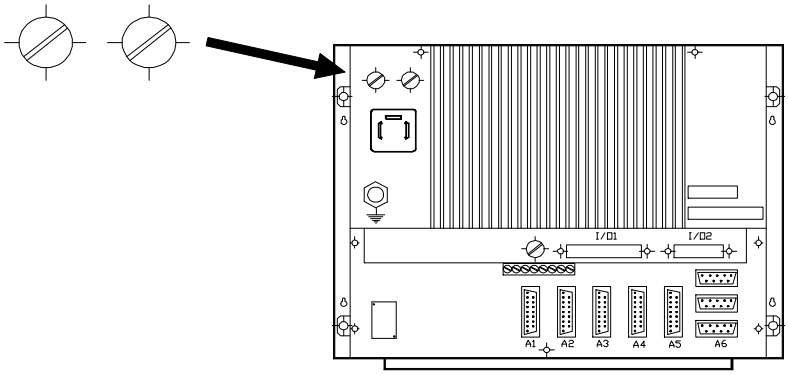
* To prevent electrical shock at the Monitor, use the proper mains AC connector at the Power Supply Module. Use 3-wire power cables (one for ground connection)



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

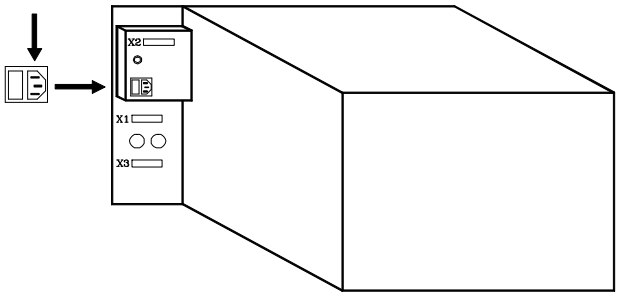
  Central Unit

  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 8025/30 M CNC

8025 M CNC OEM Manual Is directed to the machine builder or person in charge of installing and starting up the CNC.

It contains 2 manuals:

Installation Manual	describing how to isntall and set-up the CNC.
LAN Manual	describing how to instal the CNC in the Local Area Network.

Sometimes, it may contain an additional manual describing New Software Features recently implemented.

8025 M CNC USER Manual Is directed to the end user or CNC operator.

It contains 3 manuals:

Operating Manual	describing how to operate the CNC.
Programming Manual	describing how to program the CNC.
Applications Manual	describing other applications for this CNC non-specific of Milling machines

Sometimes, it may contain an additional manual describing New Software Features recently implemented.

DNC 25/30 Software Manual Is directed to people using the optional DNC communications software.

DNC 25/30 Protocol Manual Is directed to people wishing to design their own DNC communications software to communicate with the 800 without using the DNC25/30 software..

PLCI Manual To be used when the CNC has an integrated PLC.

Is directed to the machine builder or person in charge of installing and starting up the PLCI.

DNC-PLC Manual Is directed to people using the optional communications software: DNC-PLC.

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 consists of the following chapters:

Index

Comparison table of FAGOR models: 8025 M CNCs

New Features and modifications.

Introduction
Warning sheet prior to start-up:
Declaration of Conformity.
Safety conditions.
Warranty terms.
Material returning conditions.
Additional remarks.
FAGOR documentation for the 8025 M CNC.
Manual contents.

Chapter 1 CNC configuration.
Indicates the possible compositions: modular and compact.
Description and dimensions of the Central Unit.
Description and dimensions of the Monitor.
Description and dimensions of the Operator Panel.
Detailed description of all the connectors.

Chapter 2 Machine and Power connection
Indicates how to connect the main AC power
The ground connection.
The characteristics of the digital inputs and outputs.
The characteristics of the analog output.
The characteristics of the feedback inputs.
CNC set-up and start-up.
System input/output test.
Emergency input and output connection.

Chapter 3 Machine parameters.
How to operate with the machine parameters.
How to set the machine parameters.
Detail description of the general machine parameters.

Chapter 4 Machine parameters for the axes.
Detail description of the machine parameters for the axes.

Chapter 5 Machine Parameters for the spindle.
Detail description of the machine parameters for the spindle.

Chapter 6 Concepts.
Axes and coordinate systems. Nomenclature and selection.
Feedback systems, resolution.
Axis and gain adjustment.
Reference systems; Reference points, search and adjustment.
Software axis travel limits.
Acceleration / deceleration.
Unidirectional approach.
Spindle: speed control, range change.
Tools and tool magazine.
Treatment of the «Feed-hold» and «M-done» signals.
M, S, T auxiliary function transfer.
Pallet work.

Appendix Technical characteristics of the CNC. Enclosures.
Recommended probe connection circuits.
CNC inputs and outputs.
2-digit BCD spindle output conversion table.
Machine parameters. Summary chart, sequential list and setting chart.
Auxiliary «M» functions. Setting chart.
Leadscrew error compensation and cross compensation tables.
Maintenance.

Error codes.

1. CONFIGURATION OF THE CNC

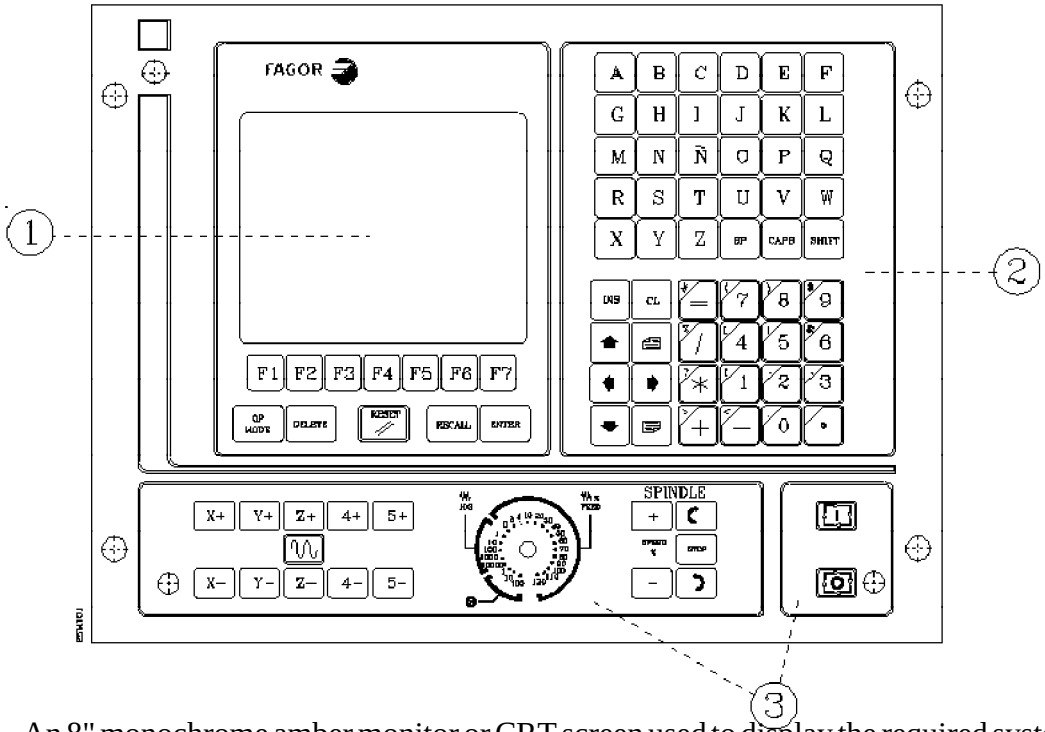
Atention:

The CNC is prepared to be used in Industrial Environments, especially on milling machines, lathes, etc. It can control machine movements and devices.

It can control machine movements and devices.

1.1 8025 CNC

The 8025 CNC is an enclosed compact module whose front view offers:



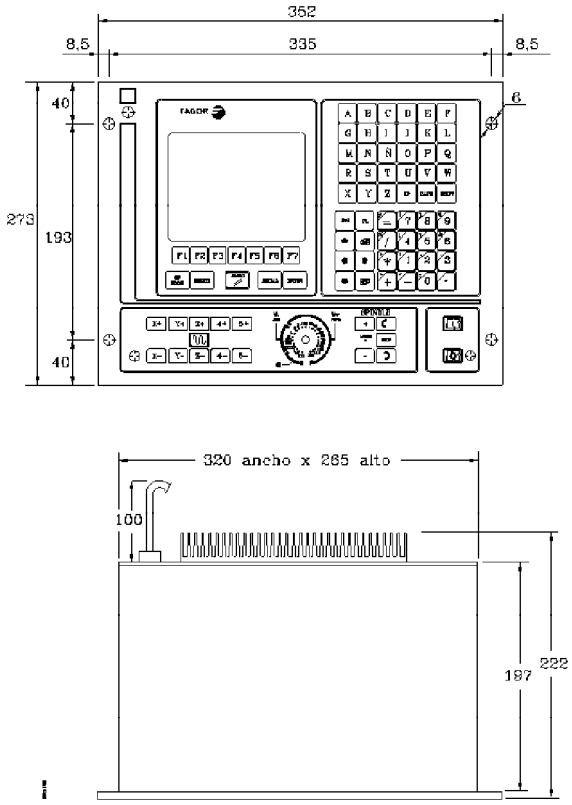
1. An 8" monochrome amber monitor or CRT screen used to display the required system information.

2. A keyboard which permits communications with the CNC; being possible to request information or change the CNC status by generating new instructions.

3. An operator panel containing the necessary keys to work in JOG mode as well as the Cycle Start/Stop keys.

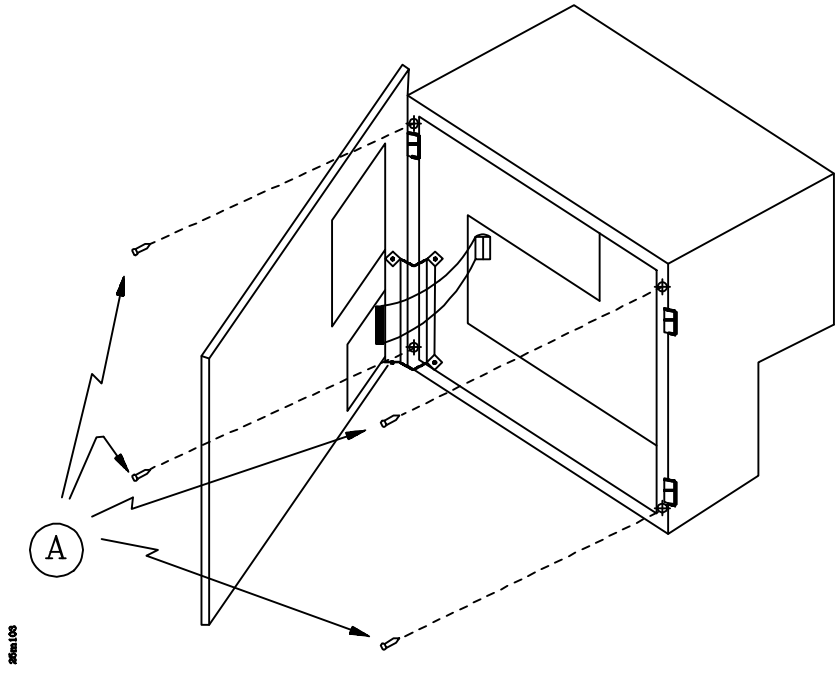
1.1.1 DIMENSIONS AND INSTALLATION OF THE 8025 CNC

This CNC, usually mounted on the machine pendant, has 4 mounting holes.



When installing it, leave enough room to swing the FRONT PANEL open in order to allow future access to its interior.

To open it, undo the 4 allen-screws located next to the CNC mounting holes.



1.2 8030 CNC

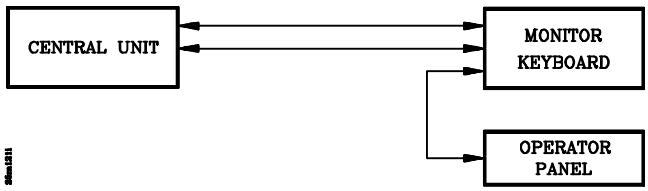
This model CNC consists of 3 independent interconnected modules. These modules can be mounted on different locations and they are:

- CENTRAL UNIT
- MONITOR/KEYBOARD
- OPERATOR PANEL

The OPERATOR PANEL module is connected to the MONITOR/KEYBOARD module via a cable supplied with that module.

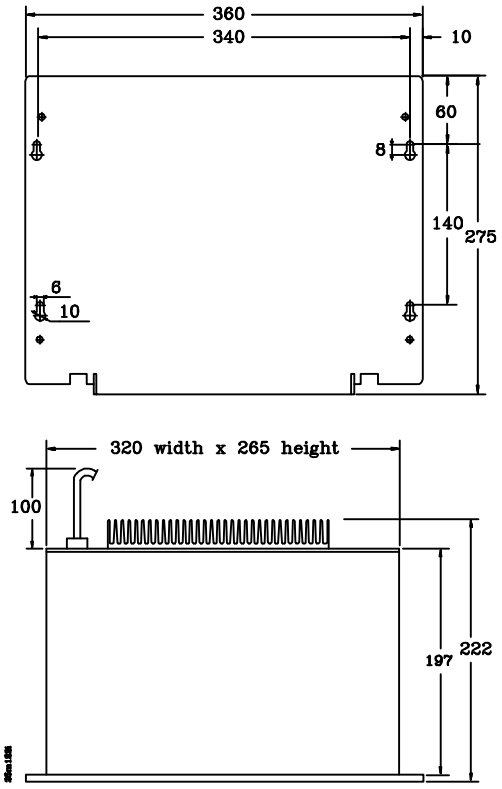
These two modules will be placed next to each other and must be connected with the CENTRAL UNIT module which could be located somewhere else. The two cables used to connect them together are also supplied with these modules. Their maximum length is 25 meters (82 feet) and they are referred to as:

- Video cable.
- Keyboard cable.



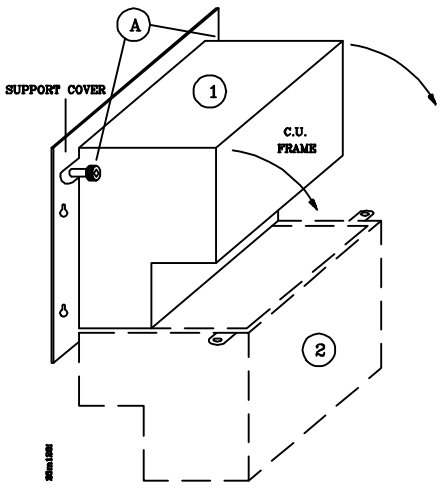
1.2.1 CENTRAL UNIT OF THE 8030 CNC

The CENTRAL UNIT is usually mounted in the electrical cabinet (machine enclosure) and it is secured by means of the mounting holes located on the support cover.

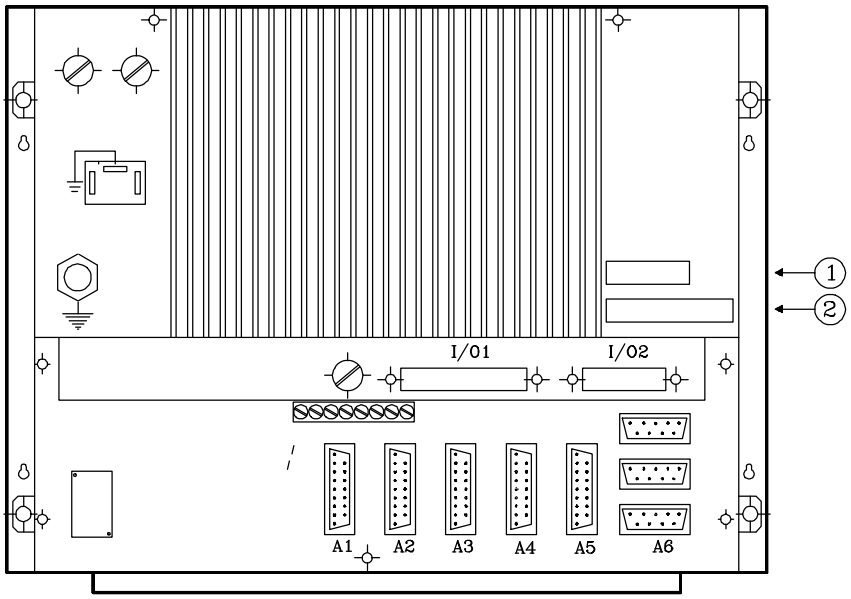


When installing it, observe enough clearance to swing the CENTRAL UNIT open in case of future inside manipulation.

To swing it open, once the support cover is secured on the machine enclosure, undo the two knurled nuts on top and swing it open while holding the body of the CENTRAL UNIT.



The CENTRAL UNIT has two connectors to connect it with the MONITOR/KEYBOARD module by means of the video and keyboard signal cables.



25m124

1.- 15-pin SUB-D type female connector for for video signals.

2.- 25-pin SUB-D type female connector for keyboard signals.

1.2.1.1 KEYBOARD CONNECTOR

It is a 25-pin SUB-D type female connector to connect the CENTRAL UNIT module to the MONITOR/KEYBOARD module.

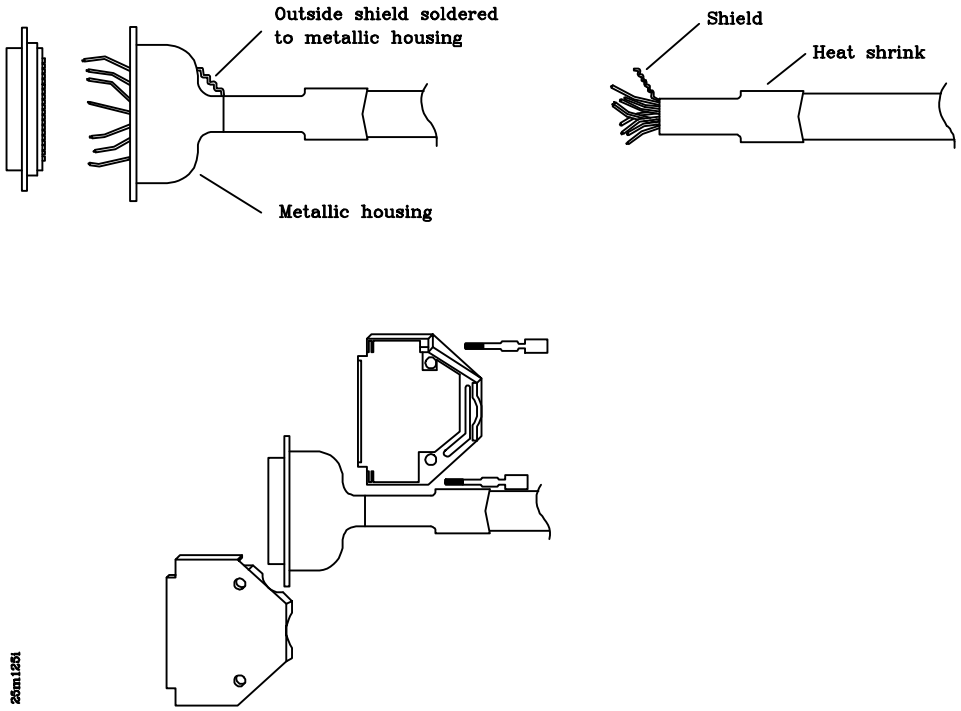
FAGOR AUTOMATION provides the cable required for this connection. It comes with a 25-pin SUB-D type male connector at each end.

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

PIN	SIGNAL
1	GND
2	C9
3	C11
4	C13
5	C15
6	C1
7	C3
8	C5
9	C7
10	D1
11	D3
12	D5
13	D7
14	C8
15	C10
16	C12
17	C14
18	C0
19	C2
20	C4
21	C6
22	D0
23	D2
24	D4
25	D6
Metal hood	Shield

The supplied cable has 25 wires (25 x 0.14mm²) with overall shield and acrylic cover. Its maximum length must be 25 meters (82 feet).

The cable shield is soldered to the metal hoods (housings) of both connectors and connected to pin 1 at both the CENTRAL UNIT and the MONITOR/KEYBOARD connectors.



1.2.1.2 VIDEO CONNECTOR

It is a 15-pin SUB-D type female connector used to interconnect the CENTRAL UNIT module and the MONITOR/KEYBOARD module.

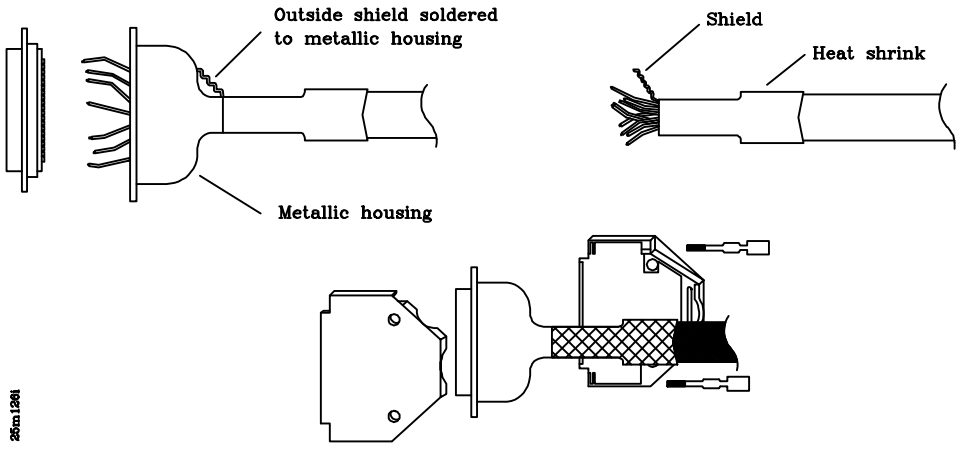
FAGOR AUTOMATION provides the cable required for this connection. It comes with a 15-pin SUB-D type male connector at one end and a 15-pin SUB-D type female connector at the other.

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

PIN	SIGNAL
1	GND
2	H
3	V
4	I
5	R
6	G
7	B
8	not connected
9	not connected
10	H
11	V
12	I
13	R
14	G
15	B
Metal hood	shield

The supplied cable has 6 twisted-pairs of wires (6 x 2 x 0.34mm²) with overall shield and acrylic cover. It has a specific impedance of 120 Ohm. Its maximum length must be 25 meters (82 feet).

The cable shield is soldered to the metal hoods (housings) of both connectors and connected to pin 1 at both the CENTRAL UNIT and MONITOR/KEYBOARD connectors.

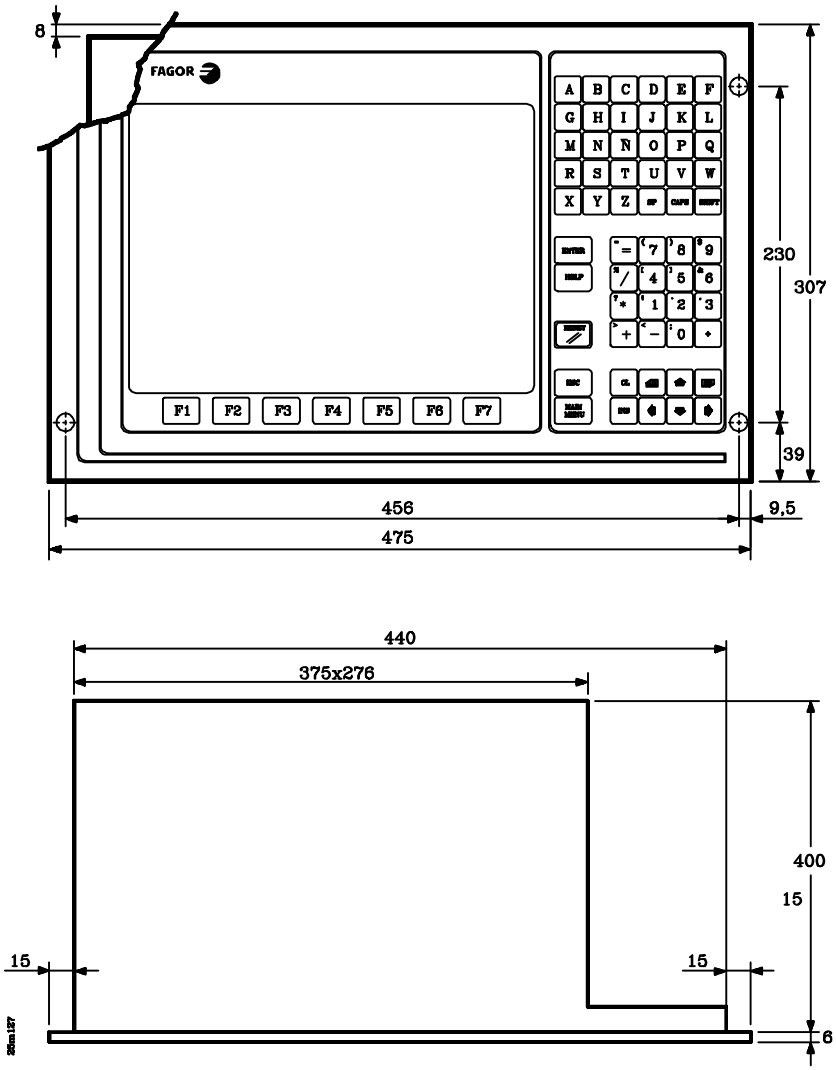


1.2.2 MONITOR/KEYBOARD OF THE 8030 CNC

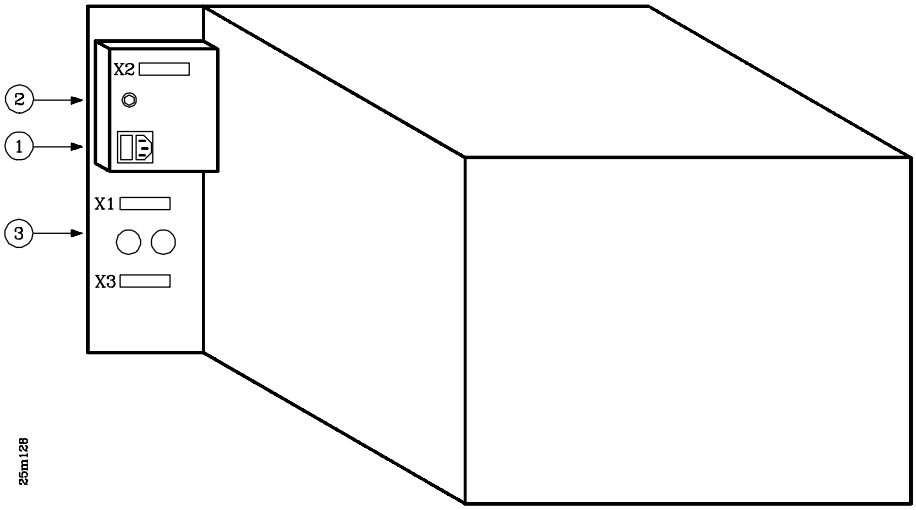
This module can be mounted on the machine pendant and it lets the operator get the necessary information at the MONITOR as well as operate the CNC by means of its KEYBOARD and OPERATOR PANEL.

This module has the connectors to connect it with the CENTRAL UNIT module.

1.2.2.1 DIMENSIONS OF THE MONITOR/KEYBOARD



1.2.2.2 ELEMENTS OF THE MONITOR/KEYBOARD



X1 25-pin SUB-D type female connector for keyboard signals.

X2 15-pin SUB-D type male connector for video signals.

X3 15-pin SUB-D type female connector to connect the MONITOR/KEYBOARD module to the OPERATOR PANEL module.

1.- A.C. power plug. Use the plug supplied with the unit to connect it to A.C. power and ground.

2.- Ground terminal. Used for general machine ground connection. Metric 6 screw.

3.- Buzzer

Atention:

Do not manipulate inside this unit

Only personnel authorized by Fagor Automatin 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.

1.2.2.3 CONNECTORS AND MONITOR/KEYBOARD INTERFACE

Connectors X1, X2

They are described in the chapter corresponding to the CENTRAL UNIT.

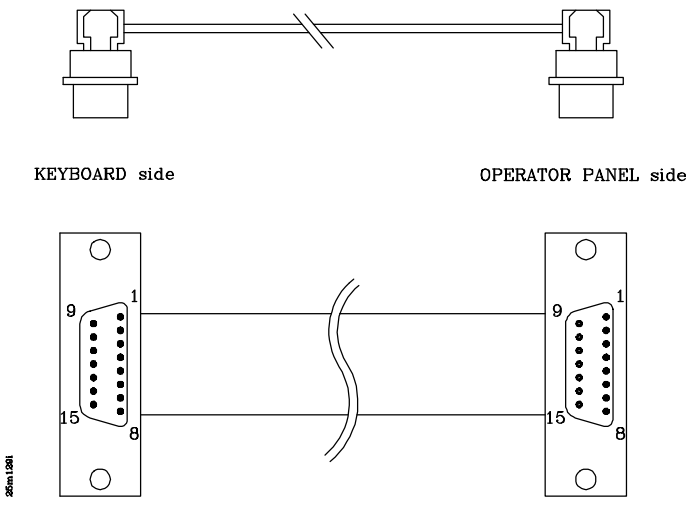
Connector X3

It is a 15-pin SUB-D type female connector used to connect the MONITOR/ KEYBOARD with the OPERATOR PANEL.

FAGOR AUTOMATION supplies the cable required for this connection. It is a 250mm-long 15-wire ribbon cable.

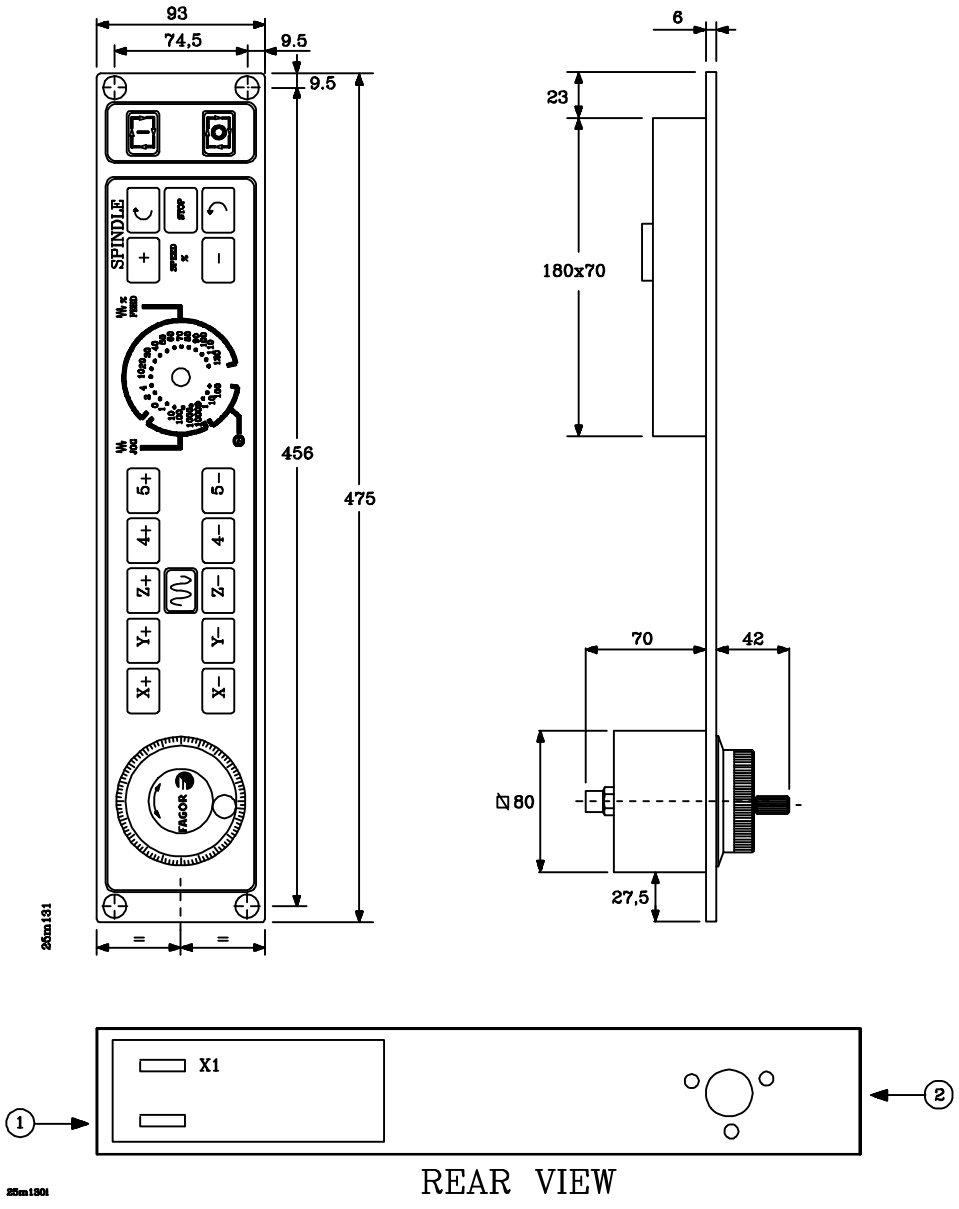
To obtain a greater distance between the Monitor/Keyboard and the Operator Panel, replace this cable with a round 15-conductor cable (15 x 0.14 mm²) with overall shield and acrylic rubber cover. The length of this cable plus the length of the one used between the Central Unit and the Keyboard (X1) must not exceed 25 meters (82 feet).

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	C14



1.2.3 OPERATOR PANEL OF THE 8030 CNC

This module is connected to the MONITOR/KEYBOARD module via a ribbon cable and it contains the JOG keys, Feedrate Override knob, Cycle Start and Stop keys, spindle keys as well as an Emergency-stop push-button (mushroom) or an optional electronic handwheel.



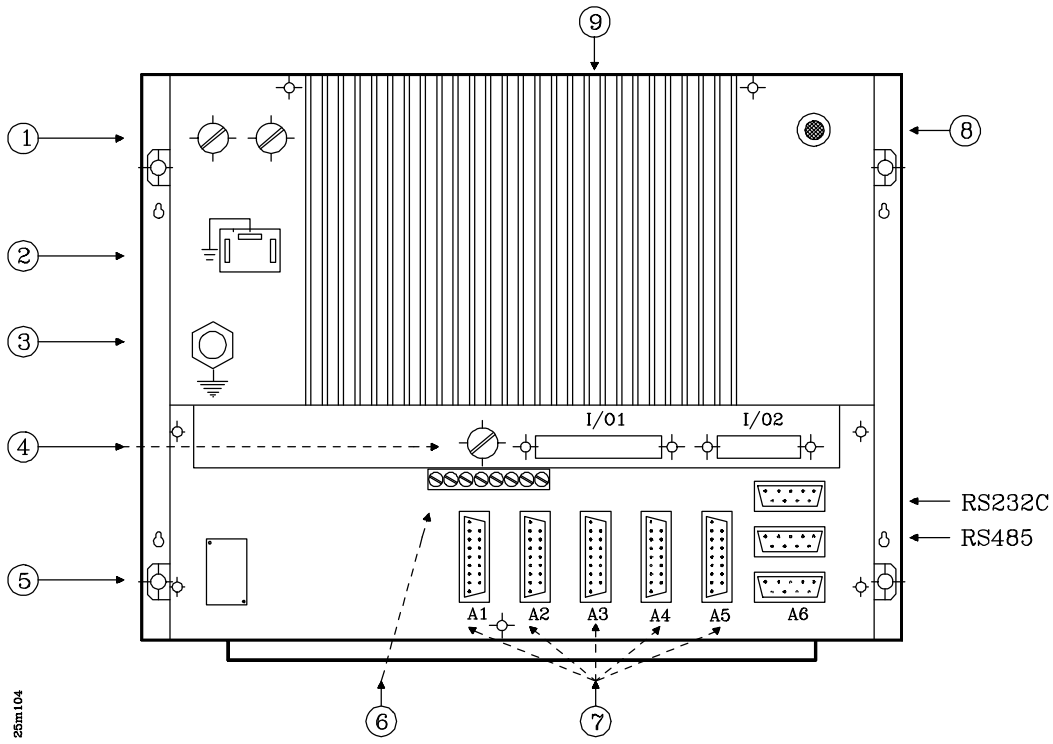
X1 15-pin SUB-D type female connector to connect the MONITOR/KEYBOARD module to the OPERATOR PANEL module.

It is described in the chapter corresponding to the MONITOR/KEYBOARD.

1.- Not being used at this time.

2.- Optional mounting location for the E-Stop button or Electronic handwheel.

1.3 CONNECTORS AND 8025/8030 INTERFACE



A1 **15-pin SUB-D type female connector** to connect the X axis feedback system. It accepts sine-wave signal.

A2 **15-pin SUB-D type female connector** to connect the Y axis feedback system. It accepts sine-wave signal.

A3 **15-pin SUB-D type female connector** to connect the Z axis feedback system. It accepts sine-wave signal.

A4 **15-pin SUB-D type female connector** to connect the W axis feedback system. It accepts sine-wave signal.

A5 **15-pin SUB-D type female connector** to connect the 5th axis (V) feedback system. It **does not** accept sine-wave signal.

When using the spindle encoder and an electronic handwheel, the CNC will only control up to 4 axes. This connector will then be used for the spindle encoder or the electronic handwheel (the other device will be connected to A6).

A6 **9-pin SUB-D type female connector** to connect the spindle encoder or an electronic handwheel and a touch probe. It **does not** accept sine-wave signal.

RS485 **9-pin SUB-D type female connector** to connect the RS485 serial line.

RS232C **9-pin SUB-D type female connector** to connect the RS232C serial line.

I/O1 **37-pin SUB-D type female connector** to interface with the electrical cabinet offering 10 digital inputs, 16 digital outputs and 4 analog outputs for servo drives (range: ±10 V.).

I/O2 **25-pin SUB-D type female connector** to interface with the electrical cabinet offering 16 digital outputs and 2 analog outputs for servo drives (range: ±10V.).

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

2- **AC power connector** To power the CNC. It must be connected to the power transformer and to ground.

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

4- **Fuse.** 3.15Amp./250V fast fuse (F) to protect the internal I/O circuitry of the CNC.

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

6- **Adjustment potentiometers for the analog outputs**. ONLY TO BE USED BY THE TECHNICAL SERVICE DEPARTMENT.

7- **10 dip-switches**. There are 2 under each feedback connector (A1 thru A5) and they are utilized to set the CNC according to the type of feedback signal being used.

8 **CRT brightness adjustment potentiometer**

9 **Heat-sink**.

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.

1.3.1 CONNECTORS A1, A2, A3, A4

They are 15-pin SUB-D type female connectors used to connect the feedback signals.

* Connector A1 for X axis feedback signals.
* Connector A2 for Y axis feedback signals.
* Connector A3 for Z axis feedback signals.
* Connector A4 for W axis feedback signals.

The cable must have overall shield. 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 square-wave feedback signals
5 6	$\overline{Io}$ Io	Machine Reference Signals (marker pulses)
7 8	Ac Bc	Sine-wave feedback signals
9 10 11 12 13 14	+5V. 0V. -5V. 	Power to feedback system. Not connected. Power to feedback system. Not connected. Power to feedback system. Not connected.
15	CHASSIS	Shield

Atention:

When using square-wave rotary encoders, their signals must be TTL compatible. Encoders with open collector outputs MUST NOT be used.

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.

1.3.1.1 DIP-SWITCHES FOR CONNECTORS A1, A2, A3, A4

There are 2 dip-switches below each feedback input connector (A1 thru A4) to set the CNC according to the type of feedback signal being used.

Switch 1 indicates whether the feedback signal is sine-wave or square-wave and switch 2 indicates whether the feedback signal is single- or double-ended (differential).

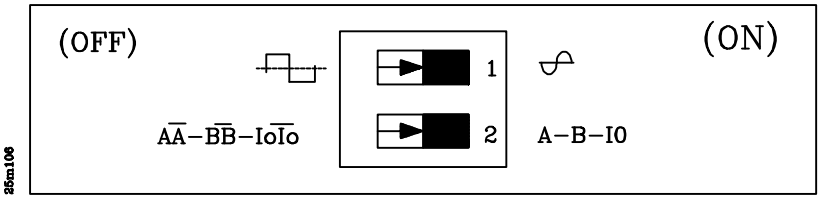
The possible types of feedback signals to be used at connectors A1 thru A4 are:

* Sine-wave (Ac, Bc, Io)
* Single-ended square-wave (A, B, Io)
* Double-ended (differential) square-wave (A, $\overline{A}$, B, $\overline{B}$, Io, $\overline{Io}$)

To select the type of signal for each axis, use the switch combinations below:

Dip-switch 1	Dip-switch 2	SIGNAL AND FUNCTION
ON	ON	Single-ended sine-wave signal (Ac,Bc,Io)
ON	OFF	Double-ended sine-wave signal **"Not allowed"**
OFF	ON	Single-ended square-wave signal (A,B,Io)
OFF	OFF	Double-ended square-wave (A, $\overline{A}$, B, $\overline{B}$, Io, $\overline{Io}$)

There is a label next to each dip-switch pair indicating the meaning of each switch.



1.3.2 CONNECTOR A5

It is a 15-pin SUB-D type female connector for the 5th axis (V) feedback signal.
It does not accept sine-wave signals.

When using the spindle encoder and an electronic handwheel, the CNC will only control up to 4 axes. This connector will then be used for the spindle encoder or the electronic handwheel (the other device will be connected to A6).

The cable must have overall shield. 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 $\overline{B}$	Double-ended square-wave signal.
5 6	Io $\overline{Io}$	Machine Reference signals (marker pulse)
7 8	Micro Io 0V.	"V" axis home switch input. "V" axis home switch 0V input. (elec.cabinet)
9 10 11 12 13 14	+5V. 0V. -5V. 	Power to feedback system. Not connected. Power to feedback system. Not connected. Power to feedback system. Not connected.
15	CHASSIS	Shield.

Atention:

When using square-wave rotary encoders, their signals must be TTL compatible. Encoders with open collector outputs MUST NOT be used.

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.

1.3.2.1 DIP-SWITCHES FOR CONNECTOR A5

There are 2 dip-switches below this feedback input connector to set the CNC according to the type of feedback signal being used.

Switch 1 indicates whether the feedback signal is sine-wave or square-wave and switch 2 indicates whether the feedback signal is single- or double-ended (differential).

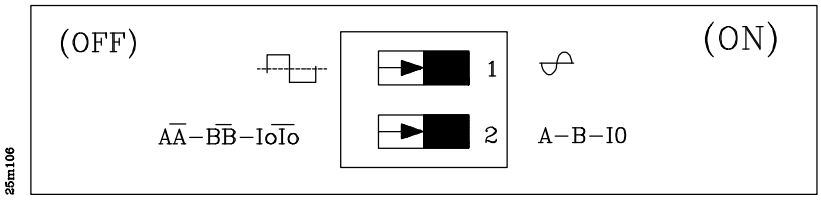
The possible types of feedback signals to be used at connector A5 are:

* Single-ended square-wave (A, B, Io)
* Double-ended (differential) square-wave (A, $\overline{A}$, B, $\overline{B}$, Io, $\overline{Io}$)

To select the type of signal for each axis, use the switch combinations below:

Dip-switch 1	Dip-switch 2	SIGNAL AND FUNCTION
ON	ON	Single-ended sine-wave signal **"Not allowed"**
ON	OFF	Double-ended sine-wave signal **"Not allowed"**
OFF	ON	Single-ended square-wave signal (A,B,Io)
OFF	OFF	Double-ended square-wave (A, $\overline{A}$, B, $\overline{B}$, Io, $\overline{Io}$)

There is a label next to each dip-switch pair indicating the meaning of each switch.



1.3.3 CONNECTOR A6

It is a 9-pin SUB-D type female connector to connect the spindle encoder or the electronic handwheel and a touch probe. **It does not take sine-wave signals.**

The cable must have overall shield. 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.

There are two probe inputs (5V and 24V) and the 0V of the external power supply must be connected to the "probe 0V input" (pin 8).

The appendix of the manual includes information about these probe inputs as well as recommended probe connection diagrams.

All cable shields must be connected to ground ONLY at the CNC end through the connector leaving the other end of the cable not connected. The wires of a shielded cable must not be unshielded (sticking out) for more than 75mm (about 3 inches).

PIN	SIGNAL AND FUNCTION	
1 2	A B	Square-wave signals from the spindle encoder or from the electronic handwheel
3	Io	Home marker pulse (Machine Reference)
4 5	+5V. 0V.	Power to spindle encoder or handwheel
6 7 8	PROB 5 PROB 24 0 PROB	Probe input: 5 V. TTL Probe input: 24 Vcc Probe input: 0 V.
9	CHASSIS	Shield.

Atention:

When using square-wave rotary encoders, their signals must be TTL compatible. **Encoders with open collector outputs MUST NOT be used.**

When using a FAGOR 100P handwheel, the axis selector signal must be connected to pin 3.

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.

1.3.3.1 MACHINE WITH "V" AXIS" AND HANDWHEEL OR SPINDLE ENCODER

When using a "V" axis, machine parameter P616(4) must be set to "1". In this case, it is possible to use connector A6 to connect the electronic handwheel or the spindle encoder; but not both at the same time.

A1	A2	A3	A4	A5	A6
X	Y	Z	W	V	S
X	Y	Z	W	V	Handwheel

Machine parameter P800 must also be set with the corresponding value to indicate which one of them is being connected.

1.3.3.2 WITHOUT "V AXIS" AND WITH ELECTRONIC HANDWHEEL OR SPINDLE ENCODER

When the machine does not have a "V" axis, machine parameter P616(4) must be set to "0". In this case, it is possible to connect the electronic handwheel or the spindle encoder or both at the same time.

It is also possible to select the connector (A5 or A6) where each device is being connected.

A1	A2	A3	A4	A5	A6
X	Y	Z	W	Handwheel	S
X	Y	Z	W	S	Handwheel

1.3.4 RS232C CONNECTOR

9-pin SUB-D type female connector 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	—-	Not connected
9	DTR	Data Terminal Ready

SUGGESTIONS FOR THE RS232C INTERFACE

* **Connect/disconnect peripheral.**

The CNC must be powered off when connecting or disconnecting any peripheral through this connector.

* **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).**

  For greater distances, it is suggested to intercalate RS232C-to-RS422A signal converters (and vice-versa). Contact the corresponding distributor.

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

  Use shielded 7-conductor cable of 7*0.14mm² section.

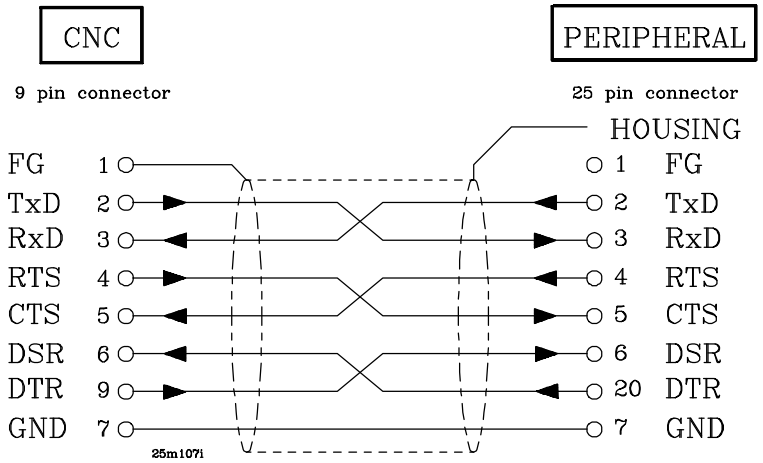
* **Transmission speed (baudrate)**. The baudrate normally used with peripherals is 9600 baud.

  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.

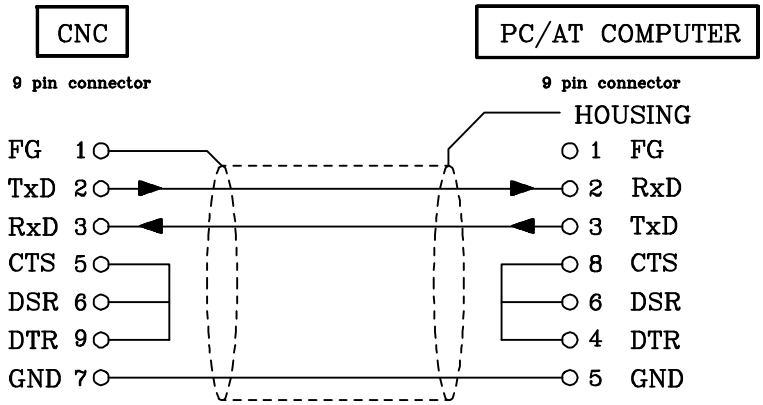
RECOMMENDED CONNECTIONS FOR THE RS232C INTERFACE

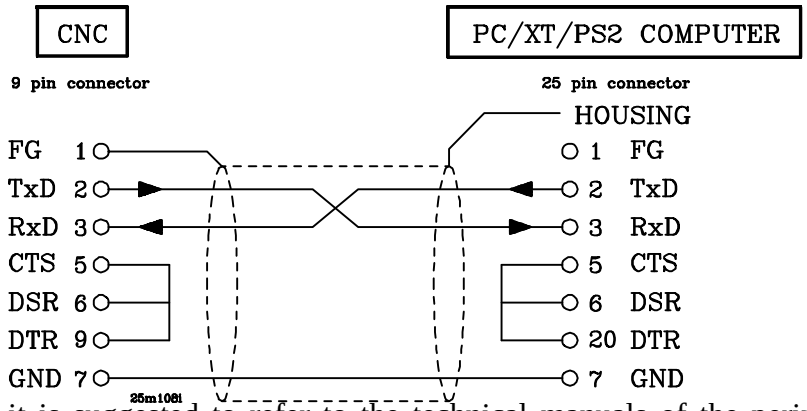
* Complete connection



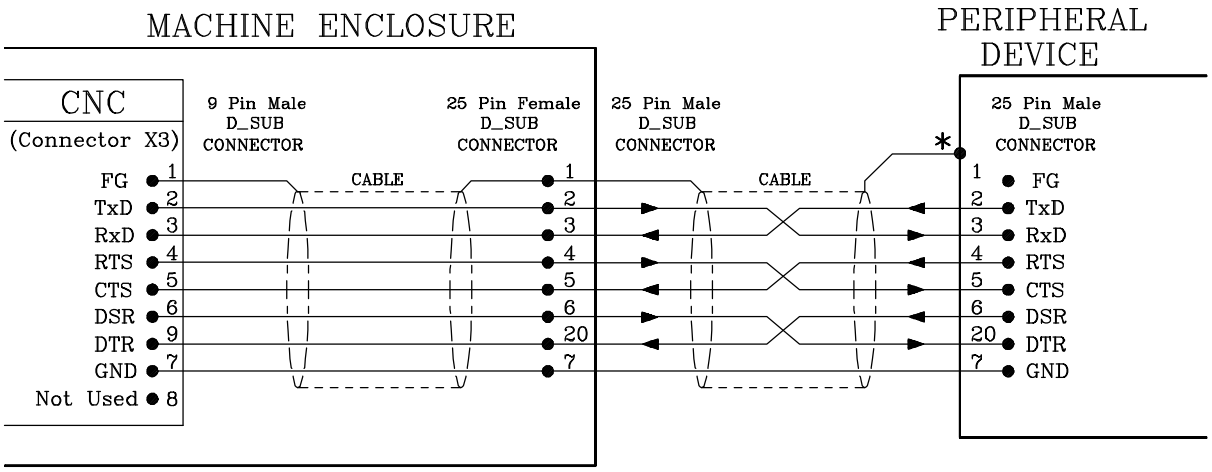
* Simplified connection

To be used when the peripheral or the computer meets one of the following requirements:
- It does not have the RTS signal.
- It is connected via DNC.
- The receiver can receive data at the selected baudrate.

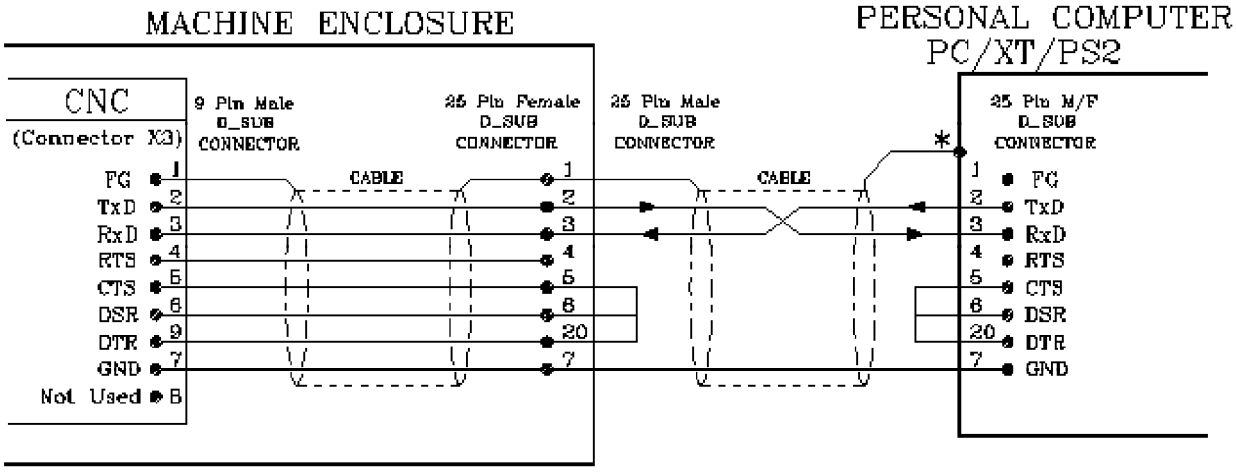




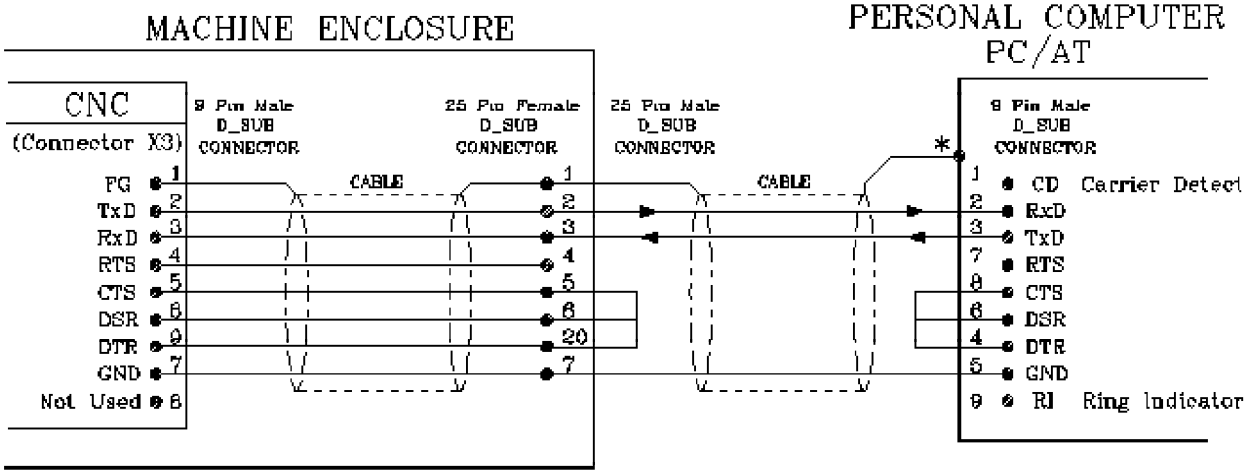
Nevertheless, it is suggested to refer to the technical manuals of the peripheral equipment in case there should be any discrepancy.



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

1.3.5 RS485 CONNECTOR

It is a 9-pin SUB-D type female connector to connect the RS485 serial line.

This serial line is used to integrate the CNC into the FAGOR LOCAL AREA NETWORK (LAN) in order to communicate with other FAGOR CNCs and PLCs (FAGOR PLC 64).

PIN	SIGNAL	FUNCTION
1	---	*Not connected*
2	---	*Not connected*
3	TxD	Transmit Data
4	---	*Not connected*
5	---	*Not connected*
6	---	*Not connected*
7	---	*Not connected*
8	TxD	Transmit Data
9	---	*Not connected*

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.

For better immunity of the RS485 serial line against conducted electromagnetic disturbances, it is recommended to solder the cable mesh to the metal hood of the connector.

1.3.5.1 RECOMMENDED CABLE FOR THE RS485

TECHNICAL CHARACTERISTICS

CABLE "TWINAXIAL"

SPECIFICATIONS		
Conductor	Type: Material: Resistance:	02 AWG twisted 7x28 Copper (only one stained wire) Max 11 L per every 305m. (1000 ft)
Insulator	Material:	Teflon
Shields	Material Type Cover Resistance	Stained copper Braid 34 AWG. 8 ends / 16 carriers Minimum 95% Maximum 3L per every 305m. (1000 ft)
Covering	Material: Outside diameter	Teflon Nominal 7mm. (0.257inches)
Capacitance		Maximum 53,1 pF/m (16.2 pF/ft)
Impedance		107± 5% Ohm at 1 MHz.

1.3.6 CONNECTOR I/O 1

It is a 37-pin SUB-D type female connector to interface with the electrical cabinet.

Pin	SIGNAL AND FUNCTION	
1	0V.	Input from external power supply
2	T Strobe	Output. The BCD outputs represent a tool code.
3	S Strobe	Output. The BCD outputs represent a spindle speed code.
4	M Strobe	Output. The BCD outputs represent an M code.
5	Emergency	Output.
6	W Enable Threading ON	Output.
7	Z Enable	Output.
8	Y Enable	Output.
9	X Enable	Output.
10	X home switch	Input from machine reference switch.
11	Y home switch	Input from machine reference switch.
12	Z home switch	Input from machine reference switch.
13	W home switch	Input from machine reference switch.
14	Emergency Stop	Input.
15	Feed Hold Transfer inhibit M-done	Input.
16	Stop Emergency subrout.	Input.
17	Start Rapid JOG Enter in Play-back	Input
18	Block Skip	Conditional Input
19	DRO	Input. The CNC acts as a DRO
20	MST80	BCD coded output, weight: 80
21	MST40	BCD coded output, weight: 40
22	MST20	BCD coded output, weight: 20
23	MST10	BCD coded output, weight: 10
24	MST08	BCD coded output, weight: 8
25	MST04	BCD coded output, weight: 4
26	MST02	BCD coded output, weight: 2
27	MST01	BCD coded output, weight: 1
28	CHASSIS	Connect all cable shields to this pin.
29	24V.	Input from external power supply.
30	±10V	Analog output for X axis servo drive.
31	0V.	Analog output for X axis servo drive.
32	±10V	Analog output for Y axis servo drive.
33	0V.	Analog output for Y axis servo drive.
34	±10V	Analog output for Z axis servo drive.
35	0V.	Analog output for Z axis servo drive.
36	±10V	Analog output for the spindle drive.
37	0V.	Analog output for the spindle drive.

Atention:

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.

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.

1.3.6.1 INPUTS OF CONNECTOR I/O 1

0 V. Pin 1

INPUT from external power supply.

X AXIS HOME SWITCH Pin 10

This INPUT must be high (24V) as long as the machine reference switch for the X axis is pressed.

Y AXIS HOME SWITCH Pin 11

This INPUT must be high (24V) as long as the machine reference switch for the Y axis is pressed.

Z AXIS HOME SWITCH Pin 12

This INPUT must be high (24V) as long as the machine reference switch for the Z axis is pressed.

W AXIS HOME SWITCH Pin 13

This INPUT must be high (24V) as long as the machine reference switch for the W axis is pressed.

EMERGENCY STOP Pin 14

This INPUT must be normally high (24V).

When set low (0V), the CNC deactivates the axis enables and analog voltages, it interrupts the part program execution and it displays ERROR 64 on the CRT.

It **does not** imply an emergency output (pin 5 of this connector).

FEED HOLD / TRANSFER INHIBIT / M-DONE Pin 15

This INPUT must be normally high (24V) and its meaning depends on the type of block or function being executed at the time.

* If while moving the axes this signal (FEED-HOLD) is set low (0V), the CNC maintains the spindle turning and stops the axes bringing their analog voltages to 0V while maintaining their enables active.

  When this input is brought back high (24V), the axes will resume their movements.

* If while executing a motionless block this signal (TRANSFER INHIBIT) is set low (0V), the CNC interrupts the program execution at the end of the block currently in execution.

  When this signal is brought back high, the CNC resumes program execution.

* The "M-DONE" signal is used when machine parameter P605(5) is set to "1".

  The CNC waits for the electrical cabinet to execute the requested miscellaneous M function. In other words, it waits for the "M-done" input to be set high (24V).

STOP/ EMERGENCY SUBROUTINE Pin 16

This INPUT must be normally high (24V) and its meaning depends on the setting of machine parameter "P727".

* P727= 0. There is **no** emergency subroutine.

  When this input is set low (0V), the CNC interrupts the program execution just as if the  key were pressed at the OPERATOR PANEL.

  To resume program execution, it is necessary to bring this input back high (24V) and press the  key at the OPERATOR PANEL.

* P727 other than "0". **There is** an emergency subroutine.

  When a down-flank (trailing edge or high-to-low transition) of this signal (EMERGENCY SUBROUTINE) is detected, the CNC interrupts the execution of the current program and "jumps" to execute the subroutine whose number is indicated by machine parameter P727.

START / RAPID TRAVERSE / ENTER Pin 17

This INPUT must be normally low (0V) and its meaning depends on the type of operation selected.

* If an up-flank (leading edge or low-to-high transition) of this signal (START) is detected while in Automatic, Single Block or Dry-Run mode, the CNC considers that the external CYCLE START key is pressed and it behaves as if the key were pressed at the OPERATOR PANEL.

  However, to disable the key of the OPERATOR PANEL in order to only use this input, set machine parameter P618(1) to "1".

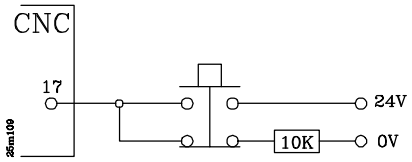
* When machine parameter P609(7) has been set to "1" and this input (RAPID TRAVERSE) is high (24V), the CNC acts as if the key were pressed.

  The CNC will perform all G01, G02 and G03 movements at 200% of the programmed feedrate F. If the resulting feedrate is greater than the maximum established by machine parameter P708, the CNC will issue the corresponding error message.

  Also, in the JOG mode and while this input is maintained high (24V), all movements will be carried out in rapid (G00).

* If while in PLAY BACK mode and being machine parameter P610(3) set to "1", the CNC detects an up-flank (leading edge or low-to-high transition) at this input, it acts as if the [ENTER] key were pressed.

While inactive, this input must be connected to 0V through a 10KOhm resistor.



BLOCK SKIP (Conditional input) Pin 18

Every time the CNC executes the miscellaneous function M01 (conditional stop), it analyzes the status of this input. If high (24V), the CNC will interrupt the execution of the program.

By the same token, every time the CNC must execute a conditional block, it will analyze the status of this input and it will execute the block if this input is high (24V).

When the JOG mode is selected, the CNC analyzes the status of this input. If active, (high) the CNC ignores the key.

DRO (DRO mode) Pin 19

If this input is set high (24V) while in the JOG mode, the CNC acts as a DRO.

1.3.6.2 OUTPUTS OF CONNECTOR I/O 1

T Strobe Pin 2

The CNC sets this output high (24V) whenever it sends a tool code (T function) via the BCD outputs (pins 20 thru 27).

S Strobe Pin 3

The CNC sets this output high (24V) whenever it sends a spindle speed code (S function) via the BCD outputs (pins 20 thru 27).

M Strobe Pin 4

The CNC sets this output high (24V) whenever it sends an M function code via the BCD outputs (pins 20 thru 27).

EMERGENCY Pin 5

The CNC activates this output whenever it detects an alarm condition or internal emergency.

This output is normally high (24V) or low (0V) depending on the setting of machine parameter P605(8).

W AXIS ENABLE / THREADING ON Pin 6

The function of this OUTPUT depends on the axes controlled by the CNC.

When the W axis is used, the CNC sets this output (W ENABLE) high (24V) whenever the W axis drive must be enabled.

When the W axis is **not** used (3-axis machine, machine parameter P11=0), the CNC sets this output (THREADING ON) high (24V) whenever an electronic threading block (G33) is being executed.

Z AXIS ENABLE Pin 7

The CNC sets this output high (24V) to enable the Z axis servo drive.

Y AXIS ENABLE Pin 8

The CNC sets this output high (24V) to enable the Y axis servo drive.

X AXIS ENABLE Pin 9

The CNC sets this output high (24V) to enable the X axis servo drive.

MST80 Pin 20
MST40 Pin 21
MST20 Pin 22
MST10 Pin 23
MST08 Pin 24
MST04 Pin 25
MST02 Pin 26
MST01 Pin 27

The CNC uses these outputs to indicate to the electrical cabinet the M, S or T function that has been selected.

This information is BCD coded and the significance (weight) of each output is expressed by the corresponding mnemonic.

For example, to select the first spindle speed range, the CNC sends the M41 code out to the electrical cabinet.

MST80	MST40	MST20	MST10	MST08	MST04	MST02	MST01
0	1	0	0	0	0	0	1

Together with these signals, the CNC will activate the "M Strobe", "T Strobe" or "S Strobe" output to indicate the type of function being selected.

CHASSIS Pin 28

This pin must be used to connect all cable shields to it.

Analog voltage for X ±10V. Pin 30
Analog voltage for X 0V. Pin 31

These outputs provide the analog voltage for the X axis servo drive. The cable used for this connection must be shielded.

Analog voltage for Y ±10V. Pin 32
Analog voltage for Y 0V. Pin 33

These outputs provide the analog voltage for the Y axis servo drive. The cable used for this connection must be shielded.

Analog voltage for Z ±10V. Pin 34
Analog voltage for Z 0V. Pin 35

These outputs provide the analog voltage for the Z axis servo drive. The cable used for this connection must be shielded.

Spindle analog voltage ±10V. Pin 36
Spindle analog voltage 0V. Pin 37

These outputs provide the analog voltage for the spindle drive. The cable used for this connection must be shielded.

1.3.7 CONNECTOR I/O 2

It is a 25-pin SUB-D type female connector to interface with the electrical cabinet.

PIN	SIGNAL AND FUNCTION	
1	0V.	Input from external power supply.
2	0V.	Input from external power supply.
3	Output M1	Value of bit 1 of the decoded M function table.
4	Output M2	Value of bit 2 of the decoded M function table.
5	Output M3	Value of bit 3 of the decoded M function table.
6	Output M4	Value of bit 4 of the decoded M function table.
7	Output M5	Value of bit 5 of the decoded M function table.
8	Output M6	Value of bit 6 of the decoded M function table.
9	Output M7	Value of bit 7 of the decoded M function table.
10	Output M8	Value of bit 8 of the decoded M function table.
11	Output M9	Value of bit 9 of the decoded M function table.
12	Output M10 V Enable	Value of bit 10 of the decoded M function table.
13	Output M11 Addit. data	Value of bit 11 of the decoded M function table.
14	0V	Analog voltage output for V axis servo drive.
15	±10V.	Analog voltage output for V axis servo drive.
16	CHASSIS	Connect all cable shields to this pin.
17	0V	Analog voltage output for W axis servo drive.
18	±10V.	Analog voltage output for W axis servo drive.
19	24V.	Input from external power supply.
20	24V.	Input from external power supply.
21	JOG	Output. JOG mode is selected.
22	Output M15 Magaz. Rot.	Value of bit 15 of the decoded M function table.
23	Output M14 Reset	Value of bit 14 of the decoded M function table.
24	Output M13 Cycle On Automatic G00	Value of bit 13 of the decoded M function table.
25	Output M12 Vertical axis	Value of bit 12 of the decoded M function table.

Atention:

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.

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.

1.3.7.1 OUTPUTS OF CONNECTOR I/O 2

"Decoded M" outputs Pins 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 22, 23, 24, 25

These OUTPUTS provide the values indicated at the table corresponding to the selected M function.

For example: If the table corresponding to function M41 has been set as follows:

M41 100100100100100 (outputs to be activated)
00100100100100100 (outputs to be deactivated)

Every time this M41 function is executed, the CNC will act as follows:

	M01	M02	M03	M04	M05	M06	M07	M08	M09	M10	M11	M12	M13	M14	M15
Pin I/O2	3	4	5	6	7	8	9	10	11	12	13	25	24	23	22
at 24V	x			x			x			x			x		
at 0V			x			x			x			x			x
Not modified		x			x			x			x			x	

Outputs M10 / V axis Enable Pin 12

This output provides the value of bit 10 of the decoded table corresponding to the selected M function.

When the V axis is being used, this output will be utilized as Enable signal for this axis.

Therefore, When having a V axis, be careful not to set the bit of the decoded M table which corresponds to this M10 output since the CNC will activate it in both cases.

Outputs M11 / Additional data Pin 13

This output provides the value of bit 11 of the decoded table corresponding to the selected M function.

When operating with M06 and RANDOM tool magazine, the CNC will set this output high (24V) whenever a SPECIAL TOOL is selected.

If the tool magazine being used is NOT RANDOM and the M06 function requires a special treatment (prior tool positioning, etc.), machine parameter P603(2) must be set to "1" and the CNC will set this output high (24V) every time M06 is selected.

Care must be taken when having one of these options not use the bit of the decoded M table corresponding to this output M11 since the CNC will activate it in both cases.

V axis analog voltage ±10V. Pin 15
V axis analog voltage 0V. Pin 14

These outputs provide the analog voltage for the V axis servo drive. The cable used for this connection must be shielded.

W axis analog voltage ±10V. Pin 18
W axis analog voltage 0V. Pin 17

These outputs provide the analog voltage for the W axis servo drive. The cable used for this connection must be shielded.

JOG Pin 21

The CNC sets this OUTPUT high (24V) whenever the JOG mode is selected.

Outputs M15 / Tool magazine rotating direction Pin 22

This OUTPUT provides the value of bit 15 of the decoded M table corresponding to the selected M function.

If machine parameter P605(7) is set so the tool magazine rotates in the quickest direction, this output will indicate the rotating direction. If the tool magazine is turning in the positive direction (counting up), this output will be set low (0V) and if it is turning in the negative direction (counting down), this output will be set high (24V).

Care must be taken, when having this option, not to use the bit of the decoded M table corresponding to this output M15 since the CNC will activate it in both cases.

Outputs M14 / RESET Pin 23

This OUTPUT provides the value of bit 14 of the decoded M table corresponding to the selected M function.

If machine parameter P609(3) is set to "1" to provide a RESET pulse, this positive reset pulse will be output every time the CNC executes a RESET.

Care must be taken, when having this option, not to use the bit of the decoded M table corresponding to this output M14 since the CNC will activate it in both cases.

Outputs M13 / CYCLE ON / AUTOMATIC / G00 Pin 24

This OUTPUT provides the value of bit 13 of the decoded M table corresponding to the selected M function.

If machine parameter P611(1) is set to "1" so the CNC provides the status of the CYCLE ON signal, this OUTPUT will be set high (24V) every time a part-program block is being executed.

If machine parameter P611(6) is set to "1" so the CNC provides the status of the AUTOMATIC signal, this OUTPUT will be set high (24V) whenever the AUTOMATIC mode of operation is selected.

If machine parameter P613(4) is set to "1" so the CNC provides the status of the G00 signal, this OUTPUT will be set high (24V) whenever the CNC is executing a rapid positioning move (G00).

Care must be taken, when having one of these options, not to use the bit of the decoded M table corresponding to this output M13 since the CNC will activate it in both cases.

Outputs M12 / Vertical axis movement Pin 25

This OUTPUT provides the value of bit 12 of the decoded M table corresponding to the selected M function.

If machine parameter P613(2) is set to "1" in order for the CNC to provide the status of the vertical axis movement, this output will indicate the direction of that movement. If the axis is moving in the positive direction (counting up), this output will be set low (0V) and it will be set high (24V) if moving in the negative direction (counting down).

Care must be taken, when having this option, not to use the bit of the decoded M table corresponding to this output M12 since the CNC will activate it in both cases.

2. *POWER AND MACHINE INTERFACE*

Atention:

Power switch

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

Intall 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 INTERFACE

The rear of the 8025 CNC has a three-prong connector for AC and ground connection.

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

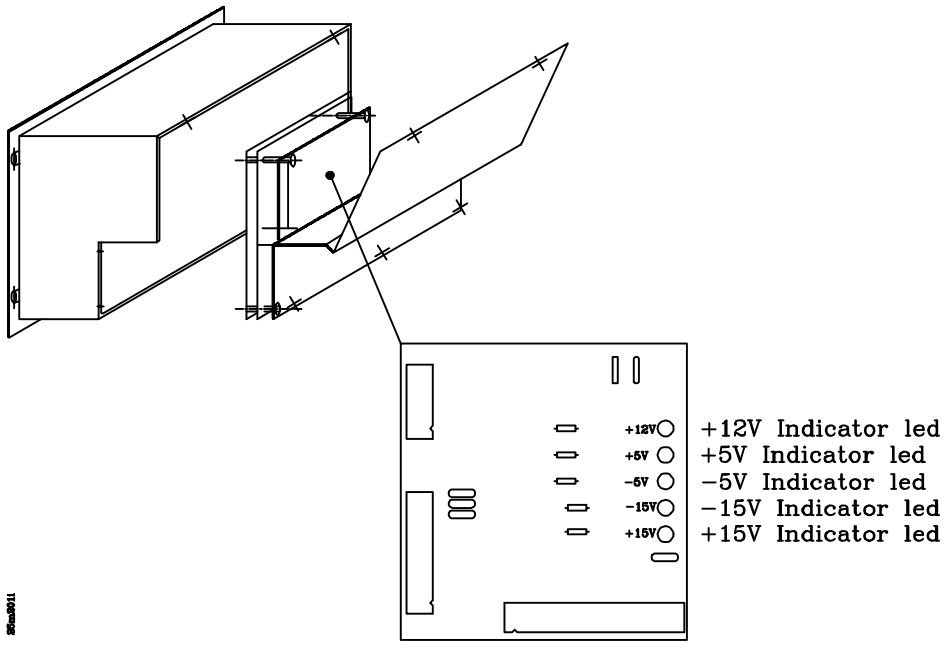
The power outlet to connect the equipment must be near it and easily accessible.

In case of overload or overvoltage, it is recommended to wait for 3 minutes before powering the unit back up in order to prevent any possible damage to the power supply.

2.1.1 INTERNAL POWER SUPPLY

Inside the 8025 CNC there is a power supply providing the required voltages.

Besides the 2 outside AC power fuses (one per line), it has a 5 Amp. fuse inside to protect it against overcurrent.



2.2 MACHINE INTERFACE

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.

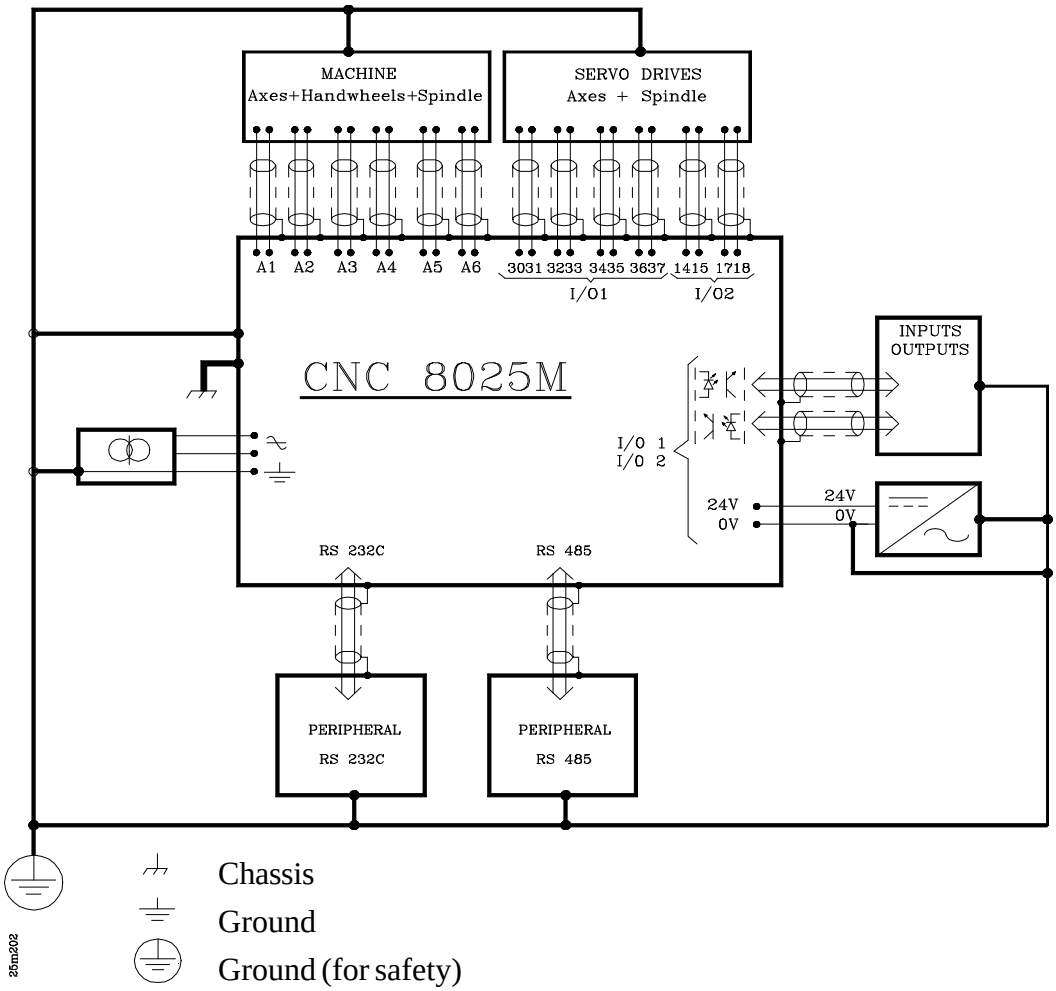
Even when a proper **ground** connection reduces the effects of electrical interference (noise), the signal cables require additional protection.
This is generally achieved by using twisted-pair cables which are also covered with anti-static shielding mesh-wire. This shield must be connected to a specific point avoiding **ground loops** that could cause undesired effects. This connection is usually done at one of the CNC's ground points.

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 not smaller than 8 mm².

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 CNC has several optocoupled digital outputs which may be used to activate relays, deacons, etc.

These digital outputs, with galvanic isolation 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 D.C.
Minimum voltage value	+18 V D.C.
Output voltage	Vcc.- 2V
Maximum output current	100 mA.

All outputs are protected by means of:

Galvanic isolation by optocouplers.
External 3A fuse for protection against output overload (greater than 125mA), external power supply overvoltage (over 33V DC) and against reverse connection of the external power supply.

2.2.3 DIGITAL INPUTS.

The digital inputs of the CNC are to used to "read" external devices.

All of them are galvanically isolated from the outside world by optocouplers.

The electrical characteristics of these inputs are:

Nominal voltage value	+24 V DC
Maximum voltage value	+30 V.
Minimum voltage value	+18 V.
High threshold voltage (logic state 1) over	+18V.
Low threshold voltage (logic state 0) under	+5V.
Typical input consumption	5 mA.
Maximum consumption per input	7 mA.

All inputs are protected by means of:

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

Atention:

The external 24V power supply used for the digital 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 CNC has 6 analog outputs which could be used to command servo drives, spindle drives and other devices.

The electrical characteristics of these outputs are:

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

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

Atention:

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

2.2.5 FEEDBACK INPUTS

The feedback inputs are used to receive sine-wave, single-ended and double-ended square-wave signals coming from linear or rotary transducers (encoders).

Connector A1 is used for the X axis feedback signals and it accepts sine-wave and double-ended (differential) square-wave signals.

Connector A2 is used for the Y axis feedback signals and it accepts sine-wave and double-ended (differential) square-wave signals.

Connector A3 is used for the Z axis feedback signals and it accepts sine-wave and double-ended (differential) square-wave signals.

Connector A4 is used for the W axis feedback signals and it accepts sine-wave and double-ended (differential) square-wave signals.

Connector A5 is used for the 5th axis (V) feedback signals and it accepts double-ended (differential) square-wave signals.

Connector A6 is used for the spindle encoder or for the electronic handwheel and it accepts single-ended (not differential) square-wave signals.

The electrical characteristics of these inputs are:

Sine-wave signals	Supply voltage	±5V.±5%
	Maximum counting frequency	25KHz.
Square-wave signals	Supply voltage	±5V.±5%
	Maximum counting frequency	200KHz.

It is recommended to use shielded cables for their connection connecting the shield to the corresponding pin of the connector.

2.3 SET-UP

2.3.1 GENERAL CONSIDERATIONS

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

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 external power supply used for the digital inputs and outputs 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 short-circuits 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 feedback dip-switches for each axis are set according to the type of feedback signal being used.

Verify that the E-STOP button is pressed.

2.3.3 CONNECTION

Verify that the AC power is correct.

Being the CNC disconnected, 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 tacho is operating properly.

Connect the AC power to the CNC. The CRT will show the model number and the available software (for example: CNC8025-MS).

After a self-test, the CNC will show the message: "GENERAL TEST PASSED". If there is any problem, the CNC will display the corresponding error message.

2.3.4 SYSTEM INPUT/OUTPUT TEST

This CNC offers a work mode which allows the possibility to activate or deactivate each one of the logic inputs and outputs of the CNC.

To do this, press the following keystroke sequence:

[OP MODE]
[9] (SPECIAL MODES)
[0] (TEST)

After the self-test, the CNC will show at the bottom of the screen a series of options which may be selected by means of the corresponding softkey.

By pressing the **[IN/OUT]** softkey, it will show the status of the logic inputs and it will be possible to change the status of the logic outputs.

Logic inputs

INPUT	PIN	FUNCTION
A	17 (I/O 1)	START
B	16 (I/O 1)	STOP
C	15 (I/O 1)	FEEDHOLD
D	14 (I/O 1)	EMERGENCY STOP
E	13 (I/O 1)	W axis home switch
F	12 (I/O 1)	Z axis home switch
G	11 (I/O 1)	Y axis home switch
H	10 (I/O 1)	X axis home switch
I	19 (I/O 1)	DRO mode
J	18 (I/O 1)	Block skip (conditional stop)
K		To be used only by the technical service
L		To be used only by the technical service
M		To be used only by the technical service
N		To be used only by the technical service

The CNC will show at all times and dynamically the status of all these inputs. To check a specific one, just actuate on the external push-button or switch observing its behavior on the CRT.

The value of "1" on the screen indicates that the corresponding input is receiving 24V DC and a "0" indicates that it doesn't.

Logic outputs

OUTPUT	ROW 1 PIN/FUNCTION	ROW 2 PIN/FUNCTION
A	(2 I/O 1) T Strobe	(3 I/O 2) Output 1, decoded M
B	(3 I/O 1) S Strobe	(4 I/O 2) Output 2, decoded M
D	(5 I/O 1) Emergency	(6 I/O 2) Output 4, decoded M
E	(6 I/O 1) W Enable	(7 I/O 2) Output 5, decoded M
F	(7 I/O 1) Z Enable	(8 I/O 2) Output 6, decoded M
G	(8 I/O 1) Y Enable	(9 I/O 2) Output 7, decoded M
H	(9 I/O 1) X Enable	(10 I/O 2) Output 8, decoded M
I	(27 I/O 1) MST01	(11 I/O 2) Output 9, decoded M
J	(26 I/O 1) MST02	(12 I/O 2) Output 10, decoded M
K	(25 I/O 1) MST04	(13 I/O 2) Output 11, decoded M
L	(24 I/O 1) MST08	(25 I/O 2) Output 12, decoded M
M	(23 I/O 1) MST10	(24 I/O 2) Output 13, decoded M
N	(22 I/O 1) MST20	(23 I/O 2) Output 14, decoded M
O	(21 I/O 1) MST40	(22 I/O 2) Output 15, decoded M
P	(20 I/O 1) MST80	(21 I/O 2) CNC in JOG mode

To check one of these outputs, select it with the cursor which may be moved by means of the right and left arrow keys.

Once the desired output is selected, press "1" to activate it and "0" to deactivate it. The CRT will show the status change.

It is possible to have several outputs active at the same time providing 24V at their corresponding pins.

Once the INPUT/OUTPUT test is completed, disconnect the electrical cabinet and, then, connect the input/output connectors as well as the feedback systems of the axes to the CNC.

Then, connect the electrical cabinet and the CNC to AC power and activate the servo drives.

2.4 EMERGENCY INPUT/OUTPUT CONNECTION

The Emergency Input of the CNC is called EMERGENCY STOP (E-STOP) and corresponds to pin 14 of connector I/O1. This input must normally have 24V DC.

The CNC processes this signal directly, therefore, whenever these 24V disappear, it will issue EXTERNAL EMERGENCY ERROR (Error 64), it will deactivate the axes enables and cancel the analog voltages for all the axes and the spindle.
It does **NOT** imply the emergency output (pin5).

The electrical cabinet interface must take into account all the external elements that could cause this error.

For example, some of these elements may be:

* The E-Stop button has been pressed.

* An axis travel limit switch has been pressed.

* An axis servo drive is not ready.

On the other hand, whenever a CNC detects an internal emergency error, it will activate the EMERGENCY OUTPUT at pin 5 of connector I/O1.

This output will be normally high or low depending on the setting of machine parameter P605(8).

There are some of the internal causes that can activate this output:

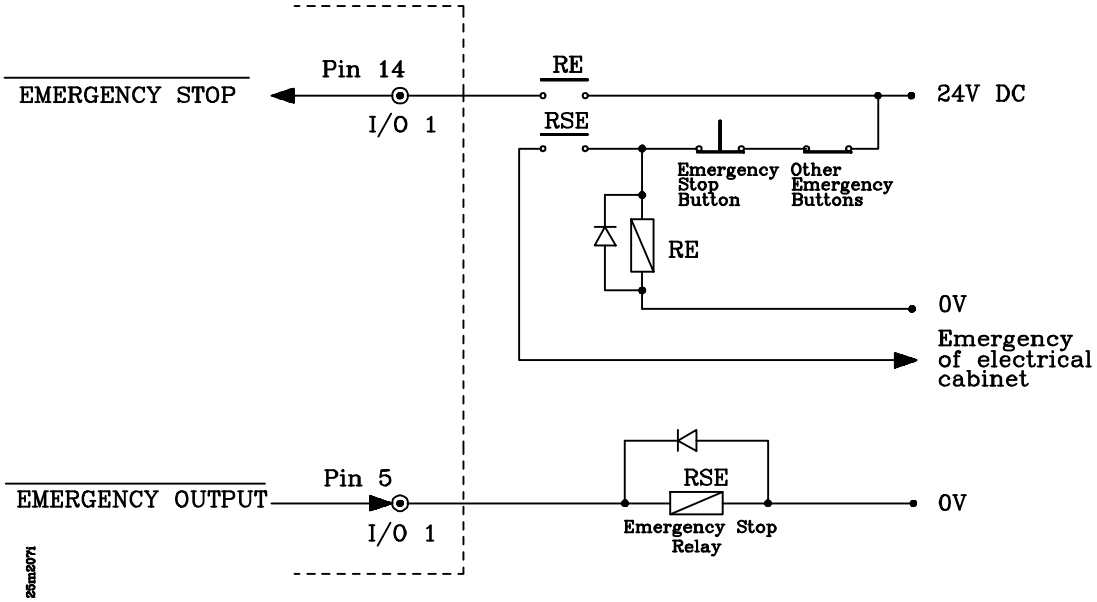
* An excessive axis following error has occurred.

* An axis feedback error has occurred.

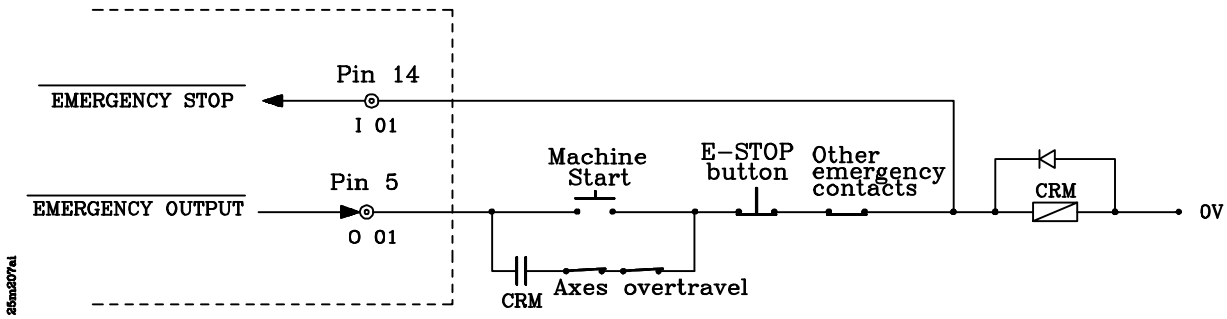
* There is erroneous data on the machine parameter table.

The recommended connection when P605(8)= 1 (output normally HIGH) is:

European Style:

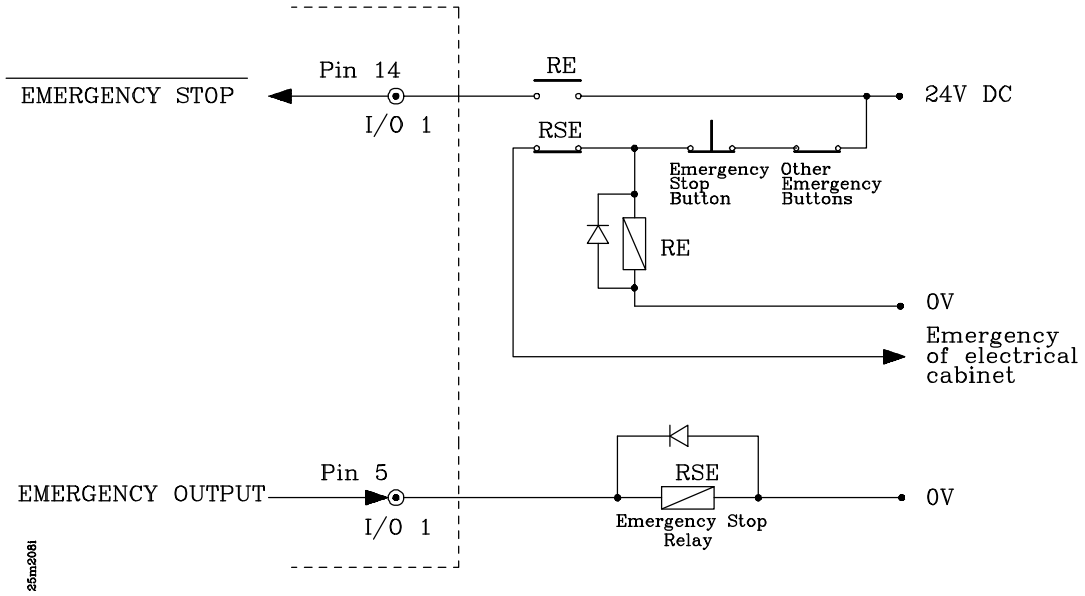


USA Style:

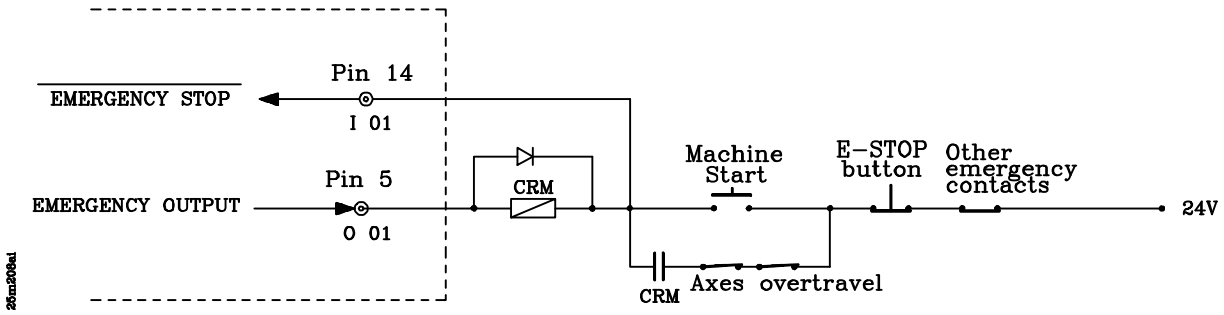


The recommended connection when P605(8)= 0 (output normally LOW) is:

European Style:



USA Style:



3. MACHINE PARAMETERS

Atention:

All unused machine parameters must be set to "0" to guarantee the proper functioning of this CNC.

It is recommended to save the machine parameters of the CNC at a peripheral device or computer in order to be able to recover them after their accidental loss.

Please note that some of the machine parameters mentioned here are described in greater detail in the chapter on "CONCEPTS" in this manual.

3.1 INTRODUCTION

On power-up, the CNC performs a system hardware test. When completed, it displays the model name and the message "GENERAL TEST PASSED" when successful and the corresponding error message if otherwise.

In order for the machine-tool to be able to properly execute the programmed instructions and recognize the interconnected elements, the CNC must "know" the specific data for the machine such as feedrates, acceleration ramps, feedback devices, etc.

This data is determined by the machine manufacturer and may be input via keyboard or via the RS232C serial line by setting the machine parameters.

To lock or unlock access to machine parameters, decoded "M" function table and to the leadscrew error compensation tables, proceed as follows:

* Press the **[OP MODE]** key.
* Press **[6]** to select the Editing mode.
* Press the softkey for **[LOCK/UNLOCK]**. The screen will show the word: "CODE:" (password).
* Key in "**PKJIY**" and press **[ENTER]** to lock the access or key in "**PKJIN**" and press **[ENTER]** to unlock the access.

When access to machine parameters is locked, **only** those regarding serial line communications via RS232C may be changed.

CAUTION when using a CNC with an integrated PLC (CNC+PLCI)

When using this access locking code, the machine parameters, the decoded "M" function table and the leadscrew error compensation tables are stored in EEPROM memory.

When using the access unlocking code, it recovers these previously stored tables from the EEPROM memory.

Therefore, **one must be careful and lock these tables <u>before</u> unlocking them.** Otherwise, the factory set values or other prelocked values, may be restored overwriting the ones the manufacturer entered but did not lock.

To access the machine parameter table via keyboard, press the following keystroke sequence:

[OP MODE]	Shows the various operating modes
[9]	Special modes
[1]	General parameters

3.2 OPERATING WITH PARAMETER TABLES

Once the machine parameter table has been selected, the operator may view the following or previous pages by means of the up and down arrow keys.

To view a particular parameter, key in the desired parameter number and press **[RECALL]**. The CNC will display the page corresponding to that parameter.

To EDIT a parameter, key in the desired number, press **[=]** and key in the value to be assigned to that parameter.

Depending on the type of machine parameter selected, it could be assigned one of the following types of values:

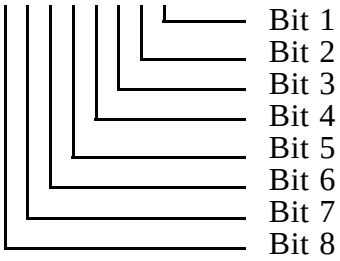
* A number	P111 = 30000
* A group of 8 bits	P602 = 00001111
* A character	P105 = Y

Once the value of the parameter has been keyed in, press **[ENTER]** so it is entered on the table.

If when pressing **[=]**, the parameter being edited disappears from the screen, it means that the machine parmeters are locked, therefore protected against modifications.

Every time a parameter bit is mentioned while describing the different machine parameters, refer to this nomenclature:

P602 = 0 0 0 0 1 1 1 1



3.3 GENERAL MACHINE PARAMETERS

P5 AC frequency

Possible values: 50 Hz. and 60 Hz.

P99 Language

Determines the language used by the CNC to show texts and messages on the screen.

0 = Spanish.
1 = German.
2 = English.
3 = French.
4 = Italian.

P13 Measuring units (mm/inches)

It determines the measuring units assumed by the CNC for machine parameters, tool tables and work units at power-up, after executing M02 or M30 and after RESET.

0 = Millimeters (G71).
1 = Inches (G70).

P6 Theoretical or Real display

It determines whether the CNC will display the real axis position or the theoretical position.

0 (REAL)= The CNC displays the real position values (coordinates).
1 (THEO)= The CNC displays the theoretical position values (ignoring the following error).

P802 Protected program

It indicates the number of the program to be protected against being read or edited.

It is given by an integer between 0 and 9999. If "0" is assigned, the CNC will interpret that no program is to be protected.

It is recommended to use this parameter to protect a program which contains the subroutines associated with functions M06, M22, M23, M24, M25 and G74, as well as those which should remain unseen by the operator.

The protected program will not be listed on the program directory and when requesting a subroutine defined in this program, the CNC will show the text: “P????”.

P619(1), P619(2) Monitor display color combination

These parameters are used by the CNC to select the color combination on a color monitor. The possible values are:

P619(2)	P619(1)	Display color
0	0	Monochrome
0	1	Combination 1
1	0	Combination 2

3.3.1 MACHINE PARAMETERS FOR AXIS CONFIGURATION

This CNC has 6 feedback inputs, A1 through A6, and the parameters indicated below can be used to set the CNC for the type of machine being installed.

The possible axis combinations offered by this CNC are:

A1	A2	A3	A4	A5	A6
X	Y	Z	W	V	S
X	Y	Z	W	V	Handwheel
X	Y	Z	W	S	Handwheel
X	Y	Z	W	Handwheel	S

The "S" letter indicates that the feedback input is used to control the spindle. However, it is possible to control the spindle without having to use that feedback device since the CNC provides the corresponding analog voltage output at pins 36 and 37 of connector I/O1.

P11 Number of axes controlled by the CNC

This CNC can interpolate up to 3 axes simultaneously; therefore, the W axis will be incompatible the X, Y or Z axis as selected by this parameter.

0 = The CNC does not control the W axis.
X = The CNC controls the W axis making it incompatible with the X axis
Y = The CNC controls the W axis making it incompatible with the Y axis
Z = The CNC controls the W axis making it incompatible with the Z axis

P600(4) Machine type (mill or boring mill)

Depending on the type of machine available, the CNC assumes the Z or Y axis as the vertical axis of the machine.

0 = Milling machine. Z as vertical axis.
1 = Boring mill. Y as vertical axis.

P616(4) The CNC controls the V axis

0 = The CNC does not control the V axis.
1 = The CNC controls the V axis.

P612(1) Connector A6. Electronic handwheel or spindle

It indicates whether the handwheel or the spindle encoder is connected to connector A6. When not having a fifth axis "V" (4-axis machine), connector A5 may be used to connect the other device.

0 = Spindle encoder connected to A5 and Electronic handwheel to A6.
1 = Spindle encoder connected to A6 and Electronic handwheel to A5.

P617(5), P605(6), P617(4), P611(4), P617(3) The X, Y, Z, W, V axis is a DRO axis.

It indicates whether the corresponding axis is treated as a normal axis (controlled by the CNC) or a DRO axis (moved externally).

0 = Normal axis.
1 = DRO axis.

P618(6), P618(5), P618(4), P618(3), P618(7) Display of the X, Y, Z, W, V axis

It indicates whether the corresponding axis is displayed on the CRT or not.

0 = It is displayed.
1 = It is **not** displayed.

P600(3), P616(3) W, V axis, normal or positioning-only

It determines the type of axis being used. Normal or positioning-only.

A positioning-only axis does not admit circular interpolation nor tool radius compensation.

0 = Normal axis.
1 = Positioning-only axis.

P600(1), P616(1) W, V axis, linear or rotary

It determines whether the axis is linear or rotary.

The position of a rotary axis is shown in degrees, thus not being affected by the mm/inch unit conversion. It does not admit tool radius compensation nor circular interpolation.

0 = Linear axis.
1 = Rotary axis.

Note: Parameter P604(1) for W and P616(7) for V indicating feedback pulse units must be set to "0" (mm) when rotary axis.

P600(2), P616(2) W, V rotary HIRTH axis

It determines whether it is a rotary axis with HIRTH toothing or not.

A HIRTH axis must be set as rotary (P600(1)= 1, P616(1)= 1) and it will only admit whole degree movements between 0° and 360°.

0 = It is **not** a HIRTH axis.
1 = It **is** a HIRTH axis.

The feedback resolution for a HIRTH axis must be in thousandths of a degree.

P606(1) W axis rotary ROLLOVER

This parameter is used when the W axis is rotary (P600(1) = 1) and its displayed position value is wanted to reset to 0° every time it reaches 360°.

0 = It is **not** ROLLOVER.
1 = It **is** ROLLOVER. **Position value rolls over from 359° to 0° and vice versa**.

P619(8), P620(6) W, V axis rotary ROLLOVER positioning via shortest path

These parameters will be used when the corresponding axis is rotary ROLLOVER (P600(1)=1, P606(1)=1 and P616(1)= 1 respectively) and their programmed moves are to be carried out in the shortest direction.

0 = The moves are **not** carried out in the quickest direction.
1 = The moves **are** carried out in the quickest direction.

P617(7) GANTRY axis

It determines whether the machine has a GANTRY axis or not.

0 = There is **no** GANTRY axis.
1 = There **is** a GANTRY axis.

With this CNC it is possible to have a pair of GANTRY axes:

* On 5-axis machines, it will consist of the V axis and its associated axis which will be indicated by machine parameter P11.

* On 4-axis machines, it will consist of the W axis and its associated axis which will be indicated by machine parameter P11.

When having a GANTRY axis, the CNC will not display the V or W axis and it will not be possible to program it.

Also, when programming a movement of the main axis, the one set by parameter P11, the CNC will apply the same move to both the main and the GANTRY axes.

P805 Maximum coupling (slaving) following error for GANTRY axes.

It sets the maximum position difference tolerated between two GANTRY axes as well as between two axes slaved by program (G77).

It is expressed in microns regardless of the type of work units being used.

Possible value: 0 thru 9999 microns

3.3.2 INPUT/OUTPUT PARAMETERS

P605(8) Normal status of the Emergency output (pin 5 connector I/O 1)

It determines whether the emergency output is normally low or high.

0 = Normally low (0V). An emergency situation will set this output high (24V)

1 = Normally high (24V). An emergency situation will set this output low (0V)

P609(7) Pin 17 of connector I/O 1 as RAPID TRAVERSE (fast feed)

It determines whether the signal input at pin 17 of connector I/O1 is treated as EXTERNAL CYCLE START or RAPID TRAVERSE.

0 = It is treated as EXTERNAL CYCLE START.
1 = It is treated as RAPID TRAVERSE.

If set as Rapid Traverse and while this input is active, the CNC will carry out all G01, G02 and G03 moves at 200% of the programmed feedrate F.

By the same token, in the JOG mode and while this input is kept active, the CNC will jog the axes in rapid G00.

P610(3) Pin 17 of connector I/O 1 as ENTER in PLAY-BACK mode

It determines whether or not the signal input at pin 17 of connector I/O1 is treated as the ENTER key while in the PLAY-BACK mode.

0 = It is **not** treated as the ENTER key.
1 = It **is** treated as the ENTER key.

P605(7) Pin 22 of connector I/O 2 as "tool magazine turning direction"

It indicates, on machines with automatic tool changer, whether or not pin 22 of connector I/O2 is used to indicate the turning direction of the tool magazine.

0 = It is **not** used as indicator of **tool magazine turning direction.**
1 = It **is** used as indicator of **tool magazine turning direction.**

If this parameter is set to "1", the output will go low (0V) to indicate the positive turning direction (count-up) and it will go high (24V) to indicate the negative turning direction (count-down).

It must be borne in mind that this pin is also used as output 15 of the decoded M functions; therefore, it should not be set on the decoded M function table when this parameter is set to indicate tool magazine turning direction (set to "1").

P609(3) Pin 23 of connector I/O 2 as RESET output

Indicates whether there is or not a RESET output via pin 23 of connector I/O2.

0 = It is **not** used as RESET output.
1 = It **is** used as RESET output.

It must be borne in mind that this pin is also used as output 14 of the decoded M functions; therefore, it should not be set on the decoded M function table when this parameter is set to output a RESET signal (set to "1").

P611(1) Pin 24 of connector I/O 2 as CYCLE ON
P611(6) Pin 24 of connector I/O 2 as AUTOMATIC
P613(4) Pin 24 of connector I/O 2 as G00

They indicate whether there is or not a CYCLE ON, AUTOMATIC or G00 indicating output via pin 24 of connector I/O2.

The CYCLE ON signal will be active whenever the CNC is executing a block.

The AUTOMATIC signal will be active as long as the AUTOMATIC mode of operation is selected.

The G00 signal will be active as long as the CNC is moving an axis in rapid (G00).

It must be borne in mind that this pin is also used as output 13 of the decoded M functions; therefore, it should not be set on the decoded M function table when this parameter is set for the output to indicate CYCLE ON, AUTOMATIC or G00. (set to "1").

0 = Used as output 13 of decoded M functions.
1 = Used as output indicating CYCLE ON, AUTOMATIC or G00, **and** output 13 of decoded M functions.

When setting two or three of these parameters to "1", the CNC will only output one of them "CYCLE ON" having the highest priority and "G00" the lowest.

CYCLE ON -> AUTOMATIC -> G00

P613(2) Pin 25 of connector I/O 2 as "Vertical axis movement" indicator output

It determines whether or not pin 25 of connector I/O2 is used to indicate the direction of the vertical axis movement. This output will be low (0V) for positive direction (count-up) or high (24V) for negative direction (count-down).

0 = It is **not** used as vertical axis moving direction indicator output.
1 = It **is** used as vertical axis moving direction indicator output.

It must be borne in mind that this pin is also used as output 12 of the decoded M functions; therefore, it should not be set on the decoded M function table when this parameter is set for the output to be used as vertical axis moving direction indicator (set to "1").

P617(8) M functions output in BCD or BINARY code

It determines whether the M function is output in BCD or Binary code via pins 20 thru 27 of connector I/O1.

0 = M function output in BCD code
1 = M function output in BINARY code

The significance or weight of each pin in both cases is as follows:

Pin	M in BCD WEIGHT	M in BINARY WEIGHT
27	1	1
26	2	2
25	4	4
24	8	8
23	10	16
22	20	32
21	40	64
20	80	128

For example: Depending on the type of code selected, the CNC will output the M41 as follows:

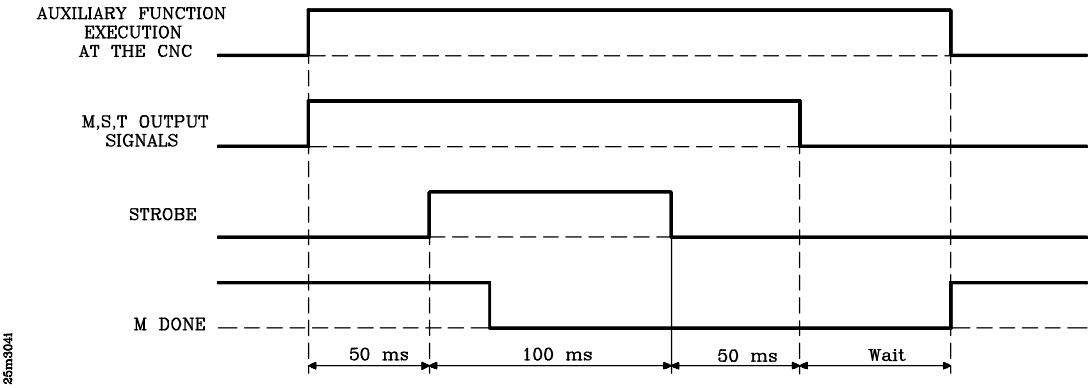
Pin	20	21	22	23	24	25	26	27
BCD	0	1	0	0	0	0	0	1
Binary	0	0	1	0	1	0	0	1

P605(5) The CNC waits for the down flank (trailing edge) at M-DONE input

It indicates whether it is necessary or not to wait for the down flank (24V-to-0V transition) of the M-DONE signal (at pin 15 of connector I/O 1) in response to an "S STROBE", "T STROBE" or "M STROBE" so the CNC resumes the execution of such functions.

“P605(5)=0”

The CNC will send out to the electrical cabinet the BCD signals corresponding to the M, S or T code for a period of 200 milliseconds. Then, if the "M-DONE" signal is low (0V), it will wait for it to be set high (24V) in order to consider the M, S or T function done (completed).



“P605(5)=1”

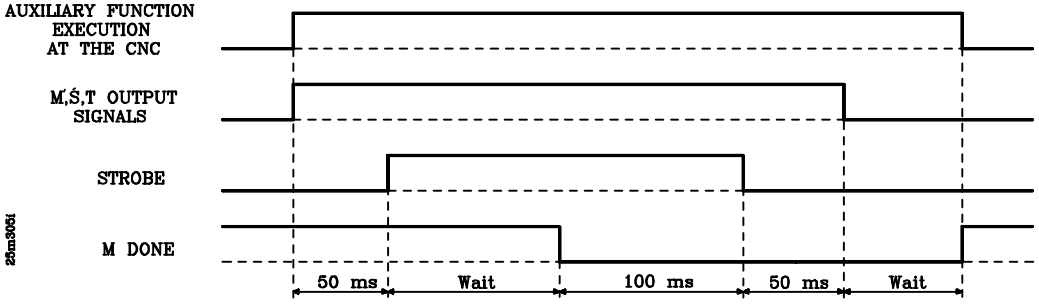
50 milliseconds after having sent the M, S or T BCD signals out to the electrical cabinet, it sends out the corresponding "Strobe" signal.

Then, if the "M-DONE" signal is high (24V), the CNC waits for it to be set low (0V).

Once the "M-done" signal is set low, the CNC continues maintaining the "Strobe" signal active for another 100 milliseconds.

After deactivating the Strobe signal, the M, S T BCD code signals are kept active for another 50 milliseconds.

After that time and if the "M-DONE signal is low, the CNC will wait until it becomes high so it can consider the auxiliary function M, S or T completed.



P609(5) Decoded M function code NOT output in BCD or BINARY.

When executing an M function which is decoded on the table, the CNC activates the decoded outputs of connector I/O2.

This parameter determines whether or not the CNC activates the M-BCD outputs of connector I/O1 (pins 20 thru 27) **besides** the decoded M outputs of connector I/O2.

0 = It **also** outputs the M function in BCD or BINARY code.
1 = It **does not** output the M function in BCD or BINARY code.

P602(8), P602(7), P602(6), P602(5), P603(1) Feedback alarm cancellation of the X, Y, Z, W and 5th axis respectively

The CNC will show the axis feedback alarm when not receiving all its corresponding feedback signals or when any of them is not within the permitted levels.

This parameter indicates whether this feedback alarm is to be cancelled or not.

0 = The feedback alarm for the corresponding axis is **not** cancelled.
1 = The feedback alarm for the corresponding axis **is** cancelled.

If the feedback system being used only utilizes 3 square-wave signals (A, B and Io), the corresponding parameter must be set to "1" (feedback alarm for that axis cancelled).

It must be borne in mind that the 5th axis might be the V axis or the Spindle.

3.3.3 HANDWHEEL PARAMETERS

P613(1) Electronic handwheel FAGOR 100P

It indicates whether the electronic handwheel being used is a FAGOR 100P model with axis selector button.

0 = It is **not** a FAGOR 100P model.
1 = It **is** a FAGOR 100P model.

P612(2) Counting direction of the electronic handwheel

It indicates the counting direction of the electronic handwheel. If correct, leave it as is and change it if otherwise.

Possible values: “1” and “0”.

P612(3) Feedback units of the electronic handwheel

It indicates whether the pulses received from the electronic handwheel are considered to be in millimeters or inches.

0 = Millimeters.
1 = Inches.

P612(4), P612(5) Feedback resolution of the electronic handwheel

They indicate the counting resolution of the electronic handwheel.

Possible values with square-wave signals:

1 = Resolution of 0.001 mm, 0.0001 inch or 0.001°.
2 = Resolution of 0.002 mm, 0.0002 inch or 0.002°.
5 = Resolution of 0.005 mm, 0.0005 inch or 0.005°.
10 = Resolution of 0.010 mm, 0.0010 inch or 0.010°.

The units being used depend on the setting of parameter P612(3) and on whether it is a linear or rotary axis.

To set the type of resolution, use the following chart:

P612(5)	P612(4)	Resolution
0	0	1
0	1	2
1	0	5
1	1	10

P612(6) Multiplying factor for Electronic handwheel signals

It indicates the "x2" or "x4" multiplying factor to be applied to the feedback signals provided by the electronic handhweel.

0 = "x4" factor being applied.
1 = "x2" factor being applied.

Example:

If the electronic handwheel has been set as follows:

P612(3) = 0	Millimeters
P612(4) = 0 y P612(5) = 0	Resolution 0.001 mm.
P612(6) = 0	Multiplying factor of "x4"

And the Feedrate Override Switch is positioned at "x100".

The selected axis will move 0.001mm x4 x100 = 0.4mm per pulse received.

P625(7) Electronic handwheel managed by the PLC

It indicates whether the CNC assumes the handwheel positions of the manual feedrate override switch or the PLCI outputs O45 and O46 or Marks M13, M14 of the PLC64 when jogging the axes with the handwheel.

0 = Assumes the Manual Feedrate Override Switch positions.

1= Assumes the setting of PLCI outputs O45 and O46 or Marks M13 and M14 of the PLC64.

O45 M13	**O46 M14**	
0	0	Assumes MFO switch settings
1	0	Equivalent to x1 of MFO switch
0	1	Equivalent to x10 of MFO switch
1	1	Equivalent to x100 of MFO switch

3.3.4 TOUCH PROBE PARAMETERS

P612(7) Pulse type of the touch probe

It indicates whether the probe functions of the CNC are active high (positive pulse) or low (negative pulse) with the probe signal received at connector A6.

0 = Negative pulse (0V.).
1 = Positive pulse (5V. or 24V.).

P720 M function associated with the probing movement (G75)

It indicates the M function that is executed when a probing move is carried out (G75).

It is defined by an integer between 0 and 99. If set to "0", no miscellaneous M function will be executed.

The CNC executes the selected M function before starting the execution of G75.

The selected M function may be used, for example, to activate an infrared-based probe.

P804 Probing feedrate in JOG mode

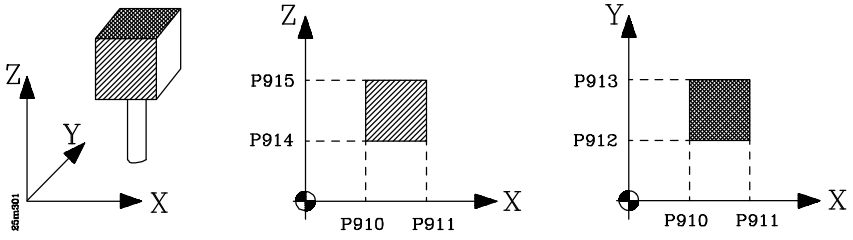
It indicates the probing feedrate used when calibrating and loading the tool length by means of a touch probe in JOG mode.

Possible values: 1 thru 65.535 mm./minute (degrees/minute).
1 thru 25.800 tenths-of-inch/minute.

P910 Minimum X coordinate of the touch probe
P911 Maximum X coordinate of the touch probe
P912 Minimum Y coordinate of the touch probe
P913 Maximum Y coordinate of the touch probe
P914 Minimum Z coordinate of the touch probe
P915 Maximum Z coordinate of the touch probe

They determine the position the table-top probe occupies for tool calibration.

These coordinates are absolute and referred to Machine Reference Zero.



Possible values: ± 8388.607 millimeters.
± 330.2599 inches.

P621(6) No error is issued when probing (G75)

It indicates whether the CNC issues "error 65" or not when the probe reaches the target position without sending the signal to the CNC during a probing move (G75).

The CNC interrupts the program whenever error 65 is issued.

0 = Error 65 is issued interrupting the program.
1 = Error 65 is not issued and it does not interrupt the program.

3.3.5 TOOL PARAMETERS

The section on "tools and tool magazine" in the chapter on "concepts" of this manual describes how these parameters may be used.

P701 Number of tool positions in the tool magazine

It indicates the number of tool positions in the tool magazine.

It is given by an integer between 0 and 98.

When the machine does not have an automatic tool changer, this parameter must be set to "98".

P743 Subroutine associated with the T function

It indicates the standard subroutine (not parametric) that will be executed when the block in execution contains a T function. In other words, every time a tool is selected in the part-program.

It is defined by an integer between 0 and 99. If set to "0", no subroutine will be executed.

This way, it will be possible to define the corresponding standard subroutine to select the desired tool.

P625(4) The associated subroutine is executed before the T function

It determines whether the subroutine associated with the T function is executed before or after the T function.

0 = It is executed **after** the T function.
1 = It is executed **before** the T function.

When setting this parameter to "1", the following considerations must be observed:

* The T function must be programmed alone in a block.
* When executing the T function in the Teach-in mode, the CNC will not execute the associated subroutine.

P626(1) The CNC displays the tool tip or tool base position

It indicates whether the CNC displays the tool base or tool tip position when working with tool length compensation (G43).

0 = It displays the tool **base** position.
1 = It displays the tool **tip** position.

Atention:

When **not** working with tool length compensation (G44), the CNC always displays the **tool base** position.

P601(5) Machining Center

It indicates whether it is a MACHINING CENTER or not.

0 = It is **not** a Machining Center.
1 = It **is** a Machining Center.

When it is a machining center, the CNC selects, in the tool magazine, the tool indicated by the T function and it will, then, be necessary to execute an "M06" to perform the tool change.

P601(1) RANDOM tool magazine

It indicates whether the tool magazine is or not RANDOM.

0 = The tool magazine is **not** RANDOM.
1 = The tool magazine **is** RANDOM.

If this parameter is set as RANDOM, the CNC will consider it to be a machining center **regardless** of the setting of parameter "P601(5)" (machining center).

P709 Subroutine associated with function M06

It indicates the standard subroutine (not parametric) that will be executed when executing an M06 function.

It is defined by an integer between 0 and 99. If set to "0", no subroutine will be executed.

This way, it will be possible to define the corresponding standard subroutine to carry out the desired tool change

P618(2) M06 executed before or after the subroutine

It determines whether the CNC outputs the M06 before or after executing its associated subroutine (parameter P709).

0 = M06 output before associated subroutine.
1 = M06 output after associated subroutine.

P601(8) Function M06 interrupts program execution

It indicates whether function M06 interrupts the program or not.

0 = It does **not** interrupt program execution.
1 = It interrupts program execution.

P702 First axis to move when executing function M06
P703 Second axis to move when executing function M06
P704 Third axis to move when executing function M06
P705 Fourth axis to move when executing function M06

These parameters indicate the order the axes will move when executing an M06 function.

It is set by an integer between 0 and 5.

0 = No axis moves.
1 = X axis.
2 = Y axis.
3 = Z axis.
4 = W axis.
5 = V axis.

If parameter "P702" is set to "0", no axis will move regardless of the setting of the other three parameters.

P900 Tool change position of the first axis when executing M06
P901 Tool change position of the second axis when executing M06
P902 Tool change position of the third axis when executing M06
P903 Tool change position of the fourth axis when executing M06

These parameters indicate the tool change position of the axes when executing an M06. The order of their movements are established by parameters “P702, P703, P704 and P705”.

These coordinates are absolute and referred to Machine Reference Zero of the corresponding axis.

Possible values: ± 8388.607 millimeters.
± 330.2599 inches.

P621(7) Function M06 implies M19 execution

Indicates whether the CNC executes or not function M19 when executing an M06.

0 = M06 implies M19 execution.
1 = M06 does not imply M19 execution.

Function M19 consists in two stages: Home search on the spindle and spindle orient to the position indicated by machine parameter "P916".

P615(8) In M06, the M19 is executed while moving the axis.

This parameter is used when a tool change involves the movement of the axes and spindle orientation (M19) and the spindle takes a long time to be homed (first stage of M19 execution).

By means of this parameter, it is possible to select that the movement of the first axis and the homing of the spindle are carried out simultaneously.

0 = The first axis does not move until function M19 is completed.
1 = The first axis move and the spindle homing are performed at the same time.

P603(2) Special sequence with M06

This parameter is used when the tool magazine is NOT RANDOM and the M06 requires special treatment (such as previous tool magazine positioning, etc.).

It indicates whether the CNC executes a normal or special sequence when executing an M06.

0 = Normal sequence with M06.
1 = Special sequence with M06.

The special M06 sequence is carried out as follows:

* The CNC activates the output at pin 13 of connector I/O2 when executing M06.

* Without waiting for the up-flank (leading edge or 0V-to-24V transition) at the M-DONE input (pin 15 of connector I/O1), the CNC outputs a T function indicating the tool pocket number where the tool which was at the spindle must be deposited.

* Once the execution of this T function is ended, the M-DONE input must be set high (24V).

The CNC will consider the SPECIAL M06 SEQUENCE completed when it detects this up-flank at the M-DONE input.

3.3.6 MACHINE PARAMETERS FOR THE RS232C SERIAL LINE

P0 Transmission speed (baudrate)

It determines the transmission baudrate used in communications between the CNC and the peripheral devices.

It is given by an integer (9600 maximum) and in baud units.

Typical values:

110, 150, 300, 600, 1200, 2400, 4800, 9600

P1 Data bits per transmitted character

It determines the number of data bits used in each transmitted character.

Possible values:

7 = Only the 7 least significant bits (out of 8) are used. Assign this value when transmitting standard ASCII characters.

8 = All 8 bits of the transmitted character are used. Assign this value when transmitting special characters (ASCII code over 127).

P2 Parity

It determines the type of parity check used in the transmission.

Possible values:

0 = None.
1 = ODD parity.
2 = EVEN parity.

P3 Stop bits

It determines the number of stop bits used at the end of the transmitted word.

Possible values:

1 = 1 stop bit.
2 = 2 stop bits.

P607(3) DNC

It indicates whether the CNC can work with the DNC protocol or not.

0 = DNC function **not** available.
1 = DNC function **available**.

P607(4) Type of communication, FAGOR Floppy Disk Unit or Cassette

P607(4) =1 Communication with a FAGOR Floppy Disk Unit. The CNC uses the settings of machine parameters P0, P1, P2 and P3.

P607(4)=0 Communication with a FAGOR Cassette reader reader/recorder. The CNC ignores the setting of parameters P0, P1, P2 and P3 and it uses the following internal setting for the FAGOR Cassette reader/recorder:

Baudrate = 13,714 Baud
Number of data bits = 7 bits
Parity = Even
Stop bits = 1

P607(5) DNC protocol active on power-up

It indicates whether the DNC protocol is active on CNC power-up or not.

0 = DNC **not** active on power-up.
1 = DNC **active** on power-up.

P607(6) The CNC does not abort DNC communication (program debugging)

The CNC offers a safety system that aborts DNC communications whenever:

* More than 30 seconds elapse without receiving a character while in the reception mode.

* More than 3 incorrect acknowledgments or non-acknowledgments occur in a row while in transmission mode.

This parameter can be used in order to be able to debug a user communications program without the CNC aborting the communication.

0 = The CNC aborts communications.
1 = The CNC does **not** abort communications (Debug mode).

P607(7) Status report by interruption

It indicates whether the "status report by interruption" is active or not while in DNC mode.

0 = It is **not** active.
1 = It is active.

A more detailed explanation on this function can be found in the "DNC COMMUNICATIONS PROTOCOL FOR THE 8025 CNC" manual.

3.3.7 JOG PARAMETERS

P606(3) M30 when switching to JOG mode

It indicates whether the CNC must generate an M30 automatically or not when switching to the JOG mode.

0 = M30 is **not** generated.
1 = M30 **is** generated.

P803 Axis feedrate when selecting the JOG mode

It defines the feedrate F assumed by the CNC when in JOG mode. This feedrate will be the same for all the axes.

Possible values: 1 thru 9.999 mm./minute (degrees/minute).
1 thru 3.936 tenths-of-inch/minute.

If this parameter is set to "0", the feedrate for each axis will be the maximum one established by machine parameters P110, P210, P310, P410 and P510.

P12 Continuous or pulsating axis jog

It indicates whether the axes are jogged **while** their corresponding jog keys are pressed (pulsating) or their movements are maintained until the CYCLE STOP key or another jog key is pressed (continuous).

Y = Pulsating mode. The axis is jogged **as long as** its corresponding jog key is maintained pressed.

N = Continuous mode. The axis starts moving when its corresponding jog key is pressed and it stops when the CYCLE STOP key or another jog key is pressed. In this latter case, the CNC will move the new selected axis in the chosen direction until the CYCLE STOP key or another jog key is pressed.

P609(6) Maximum incremental JOG move

It indicates the maximum distance the axes can be jogged when selecting one of the JOG positions of the Feedrate Override Switch on the operator panel (positions 1, 10, 100, 1000, 10000).

0 = Limited to 10 mm. or 1 inch.
1 = Limited to 1 mm. or 0.1 inch.

3.3.8 PARAMETERS RELATED TO THE EMERGENCY SUBROUTINE

P727 EMERGENCY subroutine

It indicates the number of the standard subroutine (not parametric) that will be executed when activating the EMERGENCY SUBROUTINE input (pin 16 of connector I/O1).

It is defined by an integer between 0 and 99. If set to "0", no emergency subroutine will be executed.

P621(3) Repetitive EMERGENCY subroutine

This parameter will be taken into account if the Emergency subroutine input (pin 16 of connector I/O1) is activated while the Emergency subroutine (P727) was already being executed.

It indicates whether the Emergency subroutine must be restarted every time the emergency subroutine input is activated or the status of the emergency subroutine input is ignored while the emergency subroutine is being executed.

0 = The status of the **emergency subroutine input is ignored** while the emergency subroutine is being executed.
1 = The **Emergency subroutine is restarted** every time the emergency subroutine input is activated.

P619(5) The Emergency subroutine executes M00

It indicates whether the CNC must execute an M00 after the Emergency subroutine or not.

Function M00 interrupts program execution and is not output.

0 = M00 **is** executed.
1 = M00 is **not** executed.

P619(4) Coordinate assignment to arithmetic parameter in Emergency subroutine

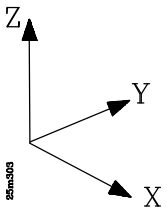
It indicates the coordinates to be assigned to an arithmetic parameter when executing a "P0=X" type block in the emergency subroutine.

0 = It assigns the coordinates of the beginning point of the block interrupted by the emergency.
1 = It assigns the coordinates of the point where the emergency input was activated.

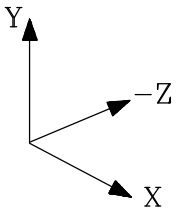
If at the beginning of the emergency subroutine we program the block: “P0=X P1=Y P2=Z”, and after performing all the emergency operations we program, inside the emergency subroutine, a block with movement to point "XP0 YP1 ZP2", the tool will return to the point of program interruption or to the beginning point of the interrupted block.

3.3.9 PARAMETERS RELATED TO OPERATING AND PROGRAMMING MODES

P609(8) Graphic representation of coordinate system

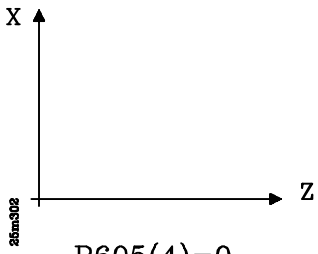


P609(8)=0
Mill model



P609(8)=1
Boring Mill model

P605(4) XZ plane representation



P605(4)=0

P605(4)=1

P611(3) Z axis represented as Z + W axes

It indicates whether the Z axis graphic representation corresponds only to Z axis movements or to the combined movements of the Z and the W axes.

0 = Normal representation. The Z axis graphic representation corresponds only to Z axis movements
1 = Special representation. The Z axis graphic representation corresponds to the combined movements of the Z and the W axes.

In order to use the special representation, the W axis must be set as linear "P601(1)=0" and incompatible with the Z axis "P11=Z".

P618(1) Disabling the CYCLE START key

It indicates whether the CYCLE START key of the operator panel is cancelled (ignored by the CNC) or not.

0 = The CYCLE START key is **not disabled.**
1 = The CYCLE START key **is disabled** (ignored by the CNC).

P625(6) Spindle inhibit via PLC.

To stop the spindle via PLC, it is possible to:
* Cancel (disable) the drive enable.
* Send the M05 code out to the CNC.
* Use the O44 signal of the PLCI or the M12 signal of the PLC64 to disable or re-enable the Spindle.

This machine parameter, P625(6), indicates whether or not O44 (at the PLCI) or M12 (at the PLC64) are used to enable or disable the spindle.

0 = They **are not** used.
1 = They **are** used.

When the CNC receives the spindle inhibiting signal, (O44 =1) or (M12=1), it outputs an analog voltage of 0V; but it does not change the current spindle conditions such as selected gear, rotating direction, etc.

When the spindle is re-enabled, (O44 =0) or (M12=0), the CNC outputs the corresponding spindle analog voltage again.

P606(2) Maximum value of the Manual Feedrate Override

It indicates the maximum feedrate override value to be selected by the Manual Feedrate Override Switch at the operator panel.

0 = Possible up to 120%.
1 = Limited to 100% even when selecting the 110% and 120% switch positions.

P4 The Manual Feedrate Override switch active in G00

It indicates whether it is possible or not to override the axis feedrate by this switch when moving in G00 (rapid positioning).

NO The feedrate override switch is ignored when in G00.

YES The feedrate override switch is active (not ignored) when in G00 applying a range from 0% to **100%** of the maximum feedrate set by machine parameters P111, P211, P311, P411 and P511 even at 110% and 120% positions.

P610(2) Vectored G00

It indicates whether the G00 moves (rapid positioning) are vectored (interpolated-all axes reaching the final position at the same time-) or not.

0 = G00 not vectored (not interpolated). Each axis moves at its fastest feedrate reaching the target point at different times.
1 = Vectored G00 (interpolated). All the axes involved in the move reach the target point at the same time. Their calculated feedrates are based on the maximum feedrate of the slowest axis.

P613(5) G05 or G07 active on power up

It indicates whether the CNC assumes function G05 (round corner) or G07 (square corner) on power-up, after M02, M30, EMERGENCY or RESET.

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

P715 Dwell between blocks in G07 (square corner)

It defines the dwell applied to motion blocks in G07.

It is given by an integer between 0 and 255.

Value 0 = No dwell.
Value 1 = 10 msec.
Value 10 = 100 msec.
Value 255 = 2550 msec.

P611(5) Feedrate units in G94

It determines the F programming units when function G94 is active.

0 = 1 mm./minute or 0.1 inch/minute.
1 = 0.1 mm./minute or 0.01 inches/minute.

If parameter "P611(5)=1", it is working in mm and F0.1 is programmed, the applied feedrate will be F0.01 mm/min.

It must be borne in mind that the machine parameters corresponding to the maximum programmable feedrate **F0** (P110/210/310/410/510), the maximum feedrate in **G00** (P111/211/311/411/511), the **home searching** feedrate (P112/212/312/412/512) and the **unidirectional approach** feedrate (P801) **are not affected by this parameter**. They are expressed in 1 mm/min or 0.1 inch/min units.

P607(8) G53 zero offset applied on RESET

It determines whether the CNC applies the G53 zero offset (selected on the zero offset table) when executing a RESET.

0 = G53 is **not** applied.
1 = G53 **is** applied.

P619(7) G59 as additive zero offset

It determines if function G59 is applied as regular zero offset or as an additive zero offset which will be added to the one currently selected. **It does not affect G53.**

0 = G59 acts as a **regular** zero offset.
1 = G59 acts as an **additive** zero offset.

If "P619(7)=1" and one of functions G54, G55, G56, G57 or G58 is executed, the CNC will apply a zero offset equal to the sum of their corresponding table values plus that of G59.

P607(2) The spindle turning reversal in G84 generates M05

It determines whether the CNC generates an M05 (stops the spindle) when reversing the spindle turning direction in the tapping canned cycle (G84).

0 = G84 **with** M05.
1 = G84 with**out** M05.

P610(1) FEED-HOLD in G84 and G47

It indicates whether the CNC stops the movement of the axes while the FEED-HOLD input is active during the tapping cycle (G84) and during G47 (single block treatment).

0 = The FEED-HOLD input does not stop the axes.
1 = The FEED-HOLD input stops the axes.

P613(8) Arithmetic parameters P150 thru P254 read-only

It indicates whether arithmetic parameters P150 thru P254 are read/write or read-only when the machine parameters are locked (code: PKJIY).

0 = They are always read/write.
1 = **When** the **machine parameters** are **locked**, these **arithmetic parameters** are **read-only; otherwise**, they are **read/write.**

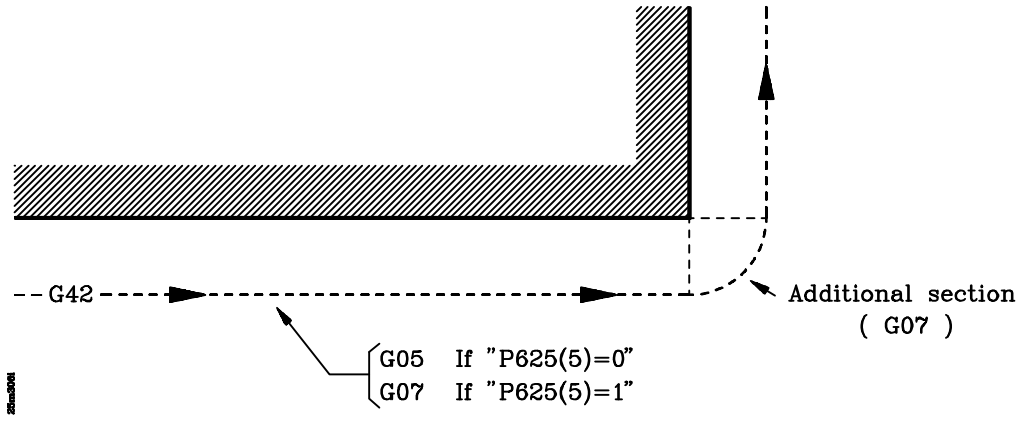
P618(8) Function P1=0X takes into account work units

It indicates whether or not the work units (mm or inches) are taken into account or ignored when executing a "P1=0X" type block.

0 = The **work units** are ignored. The axis position with respect to the machine reference zero is **always** taken **in millimeters.**
1 = The **work units** are not ignored. The axis position with respect to the machine reference zero is taken in the work units **currently active** (mm or inches).

P625(5) Type of compensation in sections programmed in G07

The CNC takes this parameter into account when tool radius compensation (G41 or G42) must be applied on a section programmed in G07 (square corner) which requires an additional circular section.



When the profile has been programmed in G05 (round corner), the whole compensated path will be done in G05.

P626(4) Using function G64 for multiple machining in an arc.

This function utilizes part of user memory for its execution. The CNC reserves this memory section on power-up.

This parameter indicates whether this G64 function will be used or not.

0 = Yes, function G64 will be used.
1 = No, function G64 will not be used.

When setting P626(4)=1, the whole user memory is available (free) and the CNC does not reserve any part of it.

4. MACHINE PARAMETERS FOR THE AXES

Observe that some of the parameters mentioned in this chapter are also described in more detail in the chapters on "power and machine interface" and "concepts" in this manual.

P100, P200, P300, P400, P500 Sign of the analog voltage for X, Y, Z, W, V axes

It determines the sign of the analog voltage for the axis servo drive. If correct, leave it as is and change it if otherwise.

Possible values: “Y” and “N”.

P101, P201, P301, P401, P501 Sign of the X, Y, Z, W, V axis feedback

It determines the counting direction of the axis. If correct, leave it as is and change it if otherwise.

Possible values: “Y” and “N”.

Observe that when changing this parameter, the one corresponding to the sign of the analog voltage must also be changed to prevent the axis from running away (P100, P200, P300, P400, P500).

P102, P202, P302, P402, P502 X, Y, Z, W, V axis jogging direction

It establishes the axis jogging direction by means of the JOG keys of the operator panel.

It determines the counting direction of the axis. If correct, leave it as is and change it if otherwise.

Possible values: “Y” and “N”.

4.1 PARAMETERS RELATED TO AXIS RESOLUTION

The section on "Axis resolution" in the chapter on "concepts" of this manual describes how these parameters may be used.

P103, P203, P303, P403, P503 X, Y, Z, W, V axis feedback resolution

They indicate the counting resolution for the axis.
Possible values for square-wave signals:

1 = Resolution of 0.001 mm, 0.0001 inch or 0.001°.
2 = Resolution of 0.002 mm, 0.0002 inch or 0.002°.
5 = Resolution of 0.005 mm, 0.0005 inch or 0.005°.
10 = Resolution of 0.010 mm, 0.0010 inch or 0.010°.

The units used depend on the setting of the following parameters:

P600(1)	The W axis is linear or rotary.
P616(1)	The V axis is linear or rotary.
P604(4,3,2,1) and P616(7)	In millimeters or inches.

P622(1), P622(2), P622(3), P622(4), P622(5) Counting resolution for X, Y, Z, W, V axis with sine-wave feedback

When using sine-wave feedback signals, the CNC considers these parameters as well as P103, P203, P303, P403 and P503 to set the axis resolution.

Possible values for P103, P203, P303, P403 and P503 with P622(1), P622(2), P622(3), P622(4), P622(5) = 0:

5 = Resolution of 0.001 mm, 0.0001 inch or 0.001°.
10 = Resolution of 0.002 mm, 0.0002 inch or 0.002°.

Possible values for P103, P203, P303, P403 and P503 with P622(1), P622(2), P622(3), P622(4), P622(5) = 1:

1 = Resolution of 0.001 mm, 0.0001 inch or 0.001°.
2 = Resolution of 0.002 mm, 0.0002 inch or 0.002°.
5 = Resolution of 0.005 mm, 0.0005 inch or 0.005°.
10 = Resolution of 0.010 mm, 0.0010 inch or 0.010°.

P604(4), P604(3), P604(2), P604(1), P616(7) X, Y, Z, W, V feedback units

They indicate the units of the feedback pulses for the corresponding axis.

0 = Millimeters or degrees
1 = Inches.

P106, P206, P306, P406, P506 X, Y, Z, W, V axis feedback signal type

They indicate the type of feedback signals being used.
Y = Sine-wave feedback signals
N = Square-wave feedback signals

The CNC **always** applies a x5 multiplying factor to the sine-wave feedback signals.

P604(8), P604(7), P604(6), P604(5), P616(8) Multiplying factor for X, Y, Z, W, V axis feedback signals

It indicates whether the CNC applies a x2 or x4 multiplying factor to the feedback signals of the axes.

0 = It applies a x4 factor.
1 = it applies a x2 factor.

When using FAGOR linear transducers (scales), set the corresponding parameters to "0".

Setting examples for the X axis:

Using square-wave linear transducers (scales):

Since the CNC applies either a x2 or x4 multiplying factor, a linear transducer must be selected which has a signal period of twice or four times the desired resolution.

Desired resolution		P103	P604(8)	Feedback Signal period	FAGOR Linear transducer
P604(4)=0	P604(4)=1				
0.001 mm	0.0001 inch	1	x2 P604(8)=1	0.002 mm	
			x4 P604(8)=0	0.004 mm	CX, CVX, MX
0.002 mm	0.0002 inch	2	x2 P604(8)=1	0.004 mm	CX, CVX, MX
			x4 P604(8)=0	0.008 mm	
0.005 mm	0.0005 inch	5	x2 P604(8)=1	0.010 mm	
			x4 P604(8)=0	0.020 mm	CT, CVT, MT, MVT, FT
0.010 mm	0.0010 inch	10	x2 P604(8)=1	0.020 mm	CT, CVT, MT, MVT, FT
			x4 P604(8)=0	0.040 mm	

Using sine-wave linear transducers and P622(1)=1:

Besides the x2 or x4 selected by P604(8), the CNC applies an additional x5 factor to the sinewave signals. Therefore, a transducer must be chosen which has a feedback signal period 10 or 20 times the desired resolution.

If parameter P622(1)=1, it is possible to obtain resolution of 1, 2, 5 and 10 microns or ten-thousandths of an inch.

Desired resolution		P103	P604(8)	Feedback signal period	FAGOR Linear transducer
P604(4)=0	P604(4)=1				
0.001 mm	0.0001 inch	1	x2 P604(8)=1	0.010 mm	
			x4 P604(8)=0	0.020 mm	CVS,MVS
0.002 mm	0.0002 inch	2	x2 P604(8)=1	0.020 mm	CVS,MVS
			x4 P604(8)=0	0.040 mm	
0.005 mm	0.0005 inch	5	x2 P604(8)=1	0.050 mm	
			x4 P604(8)=0	0.100 mm	FS
0.010 mm	0.0010 inch	10	x2 P604(8)=1	0.100 mm	FS
			x4 P604(8)=0	0.200 mm	

Using sine-wave linear transducers and P622(1)=0:

Besides the x2 or x4 selected by P604(8), the CNC applies an additional x5 factor to the sinewave signals. Therefore, a transducer must be chosen which has a feedback signal period 10 or 20 times the desired resolution.

If parameter P622(1)=0, it is possible to obtain resolution of 1 and 2 microns or ten-thousandths of an inch.

Desired resolution		**P103**	**P604(8)**	**Feedback signal period**	**FAGOR Linear transducer**
P604(4)=0	P604(4)=1				
0.001 mm	0.0001 inch	5	x2 P604(8)=1	0.010 mm	
			x4 P604(8)=0	0.020 mm	**CVS,MVS**
0.002 mm	0.0002 inch	10	x2 P604(8)=1	0.020 mm	**CVS,MVS**
			x4 P604(8)=0	0.040 mm	

P603(8), P603(7), P603(6), P603(5), P616(5) Binary encoder on X, Y, Z, W, V axes

It indicates whether the corresponding axis has a BINARY encoder (1024/2048 lines per turn) or not.

1 = It **is** a binary encoder.
0 = It is **not** a binary encoder.

P610(8), P610(7), P610(6), P610(5), P616(6) Equivalence of the binary encoder for X, Y, Z, W, V axes

This parameter is to be set when using a binary encoder (1024 or 2048 pulses) in place of one with 1000 or 1250 lines to obtain the desired resolution.

By setting this parameter, the CNC will adapt the encoder pulse-count as follows:

0 = It will treat the 1024-count binary encoder as a 1250-count and the 2048-count binary encoder as a 2500-count encoder.
1 = It will treat the 1024-count binary encoder as a 1000-count and the 2048-count binary encoder as a 2000-count encoder.

To calculate the axis resolution (P103, P203, P303, P403, P503) use the equivalent number of pulses selected here (1000, 1250, 2000, 2500).

The usefulness of the binary encoders is obvious since the same encoder can be utilized on two different types of leadscrews (for example 4-pitch and 5-pitch) without having to stock two different encoder models.

$$\text{Encoder} = \frac{\text{Leadscrew pitch}}{\text{Multiplying factor x Resolution}} = \frac{\text{1/4 inch/turn}}{\text{x4 x 0.0001 inch/pulse}} = \text{1250 pulses/turn}$$

$$\text{Encoder} = \frac{\text{Leadscrew pitch}}{\text{Multiplying factor x Resolution}} = \frac{\text{1/5 inch/turn}}{\text{x2 x 0.0001 inch/pulse}} = \text{1000 pulses/turn}$$

4.2 PARAMETERS RELATED TO THE ANALOG OUTPUTS

The section on "Adjustment of the axes" in the chapter on "concepts" of this manual describes how these parameters may be used.

P117, P217, P317, P417, P517 Minimum analog for X, Y, Z, W, V axes

They determine the minimum analog values for the axes.

It is given by an integer between 1 and 255.

Value 1 = 2.5 mV.
Value 10 = 25.0 mV. (10 x 2.5)
Value 255 = 637.5 mV. (255 x 2.5)

P104, P204, P304, P404, P504 Enable-Analog delay for X, Y, Z, W, V axes

They indicate whether there is 400 msec delay from the time the axis ENABLE signal is activated to when its analog voltage is output.

Y = There is delay.
N = There is no delay.

This parameter is used when the axis is not continuously controlled (held in position by the CNC) and some time it is required to deactivate certain devices such as a holding brake, etc.

P118, P218, P318, P418, P518 In-position zone (dead-band) for X, Y, Z, W, V

They define the in-position zone (to either side of the programmed coordinate value) where the CNC considers the axis to be in position.

It is always expressed in microns regardless of the work units being used.

Possible values: 0 thru 255 microns.

P105, P205, P305, P405, P505 Continuous control of the X, Y, Z, W, V axes

They determine whether or not the axis is held in position by the CNC by keeping its ENABLE signal ON when reaching its target position.

Y = It is continuously controlled (ENABLE on when in position).
N = It is not continuously controlled (ENABLE off when in position)

4.3 PARAMETERS RELATED TO TRAVEL LIMITS

The section on "Adjustment of the axes" in the chapter on "concepts" of this manual describes how these parameters may be used.

P107, P207, P307, P407, P507 Positive travel limit for X, Y, Z, W, V axes
P108, P208, P308, P408, P508 Negative travel limit for X, Y, Z, W, V axes

They establish the positive and negative travel limits for the axes. These distances are referred to machine reference zero (home).

Possible values: ± 8388.607 millimeters or degrees.
± 330.2599 inches.

If both limits are assigned the same value (for example: 0), the CNC will not allow to move the axis.

In JOG mode, and for safety reasons, it is possible to move the axis only up to 100 microns from the travel limits set by these parameters.

4.4 FEEDRATE RELATED PARAMETERS

The axis feedrate is programmed with the letter "F" and its value depends on the currently selected work units, millimeters or inches, and type of feedrate, G94 or G95.

Metric programming:

	Format	**Programming units**	**Minimum value**	**Maximum value**
G94	F 5.4	F1= 1mm/min	F0.0001 (0.0001 mm/min)	F65535.000 (63535 mm/min)
G95	F3.4	F1= 1mm/rev.	F0.0001 (0.0001 mm/rev.)	F500.000 (500 mm/rev.)

When operating in inches, we recommend setting machine parameter P615(6) to "1" so the programming units in G94 are in inches/minute.

P615(6) = 0 Programming format F1 = 0.1 inch/min.
Maintaining compatibility with older versions which did not accept decimal feedrate values.

P615(5) = 1 Programming format F1 = 1 inch/min.

	P615(6)	**Format**	**Programming units**	**Minimum value**	**Maximum value**
G94	P615(6)=0	F 5.4	F1= 0.1inch/min	F0.001 (0.0001 inch/min)	F25801.1810 (2580.1181 inch/min)
	P615(6)=1	F 5.4	F1= 1 inch/min	F0.0001 (0.0001 inch/min)	F25801.1810 (25801.1810 inch/min)
G95	------	F2.4	F1= 1 inch/rev.	F0.0001 (0.0001inch/rev.)	F19.6849 (19.6849 inch/rev.)

By the same token, when operating in inches and with rotary axes, we recommend setting machine parameter P615(7) to "1" so the programming units in G94 are in degrees/minute.

	P615(6)=1 Programming units: Inches/min		
	P615(7)	**Only rotary axis**	**Rotary axis interpolated with a linear axis**
G94	P615(7)=0	F1= 2.54°/min	F1= 1 inch/min
	P615(7)=1	F1= 1°/min	F1= 1 inch/min

These and other feedrate related parameters are described next.

P615(6) Feedrate in inches/minute

This parameter is used when working in inches (G70).

It indicates whether the axis feedrate units are inches/min. or 0.1 inch/min.

0 = Feedrate value in 0.1inch/min. (example: F10 = 1 inch/min.)
1 = Feedrate value in inches/min. (example: F10 = 10 inches/min.)

P615(7) Feedrate for rotary axes in degrees/minute

This parameter is to be used when working in inches (G70).

It indicates whether the feedrate units for rotary axes are degrees/minute or 2.54 degree/minute

0 = Feedrate units for rotary axes in 2.54 degrees/minute.
1 = Feedrate units for rotary axes in degrees/minute.

P110, P210, P310, P410, P510 Maximum programmable feedrate for X, Y, Z, W, V

They determine the maximum programmable feedrate F0 for the axes.

Possible values: 1 thru 65,535 mm./minute (degrees/minute).
1 thru 25,800 0.1inch/minute.

P111, P211, P311, P411, P511 G00 feedrate for X, Y, Z, W, V

They determine the rapid positioning feedrate (G00).

Possible values: 1 thru 65,535 mm./minute (degrees/minute).
1 thru 25,800 0.1inch/minute.

P729 Maximum feedrate F for circular interpolations

It determines the maximum feedrate for circular interpolations. This value is a function of the arc radius according to the formula:

$$F\ maximum = \frac{P729 \times Radius}{0.085}$$

It is given by an integer between 0 and 255. If set to "0", there will be no feedrate limitation.

Example:

We set parameter P729 = 17 so the feedrate on a 15mm-radius arc is limited to 3000 mm/min.

If we, then, program a 100mm-radius arc, the maximum feedrate for that arc will be:

$$F\ maximum = \frac{P729 \times Radius}{0.085} = \frac{17 \times 100}{0.085} = 20000\ mm/min$$

P708 Feedrate override when the analog voltage reaches 10V.

It indicates the Feedrate override (%) that the CNC applies when the analog voltage of an axis reaches 10V.

It is given by an integer between 0 and 128.

Value 0 = No % is applied.
Value 32 = 25 %
Value 64 = 50 %
Value 128 = 100 %

This parameter makes the CNC "wait" for the axis to catch up, reducing its analog voltage; thus preventing from issuing the corresponding following error alarms.

P714 Error if actual axis feed is not between 50% & 200% of programmed F

It indicates whether or not the CNC verifies that the actual axis feedrate is between 50% and 200% of the programmed F value.

This parameter is assigned a time period in which the axis feedrate is allowed to be out of this 50%-200% range.

It is expressed by an integer between 0 and 255.

Value 0 = The actual feedrate is not monitored in this sense.
Value 1 = 10 msec.
Value 10 = 100 msec.
Value 255 = 2550 msec.

4.5 PARAMETERS RELATED TO AXIS CONTROL

The section on "Gain adjustment " in the chapter on "concepts" of this manual describes how these parameters may be used.

P114, P214, P314, P414, P514 Proportional gain K1 for X, Y, Z, W, V axes

They set the analog output for 1 micron of following error.

It is given by an integer between 0 and 255 in such a way that a value of 64 corresponds to an analog voltage of 2.5mV.

$$\text{Analog (mV)} = K1 \times \text{Following error (microns)} \times \frac{2{,}5\text{mV.}}{64}$$

P115, P215, P315, P415, P515 Gain break-point for X, Y, Z, W, V axes

They define the following error value from where the proportional gain K2 takes over and K1 is no longer applied.

It is recommended to set these parameters to a value slightly greater than the following error corresponding to the maximum machining feedrate F0.

Value range: 1 thru 32766 microns
1 thru 12900 ten-thousandths of an inch (= 1.29 inches)

P116, P216, P316, P416, P516 Proportional gain K2 for X, Y, Z, W, V axes

They set the analog output for 1 micron of following error from the gain break-point on.

It is given by an integer between 0 and 255 in such a way that a value of 64 corresponds to an analog voltage of 2.5mV.

Analog = (K1 x Ep) + [K2 x (Following error -Ep)]

Where Ep is the value of the gain break-point.

It is recommended to set these parameters to a value between 50% and 70% of K1 in order to prevent jerky transitions between K1 and K2 or between machining feedrates and rapid positioning (G00).

Refer to the sections on "adjustment of the proportional gain" in the chapter about "MACHINE AND POWER INTERFACE".

P611(8) G00 and F00 always with proportional gain K2 from a set gain break-point of 256 microns

It determines whether the gain break-point being applied is the one set by the user or set at 256 microns.

0 = Gain break-point set by the user
1 = Gain break-point set at 256 microns.

P726 Recovery of programmed position of axes without continuous control.

It determines how the CNC behaves regarding the non-continuously controlled axes once they reach the programmed position.

It is assigned an integer value between 0 and 255.

Once the programmed position is reached, the axis is "free" since its enable signal disappears and it is no longer controlled by the CNC. However, depending on the value given to this parameter, it behaves as follows:

P726 = 0

If the axis drifts out of position a distance greater than 16 times the in-position value (P118, P218, P318, P418, P518), the CNC will issue the corresponding following error message.

P726 = other than zero.

If the axis drifts out of position a distance greater than “P726”/2 times the in-position value (P118, P218, P318, P418, P518), the CNC activates the corresponding enable signal in order to recover the drifted distance.

4.6 PARAMETERS RELATED TO MACHINE REFERENCE ZERO

The section on "Reference systems" in the chapter on "concepts" of this manual describes how these parameters may be used.

P627(1, 2, 3, 4, 5) Type of feedback marker pulse (reference mark "Io"). X, Y, Z, W, V axes.

They indicate the type of reference pulse (Io) provided by the feedback system.

0 = Regular Io
1 = Coded Io (semi-absolute)

When utilizing semi-absolute linear scales (with coded Io), there is no need for home limit switches since home may be "found" anywhere along the axis travel.

P628(1), P628(3), P628(5), P628(7), P629(1) Period of fixed Io signal (X,Y,Z,W,V)

These parameters must be set when using semi-absolute linear scales (with coded Io).

They indicate the period (pitch) of the fixed Io signal supplied by the feedback device.

0 = 20 mm.
1 = 100 mm

P628(2), P628(4), P628(6), P628(8), P629(2) Variable Io signal period increasing in the positive or negative direction (X,Y,Z,W,V)

These parameters must be set when using semi-absolute linear scales (with coded Io). They indicate whether the variable Io signal period increases in the positive or negative counting direction.

0 = Variable Io pitch increasing in the positive counting direction.
1 = Variable Io pitch increasing in the negative counting direction.

Fagor offers the following semi-absolute linear scales (with coded Io):

Scale	P627 (1)	P628(1)	P628(2)
COS	1	0	1
COC	1	0	0
COX	1	0	0
COVS	1	0	1
COVC	1	0	0
COVX	1	0	0

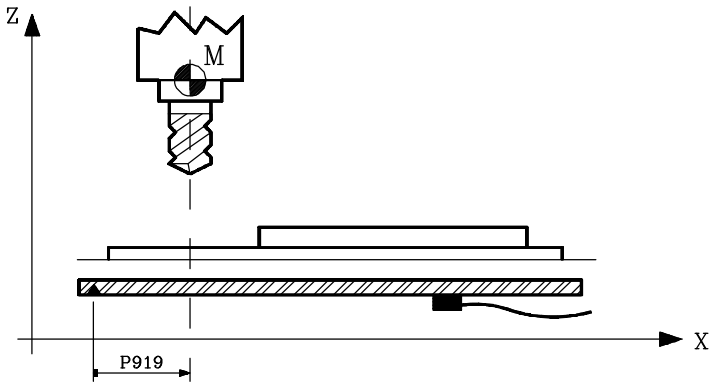
Scale	P627 (1)	P628(1)	P628(2)
MOVS	1	0	0
MOVC	1	0	0
MOVX	1	0	0
FOT	1	1	0
FOS	1	1	0
FOC	1	1	0

P919, P920, P921, P922, P923 Offset of the semi-absolute linear scale. (X,Y,Z,W,V)

These parameters must be set when using semi-absolute linear scales (with coded Io).

Semi-absolute linear scales provide their own Absolute Reference Zero mark being sufficient to move the axis a maximum distance of 20mm or 100mm (depending on the scale model) to find its absolute position with respect to this Absolute Reference Zero point.

In order for the CNC to show the coordinates referred to the Machine Reference Zero (home), machine parameters P919, P920, P921, P922 and P923 must be set with the Offset of the Machine home (M) with respect to the Absolute Reference Zero of the scale itself.



P602(4), P602(3), P602(2), P602(1), P617(2) Home switch for X,Y,Z,W,V

They indicate whether or not a home switch is being used for the axis home search.

0 = Yes, the axis has a home switch.
1 = No, the axis does not have a home switch.

P623(8), P623(7), P623(6), P623(5), P623(4) Home searching direction for X, Y, Z, W, V axes

They determine the direction of the axis move while searching the Machine Reference Point.

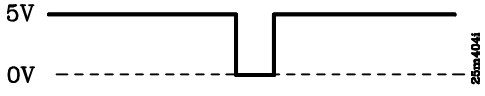
0 = Positive direction.
1 = Negative direction.

P600(8), P600(7), P600(6), P600(5), P617(1) Type of machine reference pulse for X, Y, Z, W, V axes

They define the type of reference pulse (marker, Io) of the feedback device being used.

1=Positive pulse.

0=Negative pulse.



P119, P219, P319, P419, P519 Coordinate of Machine Reference Zero. X,Y,Z,W,V

They determine the distance from this point to the Machine Reference Zero.

Possible values: ± 8388607 microns.
± 3302599 ten-thousandths of an inch.

The machine reference point is a point set by the machine manufacturer and used to synchronize the system. The CNC positions the axes at this point (physical location of the marker pulse) instead of moving all the way to the Machine Reference Zero point.

When the feedback system has coded Io, the home search may be carried out anywhere along the axis travel. Therefore, **these parameters must only be set when using leadscrew error compensation on their corresponding axes**. The amount of leadscrew error to be assigned to this Machine Reference point must be "0".

P112, P212, P312, P412, P512 1st home searching feedrate for X, Y, Z, W, V
P810, P811, P812, P813, P814 2nd home searching feedrate for X, Y, Z, W, V

They determine the feedrates used during home search.

The axis will move at the 1st feedrate until the home switch is pressed and, then, at the 2nd feedrate until the marker pulse is found on the feedback device.

Possible values: 1 thru 65,535 mm./minute (degrees/minute).
1 thru 25,800 0.1 inch/minute.

When setting the 2nd feedrate to "0", the axis will move at 100 mm/min. (about 4 inches/min).

P611(2) Home search required on power-up

It determines whether it is required or not to perform the home search on all the axes after powering the CNC up.

0 = It is not required.
1 = It is required.

Being this parameter set to "1", the CNC will issue the corresponding error message when attempting to execute a part-program in AUTOMATIC, SINGLE BLOCK or TEACH-IN mode.

P606(4) Function G74 generates an M30

It indicates whether the CNC generates an M30 automatically when executing function G74 (home search).

0 = M30 is not generated.
1 = M30 is generated.

P725 Home searching subroutine (associated with function G74)

It indicates the number of the standard subroutine (not parametric) that the CNC will execute when executing a block containing G74 (home search).

It is set by an integer between 0 and 99. If set to "0", no home search subroutine will be executed.

This way, it will be possible to define a standard subroutine to carry out the home search with the desired movements.

4.7 PARAMETERS FOR ACCELERATION/DECELERATION OF THE AXES

The section on "Gain adjustment " in the chapter on "concepts" of this manual describes how these parameters may be used.

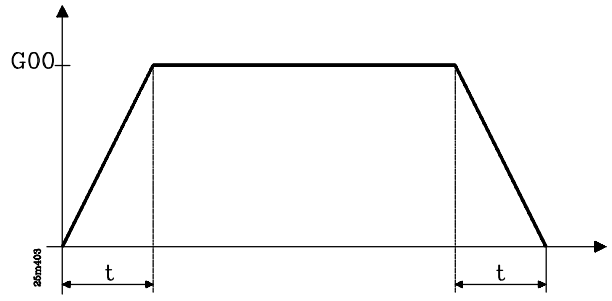
4.7.1 LINEAR ACCELERATION/DECELERATION

This type of acc./dec. is applied mainly on G00 and F00 moves although it is also possible to use it in G01 moves.

P721, P722, P723, P724, P728 ACC./DEC. Control of the X,Y, Z, W, V axes.

In order to avoid abrupt start-ups and brakes of the machine, it is possible to define some acceleration and deceleration ramps.

These parameters define the time that each axis needs to reach the positioning feedrate (machine parameters P111, P211, P311, P411, P511) while accelerating. This acceleration time will be the same as the deceleration time.



It is given by an integer between 0 and 255.

Value of 0 = There is no Acceleration/deceleration control.
Value of 1 = 0.020 seconds.
Value of 10 = 0.200 seconds. (10 x 0.02)
Value of 255 = 5.100 seconds. (255 x 0.02)

During a linear interpolation or a rapid move, the CNC applies the longest of the acc./dec. times assigned to the axes involved in the move.

Note: No acc./dec. will be applied on circular interpolations.

P613(7) ACC./DEC. in all linear interpolations (G01).

It indicates whether the acc./dec. ramps (P721, P722, P723, P724, P728) are to be applied on all linear interpolations or only when they are carried out at the maximum programmable feedrate F0 set by machine parameters P110, P210, P310, P410, P510.

0 = Acc./Dec. applied only in linear interpolations at F0.
1 = Acc./Dec. applied in **all** linear interpolation (at any feedrate).

P620(2) Acceleration/deceleration in G05 (corner rounding)

It indicates whether or not the acc./dec. ramps are applied in blocks with G05 active (corner rounding).

0 = Yes. Acc./dec. is being applied.
1 = No. Acc./dec. is not being applied.

4.7.2 BELL-SHAPED ACCELERATION/DECELERATION

This type of acceleration may be applied on all kinds of movements, G00, G01, G02, etc and for any type of feedrate F.

P624(8) Bell-shaped acceleration/deceleration

It is to be used on high-speed machines.

The acc./dec. ramps are applied onto all kinds of movement: rapid, linear interpolation, **arcs**, etc.

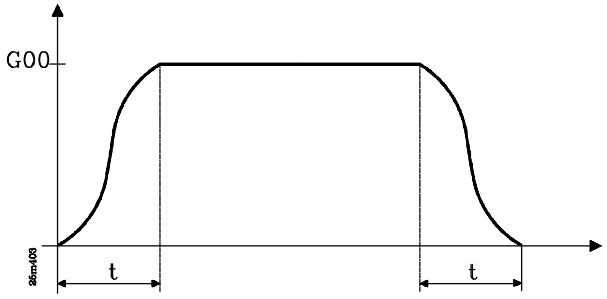
0 = No. This type of acc./dec. is not applied.
1 = Yes. This type of acc./dec. is applied.

Note that the acc./dec. ramps set by this parameter and P744 is common to all the axes.

P744 Duration of the Bell-shaped Acc./Dec. ramp

This parameter will be used when machine parameter "P624(8)=1".

It defines the time needed by the axis to reach the selected feedrate (when accelerating). This time period is the same for the deceleration stage and common to all the axes of the machine.



It is given by an integer between 0 and 255.

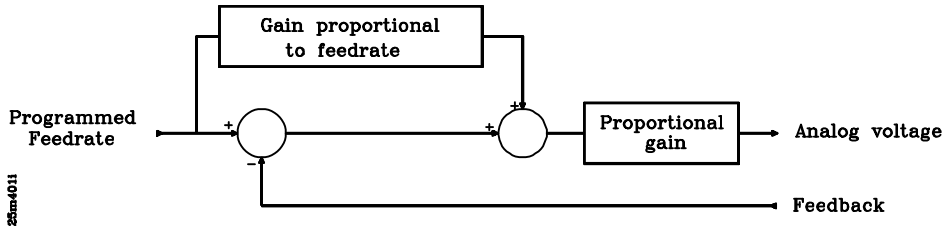
Value of 0 = There is no Acc./Dec. common to all the axes
Value of 1 = 0.010 seconds.
Value of 10 = 0.100 seconds. (10 x 0.01)
Value of 255 = 2.550 seconds. (255 x 0.01)

4.7.3 FEED-FORWARD GAIN

P732, P733, P734, P735, P736 FEED-FORWARD gain for X, Y, Z, W, V

With the Feed-forward gain, which is proportional to the feedrate, it is possible to improve the positioning loop minimizing the amount of following error. **However it is not recommended when acceleration/deceleration ramps are not being used.**

These parameters define the % of analog voltage due to the programmed feedrate. It is given by an integer between 0 and 255.



The value which will be added to the following error is (Kf x F/6) where F is the programmed feedrate and Kf is:

* The value of this parameter in the case of linear acc/dec.
  For example, for the X axis: "Kf = P732"

* An eighth of the value assigned to this parameter in the case of bell-shaped acceleration/deceleration.
  For example, for the X axis: "Kf = P732/8"

The CNC will apply the proportional gain (K1 and K2) to the value resulting from the addition of the following error plus the value selected by means of the feed-Forward gain.

When the value resulting from the addition is smaller than the value assigned to the gain break-point, the CNC will apply the following formula:

Analog = K1 x [Following error + (Kf x F/6)]

And when the value resulting from the addition is greater than the value of the gain break-point:

Analog = (K1 x Ep) + {K2 x [Following error + (Kf x F/6) - Ep]}

Where “Ep” is the gain break-point value assigned to the corresponding parameter.

4.8 PARAMETERS FOR UNIDIRECTIONAL APPROACH

The section on "Unidirectional approach" in the chapter on "concepts" of this manual describes how these parameters may be used.

P608(4), P608(3), P608(2), P608(1) Unidirectional approach for X, Y, Z, W axes

They indicate whether the unidirectional approach is to be used on G00 moves (rapid) or not.

0 = G00 moves **without** unidirectional approach.
1 = G00 moves **with** unidirectional approach.

P608(8), P608(7), P608(6), P608(5) Direction of the unidirectional approach for X, Y, Z, W axes

They indicate the direction of the unidirectional approach.

0 = Positive direction.
1 = Negative direction.

P716 Distance between the approach point and the programmed position

It indicates the factor used by the CNC to calculate the distance between the unidirectional approach point and the programmed position.

It is given by an integer between 0 and 255.

This parameter is common to all the axes and the CNC calculates this distance by multiplying this factor by the value assigned to the amount of leadscrew backlash indicated for each axis: “P109, P209, P309, P409”.

P801 Feedrate of the unidirectional approach.

It indicates the feedrate of the unidirectional approach from the approach point to the programmed position.

Possible values: 1 thru 9999 mm/minuto.
1 thru 3936 0.1inch/minute.

4.9 LEADSCREW RELATED PARAMETERS

With this CNC it is possible to compensate for leadscrew error as well as for its backlash when reversing movement direction.

4.9.1 LEADSCREW BACKLASH

P109, P209, P309, P409, P509 Backlash compensation for X, Y, Z, W, V axes

They indicate the amount of leadscrew backlash. When using linear transducers (scales), the corresponding parameter must be set to "0".

It is given in microns regardless of the work units being used.

Possible values: 0 thru 255 microns.

P624(1), P624(2), P624(3), P624(4), P624(5) Sign of backlash for X,Y,Z,W,V

They define the sign of the backlash compensation value indicated by parameters P109, P209, P309, P409, P509.

0 = Positive.
1 = Negative.

P113, P213, P313, P413, P513 Additional analog pulse for X, Y, Z, W, V axes

Additional analog pulse to make up for the leadscrew backlash when reversing movement direction.

It is given by an integer between 0 and 255.

Value 0 = There is no additional analog pulse.
Value 1 = 2.5 mV.
Value 10 = 25.0 mV. (10 x 2.5)
Value 255 = 637.5 mV. (255 x 2.5)

Every time the axis moving direction is reversed, the CNC will apply the corresponding analog voltage for that axis plus the additional analog pulse indicated in this parameter during 40 milliseconds.

When the feedback device is a rotary encoder, this parameter must be set to "0".

4.9.2 LEADSCREW ERROR

There are 4 leadscrew compensation tables with 30 points each or 2 tables of 60 points each.

For each point, it is required to define the position of the axis and the amount of error at that point.

To access these tables, press the following keystroke sequence:

[OP MODE]	Display of the various operating modes.
[9]	Access to special modes.
[3]	Access leadscrew error compensation tables.

The operator may view the following or previous pages by using the up and down arrow keys.

To view a particular parameter, key in its number and press **[RECALL]**. The CNC will show the page corresponding to that parameter.

To clear the table by setting all the parameters to 0, key in the following sequence: **[K] [J] [I] [ENTER]**.

Each parameter pair of this table represents:

Even parameter The position of the error point on the leadscrew. This position is referred to Machine Reference Zero (home).

Value range: ±8388.607 millimeters
±330.2599 inches

Odd parameter The amount of leadscrew error at that point.

Value range: ±32.766 millimeters
±1.2900 inches

When defining the compensation points on the table, the following rules must be observed:

* The even parameters are ordered according to their position along the axis. The first pair of parameters (P0 or P60) must be set for the most negative (least positive) point of the axis to be compensated.

* If all 30 points of the table are not required, set the unused ones to 0.

* For those sections outside the compensation area, the CNC will apply the compensation defined for the nearest point.

* The Machine Reference Zero point (home) must be set with an error of 0.

* The maximum difference between the error values of two consecutive compensation points must be within: ±0.127 mm (±0.0050 inches)

* The inclination of the error graph between two consecutive points cannot be greater than 3%.

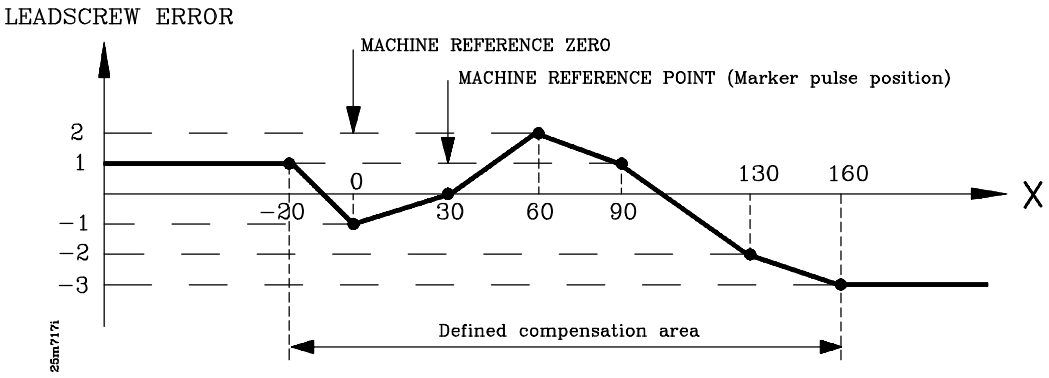
Examples: If the distance between two consecutive points is 3 mm. the maximum difference of their relevant error values can be 0.090 mm.
If the error difference between two consecutive points is the maximum (0.127mm), the distance between them cannot be smaller than 4.233mm.

To EDIT a parameter, key in its number, press **[=]**, key in the desired value and press **[ENTER]** so the new value is entered on the table.

Remember to press **[RESET]** or power the CNC off and back on once the machine parameters have been set in order for the CNC to assume their new values.

Programming example:

An X axis leadscrew is to be compensated according to the following graph in the section between X-20 and X160:



Considering that the machine reference **point** has a value of X30 (meaning that it is located 30mm from the Machine Reference **Zero**), the leadscrew error compensation parameters will be defined as follows:

P000 = X -20.000	P001 = X 0.001
P002 = X 0.000	P003 = X -0.001
P004 = X 30.000	P005 = X 0.000
P006 = X 60.000	P007 = X 0.002
P008 = X 90.000	P009 = X 0.001
P010 = X 130.000	P011 = X -0.002
P012 = X 160.000	P013 = X -0.003
P014 = X 0.000	P015 = X 0.000
P016 = X 0.000	P017 = X 0.000
" "	" "
" "	" "
P056 = X 0.000	P057 = X 0.000
P058 = X 0.000	P059 = X 0.000

P606(8), P606(7), P606(6), P606(5) Leadscrew error compensation for X,Y,Z,W

They indicate whether the CNC must apply leadscrew error compensation on the corresponding axis or not.

0 = Leadscrew error compensation **not** applied.
1 = Leadscrew error compensation **applied**.

P613(6) Number of leadscrew error compensation tables (2 or 4)

It indicates whether four 30-point tables are used or two 60-point tables.

0 = 4 compensation tables (for X, Y, Z and W)
1 = 2 compensation tables (for X and Y)

4.10 PARAMETERS RELATED TO CROSS COMPENSATION

With this CNC it is possible to compensate for a positioning error suffered by one axis when another axis moves. A typical example of cross compensation is the beam sag error compensation.

To use this cross compensation, it is necessary to define the axis causing the error and the one suffering it depending on the position of the "guilty" axis.

In this case, only the leadscrew compensation tables of the X, Y and Z axes may be used since the W axis table will now be used for cross compensation.

The axes involved in the cross compensation must be selected between X, Y and Z. The table has 30 points (P180 through P239) indicating the error position and the amount of error at each point. The way to set this table is described in the section on "Leadscrew related parameters" in this chapter.

P623(1) Cross compensation applied to X axis
P620(5) Cross compensation applied to Y axis
P620(4) Cross compensation applied to Z axis

They indicate the axis on which the cross compensation will be applied.

0 = Cross compensation is **not** applied.
1 = Cross compensation **is** applied.

COMPENSATED AXIS			
	P623(1)	**P620(5)**	**P620(4)**
X	**1**	**0**	**0**
Y	**0**	**1**	**0**
Z	**0**	**0**	**1**

When not using cross compensation, set parameters:
P623(1)=0, P620(5)=0 and P620(4)=0

P623(2), P623(3) Axis causing the cross error when moving

With cross compensation it is possible to compensate for measuring errors caused by one axis onto another.

These parameters determine which moving axis is causing the error (guilty) onto the one indicated by machine parameters "P623(1), P620(5), P620(4)."

GUILTY AXIS		
	P623(3)	**P623(2)**
X	**0**	**1**
Y	**1**	**0**
Z	**1**	**1**

4.10.1 DOUBLE CROSS COMPENSATION

When having a CNC with an integrated PLC or connected to a FAGOR PLC64, it is possible to use the "double cross compensation".

This feature is to be used when having a cross compensation between two axes obtaining two different tables applicable according to two different external conditions. For example:

* Two different temperatures. One table may be used when the machine is cold and the other one when it is hot.

* Machine with two spindles of different weights. Thus, the beam sag error will be different when using one or the other.

With this CNC, it is possible to use either one cross compensation table or two; but not both at the same time.

P625(3) Double Cross Compensation

It indicates whether one or two cross compensation tables are to be used.

0 = The CNC applies one cross compensation table.
1 = The CNC applies two cross compensation tables.

In either case, machine parameters: P623(1), P620(5), P620(4), P623(2) and P623(3) must be set accordingly.

When using double cross compensation, "P625(3)=1", the cross compensation table is divided into two 15-point tables.

These tables will be selected from the PLCI or from the PLC64 by setting bit 4 of register R155 to either "0" or "1" and by activating mark M1955.

When setting B4R155=0, the first table will be selected (parameters P180 through P209) and when setting B4R155=1, the second table will be selected (P210 through P239).

4.11 PALLET RELATED PARAMETERS

The section on "Pallet work" in the chapter on "concepts" of this manual describes how these parameters may be used.

P603(3) Machine with PALLETS

It indicates whether the machine uses PALLETS or not.

0 = PALLETS are **not** used.
1 = PALLETS **are** used.

P605(3) The CNC generates an M21 when executing M22, M23, M24 or M25

This parameter is used when the machine utilizes PALLETS [P603(3) =1].

It indicates whether the CNC must send out an M21 every time functions M22, M23, M24 or M25 are executed.

0 = Function M21 is **not** sent out.
1 = Function M21 **is** sent out.

Function M21 is generated before the axes start moving.

P605(2) The Z axis moves when executing M22, M23, M24 or M25

This parameter is used when the machine utilizes PALLETS [P603(3) =1].

It indicates whether the Z axis must move or not when executing functions M22, M23, M24 or M25.

0 = It **does not** move.
1 = It moves.

P611(7) The X axis moves when executing M22, M23, M24 or M25
P605(1) The W axis moves when executing M22, M23, M24 or M25

This parameter is used when the machine utilizes PALLETS [P603(3) =1].

It indicates whether the axis must move or not when executing functions M22, M23, M24 or M25.

0 = It moves.
1 = It **does not** move..

P607(1) Travel limits ignored when executing M06, M22, M23, M24 or M25

It indicates whether or not the travel limits of the axes are ignored when executing functions M06, M22, M23, M24 or M25.

0 = The travel limits are **not ignored** (they are active).
1 = The travel limits are **ignored.**

P710 Subroutine associated with function M22
P711 Subroutine associated with function M23
P712 Subroutine associated with function M24
P713 Subroutine associated with function M25

It indicates the number of the standard subroutine (not parametric) that will be executed when executing a block containing the corresponding pallet function (M22, M23, M24 or M25).

It is given by an integer between 0 and 99. If set to "0", no subroutine will be executed.

This permits to define a standard subroutine to move the axes in a specific way.

P904 W axis position to move when executing M22, M23, M24 or M25
P905 X axis position to move when executing M22 or M23
P906 X axis position to move when executing M24 or M25
P907 Z axis position to move when executing M22, M23, M24 or M25

These parameters are used when the machine utilizes PALLETS [P603(3) =1].

They indicate the position where the corresponding axis has to move when executing functions M22, M23, M24 or M25.

These position values must be absolute and referred to machine reference zero (home).

Possible values: ± 8388.607 millimeters.
± 330.2599 inches.

4.12 SPECIAL MACHINE PARAMETERS

P609(1) Machine travels over 8388.607 mm (330.2599 inches)

This parameter must only be set for those machine having one or more axes with a travel greater than **8388.607 mm (330.2599 inches).**

This parameter **affects both axes** even when one of them might not require this extended travel.

Possible values:

0 = Machine with **normal** axis travel **within** 8388.607 mm (330.2599 inches).
1 = Machine with **extended** axis travel **over** 8388.607 mm (330.2599 inches)

When setting this parameter to "1", the following items must be considered:

* The minimum display resolution for both axes will now be: 0.01mm or 0.001 inch.

* The programming format will now be: ±5.2 in mm and ±4.3 in inches.

* The minimum moving distance will now be: ±0.01mm and ±0.001 inch. The maximum moving distance will be: ±83886.07mm and ±3302.599 inches.

* The tool table format will be affected the same way:

  R,L ±4.2 in mm or ±3.3 in inches. Minimum value: ±0.01mm and ±0.001 inch. Maximum value: ±9999.99mm and ±393.699 inches.

  I,K ±3.2 in mm and ±2.3 in inches. Minimum value: ±0.01mm and ±0.001 inch. Maximum value: ±327.66 mm and ±12.900 inches.

* The integer values assigned to machine parameters P103, P203, P303, P403 and P503 for axis resolution now acquire new units:

  1 = 0.01 mm or 0.001 inch or 0.001° resolution.
  2 = 0.02 mm or 0.002 inch or 0.002° resolution.
  5 = 0.05 mm or 0.005 inch or 0.005° resolution.
  10 = 0.10 mm or 0.010 inch or 0.010° resolution.

* To calculate K1 and K2 and the Feed-forward gain, the following error is now expressed in 0.01 mm units (not microns) and 0.001 inch units (instead of 0.0001 inch as before).

  The maximum amount of following error permissible is now: 320mm

  That is to say that K1 and K2 gains (parameters P114, P214, P314, P414, P514, P116, P216, P316, P416, P516) must be given in "mV/0.01mm (mV/ 0.001 inches).

* Machine parameters P115, P215, P315, P415 and P515 for gain break-point are now expressed in 0.01 units (not microns) and 0.001 inch units (instead of 0.0001 inch as before).

* Machine parameters P109, P209, P309, P409, P509 (leadscrew backlash) and P118, P218, P318, P418, P518 (in-position zone) will also be given in 0.01 mm and 0.001 inch units.

  P118 = 100 means an in-position zone of 1mm (or 0.1 inch) for the X axis.

* Machine parameters P112, P212, P312, P412, P512, P810, P811, P812, P813, P814 (homing feedrate) and P801 (unidirectional approach) will also be expressed in 0.01mm/min. or 0.001 inch/min.

  P112 = 10000 assigns a feedrate of 100m/min.

Examples to calculate resolution with P609(1)=1:

Example 1: **Resolution in "mm" with square-wave encoder**

We want to obtain a 0.01mm resolution with a square-wave encoder mounted on the X axis whose leadscrew has a 5mm/turn pitch.

Since the multiplying factor applied by the CNC may be either x2 or x4 (depending on machine parameter setting). The resulting encoder line count will be:

$$\text{Number of pulses} = \frac{\text{Leadscrew pitch}}{\text{Multiplying Factor x Resolution}}$$

For a factor of x4:

$$\text{Number of pulses} = \frac{5\text{ mm}}{4 \times 0.01\text{m}} = 125 \text{ pulses/rev.}$$

P103= 1 P604(4)=0 P106=N P604(8)=0

For a factor of x2:

$$\text{Number of pulses} = \frac{5\text{mm}}{2 \times 0.01\text{ mm}} = 250 \text{ pulses/rev.}$$

P103= 1 P604(4)=0 P106=N P604(8)=1

Example 2: **Resolution in "inches" with square-wave encoder**

We would like to obtain a 0.001 inch resolution with a square-wave encoder mounted on to the X axis which has a 4-pitch leadscrew (4 turns per inch or 0.25 inch/turn).

Since the CNC always applies a multiplying factor of either x2 or x4 (selected by machine parameter), the required encoder line count (pulses per rev) in each case will be:

$$\text{Number of pulses} = \frac{\text{Leadscrew pitch}}{\text{Multiplying factor x Resolution}}$$

With a x4 factor:

$$\text{Number of pulses} = \frac{0.25}{4 \times 0.001} = \textbf{62.5*} \text{ pulses/turn}$$

*** A gear reduction will be required to achieve this line count per turn.**

P103= 1 P604(4)=1 P106=N P604(8)=1

With a x2 factor:

$$\text{Number of pulses} = \frac{0.25}{2 \times 0.001} = 125 \text{ pulses/turn}$$

P103= 1 P604(4)=1 P106=N P604(8)=1

P617(6) Axis resolution of 0.0001mm (tenth of a micron) or 0.00001 inch (10 millionths)

This parameter must only be set for those machine having one or more axes requiring this kind of resolution.

This parameter **affects both axes** even when one of them might not require this kind of resolution.

Possible values:

0 = Machine with **normal** axes with 0.001 mm or 0.0001 inch minimum resolution.
1 = Machine with **special** 0.0001mm or 0.00001 inch resolution.

When setting this parameter to “1”, the following items must be considered:

* The programming format will now be: ±3.4 in mm and ±2.5 in inches.

* The minimum moving distance will now be: ±0.0001mm and ±0.00001 inch. The maximum moving distance will be: ±838.8607mm and ±33.02599 inches.

* The tool table format will be affected the same way:

  R,L ±2.4 in mm or ±1.5 in inches. Minimum value: ±0.0001mm and ±0.00001 inch. Maximum value: ±99.9999mm and ±3.93699 inches.

  I,K ±1.4 in mm and ±0.5 in inches. Minimum value: ±0.0001mm and ±0.00001 inch. Maximum value: ±3.2766 mm and ±0.12900 inches.

* The integer values assigned to machine parameters P103, P203 and P303 for axis resolution now acquire new units:

  1 = 0.0001 mm, 0.00001 inch or 0.001° resolution
  2 = 0.0002 mm, 0.00002 inch or 0.002° resolution
  5 = 0.0005 mm, 0.00005 inch or 0.005° resolution
  10 = 0.0010 mm, 0.00010 inch or 0.010° resolution

* When calculating K1, K2 gains and the Feed-forward gain, the following error is now expressed in 0.0001 mm units (not microns) and 0.00001 inch units (instead of 0.0001 inch as before). The maximum amount of following error allowed is now: 3.20mm

  That is to say that K1 and K2 gains (parameters P114, P214, P314, P414, P514, P116, P216, P316, P416, P516) must be given in "mV/0.0001mm (mV/0.00001 inches).

* Machine parameters P115, P215, P315, P415. P515 for gain break-point are now expressed in 0.0001 units (not microns) and 0.00001 inch units (instead of 0.0001 inch as before).

* Machine parameters P109, P209, P309, P409, P509 (leadscrew backlash) and P118, P218, P318, P418, P518 (in-position zone) will also be given in 0.0001 mm and 0.00001 inch units.

  P118 = 100 means an in-position zone of 0.01mm (or 0.001 inch) for the X axis.

* Machine parameters P112, P212, P312, P412, P512, P810, P811, P812, P813, P814 (homing feedrate) and P801 (unidirectional approach feedrate) will also be expressed in 0.0001mm and 0.00001 inch units.

  P112 = 10000 Assigns a feedrate of 1m/min.

Examples to calculate resolution with P617(6)=1:

Example 1: **Resolution in "mm" with square-wave encoder**

We want to obtain a 0.0001mm resolution with a square-wave encoder mounted on the X axis whose leadscrew has a 5mm/turn pitch.

Since the multiplying factor applied by the CNC may be either x2 or x4 (depending on machine parameter setting). The resulting encoder line count will be:

$$\text{Number of pulses} = \frac{\text{Leadscrew pitch}}{\text{Multiplying Factor x Resolution}}$$

For a factor of x4:

$$\text{Number of pulses} = \frac{5\text{ mm}}{4 \times 0.0001\text{m}} = 12500 \text{ pulses/rev.}$$

P103= 1 P604(4)=0 P106=N P604(8)=0

For a factor of x2:

$$\text{Number of pulses} = \frac{5\text{mm}}{2 \times 0.0001\text{ mm}} = 25000 \text{ pulses/rev.}$$

P103= 1 P604(4)=0 P106=N P604(8)=1

Example 2: **Resolution in "inches" with square-wave encoder**

We would like to obtain a 0.001 inch resolution with a square-wave encoder mounted on to the X axis which has a 4-pitch leadscrew (4 turns per inch or 0.25 inch/turn).

Since the CNC always applies a multiplying factor of either x2 or x4 (selected by machine parameter), the required encoder line count (pulses per rev) in each case will be:

$$\text{Number of pulses} = \frac{\text{Leadscrew pitch}}{\text{Multiplying factor x Resolution}}$$

With a x4 factor:

$$\text{Number of pulses} = \frac{0.25}{4 \times 0.00001} = 6250 \text{ pulses/turn}$$

P103= 1 P604(4)=1 P106=N P604(8)=1

With a x2 factor:

$$\text{Number of pulses} = \frac{0.25}{2 \times 0.00001} = 12500 \text{ pulses/turn}$$

P103= 1 P604(4)=1 P106=N P604(8)=1

P908, P909 Collision zone between Y, Z

These parameters are used mainly on boring mills and horizontal machining centers to define a possible collision zone between the Y and Z axes.

Each one of them indicates the position value where the collision zone begins.

P908 = Y axis position value.
P909 = Z axis position value.

These position values are absolute and referred to the machine reference zero (home).

Possible values: ± 8388.607 millimeters.
± 330.2599 inches.

If one of the axes gets into this zone, the CNC will prevent the other one from getting into it.

P621(4) Beginning of a block synchronized with an independent axis (G65)

With function G65 it is possible to move one axis independently while other axes are being interpolated.

In the following program:

```
N0   G65 W100 F1
N10  G01 X10 Y10 Z5 F1000
N20  G01 X20
```

When executing block "N0", the W axis starts moving at a feedrate of F1. Then, block "N10" starts executing the XYZ interpolation at F1000 while the W axis keeps moving at F1.

If **"P621(4)=0"**, the CNC executes block "N20" once "N10" is completed regardless of whether "N0" is completed or not (W axis has reached position or not).

If **"P621(4)=1"**, the CNC waits until blocks "N0" and "N10" are completed (all axes have reached position) before executing block "N20".

P626(3) "RESCAN 200" feature from Renishaw available or not.

It indicates whether the machine uses the "**RESCAN 200**" feature from **Renishaw** or not.

0 = Not available.
1 = Available.

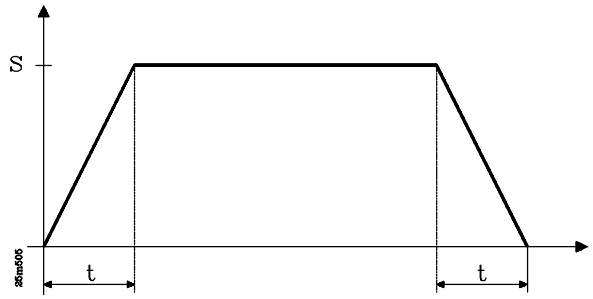
5. *SPINDLE MACHINE PARAMETERS*

Observe that some of the parameters mentioned in this chapter are also described in further detail in the chapter on "concepts" of this manual.

P815 Spindle acceleration/deceleration ramp duration

The CNC takes this parameter into account at all times when working in open or closed loop.

It establishes the time the spindle requires to reach the selected speed (acceleration stage). This time will be the same for the deceleration stage.



It is expressed by an integer between 0 and 65535.

Value of	0	= There is no acc/dec.
Value of	1	= 0.010 seconds.
Value of	10	= 0.100 seconds. (10 x 0.01)
Value of	2000	= 20 seconds. (2000 x 0.01)
Value of	4095	= 40.95 seconds (4095 x 0.01)
Value >	4095	= 40.95 seconds (4095 x 0.01)

5.1 PARAMETERS RELATED TO SPINDLE SPEED RANGE CHANGE

The section on "Spindle range change" in the chapter on "concepts" of this manual describes how these parameters may be used.

P7, P8, P9, P10 Maximum spindle speed for 1st, 2nd, 3rd and 4th RANGE.

They indicate the maximum spindle speed assigned to each range.

It is given in revolutions per minute and they accept any integer value between 0 and 9999.

The value assigned to P7 must correspond to the lowest range and that of P10 to the highest range. When not all the ranges are being used, assign the lowest range to P7 and set the unused ones to the highest speed value.

P601(6) Residual analog voltage (S) during spindle range change.

It determines whether the CNC must generate a residual analog voltage (S) during a range change.

Possible values:

0 = **No** residual analog voltage is generated.
1 = A residual analog voltage **is** generated.

P706 Value of the residual analog voltage (S).

It indicates the value of the residual analog voltage (S) for the spindle speed range change.

It is given by an integer between 1 and 255.

Value of	1	=	2.5 mV.
Value of	10	=	25.0 mV. (10 x 2.5)
Value of	255	=	637.5 mV. (255 x 2.5)

P707 Oscillation period during a spindle range change.

It indicates the oscillation time period during a spindle range change.

It is given by an integer between 0 and 255.

Value of	0	=	Continuous movement in one direction.
Value of	1	=	Continuous movement in the other direction.
Value of	2	=	20 msec. oscillation period.
Value of	10	=	100 msec. oscillation period.
Value of	255	=	2550 msec. oscillation period.

5.2 PARAMETERS FOR ANALOG SPINDLE SPEED OUTPUT

The section on "Spindle" in the chapter on "concepts" of this manual describes how these parameters may be used.

P601(4) Sign of the spindle analog output.

It determines the sign of the spindle analog output (S). If correct, leave it as is; if not, change it.

Possible values: “0” and “1”.

P610(4) Unipolar or bipolar spindle analog output.

It indicates the type of spindle analog output.

If the analog output is BIPOLAR, the CNC will generate a positive analog voltage (0 to +10V) to turn the spindle clockwise (M03) and a negative analog voltage (0 to -10V) to turn the spindle counter-clockwise (M04).

If the analog output is UNIPOLAR, the CNC will generate a positive analog voltage (0 to +10V) for either turning direction.

0 = The output must be BIPOLAR.
1 = The output must be UNIPOLAR.

Bear in mind that with machine parameter P601(4) it is possible to change the sign of the analog output and, therefore, the spindle turning direction.

P609(4) Any spindle speed change generates an S STROBE

It indicates whether or not a 200msec. S STROBE (pulse) is output at pin 3 of connector I/O1, every time a new spindle speed is selected.

0 = No S STROBE is generated.
1 = An S STROBE is generated.

5.3 PARAMETERS FOR SPINDLE SPEED OUTPUT IN BCD

The section on "Spindle" in the chapter on "concepts" of this manual describes how these parameters may be used.

P601(3) Spindle speed output in 2-digit BCD code.

It indicates whether there is a 2-digit BCD coded spindle speed output or not. If not, the CNC will output an analog voltage for the spindle.

0 = **No 2-digit BCD** coded output is used for spindle speed.
1 = A 2-digit BCD coded output is used for spindle speed.

If this parameter is set to “1”, the CNC will issue the value corresponding to the programmed spindle speed via the BCD outputs which are pins 20 thru 27 of the I/O 1 connector. It will also output an S STROBE pulse at pin 3 of connector I/O 1.

The chart below shows the BCD code corresponding to the programmable spindle speed values:

Programmed S	S BCD	Programmed 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 greater than 9999 is programmed, the CNC will assume the spindle speed corresponding to 9999.

P601(2) 4-digit BCD coded spindle speed output

It indicates whether there is a 4-digit BCD coded spindle speed output or not. If not, the CNC will output an analog voltage for the spindle.

0 = **No 4-digit BCD** coded output is used for spindle speed.
1 = A 4-digit BCD coded output is used for spindle speed.

If this parameter is set to "1", the CNC will issue the value corresponding to the programmed spindle speed via the BCD outputs which are pins 20 thru 27 of the I/O 1 connector.

The CNC will output the value corresponding to the programmed S in two stages with a 100 msec. delay between them. It will also output an S STROBE pulse at pin 3 of connector I/O 1.

Pin	1st stage	2nd stage
20 21 22 23	Thousands	Tens
24 25 26 27	Hundreds	Units

5.4 PARAMETERS USED FOR SPINDLE CONTROL

It is necessary to have a spindle feedback encoder installed in order to perform the following operations:

* Electronic threading (G33)
* Spindle orientation (M19)
* Rigid tapping cycle (G84R)

When working with spindle orientation (M19) or executing a rigid tapping cycle (G84R), the spindle must be in closed loop; in other words, the CNC must be controlling the actual spindle speed at all times providing the electrical cabinet with the analog voltage so the spindle turns at the selected speed.

The section on "Spindle " in the chapter on "concepts" of this manual describes how these parameters may be used.

P800 Number of spindle encoder pulses (line count)

It indicates the number of pulses of the spindle encoder.

It is expressed by an integer between 0 and 9999.

If this parameter is set to "0", it will mean that there is no spindle encoder and, therefore, the spindle will always work in open loop.

P609(2) Spindle counting direction

It determines the spindle counting direction. If correct, leave it as is and change it if otherwise.

Possible values: “0” and “1”.

5.5 PARAMETERS RELATED TO SPINDLE ORIENTATION (M19)

The section on "Spindle control" in the chapter on "concepts" of this manual describes how these parameters may be used.

P700 Spindle speed when working in M19

It is given in rpm with a value between 0 and 255.

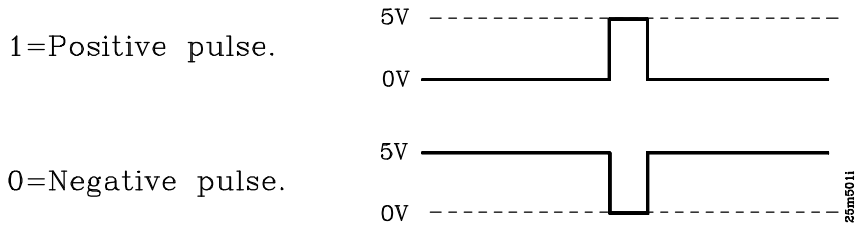
P601(7) Sign of the analog S output associated with M19

It determines the sign of the analog output associated with M19. If correct, leave it as is and change it if otherwise.

Possible values: “0” and “1”.

P612(8) Marker pulse (reference) type of the spindle encoder

It indicates the type of marker pulse (Io) of the spindle encoder used to synchronize the spindle in M19.



P619(6) Spindle orientation in both directions

It determines whether the spindle orients (M19 Sxxx) always in the same direction or in the direction indicated by the sign of the S value.

0 = The spindle always orients in the same direction..
1 = The spindle orients in the direction indicated by the sign of the S value.

When setting this parameter to "1", the positive sign coincides with the spindle homing direction.

P719 Minimum spindle analog during M19

It determines the minimum spindle analog voltage when in M19.

It is given by an integer between 0 and 255.

Value 0 = 2.5 mV.
Value 1 = 2.5 mV.
Value 10 = 25.0 mV. (10 x 2.5)
Value 255 = 637.5 mV. (255 x 2.5)

P717 In-position zone (dead band) for M19

It establishes the width of the in-position zone to both sides of the programmed coordinate so the CNC considers the spindle to be in position.

It is expressed by the number of encoder pulses between 0 and 255.

Remember that the CNC internally multiplies by 4 the number of pulses provided by the encoder.

Therefore, if the spindle encoder provides 1000 pulses per turn and P717 = 100, the in-position zone (dead band) will be:

$$\frac{360°}{1000 \times 4} \times 100 = \pm 9°$$

P718 Proportional gain K for the spindle when in M19

It set the analog voltage corresponding to 1 feedback pulse of following error of the spindle encoder.

It is defined by an integer between 0 and 255, where a value of 64 corresponds to an analog of 2.5mV.

$$\text{Analog (mV.)} = K \times \text{Following Error (pulses)} \times \frac{2.5\text{mV.}}{64}$$

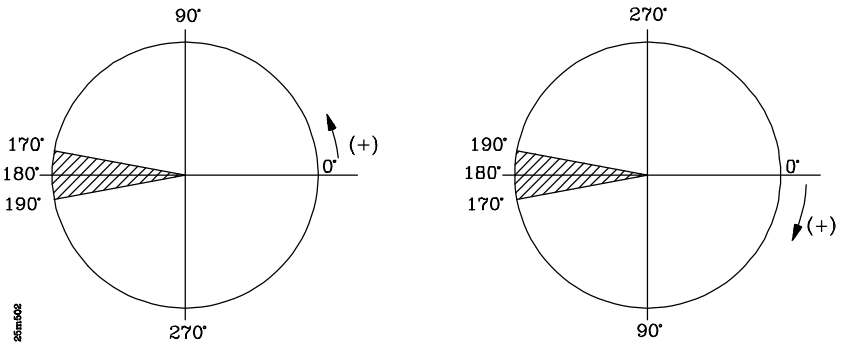
P917 Lower limit of the forbidden zone for the spindle in M19
P918 Upper limit of the forbidden zone for the spindle in M19

These parameters are applicable when the spindle can orient in both directions "P619(6)=1".

These parameters establish a forbidden zone for the spindle.

They are given in degrees with a precision of one thousandth of a degree.

Example: The illustrations below show the forbidden zone when setting:
P917 = 170 and P918 = 190.



If the spindle is positioned at 150° and we want to move it to 200°, there are two ways to program such move as described next:

M19 S200 The CNC shows the corresponding error code since this instruction would make the spindle go through the forbidden zone.

M19 S-200 The spindle will move to the desired point in the negative direction thus avoiding the forbidden zone.

If both limits are set to "0", it will mean that there is no forbidden zone.

If function M19 is executed without programming an S value, the CNC will ignore this forbidden zone.

P916 Spindle orient position when programming M19 without an "S" value

To switch from open loop (M03, M04) to closed loop, just program M19 (without S). The CNC will reduce the spindle speed below the one indicated by parameter "P700". Then, it will search home and it will position at the point indicated by parameter "P916".

It is given in degrees and with a precision of one thousandth of a degree.

5.6 PARAMETERS RELATED TO RIGID TAPPING (G84R)

General considerations:

* Rigid tapping consists in an interpolation between the spindle and the Z axis when the main plane is formed by the X and Y axes.

* There is a machine parameter for each spindle range allowing the adjustment of its acc./dec. ramp.

* When working with small taps (such as metric 3mm or 4mm, for example), it is recommended to apply smooth spindle acc./dec. ramp to prevent breaking the tap.

* In order for the tap to follow the same path in its way out, the acc./dec. times for the spindle and for the perpendicular axis must be the same.

* By the same token, the following error (lag value) of the spindle and that of the perpendicular axis must be proportional.

  For example: if when tapping at F1000mm/min, S1000 rpm (thread pitch= 1mm) we obtain:

  Z axis following error = 1mm
  Spindle following error = 360°

  It can be said that both axes are perfectly synchronized

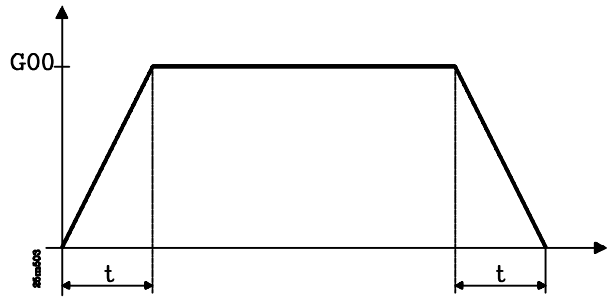
* Since the Z axis gains are different during regular machining and during rigid tapping, the CNC offers 2 machine parameters, one for each case.

P745, P747, P748, P749 ACC/DEC ramp duration for ranges 1, 2, 3 and 4

When executing a rigid tapping cycle G84R, the CNC does not apply the acc/dec ramp set by machine parameter "P815".

However, machine parameters "P745, P747, P748, P749" must be set in order to apply acc./dec. during rigid tapping cycle.

Each one of these parameters corresponds to one range and they establish the time the spindle requires to reach the selected speed (acceleration stage). This time will be the same for the deceleration stage.



It is expressed by an integer value between 0 and 255.

Value 0 = There is no acc/dec.
Value 1 = 0.020 seconds.
Value 10 = 0.200 seconds. (10 x 0.02)
Value 255 = 5.100 seconds. (255 x 0.02)

P750 Proportional gain K1 for the tapping axis

More than likely, the K1 gain set for the tapping axis in parameters (P114, P214, P314, P414, P514) is not appropriate to perform the rigid tapping operation.

In that case, this specific K1 can be used for the rigid tapping cycle and keep the other K1 for the rest.

This parameter sets the analog voltage corresponding to 1 micron of following error.

It is expressed by an integer between 0 and 255 where a value of 64 corresponds to an analog voltage of 2.5mV.

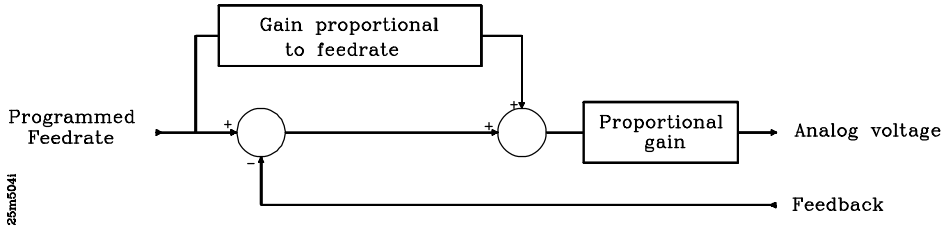
$$\text{Analog (mV.)} = P750 \times \text{Following (microns)} \times \frac{2.5\text{mV.}}{64}$$

P746 FEED-FORWARD for the spindle when Rigid Tapping

This parameter is used when executing a tapping canned cycle G84.

With the Feed-forward gain, which is proportional to the feedrate, it is possible to improve the positioning loop minimizing the amount of following error.
However it is not recommended when acceleration/deceleration ramps are not being used.

This parameter defines the % of analog voltage due to the programmed feedrate. It is given by an integer between 0 and 255.



The value which will be added to the following error is (P746 x F/6), where P746 is the feed-forward value selected and the F is the programmed feedrate.

The CNC will apply the proportional gain (P718) to the value resulting from adding the following error and the value selected by the feed-forward.

The CNC will apply the following formula:

Analog = P718 x [Following Error + (P746 x F/6)]

P625(1) The beginning of the tap is synchronized with the spindle marker pulse (Io)

This parameter is used when executing the tapping cycle G84.

0 = The beginning of the tap is **not** synchronized with the spindle Io.
1 = The beginning of the tap **is** not synchronized with the spindle Io.

When synchronizing the beginning of the tap with the spindle marker pulse (Io), it is possible to repeat any tap previously made without damaging it.

6. CONCEPTS

It is recommended to save the CNC machine parameters onto a peripheral device or PC to avoid losing them by mistake or any malfunction.

6.1 AXES AND COORDINATE SYSTEMS

Since the CNC must control the movement and positioning of the axes, it is necessary to establish the target point by means of its coordinates.

With this CNC, it is possible to use absolute and incremental coordinates along the same program.

6.1.1 NOMENCLATURE AND SELECTION OF THE AXES

The axes that can be controlled by the CNC are referred to as follows:

X, Y Main movements on the main work plane of the machine.

Z Main axis of the machine, perpendicular to the main XY plane.

V, W Auxiliary axes.

All of them must be properly defined as linear, rotary, etc. by means of their corresponding machine parameters.

With this CNC it is possible to control the following types of machines:

* 3-axis machines. X, Y, Z axes
* 4-axis machines. X, Y, Z, W axes
* 5-axis machines. X, Y, Z, W, V axes

When controlling a 4-axis or 5-axis machine, machine parameter "**P11**" must be properly set. It must also be borne in mind that the CNC may interpolate up to 3 axes at a time; therefore, the W and V axes are incompatible with each other and with one of the X, Y, Z axes set by this machine parameter (P11).

To control the **V** axis, general machine parameter **P616(4)** must be set to **"1**".

6.2 FEEDBACK SYSTEMS

The feedback inputs of this CNC are:

Connectors A1, A2, A3, A4.

They are used to connect the feedback inputs for the X, Y, Z and W axes respectively.

They admit sine-wave and double-ended (differential) square-wave signals which must be properly indicated by machine parameters “P106, P206, P306, P406” and the two dip-switches located under each feedback input.

Connector A5.

It admits double-ended (differential) square-wave signals to connect the feedback device for the 5th axis (V), the spindle encoder or the electronic handwheel.

Depending on the type of feedback being used, set machine parameters “P506, P616(4) and P800” accordingly.

Connector A6.

It admits single-ended (non-differential) signals and it is used to connect the spindle encoder or the electronic handwheel.

Depending on the type of feedback being used, machine parameters “P612(1), P616(4) and P800” must be set accordingly.

The possible axis combinations offered by this CNC are:

A1	A2	A3	A4	A5	A6
X	Y	Z	W	V	S
X	Y	Z	W	V	Handwheel
X	Y	Z	W	S	Handwheel
X	Y	Z	W	Handwheel	S

The "S" letter indicates that the feedback input is used to control the spindle. However, it is possible to control the spindle without having to use that feedback device since the CNC provides the corresponding analog voltage output at pins 36 and 37 of connector I/O1.

6.2.1 COUNTING FREQUENCY LIMITS

Sine-wave signals

The maximum counting frequency for sine-wave feedback signals is 25KHz (25,000 pulses/sec.).

Therefore, the maximum feedrate for each linear axis will depend upon the selected resolution (machine parameters "P103, P203, P303, P403, P503") and the period of the feedback signal being used.

The maximum feedrate for each rotary axis will depend upon the number of pulses per revolution.

Example 1:

When using a linear scale with a 20µm, the maximum feedrate for an axis with 1µm resolution will be:

20 µm/pulse x 25,000 pulses/sec = 500 mm/sec = 30 m/min.

Example 2:

When using a rotary table with a 3600-count sine-wave encoder the maximum speed for 0.001º resolution will be:

$$\frac{360°/\text{turn}}{3{,}600\ \text{pulses/turn}} \times 25{,}000\ \text{pulses/sec.} = 2{,}500°/\text{sec.} = 150{,}000°/\text{min.}$$

Square-wave signals

The maximum counting frequency for differential square-wave signals is 200 KHz (200,000 pulses/sec.), with a 450nsec. separation between A and B flanks (that is 90º ±20º phase shift).

Therefore, the maximum feedrate for each linear axis will depend upon the selected resolution (machine parameters "P103, P203, P303, P403, P503") and the period of the feedback signal being used.

When using FAGOR linear scales, the maximum feedrate is limited by its own characteristics to 60m/min (2362 inches/min).

When using FAGOR rotary encoders, the limitation is set by the maximum number of pulses delivered by the encoder; which is 200KHz.

6.3 AXIS RESOLUTION

The CNC has a series of machine parameters to set the resolution of each axis.

The resolution used on each axis indicates the minimum variation distinguishable by the feedback device. It is given in microns or 0.0001 inch units.

The machine parameters used to define the axis resolution are the following:

P103, P203, P303, P403, P503

They set the counting resolution for each axis.

P604(4), P604(3), P604(2), P604(1), P616(7)

They set the measuring units for each axis feedback signal (mm or inches).

P106, P206, P306, P406, P506

They set the type of feedback signal being used (square-wave or sine-wave) for each axis.

P604(8), P604(7), P604(6), P604(5), P616(8),

They indicate the multiplying factor, x2 or x4 to be applied to the feedback signals of each axis.

P622(1), P622(2), P622(3), P622(4), P622(5)

They indicate the special multiplying factor to be applied to the sine-wave feedback signals of each axis (besides the normal x5).

Example 1: **Resolution in "mm" with square-wave encoder**

We want to obtain a 2µm resolution with a square-wave encoder mounted on the X axis whose leadscrew has a 5mm/turn pitch.

Since the multiplying factor applied by the CNC may be either x2 or x4 (depending on machine parameter setting). The resulting encoder line count will be:

$$\text{Number of pulses} = \frac{\text{Leadscrew pitch}}{\text{Multiplying Factor x Resolution}}$$

For a factor of x4:

$$\text{Number of pulses} = \frac{5000\ \mu m}{4 \times 2\ \mu m} = 625 \text{ pulses/rev.}$$

P103= 2 P604(4)=0 P106=N P604(8)=0

For a factor of x2:

$$\text{Number of pulses} = \frac{5000\ \mu m}{2 \times 2\ \mu m} = 1250 \text{ pulses/rev.}$$

P103= 2 P604(4)=0 P106=N P604(8)=1

If a FAGOR encoder is chosen, its pulse output frequency is limited to 200KHz (although the CNC admits square-wave pulses with a frequency of up to 200KHz). Therefore, the maximum feedrate for this axis will be:

When using a x4 multiplying factor:

$$\text{Max. Feed} = \frac{200{,}000 \text{ pulses/sec.}}{625 \text{ pulses/rev.}} \times 5 \text{ mm/rev.} = 1600 \text{ mm/sec.} = 96 \text{ m/min.}$$

When using a x2 multiplying factor:

$$\text{Max. Feed} = \frac{200{,}000 \text{ pulses/sec.}}{1250 \text{ pulses/rev.}} \times 5 \text{ mm/rev.} = 800 \text{ mm/sec.} = 48 \text{ m/min.}$$

Example 2: **Resolution in "mm" with sine-wave encoder**

We would like to get a 2µm resolution with a sine-wave encoder mounted on to the X axis which has a 5mm/turn leadscrew pitch.

We have the following options:

P622(1)=0		
P604(8)	**Resolution**	**P103**
=1 (x2)	2 microns	10
=0 (x4)	2 microns	10

P622(1)=1		
P604(8)	**Resolution**	**P103**
=1 (x2)	2 microns	2
=0 (x4)	2 microns	2

Since the CNC always applies a x5 multiplying factor to the sine-wave feedback signals, we will need an encoder:

$$\text{Nº of pulses} = \frac{\text{Leadscrew pitch}}{5 \times \text{Multiplying factor} \times \text{Resolution}}$$

For P604(8)=1 (x2)

$$\text{Nº pulses} = \frac{5000\mu m/turn}{5 \times 2 \times 2\mu m/pulse} = 250 \text{ pulses/turn}$$

Therefore:

If P622(1)=0 => P604(4)=0 P106=Y P604(8)=1 P103= 10
If P622(1)=1 => P604(4)=0 P106=Y P604(8)=1 P103= 2

For P604(8)=0 (x4)

$$\text{Nº pulses} = \frac{5000\mu m/turn}{5 \times 4 \times 2\mu m/pulse} = 125 \text{ pulses/turn}$$

Therefore:

If P622(1)=0 => P604(4)=0 P106=Y P604(8)=0 P103= 5
If P622(1)=1 => P604(4)=0 P106=Y P604(8)=0 P103= 2

Even when choosing a FAGOR encoder which outputs up to 200KHz, the actual usable frequency is this time limited by the CNC to 25KHz for sine-wave signals. Therefore, the maximum feedrate for this example will be:

$$\text{Max. Feed} = \frac{25{,}000 \text{ pulses/sec.}}{125 \text{ pulses/rev.}} \times 5 \text{ mm/rev.} = 1000 \text{ mm/sec.} = 60 \text{ m/min.}$$

and 30m/min for 250-line encoder.

Example 3: **Resolution in "mm" with square-wave linear scale**

Considering that the CNC applies either a x2 or x4 multiplying factor (set by machine parameter), a linear scale must be chosen whose pitch is 2 or 4 times the desired resolution.

When using FAGOR linear transducers (scales) with 20µm pitch, the following resolutions may be obtained: 5µm (20/4), 10µm (20/2).

Thus:

Scale pitch	P103	P604(4)	P106	P604(8)
20µm	5	0	0	0
20µm	10	0	0	1

Since the counting frequency of the CNC is limited to 200KHz for square-wave signals, the maximum feedrate obtainable with a 20µm-pitch scale is:

Max. Feed = 20 µm/pulse x 200,000 pulses/sec. = 4000 mm/sec. = **240 m/min.**

However, if FAGOR linear scales are used, the maximum feedrate is limited (by the scales) to 60m/min. (2362 inches/min.)

Example 4: **Resolution in "mm" with sine-wave linear scales**

A sine-wave linear scale is being used with a 20µm pitch and 1µm resolution.

There are the following options:

P622(1)=0		
P604(8)	**Resolution**	**P103**
=0 (x4)	1 micron	5

P622(1)=1		
P604(8)	**Resolution**	**P103**
=0 (x4)	1 micron	1

Therefore:

If P622(1)=0 => P604(4)=0 P106=Y P604(8)=0 P103= 5
If P622(1)=1 => P604(4)=0 P106=Y P604(8)=0 P103= 1

The CNC's counting frequency is limited to 25KHz (30m/min) for sine-wave signals.

However, if FAGOR linear scales are used, the maximum feedrate is limited (by the scales) to 60m/min. (2362 inches/min.)

Example 5: **Resolution in "inches" with square-wave encoder**

We would like to obtain a 0.0001 inch resolution with a square-wave encoder mounted on to the X axis which has a 4-pitch leadscrew (4 turns per inch or 0.25 inch/turn).

Since the CNC always applies a multiplying factor of either x2 or x4 (selected by machine parameter), the required encoder line count (pulses per rev) in each case will be:

$$\text{Number of pulses} = \frac{\text{Leadscrew pitch}}{\text{Multiplying factor x Resolution}}$$

With a x4 factor:

$$\text{Number of pulses} = \frac{0.25}{4 \times 0.0001} = 625 \text{ pulses/turn}$$

P103= 1 P604(4)=1 P106=N P604(8)=0

With a x2 factor:

$$\text{Number of pulses} = \frac{0.25}{2 \times 0.0001} = 1250 \text{ pulses/turn}$$

P103= 1 P604(4)=1 P106=N P604(8)=1

If a FAGOR encoder is used, the counting frequency is limited to 200KHz by the scale (the CNC admits up to 200KHz for square-wave signals). Therefore, the maximum feedrate for this axis will be:

For x4 multiplying factor:

$$\text{Max. Feed} = \frac{200{,}000 \text{ pulses/sec.}}{625 \text{ pulses/rev}} \times 0.25 \text{ inch/rev} = 80 \text{ inches/sec} = 4800 \text{ inch/min.}$$

For x2 multiplying factor:

$$\text{Max. Feed} = \frac{200{,}000 \text{ pulses/sec.}}{1250 \text{ pulses/rev}} \times 0.25 \text{ inch/rev} = 40 \text{ inches/sec} = 2400 \text{ inch/min.}$$

Example 6: Resolution in "inches" with sine-wave encoder

We would like to get a 0.0001 inch resolution with a sine-wave encoder mounted on to the X axis which has a 4-pitch leadscrew (0.25 inch/turn).

We have the following options:

P622(1)=0		
P604(8)	**Resolution**	**P103**
=1 (x2)	0.0001 inch	5
=0 (x4)	0.0001 inch	5

P622(1)=1		
P604(8)	**Resolution**	**P103**
=1 (x2)	0.0001 inch	1
=0 (x4)	0.0001 inch	1

Since the CNC always applies a x5 multiplying factor to the sine-wave feedback signals, we will need an encoder:

$$\text{Nº of pulses} = \frac{\text{Leadscrew pitch}}{\text{5 x Multiplying factor x Resolution}}$$

For P604(8)=1 (x2)

$$\text{Nº pulses} = \frac{\text{0.25 inch/turn}}{\text{5 x 2 x 0.0001 inch/pulse}} = \text{250 pulses/turn}$$

Therefore:

If P622(1)=0 => P604(4)=0 P106=Y P604(8)=1 P103= 5
If P622(1)=1 => P604(4)=0 P106=Y P604(8)=1 P103= 1

For P604(8)=0 (x4)

$$\text{Nº pulses} = \frac{\text{0.25 inch/turn}}{\text{5 x 4 x 0.0001 inch/pulse}} = \text{125 pulses/turn}$$

Therefore:

If P622(1)=0 => P604(4)=0 P106=Y P604(8)=0 P103= 5
If P622(1)=1 => P604(4)=0 P106=Y P604(8)=0 P103= 1

Even when choosing a FAGOR encoder which outputs up to 200KHz, the actual usable frequency is this time limited by the CNC to 25KHz for sine-wave signals. Therefore, the maximum feedrate for this example will be:

$$\text{Max. Feed} = \frac{\text{25,000 pul./sec.}}{\text{125 pul./rev.}} \text{ x 0.25 inch/rev.} = \text{500 inch/sec.} = \text{3000 inch/min.}$$

and 1500 inch/min for 250-line encoder.

Example 7: Resolution in "inches" with square-wave encoder

We would like to get a 0.005° resolution with square-wave encoder mounted on the W axis.

Since the CNC applies a multiplying factor of x4 to obtain this resolution, the encoder must have the following line count (pulses):

$$\text{Nº of pulses} = \frac{\text{Degrees}}{\text{Mult. factor x Resolution}}$$

$$\text{Nº of pulses} = \frac{360°/\text{turn}}{4 \times 0.005°/\text{pulse}} = 18000 \text{ pulses/turn}$$

P403=5 P604(1)=0 P406=N P604(5)=0

If a FAGOR encoder is used, the counting frequency is limited to 200KHz by the scale (the CNC admits up to 200KHz for square-wave signals). Therefore, the maximum feedrate for this axis will be:

For x4 multiplying factor:

$$\text{Max. rpm} = \frac{200{,}000 \text{ pulses/sec.}}{18000 \text{ pulses/rev}} = 11.111 \text{rev./sec} = 666.66 \text{ rpm.}$$

Example 8: Resolution in "inches" with sine-wave encoder

We would like to get a 0.005° resolution with sine-wave encoder mounted on the W axis.

There are these options:

P622(4)=1		
P604(5)	**Resolution**	**P403**
=1 (x2)	0.005 degrees	5
=0 (x4)	0.005 degrees	5

Since the CNC always applies a multiplying factor of x5 to sine-wave feedback signals, the encoder must have the following line count (pulses):

$$\text{Nº of pulses} = \frac{\text{Degrees}}{5 \times \text{Mult. factor} \times \text{Resolution}}$$

$$\text{Nº of pulses} = \frac{360°/\text{turn}}{5 \times 4 \times 0.005°/\text{pulse}} = 3600 \text{ pulses/turn}$$

P622(4)=1 P403=5 P604(1)=0 P406=Y P604(5)=0

If a FAGOR encoder is used, the counting frequency is limited to 200KHz by the scale (the CNC admits up to 25KHz for sine-wave signals). Therefore, the maximum feedrate for this axis will be:

For x4 multiplying factor:

$$\text{Max. rpm} = \frac{25{,}000 \text{ pulses/sec.}}{3600 \text{ pulses/rev}} = 6.94 \text{ rev./sec} = 416.66 \text{ rpm.}$$

6.4 ADJUSTMENT OF THE AXES

In order to make this adjustment it is necessary to have the feedback systems for all the axes connected to the CNC.

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 axes are not CONTINUOUSLY CONTROLLED (that their enable signal is cancelled once the axis is in position). To do this, machine parameters P105, P205, P305, P405 and P505 must be set to "N".

Also, make sure that the CNC has been set to provide a delay between the axes enable and their analog voltage output. To do this, machine parameters P104, P204, P304, P404 and P504 must be set to "Y".

After the machine parameters for the axes have been properly set, proceed with their adjustment by following these suggestions:

* The axes should be adjusted one at a time.

* Connect the power output of the drive corresponding to the axis being adjusted.

* In the JOG mode, move the axis, being adjusted.

  In case of run-away, the CNC will display the relevant following error and the machine parameter corresponding to the SIGN OF THE ANALOG VOLTAGE will have to be changed. Machine parameters P100, P200, P300, P400 and P500.

* If the axis does not run away; but the direction of the move is not the desired one, the machine parameter corresponding to the COUNTING DIRECTION (P101, P201, P301, P401 and P501) will have to be changed as well as that corresponding to the SIGN OF THE ANALOG VOLTAGE (P100, P200, P300, P400 and P500).

* If the counting direction is correct but the axis moves in the opposite direction to the one indicated, change the axis machine parameter corresponding to the moving direction (P102, P202, P302, P402 and P502).

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

These adjustments are performed on axis servo drives and spindle drives.

Drift adjustment (offset)

This adjustment will be made in two stages:

Preadjustment of the drive offset

* Disconnect the analog voltage input of the drive and short-circuit it with a wire jumper.

* Turn the offset potentiometer of the drive until the voltage on the tacho terminals is 0V. This should be checked on the 200 mV DC scale of the voltmeter.

* Remove the wire jumper mentioned above.

Critical adjustment of the drive offset

* Execute a CNC program moving the axis in G00 continuously back and forth. One such program could be the following:

```
N10 G00 G90 X200
N20 X-200
N30 G25 N10
```

While the axis is moving, turn the offset potentiometer of the drive until the amounts of following error obtained in both directions are the same.

Adjustment of the maximum feedrate

It is recommended to adjust the drives so the maximum feedrate is obtained with an analog voltage of 9.5V.

Also, the maximum feedrate must be indicated in the corresponding machine parameter for that axis. Parameter P111, P211, P311, P411, P511.

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

Example for the X axis:

A motor can turn at 3000 rpm and it is attached to a 5 pitch leadscrew (1/5 inch/turn). Therefore, the maximum feedrate to be assigned to machine parameter P111 is:

Maximum feedrate (G00) = r.p.m. x leadscrew pitch

P111 = 3000 rev./min. x 1/5 inch/rev. = 600 inches/rev.

To make this adjustment, it is recommended to set P110 and P111 to the same value.

Also, run a CNC program which moves the axis in G00 continuously back and forth. One such program could be the following:

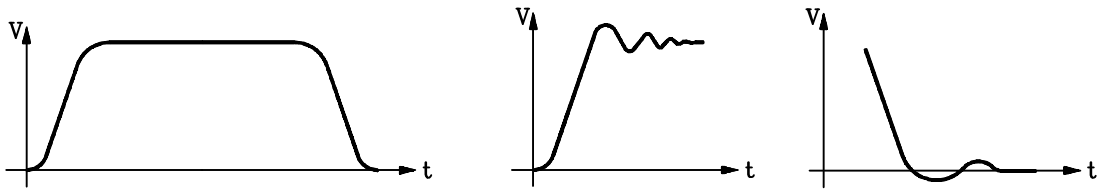
```
N10 G00 G90 X200
N20 X-200
N30 G25 N10
```

While the axis is moving, measure the analog voltage coming out of the CNC towards the servo drive and adjust the gain potentiometer **at the servo drive (never at the CNC)** until this analog voltage reaches 9.5V.

6.4.2 GAIN ADJUSTMENT

It is necessary to properly adjust the different gains for each axis in order to optimize the response of the whole system to the programmed movements.

It is recommended to use an oscilloscope in order to obtain a finer adjustment of the axes by monitoring the signals provided by the tacho. The diagram on the left corresponds to the ideal signal shape and the other ones to an unstable start-up and brake-down.



The CNC has a series of machine parameters which permit adjusting the proportional gain for each axis. These parameters are:

PROPORTIONAL GAIN K1.

Defined by parameters: P114, P214, P314, P414, P514.

PROPORTIONAL GAIN K2.

Defined by parameters: P116, P216, P316, P416, P516.

Value of the GAIN BREAK POINT

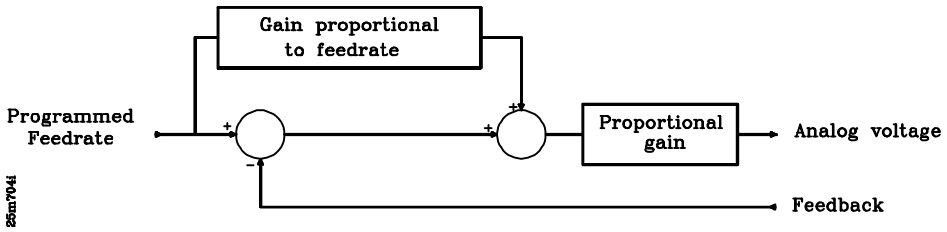
Defined by parameters: P115, P215, P315, P415, P515.

FEED-FORWARD GAIN or gain proportional to the feedrate.

Defined by parameters: P732, P733, P734, P735, P736.

The parameters corresponding to the proportional gain K1 and K2 as well as for the gain break point allow adjusting the Proportional Gain for the axis.

The parameter for the Feed-Forward gain (proportional to feedrate) will be used when acceleration/deceleration control is being applied onto the corresponding axis.



6.4.3 PROPORTIONAL GAIN ADJUSTMENT

The analog voltage supplied by the CNC to control the axis is, at all times, a function of the amount of following error; that is, the difference between the theoretical position and the real (actual) position of the axis.

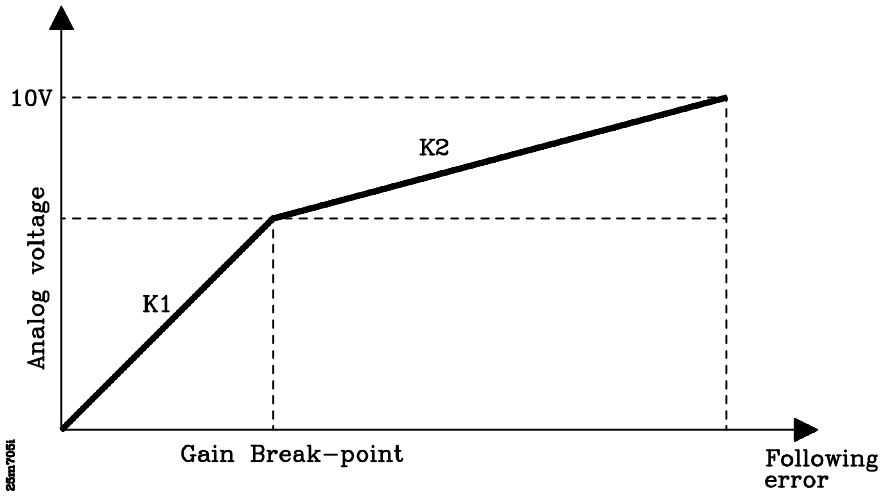
Analog output = Proportional gain “K” x Following Error

On start-up and slow-down, the following error of the axis is very small. Therefore, the proportional gain must be great in order for the axis to respond properly.

On the other hand, once the axis reaches its programmed speed, the following error is maintained practically constant and it is necessary to apply a smaller gain (K) in order to keep the system stable.

The FAGOR 8025 CNC offers two proportional gains K1 and K2 to better adjust the system as well as another parameter referred to as Gain Break point which defines the active area for each one of these gains.

The CNC applies the proportional gain K1 whenever the amount of following error for the axis is smaller than the value assigned to the machine parameter corresponding to the gain break-point.



When the amount of following error exceeds the gain break-point value, the CNC applies the K2 value.

Analog = (K1 x Ep) + [K2 x (Following Error - Ep)]

Where “Ep” is the value assigned to the gain break-point and it is given in microns.

When adjusting the proportional gain, it must be borne in mind that:

* When the amount of following error exceeds 32mm (1.2598 inches) the CNC will issue a Following error message for the corresponding axis.

* The amount of following error will decrease as the gain value increases, but the system will tend to be more unstable.

* In practice, most machines seem to respond well to what is called a unity gain (or gain of 1) which represents a following error of 1mm at a feedrate or 1m/minute or a following error of 0.001 inch at a feedrate of 1 inch/min.

  Therefore, this could be used as a practical starting point for the gain calculation described next. After analyzing the behavior of the machine for this gain, its value may be changed in order to optimize it.

6.4.3.1 CALCULATION OF K1, K2 AND GAIN BREAK-POINT

The value of K1 represents the analog voltage corresponding to 1 micron of following error. It is given by an integer between 0 and 255 in such a way that a value of 64 corresponds to an analog voltage of 2.5mV.

Therefore, the K1 value corresponding to a gain of 1 would be given by the following formulae:

$$\text{In metric (FE= 1mm for F=1m/min): } K1 = \frac{243.2}{\text{Fmax in m/min (P111 for X)}}$$

$$\text{In inches (FE= 0.001inch for F=1inch/min): } K1 = \frac{9575}{\text{Fmax in inch/min (P111 for X)}}$$

For example:

If the top feedrate for an axis is 500 inches/min, the K1 corresponding to a unity gain would be: K1 = 9575/500 = 19.15 and the value assigned to the corresponding parameter would be K1=19.

If the top feedrate for an axis is 20m/min, the K1 corresponding to a unity gain would be: K1 = 243.2/20 = 12.16 and the value assigned to the corresponding parameter would be K1=12.

The amount of following error corresponding to the GAIN BREAK-POINT is given in microns or 0.0001 inch units (by parameter P115 for the X axis, P215 for the Y axis, etc.). It is recommended to set it to a value slightly greater than the following error corresponding to the maximum machining feedrate F0 (P110, P210, P310, etc) (not positioning feedrate-P111, P211, P311, etc.-).

For example:

Let's suppose that K1 has been set for a gain of 1 (not K1=1) and that the maximum machining feedrate is 150 inches/min (P110, P210, P310). At this feedrate, the following error should be about 0.150 inch. Thus, the gain break-point value should be slightly larger than 0.150 inch; for example: P115=0.155 inch.

Or in metric:

Let's suppose that K1 has been set for a gain of 1 (not K1=1) and that the maximum machining feedrate is 5 m/min (P110, P210, P310). At this feedrate, the following error should be about 5 mm. Thus, the gain break-point value should be slightly larger than 5 mm; for example: P115=6 mm.

The machine parameter K2 gain sets the analog voltage for 1 micron of following error being applied from the gain break-point on.

It is also given by an integer between 0 and 255 and it is typically set to a value between 50% and 70% of K1 in order to avoid abrupt analog voltage changes when switching to slow machining feedrates.

To perform a practical axis adjustment at the machine, it is recommended:

1.- Adjust K1 optimizing the axis' response like the signal diagram shown earlier.

Set K1 =K2 or set the gain break-point to a large value (for example: 50000) and run a program moving the axis continuously back and forth. One such program could be the following:

```
N10 G00 G90 X200
N20 X-200
N30 G25 N10
```

2.- Set the gain-break point to the correct value.

To do this, run the previous program and watch the amount of following error reached at **maximum** feedrate and assign that value or one slightly larger to the gain break-point.

3.- Once K1 and the gain break-point have been set, change the value of K2 to one between 50% and 70% of K1.

Atention:

Once each axis has been adjusted separately. All interpolating axes should be fine adjusted together in such a way that their following errors for the same feedrate are the same in order to achieve proper interpolations between those axes in the K1 area.

6.4.4 FEED-FORWARD GAIN ADJUSTMENT.

With the Feed-Forward gain it is possible to improve the positioning loop of the axes, thus minimizing the amount of following error. **This gain must be used only when working with ACC/DEC.**

This CNC offers two types of acc/dec.:

Linear

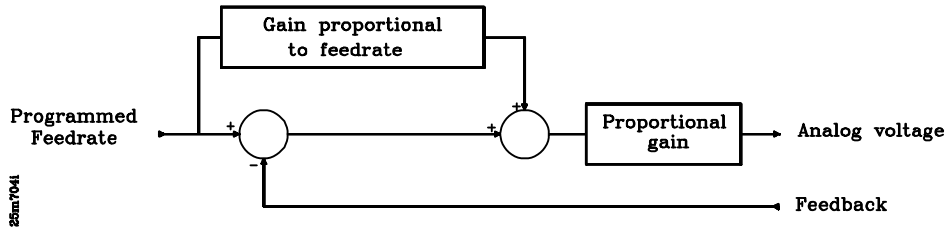
It is mainly applied onto G00 and F00 moves, although it may also be used in G01 moves.

Bell shape:

This type may be used on all kinds of movements, G00, G01, G02, etc. and with any type of feedrate F.

6.4.4.1 CALCULATION OF FEED-FORWARD GAIN

The Feed-Forward gain is proportional to the feedrate and is set by machine parameters P732, P733, P734, P735, P736 which indicate the % of analog voltage that is due to the programmed feedrate.



The value added to the following error is (Kf x F/6) where Kf is the value of Feed-Forward and F is the programmed feedrate.

The CNC will apply the proportional gain (K1 and K2) to the value resulting from adding the following error of the machine plus the value selected by the Feed-Forward.

When the result of the addition is smaller than the value of the gain break-point, the CNC will apply the formula:

Analog = K1 x [Following Error + (Kf x F/6)]

And when the result of the addition is greater than the value of the gain break-point, the CNC will apply the formula:

Analog = (K1 x Ep) + {K2 x [Following Error + (Kf x F/6) - Ep]}

Where “Ep” is the value of the gain break-point.

6.4.5 LEADSCREW ERROR COMPENSATION

With this CNC it is possible to compensate for leadscrew error as well as for its backlash when reversing movement direction.

There are 4 leadscrew compensation tables with 30 points each or 2 tables of 60 points each depending on the setting of P613(6) as shown below:

	X axis	Y axis	Z axis	W axis
P613(6)=0	P0 to P59	P60 to P119	P120 to P179	P180 to P239
P613(6)=1	P0 to P119	P120 to P239	--------	--------

To lock or unlock access to machine parameters, decoded "M" function table and to the leadscrew error compensation tables, proceed as follows:

* Press the **[OP MODE]** key.
* Press **[6]** to select the Editing mode.
* Press the softkey for **[LOCK/UNLOCK]**. The screen will show the word: "CODE:" (password).
* Key in "**PKJIY**" and press **[ENTER]** to lock the access or key in "**PKJIN**" and press **[ENTER]** to unlock the access.

When access to machine parameters is locked, **only** those regarding serial line communications via RS232C may be changed.

CAUTION when using a CNC with an integrated PLC (CNC+PLCI)

When using this access locking code, the machine parameters, the decoded "M" function table and the leadscrew error compensation tables are stored in EEPROM memory.

When using the access unlocking code, it recovers these previously stored tables from the EEPROM memory.

Therefore, **one must be careful and lock these tables before unlocking them.** Otherwise, the factory set values or other prelocked values, may be restored overwriting the ones the manufacturer entered but did not lock.

To access the leadscrew error compensation table, press the following keystroke sequence:

[OP MODE]	Display of the various operating modes.
[9]	Access to special modes.
[3]	Access leadscrew error compensation tables.

The operator may view the following or previous pages by using the up and down arrow keys.

To view a particular parameter, key in its number and press **[RECALL]**. The CNC will show the page corresponding to that parameter.

To clear the table by setting all the parameters to 0, key in the following sequence: **[K] [J] [I] [ENTER]**.

Each parameter pair of this table represents:

Even parameter The position of the error point on the leadscrew. This position is referred to Machine Reference Zero (home).

Value range: ±8388.607 millimeters
±330.2599 inches

Odd parameter The amount of leadscrew error at that point.

Value range: ±32.766 millimeters
±1.2900 inches

When defining the compensation points on the table, the following rules must be observed:

* The even parameters are ordered according to their position along the axis. The first pair of parameters (P0 or P60) must be set for the most negative (least positive) point of the axis to be compensated.

* If all 30 points of the table are not required, set the unused ones to 0.

* For those sections outside the compensation area, the CNC will apply the compensation defined for the nearest point.

* The Machine Reference Zero point (home) must be set with an error of 0.

* The maximum difference between the error values of two consecutive compensation points must be within: ±0.127 mm (±0.0050 inches)

* The inclination of the error graph between two consecutive points cannot be greater than 3%.

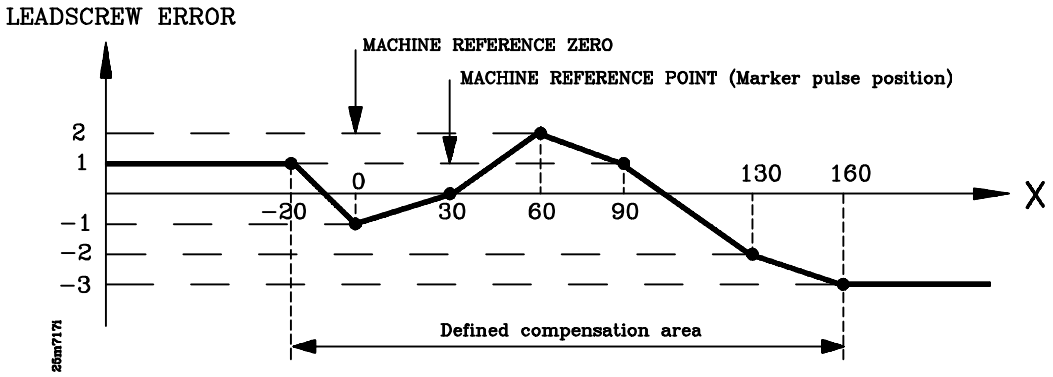
  Examples: If the distance between two consecutive points is 3 mm. the maximum difference of their relevant error values can be 0.090 mm.
  If the error difference between two consecutive points is the maximum (0.127mm), the distance between them cannot be smaller than 4.233mm.

To EDIT a parameter, key in its number, press **[=]**. Then, key in the desired value or jog the corresponding axis to the desired position and press **[ENTER]**.

Remember to press **[RESET]** or power the CNC off and back on once the machine parameters have been set in order for the CNC to assume their new values.

Programming example:

An X axis leadscrew is to be compensated according to the following graph in the section between X-20 and X160:



Considering that the machine reference **point** has a value of X30 (meaning that it is located 30mm from the Machine Reference **Zero**), the leadscrew error compensation parameters will be defined as follows:

P000	= X -20.000	P001	= X 0.001
P002	= X 0.000	P003	= X -0.001
P004	= X 30.000	P005	= X 0.000
P006	= X 60.000	P007	= X 0.002
P008	= X 90.000	P009	= X 0.001
P010	= X 130.000	P011	= X -0.002
P012	= X 160.000	P013	= X -0.003
P014	= X 0.000	P015	= X 0.000
P016	= X 0.000	P017	= X 0.000
"	"	"	"
"	"	"	"
P056	= X 0.000	P057	= X 0.000
P058	= X 0.000	P059	= X 0.000

6.5 REFERENCE SYSTEMS

6.5.1 REFERENCE POINTS

A CNC machine must have the following reference points established:

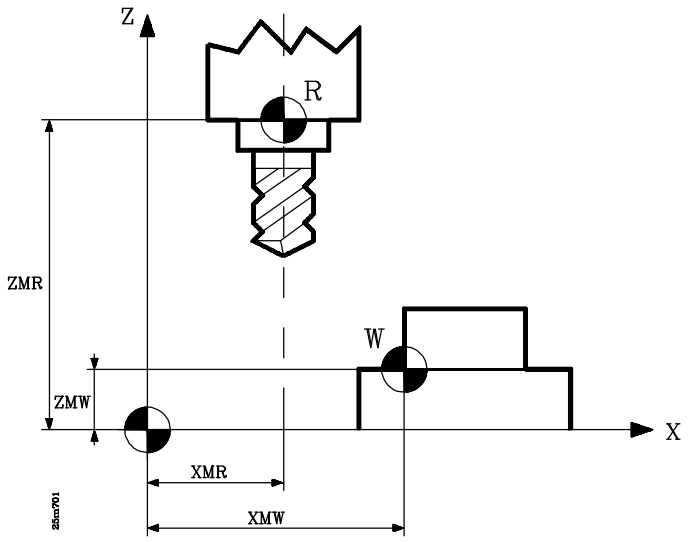
* **Machine Reference Zero** or origin point of the machine. It is set by the machine manufacturer as the origin of the coordinate system of the machine.

* **Part Zero** or origin point for the part. It is the origin point set to program the measurements of the part. It can be chosen freely by the programmer and its reference to the machine reference zero is set by means of a zero offset.

* **Machine Reference Point**. It is a reference point on the machine set by the manufacturer.

    * When the feedback system **<u>uses</u> coded Io**, this point is used **ONLY** when leadscrew error compensation is to be applied onto the axis. The amount of leadscrew error to be assigned to this point must be "0".

    * When the feedback system **<u>does not</u> use coded Io**, the CNC utilizes this point as home to synchronize the whole machine coordinate system and to apply leadscrew error compensation.



M	Machine Reference Zero
W	Part Zero
R	Machine Reference Point
XMW, YMW, ZMW, etc.	Part Zero coordinates
XMR, YMR, ZMR, etc.	Machine Reference Point coordinates

6.5.2 MACHINE REFERENCE (HOME) SEARCH

With this CNC it is possible to search home in Jog mode or by program.

Although this home search may be programmed for several axes in the same block, the CNC will home the axes one at a time in the programmed order.

Each axis is homed as follows:

The axis starts moving in the direction established by its corresponding machine parameter P623(8), P623(7), P623(6), P623(5), P623(4).

If the feedback device **does not** use coded Io:

This initial movement is carried out at the feedrate set by its corresponding machine parameter P112, P212, P312, P412, P512 until its corresponding home switch is pressed: P602(4), P602(3), P602(2), P602(1), P617(2).

Then, it will look for the marker pulse (Io) at the feedrate indicated by machine parameters P810, P811, P812, P813, P814 and, when detected, it will consider the home search concluded

If the feedback device **uses** coded Io:

This homing movement, a maximum of 20 mm or 100 mm (depending on scale model), is carried out at the feedrate set by its corresponding machine parameter P810, P811, P812, P813, P814 until its corresponding coded Io is detected.

If this **home search** is done **in JOG** mode, the previously selected **part zero** (zero offset) will be **cancelled** and the CNC will display the coordinates (position values) of the machine reference point indicated by machine parameters P119, P219, P319, P419, P519.

When the **home search** is **not** done **in JOG** mode, the previously selected **part zero** (or zero offset) will be **maintained**. Therefore, the displayed home position will be referred to that part zero (or zero offset).

6.5.2.1 HOME SEARCH ON GANTRY AXES

On GANTRY axes, one of them is the main axis and the other one the slave.

When homing them, they behave as one single axis. The CNC moves both at the same time; but only takes into account the feedback pulses from the main axis, ignoring those from the slave axis.

The home search on GANTRY axes may be carried out in JOG mode or by program as follows:

The axes start moving in the direction established by its corresponding machine parameter P623(8), P623(7), P623(6).

If the feedback device of the main axis **does not** use coded Io:

This initial movement is carried out at the feedrate set by its corresponding machine parameter P112, P212, P312 until its corresponding home switch is pressed: P602(4), P602(3), P602(2).

Then, it will look for the marker pulse (Io) at the feedrate indicated by machine parameters P810, P811, P812 and, when detected, it will consider the home search concluded

If the feedback device of the main axis **uses** coded Io:

This homing movement, a maximum of 20 mm or 100 mm (depending on scale model), is carried out at the feedrate set by its corresponding machine parameter P810, P811, P812 until its corresponding coded Io is detected.

If this **home search** is done **in JOG** mode, the previously selected **part zero** (zero offset) will be **cancelled** and the CNC will display the coordinates (position values) of the machine reference point indicated by machine parameters P119, P219, P319.

When the **home search** is **not** done **in JOG** mode, the previously selected **part zero** (or zero offset) will be **maintained**. Therefore, the displayed home position will be referred to that part zero (or zero offset).

6.5.3 ADJUSTMENT ON AXIS WITHOUT CODED Io

6.5.3.1 MACHINE REFERENCE POINT (HOME) ADJUSTMENT

The adjustment of the machine reference point (home) must be done one at a time. the recommended adjusting procedure is the following:

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

* Indicate in the corresponding Axis Machine Parameter the type of home marker pulse (machine reference pulse: Io) of the feedback system being used. Parameters P600 bits 8, 7, 6, 5 and parameter P617 bit 1.

* Set the corresponding machine parameter to indicate the direction of the home search. Parameter P623 bits 8, 7, 6, 5, 4.

* Set the machine parameters that determine the home searching feedrate until the home switch is pressed (P112, P212, P312, P412, P512) and the one until the marker pulse is detected (P810, P811, P812, P813, P814).

* The machine reference point will be assigned a value of "0". Parameter P119, P219, P319, P419, P519.

* 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 in this JOG mode. When the search is completed, the CNC will assign a "0" position value to this point (Machine Reference **Point**).

* If the desired Machine Reference **Zero** is other than the marker pulse position found earlier, move the axis to that point or to another one whose position with respect to machine reference zero is known. The CNC will display the distance between this point and the Machine Reference **Point** (marker pulse position).

  This is the distance value to be assigned to the machine parameter corresponding to the machine reference point. Parameter P119, P219, P319, P419, P519.

  P*19 = Machine coordinate of that point - CNC reading at that point.

  Example for the X axis: If the point of known coordinates is 230mm from machine zero and the CNC displays -123.5 mm, the machine reference **point** (marker pulse position) will be **353.5 mm** from the machine reference zero (arbitrary reference position or home).

  Machine parameter P119 = 230 - (-123.5) = **353.5 mm.**

* After assigning this new value to the machine parameter (P119 for the X axis), press RESET or turn the CNC off and back on in order for the CNC to assume the new values.

* The axis must be homed again in order for it to assume the correct reference values.

6.5.3.2 CONSIDERATIONS

* If at the instant the home search is initiated, the home switch is pressed, the axis will withdraw [in the opposite direction to that set by P623(8), P623(7), P623(6), P623(5), P623(4)] until releasing the home switch before starting the actual home search.

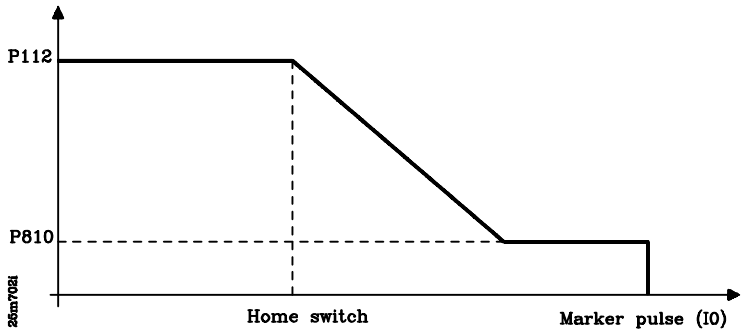
* If the axis is out of the soft travel limits (set by P107-P108, P207-P208, P307-P308, P407-P408, P507-P508), it has to be jogged into the work area (within limits) and, then, positioned at the correct side from home before starting the actual home search.

* Care must be taken when placing the home switch and when setting the home searching feedrates (P112, P212, P312, P412. P512, P810, P811, P812, P813, P814) to prevent any overshooting.

* If the selected axis does not have a home switch [P602(4), P602(3), P602(2), P602(1), P617(2)], the CNC will consider it to be pressed and it will only make the marker pulse searching move at the feedrate set by P810, P811, P812, P813, P814 until the marker pulse (Io) from the feedback device is detected; thus completing the home search.

* FAGOR linear transducers (scales) have a negative marker pulse (Io) every 50 mm ["P600(8), P600(7), P600(6), P600(5), P617(1)" = 0] and FAGOR rotary encoders output a positive marker pulse (Io) ["P600(8), P600(7), P600(6), P600(5), P617(1)" = 1] per revolution.

* The home switch will be mounted in such a way that the marker pulse “Io” is always found in the area corresponding to the second home searching feedrate (set by P810, P811, P812, P813, P814).



If there is no room for that, the first home searching feedrate (set by P112, P212, P312, P412, P512) must be reduced. This might be the case with those rotary encoders where the marker pulses are very close to each other.

6.5.4 ADJUSTMENT ON AXIS WITH CODED Io

6.5.4.1 SCALE OFFSET ADJUSTMENT

The offsets of the feedback scales must be adjusted one axis at a time. We recommend the following procedure

* Indicate by the corresponding machine parameter, the type of Io supplied by the feedback device. Parameters P600(8), P600(7), P600(6), P600(5) and P617(1).

* Indicate, also, the homing direction for each axis. Parameters P623(8), P623(7), P623(6), P623(5), P623(4).

* Indicate the homing feedrate for each axis. Parameters P810, P811, P812, P813, P814.

* Set the machine parameters for amount of OFFSET to "0". Parameters P919, P920, P921, P922, P923.

* Select the JOG mode at the CNC and home each axis. The CNC will then display the distance from the ABSOLUTE SCALE ZERO point.

* After moving the axis to MACHINE ZERO point or to another known position, take note of the position value displayed by the CNC and apply the following formula:

  Value = (CNC reading at that point) - (Machine coordinate of that point).

  Example for the X axis: If the known position is at 230 mm from machine reference zero and the CNC reads (displays) 423.5 mm, the offset of the scale will be:

  Machine parameter P919 = 423.5 -230 = 193.5 mm.

* Do this to the other axis and, after setting these scale offset parameters to their new calculated values, press **[RESET]** or turn the CNC off and back on so it assumes the new offset values.

* **All the axes must be homed <u>again</u> in order for the correct values to be assumed.**

6.5.4.2 CONSIDERATIONS

* If the axis is positioned beyond the software limits (set by machine parameters P107-P108, P207-P208, P307-P308, P407-P408, P507-508), the axis must be jogged into the work area before attempting to home it.

* When using semi-absolute linear scales (with coded Io), no home limit-switches are required.
  However, the home switch may be used as end-of-travel limit switch during the homing operation.
  If while homing the axis, this home switch is pressed, the CNC will reverse the axis and it will change the homing direction.

* Fagor semi-absolute scales provide a negative coded Io. Parameters "P600(8), P600(7), P600(6), P600(5), P617(1)" = 0.

6.5.5 SOFTWARE TRAVEL LIMITS FOR THE AXES

Once the 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 the machine parameter corresponding to the positive software travel limit. Parameter P107, P207, P307, P407, P507.

* Repeat those steps in the negative direction assigning the displayed value to machine parameter corresponding to the negative software travel limit. Parameter P108, P208, P308, P408, P508.

* Once this process is completed, press RESET or turn the CNC off and back on in order for the new values to be assumed by the CNC.

6.6 UNIDIRECTIONAL APPROACH

In order to improve repeatability in rapid moves (G00) of axes with leadscrew backlash, the CNC offers a series of machine parameters so the axis is always positioned in the same direction. These parameters are:

P608(4), P608(3), P608(2), P608(1)

The indicate whether the rapid moves (G00) of the corresponding axis are to use unidirectional approach or not.

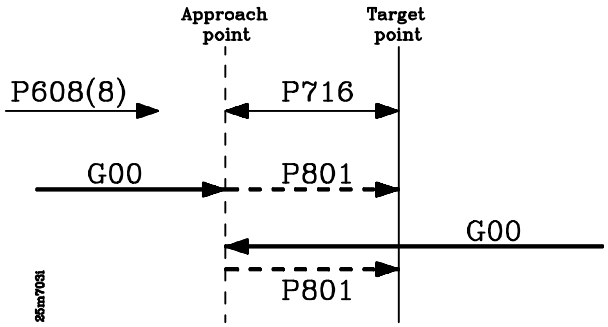
P608(8), P608(7), P608(6), P608(5)

They indicate the direction of the unidirectional approach for the corresponding axis.

P716 Indicates the distance from the approach point to the programmed point.

P801 Indicates the unidirectional approach feedrate from the approach point to the programmed position.

The CNC will calculate the approach point based on the approach distance from the programmed position value and the direction of the approach.



The positioning is done in two stages:

* Rapid move (G00) to the calculated approach point. If the movement is carried out in the direction opposite to that indicated by P608(8), P608(7), P608(6), P608(5), the axis will overshoot the programmed point (destination).

* Positioning move at the feedrate established by P801 from this point (approach) to the programmed destination point.

6.7 AUXILIARY FUNCTIONS M, S, T

M function

This CNC offers up to 100 M functions (M00 thru M99).

A program block may contain up to 7 M functions which will be sent out to the electrical cabinet every time the block is executed.

The CNC sends out to the electrical cabinet the number of the executed M function via pins 20 thru 27 of connector I/O1. Machine parameter P617(8) determines whether this value is sent out in BCD or binary code.

Also, it must be borne in mind that the CNC has an internal "decoded M" table. The way to operate with this table is described later in this chapter.

Every time an M function of the decoded-M table is executed, the CNC updates its corresponding outputs at connector I/O2.

Machine parameter P609(5) determines whether the CNC also outputs their corresponding BCD or binary coded number via pins 20 thru 27 of connector I / O1.

S function

Only to be used when the spindle speed output is in BCD (not analog). Machine parameter P601(3)=1.

Whenever a block containing a new spindle speed "S" is executed, the CNC will output the corresponding BCD code via pins 20 thru 27 of connector I/O 1.

T function

Whenever a block containing a new tool "T" is executed, the CNC will output the corresponding BCD code via pins 20 thru 27 of connector I/O 1.

This CNC also offers a tool table where the length and radius of each tool can be defined. The CNC will take these dimensions into account when machining with tool compensation (G41, G42, G43).

The way to operate with this table is described in the Operating manual. To access this table, just press the following keystroke sequence:

[OP MODE] Show the various operating modes.
[8] Access the tool table.

6.7.1 DECODED M FUNCTION TABLE

To lock or unlock access to machine parameters, decoded "M" function table and to the leadscrew error compensation tables, proceed as follows:

* Press the **[OP MODE]** key.
* Press **[6]** to select the Editing mode.
* Press the softkey for **[LOCK/UNLOCK]**. The screen will show the word: "CODE:" (password).
* Key in "**PKJIY**" and press **[ENTER]** to lock the access or key in "**PKJIN**" and press **[ENTER]** to unlock the access.

When access to machine parameters is locked, **only** those regarding serial line communications via RS232C may be changed.

CAUTION when using a CNC with an integrated PLC (CNC+PLCI)

When using this access locking code, the machine parameters, the decoded "M" function table and the leadscrew error compensation tables are stored in EEPROM memory.

When using the access unlocking code, it recovers these previously stored tables from the EEPROM memory.Therefore, **one must be careful and lock these tables before unlocking them.** Otherwise, the factory set values or other prelocked values, may be restored overwriting the ones the manufacturer entered but did not lock.

To access this table, press the following keystroke sequence:

[OP MODE]	Show the various operating modes.
[9]	Access the special modes.
[2]	Access the decoded-M function table

The CNC shows the following information for each of the M functions set in this table:

M41	100100100100100	(outputs to be activated)
	00100100100100100	(outputs to be deactivated)

* The number of the M function already set. Mxx indicates that this position is free and any other M function may be set.

* The first row has 15 characters. Each one corresponds to a decoded-M output of connector I/O2 and their value (0 or 1) indicates the following:

  0 This M function must **not activate** its decoded output at connector I/O2.
  1 This M function **must activate** its decoded output at connector I/O2.

* The second row has 17 characters. The first 15 (from left to right) correspond to the decoded-M output of I/O2 and their value (0 or 1) indicates the following:

  0 This M function must **not deactivate** its decoded output at connector I/O2.
  1 This M function **must deactivate** its decoded output at connector I/O2.

  Bit 16 determines whether the M function is executed at the beginning (if 0) or at the end of the block (if 1) where it is programmed. If this bit is set to "1" and the programmed block contains a movement of axes, the M function will be executed once the axes have reached their target position.

Bit 17 of the bottom row determines whether the CNC waits (0) or not (1) for the M-done signal from the electrical cabinet after the CNC has sent it out. See the following section for its description.

For example: If the table corresponding to function M41 has been set as follows:

M41	100100100100100	(outputs to be activated)
	00100100100100100	(outputs to be deactivated)

The CNC will behave as follows whenever a block containing function M41 is executed:

* First, before making any move, it sends out the following information:

	M01	M02	M03	M04	M05	M06	M07	M08	M09	M10	M11	M12	M13	M14	M15
Pin I/O2	3	4	5	6	7	8	9	10	11	12	13	25	24	23	22
at 24V	x			x			x			x			x		
at 0V			x			x			x			x			x
Not modified		x			x			x			x			x	

* Once this information is sent out, the CNC waits for the electrical cabinet to set the M-done signal high (24V at pin 15 of connector I/O 1).

6.7.2 M, S, T FUNCTION TRANSFER

Every time a block is executed, the CNC transfers to the electrical cabinet information of the M, S, T functions activated in it.

First, the M function information is transferred, then, that of the S functions and, then, that of the T functions.

M function:

The CNC analyzes the M functions programmed in the block and, depending on how they are defined, they will be output before or after moving the axes.

When sending more than one M function, the CNC will send them one by one starting from the first one in the block and, once it has been sent out, it will output the next one and so forth.

When sending the auxiliary M functions to the electrical cabinet, the CNC uses the BCD outputs (pins 20 thru 27 of connector I/O 1) and it activates the “M Strobe” output to “tell” the electrical cabinet to execute them.

Depending on how they are defined on the table, the CNC must wait or not for the “M-DONE” signal to consider it executed.

If an M function which is not defined on the M function table is executed, the CNC will wait for the “M-DONE” signal before resuming the execution of the program.

S function transfer:

When a new spindle speed (S) is programmed, the CNC issues the corresponding BCD code (via pins 20 thru 27 of connector I/O 1) and it activates the “S Strobe” output to “tell” the electrical cabinet to execute it.

The CNC will wait for the “M-DONE” signal to consider it executed.

If the new selected “S” involves a range change, the CNC executes first the M functions corresponding to the range change and then, it transfers the new selected spindle speed.

T function transfer:

When selecting a new tool “T”, the CNC issues the corresponding BCD code (via pins 20 thru 27 of I/O 1) and it will activate the “T Strobe” to “tell” the electrical cabinet to execute it.

The CNC will wait for the “M-DONE” signal to consider it executed.

6.7.3 M, S, T FUNCTION TRANSFER USING THE M-DONE SIGNAL

When parameter P605(5) is set to “0”, the CNC maintains the BCD outputs and the corresponding Strobe signal (M, S, T) active for 100 milliseconds.

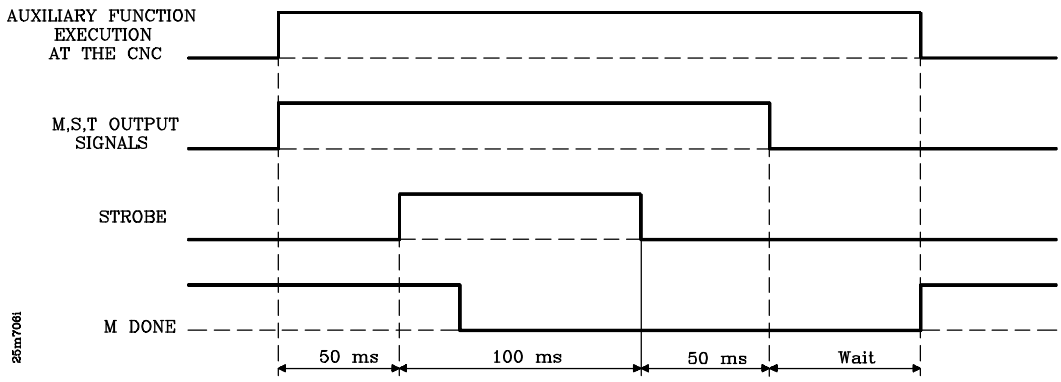
When the electrical cabinet has a device requiring the BCD signals to be active for a longer time, machine parameter P605(5) must be set to “1”.

In each case, the CNC acts as follows:

“P605(5)=0”

1.- The CNC transfers the BCD value of the selected function via pins 20 thru 27 of connector I/O 1.

50 milliseconds later, the “M Strobe” output is activated to “tell” the electrical cabinet to execute the M function.



2.- When the electrical cabinet detects the activation of the “M Strobe” signal, it must start the execution of the corresponding function either activating or not the M-done signal.

3.- The CNC will maintain the “M Strobe” signal for 100 milliseconds and the BCD signals for another 50 milliseconds.

4.- If in the decoded M function table, bit 17 of the bottom row has been set to "0", the CNC will wait for the M-done signal from the electrical cabinet to be set high (24V) indicating that the execution of that function has concluded.

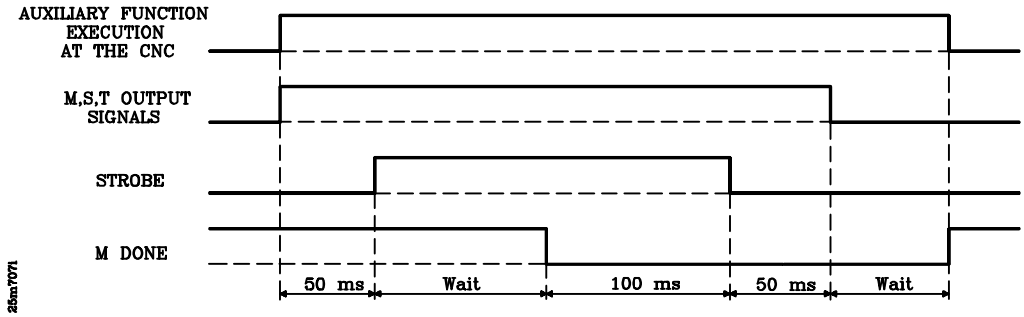
If in the decoded M function table, bit 17 of the bottom row has been set to "1"or the M-done signal has not been set low in point 2, the CNC will consider the M function executed right after the corresponding BCD signals disappear (without waiting).

“P605(5)=1”

This type of transfer is used when the electrical cabinet has a device which requires the BCD outputs from the CNC to be active for a longer period of time.

1.- The CNC sends the BCD value of the selected function via pins 20 thru 27 of connector I/O 1.

50 milliseconds later, it activates the corresponding Strobe output to “tell” the electrical cabinet to execute the required auxiliary function.



2.- When the electrical cabinet detects the activation of one of the Strobe signals, it must begin the execution of the corresponding function deactivating the M-DONE signal to let the CNC know that this M function execution has begun.

3.- The CNC will maintain the Strobe signal for another 100 milliseconds and the BCD outputs for another 150 milliseconds.

4.- If in the decoded M function table, bit 17 of the bottom row has been set to "0", the CNC will wait for the M-done signal from the electrical cabinet to be set high (24V) indicating that the execution of that function has concluded.

If in the decoded M function table, bit 17 of the bottom row has been set to "1"or the M-done signal has not been set low in point 2, the CNC will consider the M function executed right after the corresponding BCD signals disappear (without waiting).

6.8 SPINDLE

Depending on the setting of machine parameters P601(3) and P601(2), the CNC provides one of the following spindle speed outputs:

* Analog voltage (±10V) via pins 36 and 37 of connector I/O1.
* 2-digit BCD coded output via pins 20 thru 27 of connector I/O1.
* 4-digit BCD coded output via pins 20 thru 27 of connector I/O1.

Analog voltage

To use the CNC's analog voltage for the spindle drive, set P601(3) and P601(2) to "0".

The CNC will generate the analog voltage corresponding to the programmed spindle speed within ±10V.

When a unipolar analog voltage is desired (either 0 to +10V or 0 to -10V), machine parameter P610(4) must be set to "1". The sign of this analog voltage will be set by machine parameter P601(4).

When the machine has an automatic spindle range changer, machine parameter P601(1) must be set to "1". Then, whenever a new spindle speed is selected which involves a range change, the CNC will automatically generate the M function associated with the new spindle speed range M41, M42, M43 or M44.

BCD Output

When desiring a BCD coded output for spindle speed control, machine parameters P601(3) and P601(2) must be set as follows:

For 2-digit BCD outputP601(3)=1 and P601(2)=0
For 4-digit BCD outputP601(3)=0 and P601(2)=1

The CNC will issue the code corresponding to the programmed spindle speed at the BCD outputs (pins 20 thru 27 of I/O 1).

It will also activate the "S Strobe" output to indicate to the electrical cabinet that the required auxiliary function must be executed and it will wait for the "M-DONE" signal from the electrical cabinet in order to consider the data transfer has concluded.

When using a 2-digit BCD code, P601(3)=1 and P601(2)=0, The CNC will indicate the selected spindle speed according to the following conversion table:

Programmed S	S BCD	Programmed 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		

When a value greater than 9999 is programmed, the CNC will indicate the spindle speed code corresponding to 9999.

Example:

When selecting a value of S800, the CNC will issue the BCD code for S78:

	MST80	MST40	MST20	MST10	MST08	MST04	MST02	MST01
Pin	20	21	22	23	24	25	26	27
Value	0	1	1	1	1	0	0	0

When using a 4-digit BCD code, P601(3)=0 and P601(2)=1, the CNC will issue the code corresponding to the programmed S speed in two stages with a 100msec. delay between them.

It will also activate the “S STROBE” signal on each stage and it will wait for the “M-DONE” signal from the electrical cabinet at each stage.

The first stage will issue the values corresponding to the Thousands and Hundreds, and the second stage the ones corresponding to the Tens and Units. The pins corresponding to each one of them are the following:

Pin	1st stage	2nd stage
20 21 22 23	Thousands	Tens
24 25 26 27	Hundreds	Units

Example:

When selecting a value of S 1234, the CNC will show:

PIN	2 Digits (value S81)	4 digits 1st stage	4 digits 2nd stage
20 (MST80) 21 (MST40) 22 (MST20) 23 (MST10)	1 0 0 0	0 0 0 1	0 0 1 1
24 (MST08) 25 (MST04) 26 (MST02) 27 (MST01)	0 0 0 1	0 0 1 0	0 1 0 0

6.9 SPINDLE SPEED RANGE CHANGE

With this CNC, the machine can have a gear box in order to adapt the speeds and torques of the spindle motor to the various machining requirements.

Up to 4 spindle ranges may be set by means of machine parameters P7, P8, P9 and P10 specifying the maximum spindle rpm value for each one of them.

The value assigned to P7 must correspond to the lowest range (RANGE 1) and the one assigned to P10 to the highest range (RANGE 4).

When not using all 4 ranges, start the speed assignment from the lowest range up and set the unused ranges to the highest speed being used.

When the new spindle speed selected requires a range change, the CNC will execute the auxiliary M function corresponding to the new range.

The CNC uses the auxiliary functions: M41, M42, M43 and M44 to indicate to the electrical cabinet which range must be selected: RANGE 1, RANGE 2, RANGE 3 or RANGE 4).

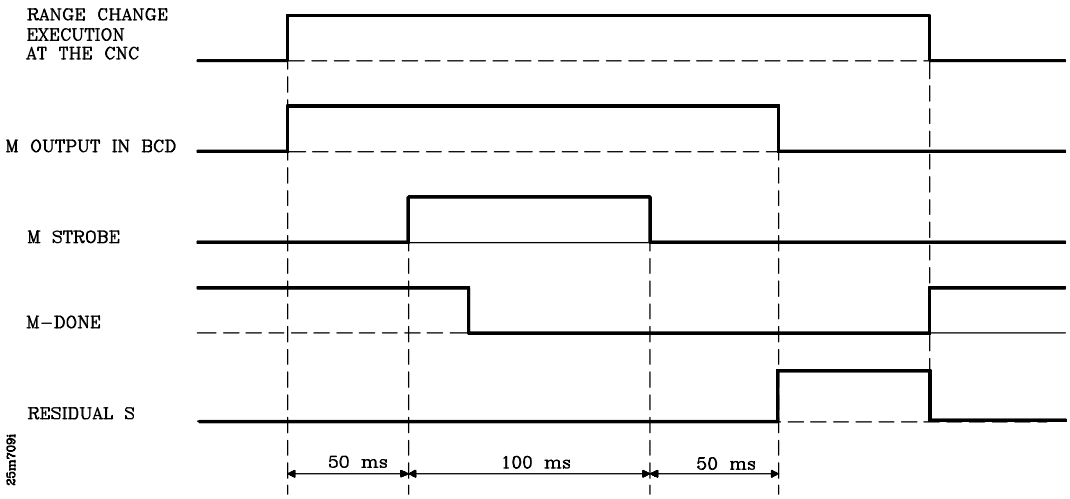
Also, in order to facilitate the range change, the CNC offers the possibility to use a residual analog output during a range change. Machine parameter for the spindle: P601(6).

The value of this residual analog voltage is defined by machine parameter P706 and the oscillation period for this residual analog voltage is set by machine parameter P707.

The automatic range change is carried out as follows:

1.- Once the range change is detected, the CNC outputs the BCD value of the corresponding M function: M41, M42, M43 or M44, via pins 20 thru 27 of connector I/O 1.

50 milliseconds later, it activates the “M Strobe” output to indicate to the electrical cabinet to execute the required M function. This signal is maintained active for 100 milliseconds.



2.- When the electrical cabinet detects the “M Strobe” signal, must deactivate the “M-DONE” input of the CNC to “tell” it that the execution of the corresponding M function has begun.

3.- The electrical cabinet will execute the required M function using the BCD outputs of the CNC (pins 20 thru 27 of connector I/O 1).

4.- After keeping the BCD outputs active for 200 milliseconds, the CNC will output the residual analog output indicated by parameter P706 if so established by parameter P601(6).

The oscillation period for this residual analog voltage is determined by machine parameter P707.

5.- Once the range change is completed, the electrical cabinet must activate the M-DONE input of the CNC to “tell” it that the requested M function has been executed.

Atention:

When the electrical cabinet has some device needing the BCD and “M Strobe” signals from the CNC active for a longer period of time, machine parameter P605(5) must be set “1” (the CNC waits for the down flank of the M-Done signal).

6.10 SPINDLE CONTROL

It is necessary to install an encoder on the spindle in order to perform the following operations:

* Electronic threading (G33).
* Spindle orientation (M19).
* Rigid tapping cycle (G84R).

The machine parameters to be set are:

P800	Number of pulses (line count) of the spindle encoder.
P609(2)	Counting direction of the spindle.

Also, to work with "spindle orientation (M19)", the following parameters must be set:

P700	Spindle speed "S" when working in M19.
P601(7)	Sign of the analog output associated with M19.
P612(8)	Type of spindle encoder reference (marker) pulse.
P619(6)	Spindle orient in both directions.
P719	Minimum spindle analog output in M19.
P717	In position zone for the spindle in M19.
P718	Spindle proportional gain in M19.
P917	Lower limit of forbidden zone for the spindle in M19.
P918	Upper limit of forbidden zone for the spindle in M19
P916	Spindle orient position when executing M19 without S.

To switch from open loop (M03, M04) to closed loop, M19 must be executed. The CNC slows down the spindle speed below the value indicated by parameter P700. Then, it carries out a home search on the spindle and it orients it to the position indicated by parameter P916.

If "M19 S" was executed, the CNC will orient the spindle to the indicated "S" position once it is homed.

The programming format for spindle orientation is: “M19 S4.3”, where:

M19	Indicates that it is a spindle positioning (orient) move in closed loop.
S4.3	Indicates the desired spindle position value. This value is expressed in degrees and referred to machine reference zero.

An “M19 S4.3”-type block is executed as follows:

* The CNC indicates to the electrical cabinet the execution of the M19 functions just as with any other M function.

* Then, the CNC orients the spindle to the indicated point at the speed set by machine parameter P700.

  The spindle orienting direction is determined by machine parameter P601(7); however, with machine parameter P619(6), it is possible to do it in both directions.

When programming function M19 by itself (without indicating a specific position), the spindle will orient to the position set by machine parameter P916.
Example:

S1000 M3	Spindle in open loop turning clockwise.
M19	Spindle in closed loop, home search and orient to "P916" position.
M19 S100	Orient to 100°
S1000	Spindle in open loop maintaining previous turning direction (M03).
M19 S200	Spindle in closed loop, home search and orient to 200°.

6.11 TOOLS AND TOOL MAGAZINE

The machine parameters related to tools and tool magazine are:

P701	Number of tool pockets in the magazine.
P743	Subroutine associated with the T function.
P625(4)	The associated subroutine is executed before the T function.
P626(1)	The CNC displays the tool tip position.
P601(5)	Machining center.
P601(1)	RANDOM tool magazine.
P709	Subroutine associated with the M06 function.
P618(2)	M06 before or after associated subroutine.
P601(8)	M06 interrupts program execution.
P702	First axis to move when executing M06.
P703	Second axis to move when executing M06.
P704	Third axis to move when executing M06.
P705	Fourth axis to move when executing M06.
P900	Position to move the first axis when executing M06.
P901	Position to move the second axis when executing M06.
P902	Position to move the third axis when executing M06.
P903	Position to move the fourth axis when executing M06.
P621(7)	M06 does not execute M19.
P615(8)	In M06, function M19 is executed while the axis is moving.
P603(2)	Special M06 sequence.

6.11.1 MACHINE WITHOUT TOOL MAGAZINE

When the machine does not have a tool magazine, set the following parameters accordingly:

P601(1)=0	It is not a RANDOM tool magazine.
P601(5)=0	It is not a machining center.

The following parameters can also be used:

P701 Indicates the highest tool number to be selected at the CNC.

In this case, since there is no tool magazine, it is recommended to set it to the largest value “P701=98”, thus avoiding errors while in execution.

P743 Indicates the number of the standard subroutine to be executed when the block contains a T function.

If set to "0", no subroutine will be executed.

6.11.2 NOT-RANDOM TOOL MAGAZINE

When the machine has a tool magazine (machining center), this magazine may be RANDOM or NOT-RANDOM.

A tool magazine is called NOT-RANDOM when each tool always occupies the same position (tool pocket) in it. Therefore, the tool number usually matches its position (pocket) number.

The following parameters must be set accordingly:

P601(5)=1 It is a machining center.
P601(1)=0 The tool magazine is not RANDOM

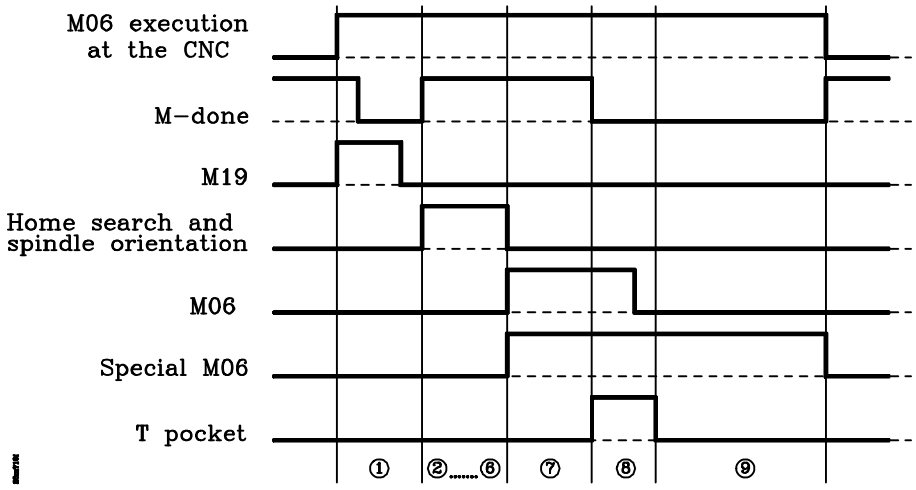
Every time a tool change is carried out, a T2.2 command must be executed to indicate the tool to be selected and then execute the M06 to actually change the tool.

T2.2 The figure located to the left of the "." indicates the tool number and the figure to the right indicates the tool offset to be applied in the machining operations.

The CNC outputs to the electrical cabinet the tool number (which is the same as the tool pocket number, in this case). This tool information transfer is done as described in the section on "M, S, T FUNCTION TRANSFER", via the BCD outputs (pins 20 thru 27 of connector I/O1) and the "T Strobe" output.

If a subroutine associated with the T function has been set (machine parameter P743), it will be executed after the T function has been transferred to the electrical cabinet.

M06 The tool changing sequence when executing this function consists of the following steps:



1.- The CNC "tells" the electrical cabinet to execute the M19 (spindle in closed loop).

If there is no spindle encoder or spindle orientation is not to be used, set machine parameter P621(7) to "1".

2.- Once the electrical cabinet has executed function M19, (indicated by activating the "M-DONE" signal), the spindle is homed.

3.- The spindle orients to the position set by machine parameter P916 and the first axis (P702) moves to the position indicated by machine parameter P900.

If so desired, it is possible to have the first axis moving and the spindle homing at the same time by setting machine parameter P615(8) to "1".

4.- Once both the spindle and the first axis are in position, the second axis (P703) moves to the point indicated by machine parameter P901.

If P703=0, this move will not take place and this operation will continue on paragraph 7.

5.- Once the second axis is in position, the third axis (P704) moves to the point indicated by machine parameter P902.

If P704=0, this move will not take place and this operation will continue on paragraph 7.

6.- Once the third axis is in position, the fourth axis (P705) moves to the point indicated by machine parameter P903.

If P705=0, this move will not take place.

7.- Once the last axis is in position, the CNC sends out function M06 for the electrical cabinet to carry out the tool change.

When function M06 has an associated subroutine (P709), machine parameter P618(2) may be set so it is executed before the M06 is sent out to the electrical cabinet.

If the tool change requires a special treatment (machine parameter P603(2)=1), the CNC will activate the SPECIAL M06 output (pin 13 of connector I/O2).

8.- Once the new tool is placed at the spindle (indicated by bringing the M-DONE signal low), the CNC will keep showing to the electrical cabinet, for 100 milliseconds, the tool pocket number where the old tool must be deposited.

9.- Once the old tool is deposited in the indicated pocket, the electrical cabinet must set the M-DONE signal high.

The CNC will consider the tool change function completed and it will deactivate the SPECIAL M06 output (pin 13 of connector I/O2) if it was on according to the setting of machine parameter P603(2).

When the M06 function has an associated subroutine (P709), it is possible to set machine parameter P618(2) so it is executed after sending the M06 to the electrical cabinet (after the old tool has been returned to its pocket).

If the M06 interrupts the execution of the program [P601(8)], it will be necessary to press CYCLE START to resume it.

6.11.3 RANDOM TOOL MAGAZINE

When the machine has a tool magazine (machining center), this magazine may be RANDOM or NOT-RANDOM.

A tool magazine is called RANDOM when the tools can occupy any position (pocket).

The following parameters must be set accordingly:

P601(5)=1 It is a machining center
P601(1)=1 It is a RANDOM tool magazine.

In a RANDOM magazine, the normal tools are interchangeable but the special ones (the larger ones which occupy more than one pocket) must always be allocated the same pocket.

Therefore, it is necessary define, in JOG mode, all the tools of the magazine and their assigned pockets.

The magazine will be managed by the CNC which will indicate to the electrical cabinet the pocket corresponding to the selected tool.

The tool currently being used will be placed in the pocket of the new selected tool except when it is a special tool in which case it will have to return to its own pocket.

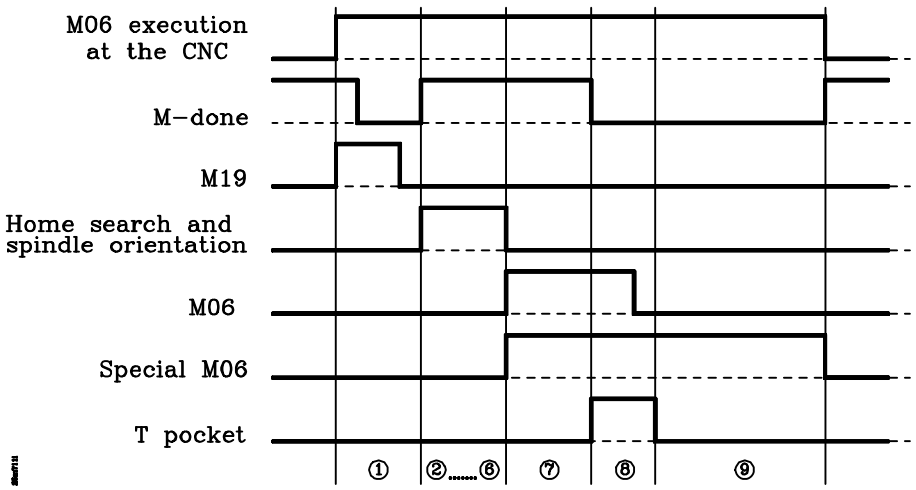
Every time a tool change is carried out, the T2.2 command must be executed to indicate the new tool to be selected and then execute the M06 to perform the actual tool change.

T2.2 The figure located to the left of the "." indicates the tool number and the figure to the right indicates the tool offset to be applied in the machining operations.

The CNC outputs to the electrical cabinet the number of the pocket occupied by the new selected tool. This tool information transfer is done as described in the section on "M, S, T FUNCTION TRANSFER", via the BCD outputs (pins 20 thru 27 of connector I/O1) and the "T Strobe" output.

If a subroutine associated with the T function has been set (machine parameter P743), it will be executed after the T function has been transferred to the electrical cabinet.

M06 The tool changing sequence when executing this function is similar to the one for a non-random magazine excepts points 7, 8 and 9 and it consists of the following steps:



1.- The CNC "tells" the electrical cabinet to execute the M19 (spindle in closed loop).

If there is no spindle encoder or spindle orientation is not to be used, set machine parameter P621(7) to "1".

2.- Once the electrical cabinet has executed function M19, (indicated by activating the "M-DONE" signal), the spindle is homed.

3.- The spindle orients to the position set by machine parameter P916 and the first axis (P702) moves to the position indicated by machine parameter P900.

If so desired, it is possible to have the first axis moving and the spindle homing at the same time by setting machine parameter P615(8) to "1".

4.- Once both the spindle and the first axis are in position, the second axis (P703) moves to the point indicated by machine parameter P901.

If P703=0, this move will not take place and this operation will continue on paragraph 7.

5.- Once the second axis is in position, the third axis (P704) moves to the point indicated by machine parameter P902.

If P704=0, this move will not take place and this operation will continue on paragraph 7.

6.- Once the third axis is in position, the fourth axis (P705) moves to the point indicated by machine parameter P903.

If P705=0, this move will not take place.

7.- Once the last axis is in position, the CNC sends out function M06 for the electrical cabinet to carry out the tool change.

When function M06 has an associated subroutine (P709), machine parameter P618(2) may be set so it is executed before the M06 is sent out to the electrical cabinet.

If the tool change requires a special treatment (machine parameter P603(2)=1), the CNC will activate the SPECIAL M06 output (pin 13 of connector I/O2).

8.- Once the new tool is placed at the spindle (indicated by bringing the M-DONE signal low), the CNC will keep showing to the electrical cabinet, for 100 milliseconds, the tool pocket number where the old tool must be deposited.

9.- Once the old tool is deposited in the indicated pocket, the electrical cabinet must set the M-DONE signal high.

The CNC will consider the tool change function completed and it will deactivate the SPECIAL M06 output (pin 13 of connector I/O2) if it was on according to the setting of machine parameter P603(2).

When the M06 function has an associated subroutine (P709), it is possible to set machine parameter P618(2) so it is executed after sending the M06 to the electrical cabinet (after the old tool has been returned to its pocket).

If the M06 interrupts the execution of the program [P601(8)], it will be necessary to press CYCLE START to resume it.

6.11.4 APPLICATION EXAMPLES

6.11.4.1 NOT-RANDOM MAGAZINE WITH TOOL CHANGER ARM

T2.2 When executing this function, the CNC sends out to the electrical cabinet the number of the pocket occupied by the new tool (which in this case is the same as the tool number).

The magazine will rotate until positioned at the indicated pocket and the new selected tool in it will be placed on the changer arm.

M06 When executing this function, the CNC will position the spindle and the axes of the machine and it will output the number of the pocket where the old tool is to be placed.

The tool magazine will rotate until positioned at this pocket while the changer arm switches tools replacing the one at the spindle with the one it had.

Finally, the old tool which was at the spindle and is now on the changer arm will be returned to its pocket.

6.11.4.2 NOT-RANDOM MAGAZINE WITHOUT CHANGER ARM

T2.2 When executing this function, the CNC sends out to the electrical cabinet the number of the pocket occupied by the new tool (which in this case is the same as the tool number).

The electrical cabinet must memorize this position for future selection.

M06 When executing this function, the CNC will position the spindle and the axes of the machine and it will output the number of the pocket where the old tool is to be placed.

In a NOT-RANDOM magazine, the tool always occupies the same pocket. Besides, the last selected tool was the one which is now at the spindle and, therefore, the currently selected tool pocket at the magazine will be the one for that tool.

This way, the tool being used will be placed in that pocket (its own).

Then, the magazine will rotate until it positions at the pocket previously indicated (by T2.2) which has been memorized by the electrical cabinet.

The tool occupying that pocket will be placed at the spindle.

6.12 PALLET WORK

When working with pallets, it is necessary to set machine parameter P603(3) to "1".

The CNC offers functions M21, M22, M23, M24 and M25 which, when working with pallets acquire the following meaning:

M21 The CNC generates this function whenever M22, M23, M24 or M25 are executed if machine parameter P605(3) has been set to "1".

M22, M23 Auxiliary functions to load (M22) and unload (M23) the part at one end of the table.

M24, M25 Auxiliary functions to load (M24) and unload (M25) the part at the other end of the table.

The machine parameters to be set when working with pallets are:

P603(3)	Machine with PALLETS
P605(3)	The CNC generates function M21 when executing M22, M23, M24 or M25
P605(2)	The Z axis moves when executing M22, M23, M24 or M25
P611(7)	The X axis moves when executing M22, M23, M24 or M25
P605(1)	The W axis moves when executing M22, M23, M24 or M25
P607(1)	Travel limits ignored when executing M06, M22, M23, M24 or M25
P710	Subroutine associated with function M22
P711	Subroutine associated with function M23
P712	Subroutine associated with function M24
P713	Subroutine associated with function M25
P904	W position when executing M22, M23, M24 or M25
P905	X position when executing M22 or M23
P906	X position when executing M24 or M25
P907	Z position when executing M22, M23, M24 or M25

Every time an M22, M23, M24 or M25 is executed, the CNC acts as follows:

1.- If machine parameter P605(3)=1, the CNC sends out an M21.

If this is not desired, set P605(3)=0.

2.- The W axis moves to the position indicated by P904.

If this move is not desired, set P605(1)=1.

3.- Once the W axis is in position, the X axis starts moving to the point indicated by machine parameter P905 for M22 or M23 or to the point indicated by P906 for M24 or M25.

If this move is not desired, set P611(7)=1.

4.- Once the X axis is in position, the Z axis starts moving to the point indicated by machine parameter P907.

If this move is not desired, set P605(2)=0.

5.- Once all the axes are positioned, the CNC sends out the selected M function (M22, M23, M24 or M25) to carry out the corresponding pallet change. This M function is transferred as any other M function.

6.- Once the pallet change has been completed, the electrical cabinet will activate the "M-DONE" signal to let the CNC know that the execution of the M function has ended.

7.- If the required auxiliary M function has an associated subroutine, this subroutine will be executed once the M function has been output to the electrical cabinet.

"P710" Subroutine associated with M22
"P711" Subroutine associated with M23
"P712" Subroutine associated with M24
"P713" Subroutine associated with M25

APPENDIX A

CNC TECHNICAL CHARACTERISTICS

GENERAL CHARACTERISTICS

Three 8-bit microprocessors
32Kbytes of part-program memory.
2 communication lines: RS232C and RS485.
6 feedback inputs up to 5 axes + spindle encoder + electronic handwheel.
Digital probe input (TTL or 24V DC)
Resolution of 0.001 mm. or 0.0001 inches.
Multiplying factor of up to x100 for sine-wave feedback signals.
Feedrates from 0.001 mm/min up to 65535 mm/min (0.0001 through 2580 inches/min)
Maximum axis travel ±8388.607 mm (330.2601 inches)
11 optocoupled digital inputs.
32 optocoupled digital outputs.
6 analog outputs: ±10V (one per axis + spindle).
Approximate weight: Compact model: 12 Kg.
Modular model: Central Unit 9Kg. Monitor 20Kg.
Maximum consumption in normal operation: Central Unit 75w. Monitor 85w

PACKAGING

Meets the "EN 60068-2-32" standard.

POWER SUPPLY

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 at 50 Hz starting from 0º and 180º (two polarities: positive and negative).
Harmonic distortion: Less than 10% of the rms voltage between low voltage conductors (sum of the 2^{nd} through the 5^{th} harmonic)

ELECTRICAL CHARACTERISTICS OF FEEDBACK INPUTS

+5V power consumption: 750 mA (250 mA per axis).
-5V power consumption: 0.3A (100 mA per axis).
Operating levels for square-wave signals:
- Maximum frequency: 200KHz.
- Maximum separation between flanks: 950 nsec.
- Phase shift: 90° ±20°
- High threshold (logic state "1"): 2.4V. < V_{IH} < 5V.
- Low threshold (logic state "0"): -5V. < V_{IL} < 0.8V.
- Vmax.: ±7 V.
- Hystheresis: 0.25 V.
- Maximum input current: 3mA.

Operating levels for sine-wave signals:
- Maximum frequency: 25KHz.
- Peak to peak voltage: 2V. < Vpp < 6V.
- Input current I_I 1mA.

ELECTRICAL CHARACTERISTICS OF DIGITAL INPUTS

Nominal voltage: +24 V DC.
Maximum nominal voltage: +30 V DC.
Minimum nominal voltage: +18 V DC.
High threshold (logic state "1"): V_{IH} >+18 V DC.
Low threshold (logic state "0"): V_{IL} < +5 V DC. or not connected.
Typical consumption per input: 5 mA.
Maximum consumption per input: 7 mA.
Protection by means of galvanic isolation by opto-couplers.
Protection against reversed connection up to -30 V DC.

ELECTRICAL CHARACTERISTICS OF DIGITAL OUTPUTS

Nominal power supply voltage: +24V DC.
Maximum nominal voltage: +30V DC.
Minimum nominal voltage: +18V DC.
Output voltage Vout = Power Supply voltage - 2 V DC.
Maximum output current: 100 mA.
Protection by means of galvanic isolation by opto-couplers.
Protection by means of external 3Amp fuse against reversed connection up to -30 V DC and overvoltage of the external power supply greater than 33V DC.

ELECTRICAL CHARACTERISTICS OF THE 5V PROBE INPUT.

Typical value: 0.25 mA. @ Vin = 5V.
High threshold (logic state "1"): 1.7V.
Low threshold (logic state "0"): 0.9V.
Maximum nominal voltage: Vimax = +15V DC.

ELECTRICAL CHARACTERISTICS OF THE 24V PROBE INPUT.

Typical value: 0.30 mA. @ Vin = 24V.
High threshold (logic state "1"): 12.5 V.
Low threshold (logic state "0"): 8.5 V.
Maximum nominal voltage: Vimax = +35 V DC.

CRT

Monitor	8" monochrome	Deflection:	90 degrees
Screen:	Anti-glare	Phosphor:	PLA (amber)
Resolution:	600 lines	Display surface:	146 x 119 mm.

SWEEP FREQUENCY

Vertical synchronism: 50-60 Hz positive Horizontal synchronism: 19.2 KHz positive

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, 50-300 Hz. and acceleration of 5g.
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

3.5V lithium battery.

Estimated life: 10 years

As from error indication (low battery), the information contained in memory will be kept for a maximum of 10 days with the CNC off. It must be replaced.

Caution, due to risk of explosion or combustion:

Do not attempt to recharge the battery.
Do not expose it to temperatures over 100 °C (232°F).
Do not short-circuit its leads.

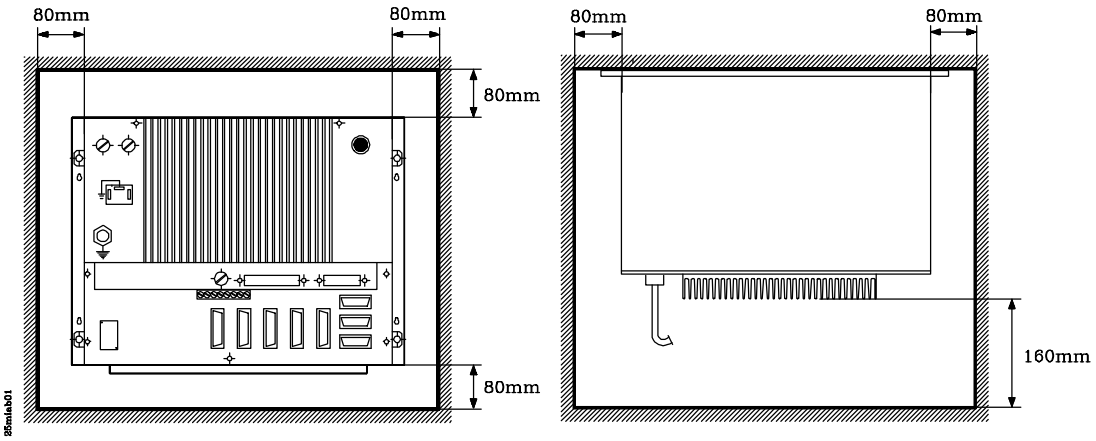
Atention:

To avoid excessive heating of internal circuits, the several ventilation slits must not be obstructed. It is also necessary to install a ventilation system which extracts hot air from the enclosure or desk supporting the CNC.

APPENDIX B

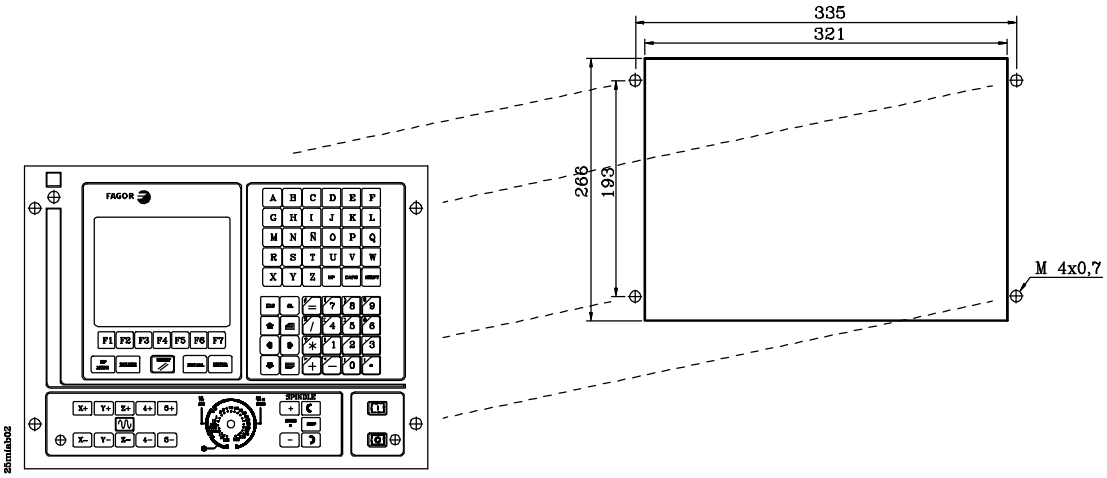
ENCLOSURES

The minimum distance between the sides of the CNC and its enclosure in order to meet the required ambient conditions must be the following:



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

The CNC must be secured as shown below (dimensions in mm):



APPENDIX C

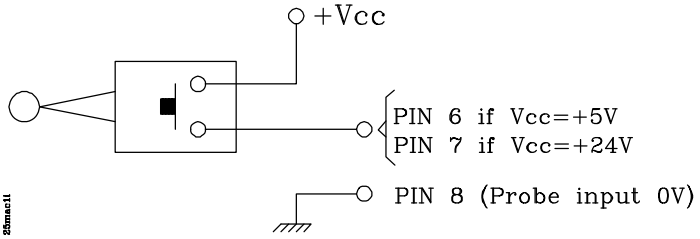
RECOMMENDED PROBE CONNECTION DIAGRAMS

The CNC offers two probe inputs located at connector A6 (pins 6 and 7); one for 5V signals and the other one for 24V signals.

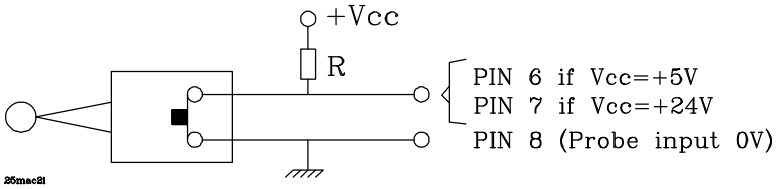
Depending on the type of connection used, machine parameter "P612(7)" must be set to indicate whether it is activated with an up-flank (leading edge) or down flank (trailing edge) of the probe signal.

DIRECT CONNECTION

- **Probe output as normally open contact:**



- **Probe output as normally closed contact:**

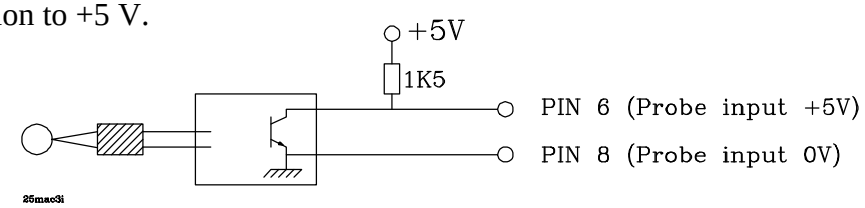


If Vcc=+5V R=10K
If Vcc=+24V R=50K

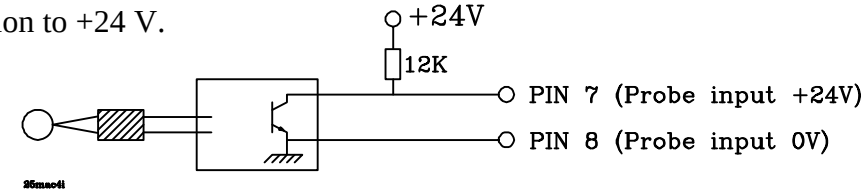
CONNECTION VIA INTERFACE

- **Interface with open-collector output:**

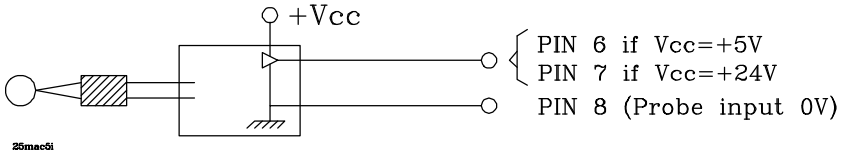
Connection to +5 V.



Connection to +24 V.



- **Interface with PUSH-PULL output:**



APPENDIX D

CNC INPUTS AND OUTPUTS

INPUTS

Pin	Connector	Function
10	I/O 1	X axis Home switch
11	I/O 1	Y axis Home switch
12	I/O 1	Z axis Home switch
13	I/O 1	W axis Home switch
14	I/O 1	Emergency stop.
15	I/O 1	/Feed hold - /Transfer inhibit - /M-done
16	I/O 1	/Stop - /Emergency subroutine
17	I/O 1	Start - rapid jog - Enter
18	I/O 1	Conditional input (block skip)
19	I/O 1	DRO mode

OUTPUTS

Pin	Connector	Function
2	I/O 1	T Strobe
3	I/O 1	S Strobe
4	I/O 1	M Strobe
5	I/O 1	Emergency
6	I/O 1	W axis enable - Threading on
7	I/O 1	Z axis enable
8	I/O 1	Y axis enable
9	I/O 1	X axis enable
20	I/O 1	MST80
21	I/O 1	MST40
22	I/O 1	MST20
23	I/O 1	MST10
24	I/O 1	MST08
25	I/O 1	MST04
26	I/O 1	MST02
27	I/O 1	MST01
30, 31	I/O 1	X axis analog output
32, 33	I/O 1	Y axis analog output
34, 35	I/O 1	Z axis analog output
36, 37	I/O 1	Spindle analog output
3	I/O 2	Decoded M01 output
4	I/O 2	Decoded M02 output
5	I/O 2	Decoded M03 output
6	I/O 2	Decoded M04 output
7	I/O 2	Decoded M05 output
8	I/O 2	Decoded M06 output
9	I/O 2	Decoded M07 output
10	I/O 2	Decoded M08 output
11	I/O 2	Decoded M09 output
12	I/O 2	Decoded M10 output - V axis enable
13	I/O 2	Decoded M11 output - Additional information
14, 15	I/O 2	V axis analog output
17, 18	I/O 2	W axis analog output
21	I/O 2	"Jog mode selected" output
22	I/O 2	Decoded M15 output - Tool magazine turning direction
23	I/O 2	Decoded M14 output - Reset
24	I/O 2	Decoded M13 output - Cycle on - Automatic - G00
25	I/O 2	Decoded M12 output - Vertical axis movement

APPENDIX E

2-DIGIT BCD CODED "S" OUTPUT CONVERSION TABLE

Programmed S	S BCD	Programmed 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		

APPENDIX F

MACHINE PARAMETER SUMMARY CHART

GENERAL MACHINE PARAMETERS

Section 3.3

P5	Mains frequency (50/60)
P99	Language (0=Spanish, 1=German, 2=English, 3=French, 4=Italian)
P13	Measuring units (0= mm, 1= inches)
P6	Theoretical (1) or Real (0) display
P802	Protected program
P619(1), P619(2)	Monitor display color combination

MACHINE PARAMETERS FOR AXIS CONFIGURATION

Section 3.3.1

P11	The machine has a W axis. 0=No, X/YZ = Yes.
P616(4)	The machine has a V axis. 0= No, 1= Yes.
P600(4)	Type of machine. 0= Mill, 1= Boring Mill.
P612(1)	Connector A6. Handwheel (0) or Spindle (1).
P617(5), P605(6), P617(4), P611(4), P617(3)	X, Y, Z, W, V as DRO axis. 0= No, 1= Yes.
P618(6), P618(5), P618(4), P618(3), P618(7)	X, Y, Z, W, V axis display. 1= No, 0= Yes.
P600(3), P616(3)	W, V axis normal (0) or positioning-only (1).
P600(1), P616(1)	W, V axis as linear (0) or rotary (1)
P600(2), P616(2)	W, V as rotary HIRTH axis. 0= No, 1= Yes.
P606(1)	W axis as rotary rollover. 0= No, 1= Yes.
P619(8), P620(6)	W, V rotary rollover via shortest path. 0= No, 1= Yes.
P617(7)	Gantry axis. 0= No, 1= Yes.
P805	Maximum coupling (slaving) error on Gantry axes (µm)

I/O RELATED MACHINE PARAMETERS

Section 3.3.2

P605(8)	Normal status of the Emergency output (pin 5 connector I/O1). 0=0V, 1=24V
P609(7)	Pin 17 of connector I/O 1 as "Rapid Jog". 0= No, 1= Yes.
P610(3)	Pin 17 of connector I/O 1 as "Enter" in PLAY BACK mode. 0= No, 1= Yes.
P605(7)	Pin 22 of connector I/O 2 as "tool magazine rotating direction". 0= No, 1= Yes.
P609(3)	Pin 23 of connector I/O 2 as "RESET". 0= No, 1= Yes.
P611(1)	Pin 24 of connector I/O 2 as "Cycle ON". 0= No, 1= Yes.
P611(6)	Pin 24 of connector I/O 2 as "Automatic". 0= No, 1= Yes.
P613(4)	Pin 24 of connector I/O 2 as "G00". 0= No, 1= Yes.
P613(2)	Pin 25 of connector I/O 2 as "Vertical move" output. 0= No, 1= Yes.
P617(8)	M function output. 0= BCD, 1= Binary
P605(5)	CNC waits for trailing edge at "M-done" input. 0= No, 1= Yes.
P609(5)	BCD or binary coded output for M functions of the "M" table. 1= No, 0= Yes.
P602(8,7,6,5), P603(1)	X, Y, Z, W, V axis feedback alarm cancellation. 0= No, 1= Yes.

HANDWHEEL MACHINE PARAMETERS

Section 3.3.3

P613(1)	Electronic handwheel model FAGOR 100P. 0= No, 1= Yes.
P612(2)	Counting direction of the electronic handwheel
P612(3)	Feedback units of the electronic handwheel. 0= mm, 1= inches.
P612(4,5)	Counting resolution of the electronic handwheel
P612(6)	Multiplying factor for electronic handwheel feedback signals. 0= x4, 1= x2
P625(7)	Handwheel managed by the PLC. 0= No, 1= Yes.

PROBE RELATED MACHINE PARAMETERS *Section 3.3.4*

P612(7)	Type of probe pulse. 0= 0V, 1= 5V or 24V.
P720	M function associated with probing (G75)
P804	Probing feedrate in Jog mode
P910	Minimum X coordinate of the probe
P911	Maximum X coordinate of the probe
P912	Minimum Y coordinate of the probe
P913	Maximum Y coordinate of the probe
P914	Minimum Z coordinate of the probe
P915	Maximum Z coordinate of the probe
P621(6)	Error to be issued while probing (G75). 1= No, 0= Yes.

TOOL RELATED MACHINE PARAMETERS *Section 3.3.5*

P701	Number of tool pockets (positions) of the tool magazine (0..98)
P743	Subroutine associated with the T function
P625(4)	The associated subroutine is executed before (1) or after (0) the T function
P626(1)	The CNC displays the tool base (0) or tool tip (1) position
P601(5)	The machine is a machining center. 0= No, 1= Yes.
P601(1)	The tool magazine is Random. 0= No, 1= Yes.
P709	Subroutine associated with M06
P618(2)	M06 executed before (0) or after (1) its associated subroutine
P601(8)	M06 interrupts the program. 0= No, 1= Yes.
P702	First axis to move when executing M06. 1=X, 2=Y, 3=Z, 4=W, 5=V.
P703	Second axis to move when executing M06. 1=X, 2=Y, 3=Z, 4=W, 5=V.
P704	Third axis to move when executing M06. 1=X, 2=Y, 3=Z, 4=W, 5=V.
P705	Fourth axis to move when executing M06. 1=X, 2=Y, 3=Z, 4=W, 5=V.
P900	Position where the first axis moves when executing M06
P901	Position where the second axis moves when executing M06
P902	Position where the third axis moves when executing M06
P903	Position where the fourth axis moves when executing M06
P621(7)	M06 does not involve M19. 1= No, 0= Yes.
P615(8)	In M06, the M19 is executed while the axis is moving. 0= No, 1= Yes.
P603(2)	Special M06 sequence. 0= No, 1= Yes.

MACHINE PARAMETERS RELATED WITH RS232C SERIAL LINE *Section 3.3.6*

P0	Communications speed (baudrate)
P1	Number of data bits per character
P2	Parity
P3	Stop bits
P607(3)	DNC
P607(4)	Communications setting for FAGOR Floppy (1) or Cassette (0).
P607(5)	DNC protocol active on power-up
P607(6)	The CNC aborts DNC communications (program debugging). 1= No, 0= Yes.
P607(7)	Status report by interruption. 0= No, 1= Yes.

JOG MODE RELATED MACHINE PARAMETERS *Section 3.3.7*

P606(3)	M30 executed when switching to Jog mode. 0= No, 1= Yes.
P803	Jogging feedrate
P12	Continuous (N) or pulsating (Y) axis jog
P609(6)	Maximum incremental Jog. 0= 10mm or 1 inch, 1= 1mm or 0.1 inch.

EMERGENCY SUBROUTINE RELATED MACHINE PARAMETERS *Section 3.3.8*

P727	Emergency subroutine
P621(3)	Repetitive emergency subroutine. 0= No, 1= Yes.
P619(5)	The emergency subroutine executes "M00" (interrupting the program). 1= No, 0= Yes.
P619(4)	Assign coordinates to arithmetic parameters in emergency subroutine. 0= Beginning point, 1= Current point

MACHINE PARAMETERS RELATED TO OPERATING AND PROGRAMMING MODES

Section 3.3.9

P609(8)	Axis orientation in graphic representation. 0 = Mill, 1= Boring Mill.
P605(4)	Axis orientation in XZ plane
P611(3)	The Z axis graphic display combined with W axis moves. 0= No, 1= Yes.
P618(1)	CYCLE START key inhibit. 0= No, 1= Yes.
P625(6)	Spindle inhibit by PLC. 0= No, 1= Yes.
P606(2)	Maximum Feedrate Override Switch value applied by the CNC. 0=120%, 1=100%
P4	Feedrate Override Switch active in G00. No, Yes.
P610(2)	Vectored (interpolated) G00. 0= No, 1= Yes.
P613(5)	G05 or G07 on CNC power-up. 0= G07, 1= G05.
P715	Dwell between G07 blocks (square corner). 1= 10 ms
P611(5)	Feedrate units in G94. 0= 1mm/min or 0.1 inch/min, 1= 0.1 mm or 0.01 inch/min.
P607(8)	G53 zero offset after Reset. 0= No, 1= Yes.
P619(7)	G59 as additive zero offset. 0= No, 1= Yes.
P607(2)	Spindle reversal in G84 requires M05. 1= No, 0= Yes.
P610(1)	Feed-Hold in G84 and G47. 0= No, 1= Yes.
P613(8)	Arithmetic parameters P150 through P254 as read-only. 0= No, 1= Yes.
P618(8)	Function P1=0X takes into account the work units. 0= No, 1= Yes.
P625(5)	Type of compensation in sections programmed in G07.
P626(4)	Use function G64, multiple machining in arc. 1= No, 0= Yes.

MACHINE PARAMETERS FOR THE AXES

P100, P200, P300, P400, P500	Sign of the analog output for X, Y, Z, W, V	*Section 4.*
P101, P201, P301, P401, P501	Counting direction for X, Y, Z, W, V	
P102, P202, P302, P402, P502	Jogging direction for X, Y, Z, W, V	

MACHINE PARAMETERS FOR AXIS RESOLUTION *Section 4.1*

P103, P203, P303, P403, P503	Counting resolution for X, Y, Z, W, V
P604(4), P604(3), P604(2), P604(1), P616(7)	Feedback units for X..V. 0= mm, 1= inches.
P106, P206, P306, P406, P506	Feedback signal type for X, Y, Z, W, V. Y= Sine, N= Squ.
P604(8), P604(7), P604(6), P604(5), P616(8)	Multiplying factor for X,Y,Z,W,V feedback. 0= x4, 1= x2.
P622(1), P622(2), P622(3), P622(4), P622(5)	Resolution for X..V sine-wave feedback
P603(8), P603(7), P603(6), P603(5), P616(5)	Binary encoder for X, Y, Z, W, V. 0= No, 1= Yes.
P610(8), P610(7), P610(6), P610(5), P616(6)	Equivalence of the binary encoder for X, Y, Z, W, V

MACHINE PARAMETERS FOR ANALOG OUTPUTS *Section 4.2*

P117, P217, P317, P417, P517	Minimum analog output for X, Y, Z, W, V. (1= 2.5mV).
P104, P204, P304, P404, P504	Dwell between Enable / Analog output for X,Y,Z,W,V. N= No, Y= Yes.
P118, P218, P318, P418, P518	In-position zone (dead-band) for X, Y, Z, W, V. (0...255µm)
P105, P205, P305, P405, P505	Continuous control of X, Y, Z, W, V. 0= No, 1= Yes.

MACHINE PARAMETERS FOR AXIS TRAVEL LIMITS *Section 4.3*

P107, P207, P307, P407, P507	Positive travel limit for X, Y, Z, W, V
P108, P208, P308, P408, P508	Negative travel limit for X, Y, Z, W, V

FEEDRATE RELATED MACHINE PARAMETERS *Section 4.4*

P615(6)	Programmed feedrate in inches/minute (1) or 0.1inch/min (0)
P615(7)	Feedrate of rotary axes in degrees/minute (1) or 2.54º/min (0)
P110, P210, P310, P410, P510	Maximum programmable feedrate X, Y, Z, W, V
P111, P211, P311, P411, P511	G00 feedrate for X, Y, Z, W, V
P729	Maximum feedrate for circular interpolations
P708	Feedrate/Override when analog output reaches 10V.
P714	Error if actual feedrate not within 50% to 200% of programmed value

MACHINE PARAMETERS FOR AXIS CONTROL *Section 4.5*

P114, P214, P314, P414, P514	Proportional gain K1 for X, Y, Z, W, V
P115, P215, P315, P415, P515	Gain break point for X, Y, Z, W, V
P116, P216, P316, P416, P516	Proportional gain K2 for X, Y, Z, W, V
P611(8)	In G00 and F00, proportional gain K2 from 256 microns on. 0= No, 1= Yes.
P726	Recovery of programmed position on "non-continuously" controlled axes

HOME SEARCH RELATED MACHINE PARAMETERS *Section 4.6*

P627(1, 2, 3, 4, 5)	Regular (0) or Coded (1) marker pulse Io.
P628(1) P628(3), P628(5), P628(7), P629(1)	Period of coded Io. 0= 20mm, 1=100 mm.
P628(2) P628(4), P628(6), P628(8), P629(2)	Variable Io increases in + direction (0) or - direction (1). X..V
P919, P920, P921, P922, P923	Offset of the semi-absolute scale (with coded Io). X..V
P602(4), P602(3), P602(2), P602(1), P617(2)	Home switch for X, Y, Z, W, V. 1= No, 0= Yes.
P623(8), P623(7), P623(6), P623(5), P623(4)	Home search direction for X, Y, Z, W, V. 0= Pos, 1= Neg.
P600(8), P600(7), P600(6), P600(5), P617(1)	Home pulse (marker) type for X, Y, Z, W, V. 1= Pos, 0= Neg.
P119, P219, P319, P419, P519	Home coordinates for X, Y, Z, W, V

P112, P212, P312, P412, P512	1st homing feedrate for X, Y, Z, W, V
P810, P811, P812, P813, P814	2nd homing feedrate for X, Y, Z, W, V
P611(2)	Mandatory home search on power-up. 0= No, 1= Yes.
P606(4)	Function G74 generates an M30. 0= No, 1= Yes.
P725	Subroutine associated with G74

ACCELERATION/DECELERATION RELATED MACHINE PARAMETERS *Section 4.7*

P721, P722, P723, P724, P728	Apply ACC/DEC onto X, Y, Z, W, V. (1= 20ms)
P613(7)	Linear ACC/DEC on all linear interpolation (G01). 0= No, 1= Yes.
P620(2)	Linear ACC/DEC in G05 (round corner). 1= No, 0= Yes.
P624(8)	Bell-shaped ACC/DEC. 0= No, 1= Yes.
P744	Bell-shaped ACC/DEC ramp duration. (1= 10 ms)
P732, P733, P734, P735, P736	FEED-FORWARD gain for X, Y, Z, W, V

MACHINE PARAMETERS FOR UNIDIRECTIONAL APPROACH *Section 4.8*

P608(4), P608(3), P608(2), P608(1)	Unidirectional approach for X, Y Z, W. 0= No, 1= Yes.
P608(8), P608(7), P608(6), P608(5)	Direction of the unidirectional approach for X, Y Z, W. (0= +, 1= -).
P716	Distance from unidirectional approach point to programmed point
P801	Unidirectional approach feedrate

LEADSCREW RELATED MACHINE PARAMETERS *Section 4.9*

P109, P209, P309, P409, P509	Leadscrew backlash for X, Y, Z, W, V. (0..255µm)
P624(1), P624(2), P624(3), P624(4), P624(5)	Sign of leadscrew backlash X, Y, Z, W, V. (0= +, 1= -).
P113, P213, P313, P413, P513	Additional analog pulse for X, Y, Z, W, V. (1= 2.5mV)
P606(8), P606(7), P606(6), P606(5)	Leadscrew error compensation X, Y, Z, W. 0= No, 1= Yes.
P613(6)	Leadscrew error compensation tables.(0= 4, 1= 2)

MACHINE PARAMETERS FOR CROSS COMPENSATION *Section 4.10*

P623(1)	Cross compensation applied onto the X axis. 0= No, 1= Yes.
P620(5)	Cross compensation applied onto the Y axis. 0= No, 1= Yes.
P620(4)	Cross compensation applied onto the Z axis. 0= No, 1= Yes.
P623(2), P623(3)	Axis causing the error onto another axis
P625(3)	Double cross compensation. 0= No, 1= Yes.

PALLET RELATED MACHINE PARAMETERS *Section 4.11*

P603(3)	Machine with pallets. 0= No, 1= Yes.
P605(3)	The CNC generates function M21 when executing M22, M23, M24 or M25. 0= No, 1= Yes.
P605(2)	The Z axis moves when executing M22, M23, M24 or M25. 0= No, 1= Yes.
P611(7)	The X axis moves when executing M22, M23, M24 or M25. 1= No, 0= Yes.
P605(1)	The W axis moves when executing M22, M23, M24 or M25. 1= No, 0= Yes.
P607(1)	Software travel limits ignored when executing M06, M22, M23, M24 or M25. 0= No, 1= Yes.
P710	Subroutine associated with function M22
P711	Subroutine associated with function M23
P712	Subroutine associated with function M24
P713	Subroutine associated with function M25
P904	Position to move the W axis when executing M22, M23, M24 or M25
P905	Position to move the X axis when executing M22 or M23
P906	Position to move the X axis when executing M24 or M25
P907	Position to move the Z axis when executing M22, M23, M24 or M25

SPECIAL MACHINE PARAMETERS *Section 4.12*

P609(1)	Machine with travels over 8 meters. 0= No, 1= Yes.
P617(6)	Resolution of 0.0001 millimeters (0.00001 inches). 0= No, 1= Yes.
P908, P909	Collision zone for Y and Z
P621(4)	Begin the execution of a block synchronized with independent axis (G65). 0= No, 1= Yes.
P626(3)	"RESCAN 200" feature from Renishaw. 0= No, 1= Yes.

SPINDLE MACHINE PARAMETERS

Section 5.

P815 ACC/DEC ramp duration for spindle speed ranges: (1= 10 ms)

MACHINE PARAMETERS FOR SPINDLE RANGE CHANGE *Section 5.1*

P7, P8, P9, P10	Maximum speed for ranges 1, 2, 3 and 4 (0..9999 rpm)
P601(6)	Residual analog voltage "S" for range change. 0= No, 1= Yes.
P706	Value of the residual analog voltage. (1=2.5 mV).
P707	Oscillation period during range change

MACHINE PARAMETERS FOR ANALOG SPINDLE SPEED OUTPUT *Section 5.2*

P601(4)	Sign of the analog spindle speed output "S"
P610(4)	Unipolar (1) or bipolar (0)analog spindle speed output "S"
P609(4)	All spindle **speed** changes generate an "S Strobe" signal. 0= No, 1= Yes.

MACHINE PARAMETERS FOR BCD-CODED SPINDLE SPEED OUTPUT *Section 5.3*

P601(3)	2-digit BCD coded output. 0= No, 1= Yes.
P601(2)	4-digit BCD coded output. 0= No, 1= Yes.

MACHINE PARAMETERS FOR SPINDLE CONTROL *Section 5.4*

P800	Number of pulses (line count) of the spindle encoder (0..9999)
P609(2)	Counting direction of the spindle encoder

MACHINE PARAMETERS FOR SPINDLE ORIENTATION (M19)

Section 5.5

P700	Spindle speed when being oriented. M19. (0..255 rpm)
P601(7)	Sign of the "S" analog output associated with M19
P612(8)	Type of marker pulse (reference) of the spindle encoder. (0= -, 1= +).
P619(6)	Spindle orientation in both directions. 0= No, 1= Yes.
P719	Minimum Spindle analog voltage for M19. (1= 2.5mV)
P717	In-position zone (dead band) for M19
P718	Proportional gain K for the spindle during M19
P917	Lower limit of forbidden zone for the spindle in M19
P918	Upper limit of forbidden zone for the spindle in M19
P916	Spindle orient position when executing M19 without "S" value

MACHINE PARAMETERS FOR RIGID TAPPING (G84R) *Section 5.6*

P745, P747, P748, P749	ACC/DEC ramp duration for the spindle in ranges 1, 2, 3, 4. (1= 20ms)
P750	Proportional gain K1 for the tapping axis
P746	Feed-forward gain for the spindle during rigid tapping
P625(1)	Beginning of tap synchronized with spindle marker pulse (Io). 0= No, 1= Yes.

APPENDIX G

SEQUENTIAL MACHINE PARAMETER LIST

P0	Communications baudrate (110, 300, 600, 1200, 2400, 4800, 9600)	*Section 3.3.6*
P1	Communications data bits (7/8)	*Section 3.3.6*
P2	Parity. (1=Odd, 2=Even)	*Section 3.3.6*
P3	Stop bits. (1/2)	*Section 3.3.6*
P4	Feedrate Override active in G00. (No/Yes)	*Section 3.3.9*
P5	Mains (AC) frequency. (50/60)	*Section 3.3*
P6	Theoretical (1) or real (0) display	*Section 3.3*
P7	Maximum spindle speed for RANGE 1. (0..9999 rpm)	*Section 5.1*
P8	Maximum spindle speed for RANGE 2. (0..9999 rpm)	*Section 5.1*
P9	Maximum spindle speed for RANGE 3. (0..9999 rpm)	*Section 5.1*
P10	Maximum spindle speed for RANGE 4. (0..9999 rpm)	*Section 5.1*
P11	The machine has a W axis. (0= No, X/Y/Z=Yes)	*Section 3.3.1*
P12	Continuous (N) or pulsating (Y) axis JOG	*Section 3.3.7*
P13	Measuring units. 0= mm, 1= inches	*Section 3.3*
P99	Language. (0= Spanish, 1= German, 2= English, 3= French, 4= Italian)	*Section 3.3*
P100	Sign of the X axis analog output	*Section 4.*
P101	Counting direction of the X axis	*Section 4.*
P102	X axis jogging direction	*Section 4.*
P103	X axis feedback (counting) resolution	*Section 4.1*
P104	Dwell between Enable and analog output for the X axis. (N= No, Y=Yes)	*Section 4.2*
P105	Continuous control of the X axis. (N= No, Y=Yes)	*Section 4.2*
P106	Type of feedback signal for the X axis. (N= Square, Y=Sine)	*Section 4.1*
P107	X axis positive travel limit	*Section 4.3*
P108	X axis negative travel limit	*Section 4.3*
P109	X axis leadscrew backlash (0..255µm)	*Section 4.9*
P110	X axis maximum programmable feedrate	*Section 4.4*
P111	X axis G00 feedrate	*Section 4.4*
P112	1st home searching feedrate for the X axis	*Section 4.6*
P113	Additional analog pulse for the X axis. (1=2.5 mV)	*Section 4.9*
P114	Proportional gain K1 for the X axis	*Section 4.5*
P115	Gain break point for the X axis	*Section 4.5*
P116	Proportional gain K2 for the X axis	*Section 4.5*
P117	Minimum X axis analog. (1=2.5mV)	*Section 4.2*
P118	In-position zone (dead band) for the X axis. (0..255µm)	*Section 4.2*
P119	X axis home coordinate	*Section 4.6*
P200	Sign of the Y axis analog output	*Section 4.*
P201	Counting direction of the Y axis	*Section 4.*
P202	Y axis jogging direction	*Section 4.*
P203	Y axis feedback (counting) resolution	*Section 4.1*
P204	Dwell between Enable and analog output for the Y axis. (N= No, Y=Yes)	*Section 4.2*
P205	Continuous control of the Y axis. (N= No, Y=Yes)	*Section 4.2*
P206	Type of feedback signal for the Y axis. (N= Square, Y=Sine)	*Section 4.1*
P207	Y axis positive travel limit	*Section 4.3*
P208	Y axis negative travel limit	*Section 4.3*
P209	Y axis leadscrew backlash (0..255µm)	*Section 4.9*
P210	Y axis maximum programmable feedrate	*Section 4.4*
P211	Y axis G00 feedrate	*Section 4.4*
P212	1st home searching feedrate for the Y axis	*Section 4.6*
P213	Additional analog pulse for the Y axis. (1=2.5 mV)	*Section 4.9*
P214	Proportional gain K1 for the Y axis	*Section 4.5*
P215	Gain break point for the Y axis	*Section 4.5*
P216	Proportional gain K2 for the Y axis	*Section 4.5*
P217	Minimum Y axis analog. (1=2.5mV)	*Section 4.2*
P218	In-position zone (dead band) for the Y axis. (0..255µm)	*Section 4.2*
P219	Y axis home coordinate	*Section 4.6*

P600(8) Type of X axis marker pulse (home). 0= Neg, 1= Pos. ..... *Section 4.6*
(7) Type of Y axis marker pulse (home). 0= Neg, 1= Pos. ..... *Section 4.6*
(6) Type of Z axis marker pulse (home). 0= Neg, 1= Pos. ..... *Section 4.6*
(5) Type of W axis marker pulse (home). 0= Neg, 1= Pos. ..... *Section 4.6*
(4) Type of machine. 0= Mill, 1= Boring mill. ..... *Section 3.3.1*
(3) W axis: normal (0) or position-only (1) ..... *Section 3.3.1*
(2) W axis: rotary HIRTH. 0= No, 1= Yes ..... *Section 3.3.1*
(1) W axis: linear (0) or rotary (1) ..... *Section 3.3.1*

P601(8) M06 interrupts the program. 0= No, 1= Yes ..... *Section 3.3.5*
(7) Sign of the analog S output associated with M19 ..... *Section 5.5*
(6) Residual analog S output during range change. 0= No, 1= Yes ..... *Section 5.1*
(5) Machining center. 0= No, 1= Yes ..... *Section 3.3.5*
(4) Sign of the spindle analog output ..... *Section 5.2*
(3) S output in 2-digit BCD code. 0= No, 1= Yes ..... *Section 5.3*
(2) S output in 4-digit BCD code. 0= No, 1= Yes ..... *Section 5.3*
(1) Random tool magazine. 0= No, 1= Yes ..... *Section 3.3.5*

P602(8) Cancellation of X axis feedback alarm. 0= No, 1= Yes ..... *Section 3.3.2*
(7) Cancellation of Y axis feedback alarm. 0= No, 1= Yes ..... *Section 3.3.2*
(6) Cancellation of Z axis feedback alarm. 0= No, 1= Yes ..... *Section 3.3.2*
(5) Cancellation of W axis feedback alarm. 0= No, 1= Yes ..... *Section 3.3.2*
(4) X axis home switch. 1= No, 0= Yes ..... *Section 4.6*
(3) Y axis home switch. 1= No, 0= Yes ..... *Section 4.6*
(2) Z axis home switch. 1= No, 0= Yes ..... *Section 4.6*
(1) W axis home switch. 1= No, 0= Yes ..... *Section 4.6*

P603(8) X axis with binary encoder. 0= No, 1= Yes ..... *Section 4.1*
(7) Y axis with binary encoder. 0= No, 1= Yes ..... *Section 4.1*
(6) Z axis with binary encoder. 0= No, 1= Yes ..... *Section 4.1*
(5) W axis with binary encoder. 0= No, 1= Yes ..... *Section 4.1*
(4) Not being used at this time (=0)
(3) Machine with pallets. 0= No, 1= Yes ..... *Section 4.11*
(2) Special M06 sequence. 0= No, 1= Yes ..... *Section 3.3.5*
(1) Cancellation of V axis feedback alarm. 0= No, 1= Yes ..... *Section 3.3.2*

P604(8) X axis feedback signal multiplying factor. (0= x4, 1= x2) ..... *Section 4.1*
(7) Y axis feedback signal multiplying factor. (0= x4, 1= x2) ..... *Section 4.1*
(6) Z axis feedback signal multiplying factor. (0= x4, 1= x2) ..... *Section 4.1*
(5) W axis feedback signal multiplying factor. (0= x4, 1= x2) ..... *Section 4.1*
(4) X axis feedback units. (0= mm, 1= inches) ..... *Section 4.1*
(3) Y axis feedback units. (0= mm, 1= inches) ..... *Section 4.1*
(2) Z axis feedback units. (0= mm, 1= inches) ..... *Section 4.1*
(1) W axis feedback units. (0= mm, 1= inches) ..... *Section 4.1*

P605(8) Normal status of Emergency output (pin 5 of connector I/O1). (0=0V, 1=24V) ..... *Section 3.3.2*
(7) Pin 22 of connector I/O 2 as "tool magazine rotating direction". 0= No, 1= Yes ..... *Section 3.3.2*
(6) Y axis as DRO axis. 0= No, 1= Yes ..... *Section 3.3.1*
(5) The CNC waits for trailing edge at "M-done" input. 0= No, 1= Yes ..... *Section 3.3.2*
(4) Axis orientation on the XZ plane ..... *Section 3.3.9*
(3) M21 generated when executing M22, M23, M24 or M25. 0= No, 1= Yes ..... *Section 4.11*
(2) The Z axis moves when executing M22, M23, M24 or M25. 0= No, 1= Yes ..... *Section 4.11*
(1) The W axis moves when executing M22, M23, M24 or M25. 1= No, 0= Yes ..... *Section 4.11*

P606(8) X axis leadscrew error compensation. 0= No, 1= Yes ..... *Section 4.9*
(7) Y axis leadscrew error compensation. 0= No, 1= Yes ..... *Section 4.9*
(6) Z axis leadscrew error compensation. 0= No, 1= Yes ..... *Section 4.9*
(5) W axis leadscrew error compensation. 0= No, 1= Yes ..... *Section 4.9*
(4) Function G74 generates an M30. 0= No, 1= Yes ..... *Section 4.6*
(3) M30 executed when switching to JOG mode. 0= No, 1= Yes ..... *Section 3.3.7*
(2) Maximum Feedrate override value applied by the CNC. 0= 120%, 1= 100% ..... *Section 3.3.9*
(1) W rotary rollover. 0= No, 1= Yes ..... *Section 3.3.1*

P607(8) G53 zero offset activated after Reset. 0= No, 1= Yes ..... *Section 3.3.9*
(7) Status report by interruption. 0= No, 1= Yes ..... *Section 3.3.6*
(6) The CNC aborts DNC communications (program debugging). 1= No, 0= Yes ..... *Section 3.3.6*
(5) DNC protocol active after power-up. 0= No, 1= Yes ..... *Section 3.3.6*
(4) Communications with FAGOR Floppy (1) or Cassette (0) ..... *Section 3.3.6*
(3) DNC. 0= No, 1= Yes ..... *Section 3.3.6*
(2) Spindle reversal with M05 in G84. 1= No, 0= Yes ..... *Section 3.3.9*
(1) Limits ignored when executing M06, M22, M23, M24 or M25. 1= No, 0= Yes ..... *Section 4.11*

P608(8) Direction of the unidirectional approach along X. 0= Positive, 1= Negative ..... *Section 4.8*
(7) Direction of the unidirectional approach along Y. 0= Positive, 1= Negative ..... *Section 4.8*
(6) Direction of the unidirectional approach along Z. 0= Positive, 1= Negative ..... *Section 4.8*
(5) Direction of the unidirectional approach along W. 0= Positive, 1= Negative ..... *Section 4.8*
(4) Unidirectional approach along X. 0= No, 1= Yes ..... *Section 4.8*
(3) Unidirectional approach along Y. 0= No, 1= Yes ..... *Section 4.8*
(2) Unidirectional approach along Z. 0= No, 1= Yes ..... *Section 4.8*
(1) Unidirectional approach along W. 0= No, 1= Yes ..... *Section 4.8*

P609(8) Axis orientation for graphic display . 0= Mill, 1= Boring mill ..... *Section 3.3.9*
(7) Pin 17 of connector I/O 1 as "rapid jog". 0= No, 1= Yes ..... *Section 3.3.2*
(6) Maximum incremental JOG move. (0= 10mm or 1 inch, 1= 1mm or 1 inch) ..... *Section 3.3.7*
(5) M functions set at M table are not output in BCD or binary. 1= No, 0= Yes ..... *Section 3.3.2*
(4) All spindle speed changes generate an S strobe output. 0= No, 1= Yes ..... *Section 5.2*
(3) Pin 23 of connector I/O 2 as "RESET". 0= No, 1= Yes ..... *Section 3.3.2*
(2) Spindle counting direction ..... *Section 5.4*
(1) Machine with axis travel over 8 meters. 0= No, 1= Yes ..... *Section 4.12*

P610(8) X axis binary encoder equivalence ..... *Section 4.1*
(7) Y axis binary encoder equivalence ..... *Section 4.1*
(6) Z axis binary encoder equivalence ..... *Section 4.1*
(5) W axis binary encoder equivalence ..... *Section 4.1*
(4) S analog output: unipolar or bipolar ..... *Section 5.2*
(3) Pin 17 of connector I/O 1 as "Enter" in Play-back mode. 0= No, 1= Yes ..... *Section 3.3.2*
(2) Vectored (interpolated) G00. 0= No, 1= Yes ..... *Section 3.3.9*
(1) Feed-Hold in G84 and G47. 0= No, 1= Yes ..... *Section 3.3.9*

P611(8) In G00 and F00 proportional gain break-point at 256 microns. 0= No, 1= Yes ..... *Section 4.5*
(7) The X axis moves when executing M22, M23, M24 or M25. 1= No, 0= Yes ..... *Section 4.11*
(6) Pin 24 of connector I/O 2 as "Automatic" output. 0= No, 1= Yes ..... *Section 3.3.2*
(5) Feed units in G94. (0=1mm/min or 0.1 inch/min, 1=0.1mm/min or 0.01inch/min) .. *Section 3.3.9*
(4) W axis as DRO. 0= No, 1= Yes ..... *Section 3.3.1*
(3) Z and W axes movements combined in Z axis graphic display. 0= No, 1= Yes ..... *Section 3.3.9*
(2) Home search required after power-up. 0= No, 1= Yes ..... *Section 4.6*
(1) Pin 24 of connector I/O 2 as "Cycle ON" output. 0= No, 1= Yes ..... *Section 3.3.2*

P612(8) Spindle marker pulse (home) type. 0= Negative, 1= Positive) ..... *Section 5.5*
(7) Type of probe pulse. (0= 0V, 1= 5V or 24V) ..... *Section 3.3.4*
(6) Multplying factor for electronic handwheel signals. (0= x4, 1= x2) ..... *Section 3.3.3*
(5) Feedback (counting) resolution of the electronic handwheel ..... *Section 3.3.3*
(4) Feedback (counting) resolution of the electronic handwheel ..... *Section 3.3.3*
(3) Feedback units of the electronic handwheel. (0= mm, 1= inches) ..... *Section 3.3.3*
(2) Counting direction of the electronic handwheel ..... *Section 3.3.3*
(1) Connector A6. Handwheel (0) or spindle (1) ..... *Section 3.3.1*

P613(8) Arithmetic parameters P150 through P254 as read-only. 0= No, 1= Yes ..... *Section 3.3.9*
(7) Acc/dec in all G01 movements. 0= No, 1= Yes ..... *Section 4.7*
(6) Leadscrew error compensation tables. (0= 4, 1= 2) ..... *Section 4.9*
(5) G05 or G07 active on power-up. (0= G07, 1= G05) ..... *Section 3.3.9*
(4) Pin 24 of connector I/O 2 as "G00" output. 0= No, 1= Yes ..... *Section 3.3.2*
(3) There is PLC64 in the LAN. 0= No, 1= Yes ..... Fagor LAN
(2) Pin 25 of connector I/O 2 as "vertical move" output. 0= No, 1= Yes ..... *Section 3.3.2*
(1) Electronic handwheel model: FAGOR 100P. 0= No, 1= Yes ..... *Section 3.3.3*

P614 CNC identification in the Fagor LAN ..... Fagor LAN

P615(8) During M06, M19 is executed while the axis is moving. 0= No, 1= Yes ..... *Section 3.3.5*
(7) Feedrate of rotary axes in degrees/minute (1) or 2.54º/min (0) ..... *Section 4.4*
(6) Feedrate of linear axes in inches/min (1) or 0.1inch/min (0) ..... *Section 4.4*
(5) The CNC occupies main node "0" in Fagor LAN. 0= No, 1= Yes ..... Fagor LAN
(4) Node Nº occupied by the CNC or number of nodes in LAN ..... Fagor LAN
(3) Node Nº occupied by the CNC or number of nodes in LAN ..... Fagor LAN
(2) Node Nº occupied by the CNC or number of nodes in LAN ..... Fagor LAN
(1) Node Nº occupied by the CNC or number of nodes in LAN ..... Fagor LAN

P616(8) Multiplying factor for V axis feedback signals. (0= x4, 1= x2) ..... *Section 4.1*
(7) V axis feedback units. 0= mm, 1= inches ..... *Section 4.1*
(6) V axis binary encoder equivalence ..... *Section 4.1*
(5) V axis with binary encoder. 0= No, 1= Yes ..... *Section 4.1*
(4) The machine has a V axis. 0= No, 1= Yes ..... *Section 3.3.1*
(3) V axis: normal (0) or positioning-only (1) ..... *Section 3.3.1*
(2) V axis: rotary HIRTH. 0= No, 1= Yes ..... *Section 3.3.1*
(1) V axis: linear (0) or rotary (1) ..... *Section 3.3.1*

P617(8) M function output in BCD (0) or Binary (1) ..... *Section 3.3.2*
(7) Gantry axis. 0= No, 1= Yes ..... *Section 3.3.1*
(6) 0.0001 millimeters (0.00001 inch) resolution. 0= No, 1= Yes ..... *Section 4.12*
(5) X as DRO axis. 0= No, 1= Yes ..... *Section 3.3.1*
(4) Z as DRO axis. 0= No, 1= Yes ..... *Section 3.3.1*
(3) V as DRO axis. 0= No, 1= Yes ..... *Section 3.3.1*
(2) V axis home switch. 1= No, 0= Yes ..... *Section 4.6*
(1) Type of V axis marker pulse (home). 0= Negative, 1= Positive ..... *Section 4.6*

P618(8) Function P1=0X considers current work units. 0= No, 1= Yes ..... *Section 3.3.9*
(7) V axis displayed. 1= No, 0= Yes ..... *Section 3.3.1*
(6) X axis displayed. 1= No, 0= Yes ..... *Section 3.3.1*
(5) Y axis displayed. 1= No, 0= Yes ..... *Section 3.3.1*
(4) Z axis displayed. 1= No, 0= Yes ..... *Section 3.3.1*
(3) W axis displayed. 1= No, 0= Yes ..... *Section 3.3.1*
(2) M06 before (0) or after (1) associated subroutine ..... *Section 3.3.5*
(1) CYCLE START key inhibited. 0= No, 1= Yes ..... *Section 3.3.9*

P619(8) W axis as rotary rollover via shortest path. 0= No, 1= Yes ..... *Section 3.3.1*
(7) G59 as additive zero offset. 0= No, 1= Yes ..... *Section 3.3.9*
(6) Spindle orientation in both directions. 0= No, 1= Yes ..... *Section 5.5*
(5) The emergency subroutine executes function M00. 0= No, 1= Yes ..... *Section 3.3.8*
(4) Coordinates assigned to arithmetic parameters in emergency subroutine.
(0= Beginning point, 1= Current point) ..... *Section 3.3.8*
(3) Analog S output proportional to actual axis feedrate. 0= No, 1= Yes ..... *Applications*
(2) Monitor color combination ..... Section 3.3
(1) Monitor color combination ..... Section 3.3

P620(8) Not being used at this time (=0)
(7) Not being used at this time (=0)
(6) V axis as rotary rollover via shortest path. 0= No, 1= Yes ..... *Section 3.3.1*
(5) Cross compensation applied to Y axis. 0= No, 1= Yes ..... *Section 4.10*
(4) Cross compensation applied to Z axis. 0= No, 1= Yes ..... *Section 4.10*
(3) Transfer inhibit and M-done inputs independent from Feed-hold. 0= No, 1= Yes ... Fagor LAN
(2) Acc/dec. in G05 (round corner). 1= No, 0= Yes ..... *Section 4.7*
(1) Marks M1801 thru M1899 to send messages to the CNC. 0= No, 1= Yes ..... LAN & PLCI

P621(8) To be used only by the Service Department of Fagor Automation
(7) Function M06 does not execute function M19. 1= No, 0= Yes ..... *Section 3.3.5*
(6) Error to be issued when probing (G75). 1= No, 0= Yes ..... *Section 3.3.4*
(5) Not being used at this time (=0)
(4) Block beginning synchronized with independent axis (G65). 0= No, 1= Yes ..... *Section 4.12*
(3) Repetitive emergency subroutine. 0= No, 1= Yes ..... *Section 3.3.8*
(2) Not being used at this time (=0)
(1) The CNC has a PLCI. 0= No, 1= Yes ..... *PLCI manual*

P622(8) Jig Grinder. 0= No, 1= Yes ..... *Applications*
(7) Transfer inhibit affects M, S, T functions. 1= No, 0= Yes ..... *Fagor LAN*
(6) Sheetmetal tracing on laser machines. 0= No, 1= Yes ..... *Applications*
(5) V axis sine-wave feedback counting resolution ..... *Section 4.1*
(4) W axis sine-wave feedback signals ..... *Section 4.1*
(3) Z axis sine-wave feedback signals ..... *Section 4.1*
(2) Y axis sine-wave feedback signals ..... *Section 4.1*
(1) X axis sine-wave feedback signals ..... *Section 4.1*

P623(8) X axis home searching direction. (0= Positive, 1= Negative) ..... *Section 4.6*
(7) Y axis home searching direction. (0= Positive, 1= Negative) ..... *Section 4.6*
(6) Z axis home searching direction. (0= Positive, 1= Negative) ..... *Section 4.6*
(5) W axis home searching direction. (0= Positive, 1= Negative) ..... *Section 4.6*
(4) V axis home searching direction. (0= Positive, 1= Negative) ..... *Section 4.6*
(3) Axis causing the cross error (cross compensation) ..... *Section 4.10*
(2) Axis causing the cross error (cross compensation) ..... *Section 4.10*
(1) Apply cross compensation onto X axis. 0= No, 1= Yes ..... *Section 4.10*

P624(8) Bell shaped acc/dec.. 0= No, 1= Yes ..... *Section 4.7*
(7) Not being used at this time (=0)
(6) Not being used at this time (=0)
(5) Sign of V axis leadscrew backlash. (0= Positive, 1= Negative) ..... *Section 4.9*
(4) Sign of W axis leadscrew backlash. (0= Positive, 1= Negative) ..... *Section 4.9*
(3) Sign of Z axis leadscrew backlash. (0= Positive, 1= Negative) ..... *Section 4.9*
(2) Sign of Y axis leadscrew backlash. (0= Positive, 1= Negative) ..... *Section 4.9*
(1) Sign of X axis leadscrew backlash. (0= Positive, 1= Negative) ..... *Section 4.9*

P625(8) To be used only by the Service Department of Fagor Automation
(7) Handwheel managed by the PLC. 0= No, 1= Yes ..... *Section 3.3.3*
(6) Spindle inhibit by the PLC. 0= No, 1= Yes ..... *Section 3.3.9*
(5) Type of compensation on G07 sections ..... Section 3.3.9
(4) Associated subroutine executed before (1) or after (0) the T function ..... *Section 3.3.5*
(3) Double cross compensation. 0= No, 1= Yes ..... *Section 4.10*
(2) Not being used at this time (=0)
(1) G84. Tap beginning synchronized with spindle Io. 0= No, 1= Yes ..... *Section 5.6*

P626(8) The machine uses non-servocontrolled open loop motors. 0= No, 1= Yes ..... *Applications*
(7) Not being used at this time (=0)
(6) Not being used at this time (=0)
(5) Not being used at this time (=0)
(4) Use G64, multiple machining in arc. 1= No, 0= Yes ..... *Section 3.3.9*
(3) "RESCAN 200" feature from Renishaw. 0= No, 1= Yes ..... *Section 4.12*
(2) Not being used at this time (=0)
(1) The CNC displays the tool base (0) or tool tip (1) position ..... *Section 3.3.5*

P627(8,7,6) Not being used at this time (=0)
(5) V axis feedback marker pulse "Io" type. 0=Regular, 1= Coded ..... Section 4.6
(4) W axis feedback marker pulse "Io" type. 0=Regular, 1= Coded ..... Section 4.6
(3) Z axis feedback marker pulse "Io" type. 0=Regular, 1= Coded ..... Section 4.6
(2) Y axis feedback marker pulse "Io" type. 0=Regular, 1= Coded ..... Section 4.6
(1) X axis feedback marker pulse "Io" type. 0=Regular, 1= Coded ..... Section 4.6

P628(2) W axis variable Io increases in positive (0) or negative (1) direction ..... Section 4.6
(1) W axis coded Io period. (0= 20mm, 1= 100mm) ..... Section 4.6
(2) Z axis variable Io increases in positive (0) or negative (1) direction ..... Section 4.6
(1) Z axis coded Io period. (0= 20mm, 1= 100mm) ..... Section 4.6
(2) Y axis variable Io increases in positive (0) or negative (1) direction ..... Section 4.6
(1) Y axis coded Io period. (0= 20mm, 1= 100mm) ..... Section 4.6
(2) X axis variable Io increases in positive (0) or negative (1) direction ..... Section 4.6
(1) X axis coded Io period. (0= 20mm, 1= 100mm) ..... Section 4.6

P629(8,7,6,5,4,3) Not being used at this time (=0)
(2) V axis variable Io increases in positive (0) or negative (1) direction ..... Section 4.6
(1) V axis coded Io period. (0= 20mm, 1= 100mm) ..... Section 4.6

P700 Spindle speed S when in M19. (0..255 rpm) ..... *Section 5.5*
P701 Number of tool pockets in tool magazine. (0..98) ..... *Section 3.3.5*
P702 First axis to move when executing M06. (1=X, 2=Y, 3=Z, 4=W, 5=V) ..... *Section 3.3.5*
P703 Second axis to move when executing M06. (1=X, 2=Y, 3=Z, 4=W, 5=V) ..... *Section 3.3.5*
P704 Third axis to move when executing M06. (1=X, 2=Y, 3=Z, 4=W, 5=V) ..... *Section 3.3.5*
P705 Fourth axis to move when executing M06. (1=X, 2=Y, 3=Z, 4=W, 5=V) ..... *Section 3.3.5*
P706 Value of the residual S analog voltage. (1=2.5mV) ..... *Section 5.1*
P707 Oscillation period during range change ..... *Section 5.1*
P708 Feedrate override When analog voltage reaches 10V. ..... *Section 4.4*
P709 Subroutine associated with M06 ..... *Section 3.3.5*
P710 Subroutine associated with M22 ..... *Section 4.11*
P711 Subroutine associated with M23 ..... *Section 4.11*
P712 Subroutine associated with M24 ..... *Section 4.11*
P713 Subroutine associated with M25 ..... *Section 4.11*
P714 Error if actual feedrate is not within 50% and 200% of programmed value ..... *Section 4.4*
P715 Dwell between blocks in G07 (square corner). (1=10ms) ..... *Section 3.3.8*
P716 Distance between unidirectional approach point and programmed point ..... *Section 4.8*
P717 Spindle in-position zone during M19 ..... *Section 5.5*
P718 Spindle proportional gain K during M19 ..... *Section 5.5*
P719 Minimum spindle analog for M19. (1=2.5mV) ..... *Section 5.5*
P720 M associated to probing (G75) ..... *Section 3.3.4*
P721 ACCELERATION/DECELERATION for X. (1=20ms) ..... *Section 4.8*
P722 ACCELERATION/DECELERATION for Y. (1=20ms) ..... *Section 4.8*
P723 ACCELERATION/DECELERATION for Z. (1=20ms) ..... *Section 4.8*
P724 ACCELERATION/DECELERATION for W. (1=20ms) ..... *Section 4.8*
P725 Subroutine associated with function G74 ..... *Section 4.7*
P726 Recovery of programmed position on axes "without continuous control" ..... *Section 4.6*
P727 Emergency subroutine ..... *Section 3.3.7*
P728 ACCELERATION/DECELERATION for V. (1=20ms) ..... *Section 4.8*
P729 Maximum feedrate for circular interpolations ..... *Section 4.5*
P730 Nº of the node receiving the M, S, T functions ..... Fagor LAN
P731 Nº of the register of node P730 receiving the M, S, T functions ..... Fagor LAN
P732 FEED-FORWARD gain for X ..... *Section 4.7*
P733 FEED-FORWARD gain for Y ..... *Section 4.7*
P734 FEED-FORWARD gain for Z ..... *Section 4.7*
P735 FEED-FORWARD gain for W ..... *Section 4.7*
P736 FEED-FORWARD gain for V ..... *Section 4.7*
P737 Group of marks used by the CNC to send its internal data ..... Fagor LAN
P738 Group of marks used by the CNC to update the status of its connectors ..... Fagor LAN
P739 Group of marks used by the CNC to update its internal data ..... Fagor LAN
P740 Group of marks used by the CNC to update its additional internal data ..... Fagor LAN

P908 Collision zone between Y, Z ..... *Section 4.12*
P909 Collision zone between Y, Z ..... *Section 4.12*
P910 Minimum X coordinate of the probe ..... *Section 3.3.4*
P911 Maximum X coordinate of the probe ..... *Section 3.3.4*
P912 Minimum Y coordinate of the probe ..... *Section 3.3.4*
P913 Maximum Y coordinate of the probe ..... *Section 3.3.4*
P914 Minimum Z coordinate of the probe ..... *Section 3.3.4*
P915 Maximum Z coordinate of the probe ..... *Section 3.3.4*
P916 Spindle orient position when executing M19 without "S" ..... *Section 5.5*
P917 Lower limit of the forbidden zone for the spindle in M19 ..... *Section 5.5*
P918 Upper limit of the forbidden zone for the spindle in M19 ..... *Section 5.5*
P919 Offset of the X axis coded Io ..... *Section 4.6*
P920 Offset of the Y axis coded Io ..... *Section 4.6*
P921 Offset of the Z axis coded Io ..... *Section 4.6*
P922 Offset of the W axis coded Io ..... *Section 4.6*
P923 Offset of the V axis coded Io ..... *Section 4.6*

P924 thru P952 Not being used at this time (=0).

APPENDIX H

MACHINE PARAMETER SETTING CHART

Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE
P0		P3		P6		P9		P12	
P1		P4		P7		P10		P13	
P2		P5		P8		P11		P99	

Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE
P100		P200		P300		P400		P500	
P101		P201		P301		P401		P501	
P102		P202		P302		P402		P502	
P103		P203		P303		P403		P503	
P104		P204		P304		P404		P504	
P105		P205		P305		P405		P505	
P106		P206		P306		P4306		P506	
P107		P207		P307		P407		P507	
P108		P208		P308		P408		P508	
P109		P209		P309		P409		P509	
P110		P210		P310		P410		P510	
P111		P211		P311		P411		P511	
P112		P212		P312		P412		P512	
P113		P213		P313		P413		P513	
P114		P214		P314		P414		P514	
P115		P215		P315		P415		P515	
P116		P216		P316		P416		P516	
P117		P217		P317		P417		P517	
P118		P218		P318		P418		P518	
P119		P219		P319		P419		P519	

Parameter	VALUE	Parameter	VALUE	Parameter	VALUE
P600		P612		P624	
P601		P613		P625	
P602		P614		P626	
P603		P615		P627	
P604		P616		P628	
P605		P617		P629	
P606		P618		P630	
P607		P619		P631	
P608		P620		P632	
P609		P621		P633	
P610		P622		P634	
P611		P623		P635	

Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE
P700		P711		P722		P733		P744	
P701		P712		P723		P734		P745	
P702		P713		P724		P735		P746	
P703		P714		P725		P736		P747	
P704		P715		P726		P737		P748	
P705		P716		P727		P738		P749	
P706		P717		P728		P739		P750	
P707		P718		P729		P740		P751	
P708		P719		P730		P741		P752	
P709		P720		P731		P742		P753	
P710		P721		P732		P743			

Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE
P800		P805		P810		P815		P820	
P801		P806		P811		P816		P821	
P802		P807		P812		P817		P822	
P803		P808		P813		P818		P823	
P804		P809		P814		P819			

Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE
P900		P911		P922		P933		P944	
P901		P912		P923		P934		P945	
P902		P913		P924		P935		P946	
P903		P914		P925		P936		P947	
P904		P915		P926		P937		P948	
P905		P916		P927		P938		P949	
P906		P917		P928		P939		P950	
P907		P918		P929		P940		P951	
P908		P919		P930		P941		P952	
P909		P920		P931		P942		P953	
P910		P921		P932		P943			

APPENDIX I

DECODED "M" FUNCTION SETTING CHART

Auxiliary Function	VALUE
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	

APPENDIX J

LEADSCREW ERROR COMPENSATION SETTING CHART

Position		ERROR	
P0		P1	
P2		P3	
P4		P5	
P6		P7	
P8		P9	
P10		P11	
P12		P13	
P14		P15	
P16		P17	
P18		P19	
P20		P21	
P22		P23	
P24		P25	
P26		P27	
P28		P29	
P30		P31	
P32		P33	
P34		P35	
P36		P37	
P38		P39	
P40		P41	
P42		P43	
P44		P45	
P46		P47	
P48		P49	
P50		P51	
P52		P53	
P54		P55	
P56		P57	
P58		P59	

Position		ERROR	
P60		P61	
P62		P63	
P64		P65	
P66		P67	
P68		P69	
P70		P71	
P72		P73	
P74		P75	
P76		P77	
P78		P79	
P80		P81	
P82		P38	
P84		P85	
P86		P87	
P88		P89	
P90		P91	
P92		P93	
P94		P95	
P96		P97	
P98		P99	
P100		P101	
P102		P103	
P104		P105	
P106		P107	
P108		P109	
P110		P111	
P112		P113	
P114		P115	
P116		P117	
P118		P119	

Position		ERROR	
P120		P121	
P122		P123	
P124		P125	
P126		P127	
P128		P129	
P130		P131	
P132		P133	
P134		P135	
P136		P137	
P138		P139	
P140		P141	
P142		P143	
P144		P145	
P146		P147	
P148		P149	
P150		P151	
P152		P153	
P154		P155	
P156		P157	
P158		P159	
P160		P161	
P162		P163	
P164		P165	
P166		P167	
P168		P169	
P170		P171	
P172		P173	
P174		P175	
P176		P177	
P178		P179	

Position		ERROR	
P180		P181	
P182		P183	
P184		P185	
P186		P187	
P188		P189	
P190		P191	
P192		P193	
P194		P195	
P196		P197	
P198		P199	
P200		P201	
P202		P203	
P204		P205	
P206		P207	
P208		P209	
P210		P211	
P212		P213	
P214		P215	
P216		P217	
P218		P219	
P220		P221	
P222		P223	
P224		P225	
P226		P227	
P228		P229	
P230		P231	
P232		P233	
P234		P235	
P236		P237	
P238		P239	

APPENDIX K

CROSS COMPENSATION SETTING CHART

AXIS CAUSING THE ERROR:		AXIS SUFFERING THE ERROR: (To be compensated)	
Position		ERROR	
P180		P181	
P182		P183	
P184		P185	
P186		P187	
P188		P189	
P190		P191	
P192		P193	
P194		P195	
P196		P197	
P198		P199	
P200		P201	
P202		P203	
P204		P205	
P206		P207	
P208		P209	
P210		P211	
P212		P213	
P214		P215	
P216		P217	
P218		P219	
P220		P221	
P222		P223	
P224		P225	
P226		P227	
P228		P229	
P230		P231	
P232		P233	
P234		P235	
P236		P237	
P238		P239	

APPENDIX L

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 of the CNC are resistant to :

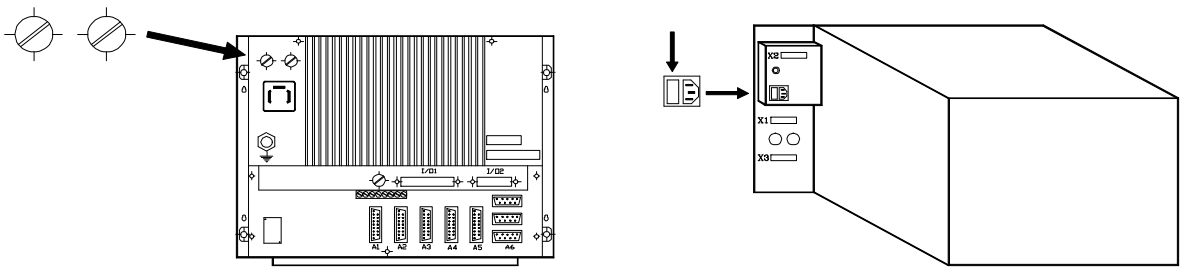
1.- Grease and mineral oils
2.- Bases and bleach
3.- Dissolved detergents
4.- Alcohol

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 Central Unit has two 3,15Amp./250V fast fuses (F), one per AC line.
See the label on the Monitor since it depends on the model.



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***
Central Unit	8030 MG 8030 MGI 8030 MS 8030 MSI	83360002 83360003 83360000 83360006	Fagor Automation	
14" Color Monitor		83420005	Fagor Automation	
Operator Panel		80300010	Fagor Automation	
Cable set for the Monitor	de 5 m de 10 m de 15 m	83640000 83640001 83640002	Fagor Automation	
Mains cable 3x0,75		11313000	Fagor Automation	
3,15A/250V Fuse		12130015	Schurter Wickmann	FST-034-1521 Ref. 19115
English Manual	OEM USER	83750039 8G900057	Fagor Automation	

ERROR CODES

001 This error occurs in the following cases:

> When the first character of the block to be executed is not an "N".

> When while BACKGROUND editing, the program in execution calls a subroutine located in the program being edited or in a later program.

The order in which the part-programs are stored in memory are shown in the part-program directory. If during the execution of a program, a new one is edited, this new one will be placed at the end of the list.

002 Too many digits when defining a function in general.

003 A negative value has been assigned to a function which does not accept the (-) sign or an incorrect value has been given to a canned cycle parameter.

004 A canned cycle has been defined while function G02, G03 or G33 was active.

005 Parametric block programmed wrong.

006 There are more than 10 parameters affected in a block.

007 Division by zero.

008 Square root of a negative number.

009 Parameter value too large

010 M41, M42, M43 or M44 has been programmed.

011 More than 7 "M" functions in a block.

012 This error occurs in the following cases:

> Function G50 is programmed wrong

> Tool dimension values too large.

> Zero offset values (G53/G59) too large.

013 Cycle defined incorrectly.

014 A block has been programmed which is incorrect either by itself or in relation with the program history up to that instant.

015 Functions G20, G21, G22, G23, G24, G25, G26, G27, G28, G29, G30, G31, G32, G50, G52, G53, G54, G55, G56, G57, G58, G59, G72, G73, G74, G92 and G93 must be programmed alone in a block.

016 The called subroutine or block does not exist or the block searched by means of special function F17 does not exist.

017 This error is issued in the following cases:

> Negative or too large thread pitch value.

> Function G95 or M19 has been used with machine parameter "P800=0".

018 Error in blocks where the points are defined by means of angle-angle or angle-coordinate.

019 This error is issued in the following cases:

> After defining G20, G21, G22 or G23, the number of the subroutine it refers to is missing.

> The "N" character has not been programmed after function G25, G26, G27, G28 or G29.

> Too many nesting levels.

020 The axes of the circular interpolation are not programmed correctly.

021 There is no block at the address defined by the parameter assigned to F18, F19, F20, F21, F22.

022 An axis is repeated when programming G74.

023 K has not been programmed after G04.

024 The decimal point is missing when programming T2.2 or N2.2.

025 Error in a definition block or subroutine call, or when defining either conditional or unconditional jumps.

026 This error is issued in the following cases:

> Memory overflow.

> Not enough free tape or CNC memory to store the part-program.

027 I/J/K has not been defined for a circular interpolation or thread.

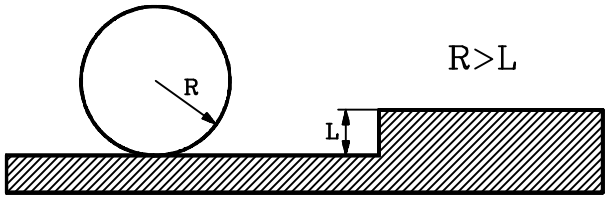
028 An attempt has been made to select a tool offset at the tool table or a non-existent external tool (the number of tools is set by machine parameter).

029 Too large a value assigned to a function.

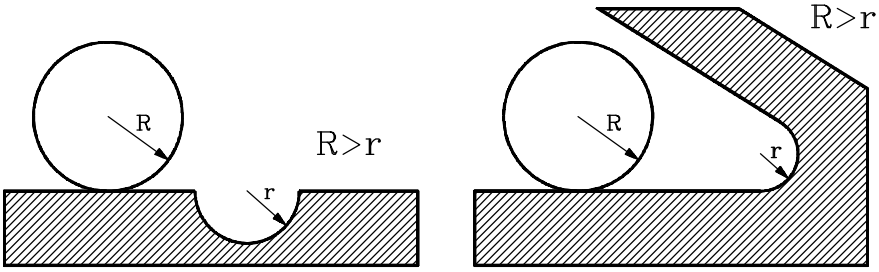
This error is often issued when programming an F value in mm/min (inch/min) and, then, switching to work in mm/rev (inch/rev) without changing the F value.

030 The programmed G function does not exist.

031 Tool radius value too large.



032 Tool radius value too large.



033 A movement of over 8388 mm or 330.26 inches has been programmed.

Example: Being the X axis position X-5000, if we want to move it to point X5000, the CNC will issue error 33 when programming the block N10 X5000 since the programmed move will be:
5000 - (-5000) = 10000 mm.

In order to make this move without issuing this error, it must be carried out in two stages as indicated below:

```
N10 X0          ; 5000 mm move
N10 X5000       ; 5000 mm move
```

034 S or F value too large.

035 Not enough information for corner rounding, chamfering or compensation.

036 Repeated subroutine.

037 Function M19 programmed incorrectly.

038 Function G72 or G73 programmed incorrectly.

It must be borne in mind that if G72 is applied only to one axis, this axis must be positioned at part zero (0 value) at the time the scaling factor is applied.

039 This error occurs in the following cases:

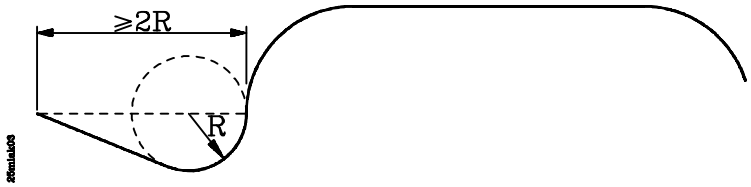
> More than 15 nesting levels when calling subroutines.

> A block has been programmed which contains a jump to itself. Example: N120 G25 N120.

040 The programmed arc does not go through the defined end point (tolerance 0.01mm) or there is no arc that goes through the points defined by G08 or G09.

041 This error is issued when programming a tangential entry as in the following cases:

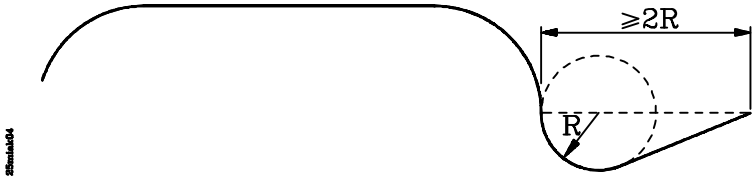
> There is no room to perform the tangential entry. A clearance of twice the rounding radius or greater is required.



> If the tangential entry is to be applied to an arc (G02, G03), The tangential entry must be defined in a linear block.

042 This error is issued when programming a tangential exit as in the following cases:

> There is no room to perform the tangential exit. A clearance of twice the rounding radius or greater is required.



> If the tangential exit is to be applied to an arc (G02, G03), The tangential exit must be defined in a linear block.

043 Polar origin coordinates (G93) defined incorrectly.

044 Canned cycle defined wrong.

045 Function G36, G37, G38 or G39 programmed incorrectly.

046 Polar coordinates defined incorrectly.

047 A zero movement has been programmed during radius compensation or corner rounding.

048 W axis programmed wrong.

049 Chamfer programmed incorrectly.

050 Functions M06, M22, M23, M24, M25 must be programmed alone in a block.

051 * A tool or pallet change cannot be performed without being in the change position.

052 * The requested tool is not in the magazine.

053 * This error occurs when having a machining center and 2 different external Ts have been programmed in a row without programming an M06 in between.

054 There is no disk in the FAGOR Floppy Disk Unit, there is no tape in the cassette reader or the reader head cover is open.

055 Parity error when reading or writing a floppy or cassette.

056 This error comes up in the following cases:

> When the memory is locked and an attempt is made to generate a CNC program by means of function G76.

> When trying to generate program P99999 or a protected program by means of function G76.

> If function G76 is followed by function G22 or G23.

> If there are more than 70 characters after G76.

> If function G76 (block content) has been programmed without having programmed G76 P5 or G76 N5 before.

> If in a G76 P5 or G76 N5 type function does not contain the 5 digits of the program number.

> If while a program is being generated (G76 P5 or G76 N5), its program number is changed without cancelling the previous one.

> If while executing a G76 P5 type block, the program referred to is not the one edited. In other words, that another one has been edited later or that a G76 P5 type block is executed while a program is being edited in background.

057 Write-protected floppy disk or cassette.

058 Irregular floppy drive motion or sluggish tape transport.

059 Communication error between the CNC and the FAGOR Floppy Disk Unit or between the CNC and the cassette reader.

060 Internal CNC hardware error. Consult with the Technical Service Department.

061 Battery error.

The memory contents will be kept for 10 more days (with the CNC off) from the moment this error occurs. The whole battery module located on the back must be replaced. Consult with the Technical Service Department.

Due to danger of explosion or combustion: do not try to recharge the battery, do not expose it to temperatures higher than 100°C (232°F) and do not short the battery leads.

064 * External emergency input (pin 14 of connector I/O1) is activated.

065 * This error comes up in the following cases:

> If while probing (G75) the programmed position is reached without receiving the probe signal.

> If while executing a probing canned cycle, the CNC receives the probe signal without actually carrying out the probing move itself (collision).

066 * X axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

067 * Y axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

068 * Z axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

069 * W axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

070 ** X axis following error.

071 ** Y axis following error.

072 ** Z axis following error.

073 ** W axis following error.

074 ** Spindle speed value too large.

075 ** X axis feedback error. Connector A1.

076 ** Y axis feedback error. Connector A2.

077 ** Z axis feedback error. Connector A3.

078 ** W axis feedback error. Connector A4.

079 ** Spindle feedback error. Connector A5.

080 ** Handwheel feedback error. Connector A5.

081 ** V axis feedback error. Connector **A**5.

082 ** Parity error in general parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

083 ** Parity error in V axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

084 * V axis travel limit overrun.

085 ** V axis following error.

086 Not being used at this time.

087 ** Internal CNC hardware error. Consult with the Technical Service Department.

088 ** Internal CNC hardware error. Consult with the Technical Service Department.

089 * All the axes have not been homed.

This error comes up when it is mandatory to search home on all axes after power-up. This requirement is set by machine parameter.

090 ** Internal CNC hardware error. Consult with the Technical Service Department.

091 ** Internal CNC hardware error. Consult with the Technical Service Department.

092 ** Internal CNC hardware error. Consult with the Technical Service Department.

093 ** Internal CNC hardware error. Consult with the Technical Service Department.

094 Parity error in tool table or zero offset table G53-G59. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

095 ** Parity error in W axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

096 ** Parity error in Z axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

097 ** Parity error in Y axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

098 ** Parity error in X axis parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

099 ** Parity error in M table. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

100 ** Internal CNC hardware error. Consult with the Technical Service Department.

101 ** Internal CNC hardware error. Consult with the Technical Service Department.

105 This error comes up in the following cases:
> A comment has more than 43 characters.
> A program has been defined with more than 5 characters.
> A block number has more than 4 characters.
> Strange characters in memory.

106 ** Inside temperature limit exceeded.

107 ** Error in W axis leadscrew error compensation parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

108 ** Error in Z axis leadscrew error compensation parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

109 ** Error in Y axis leadscrew error compensation parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

110 ** Error in X axis leadscrew error compensation parameters. The CNC resets the serial line RS232C machine parameters: "P0= 9600", "P1=8", P2=0", "P3=1", "P607(3)=1", "P607(4)=1", "P607(5)=1".

111 * FAGOR LAN line error. Hardware installed incorrectly.

112 * FAGOR LAN error. It comes up in the following instances:

> When the configuration of the LAN nodes is incorrect.
> The LAN configuration has been changed. One of the nodes is no longer present (active).

When this error occurs, access the LAN mode, editing or monitoring, before executing a program block.

113 * FAGOR LAN error. A node is not ready to work in the LAN. For example:

> The PLC64 program is not compiled.
>A G52 type block has been sent to an 82CNC while it was in execution.

114 * FAGOR LAN error. An incorrect command has been sent out to a node.

115 * Watch-dog error in the periodic module.

This error occurs when the periodic module takes longer than 5 milliseconds.

116 * Watch-dog error in the main module.

This error occurs when the main module takes longer than half the time indicated in machine parameter "P741".

117 * The internal CNC information requested by activating marks M1901 thru M1949 is not available.

118 * An attempt has been made to modify an unavailable internal CNC variable by means of marks M1950 thru M1964.

119 Error when writing machine parameters, the decoded M function table and the leadscrew error compensation tables into the EEPROM memory.

This error may occur when after locking the machine parameters, the decoded M function table and the leadscrew error compensation tables, one tries to save this information into the EEPROM memory.

120 Checksum error when recovering (restoring) the machine parameters, the decoded M function table and leadscrew error compensation tables from the EEPROM memory.

150 Incoherent data in the 512 Kb memory.

When this error occurs, save as many programs as you can into the Floppy Disk Unit, peripheral or PC.

Then, proceed as follows to format the 512 Kb memory (when doing this, all part-programs stored in this memory will be lost).

Press **[OP MODE] [6]** to select the Editing mode.
Press **[LOCK/UNLOCK]** the screen displays the text: CODE:
Key in: **FM512** and press **[ENTER]**

Once the 512 Kb memory is formatted, recover (restore) the programs you saved into the Floppy Disk Unit, peripheral or PC.

151 Defective 512 Kb memory. Consult with the Technical Service department.

152 Not enough available free space in the 512 Kb memory.

Atention:

The **ERRORS** indicated with "*" behave as follows:

They stop the axis feed and the spindle rotation by cancelling the Enable signals and the analog outputs of the CNC.

They interrupt the execution of the part-program of the CNC if it was being executed.

The **ERRORS** indicated with "**" besides behaving as those with an "*", they activate the INTERNAL EMERGENCY OUTPUT.

FAGOR 8025/8030 CNC

Models: M, MG, MS, GP

FAGOR LOCAL AREA NETWORK

Ref. 9701 (in)

ABOUT THE INFORMATION IN THIS MANUAL

This manual is addressed to the machine manufacturer and must only be used when installing the 8025 CNC in the FAGOR Local Area Network (LAN).

Chapter 1 "FAGOR LAN Configuration" explains what this LAN is all about and how to interface its various components or elements.

Chapter 2. "The 8025 CNC in the FAGOR LAN" describes:

* How to set the 8025 CNC to operate in the LAN.
* What internal information of this CNC is accessible to the other FAGOR LAN elements.
* How to access the information of the other FAGOR LAN elements from this CNC.

To find out, in further detail, how to set up and use any other element of the FAGOR LAN , we recommend to read its corresponding manual.

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

FAGOR AUTOMATION, S. Coop. Ltda. reserves the right to modify the contents of this manual without prior notice.

INDEX

Section Page

1. FAGOR LOCAL AREA NETWORK CONFIGURATION

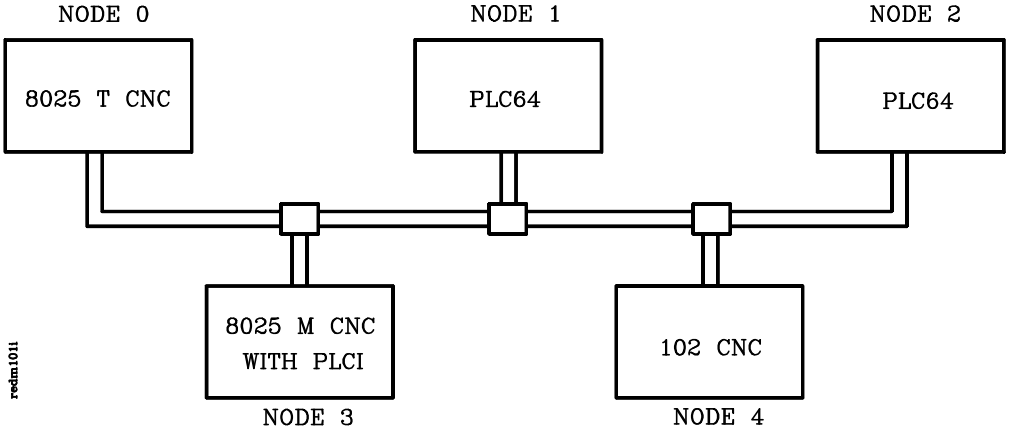
1.1 INTRODUCTION

The FAGOR Local Area Network (LAN) is a Token-Passing-Bus type communication network that allows the interconnection of up to 15 elements (nodes).

The different elements that may be interconnected via the FAGOR LAN are:

CNC 82
CNC 101S, 102, 102S
CNC 800MG (with or without integrated PLC)
CNC 800 T or TG (with or without integrated PLC)
CNC 8025 GP
CNC 8025 M, MG or MS (with or without integrated PLC)
CNC 8025 T, TG or TS (with or without integrated PLC)
CNC 8025 P, PG or PS (with or without integrated PLC)
PLC64

Example:



Each element occupies a NODE of the network and all of them have a number of machine parameters to configure the network.

The numbering of the nodes must begin by "0" and it must be sequential. Node "0" is the Main Node and the element connected to it sets or determines the total number of nodes being installed in the LAN.

This LAN utilizes the ROTARY MASTER system; that is, the different nodes temporarily assume the master function so they can communicate directly with each other.

1.2 LAN INTERFACE

The various elements must be interconnected via an RS485 serial line.

The RS485 connector for each element is a SUB-D type 9-pin female connector with the following pin-out:

PIN	SIGNAL	FUNCTION
1	FG	Shield
2	—-	Not connected
3	TxD	Transmit Data
4 5 6 7	—- —- —- —-	Not connected Not connected Not connected Not connected
8	TxD	Transmit Data
9	—-	Not connected

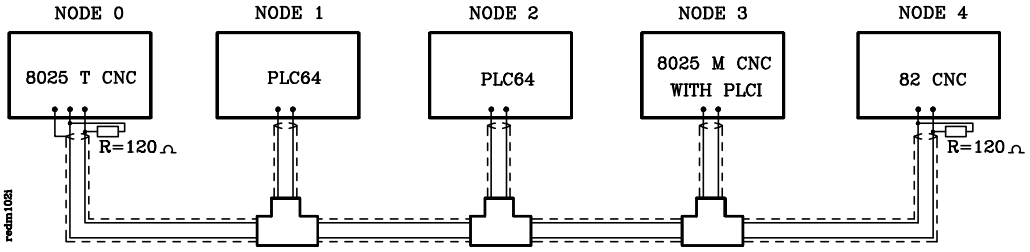
The nodes must be connected via a "TWINAXIAL" cable which must also meet the following requirements:

Conductor	Type: Material: Resistance:	02 AWG twisted pair 7x28 Copper (only 1 stained conductor) Maximum 11 L every 305m. (1000 ft)
Insulation	Material:	Teflon
Shield	Material Type Covering Resistance	Stained copper Twisted 34 AWG. 8 ends / 16 carriers Minimum 95% Maximum 3L every 305m. (1000 ft)
Cover	Material: Outside Diameter	Teflon Nominal 7mm. (0.257 inch)
Capacitance		Maximum 53,1 pF/m (16.2 pF/ft)
Impedance		107± 5% Ohms at 1 MHz.

To interconnect the nodes follow these considerations:

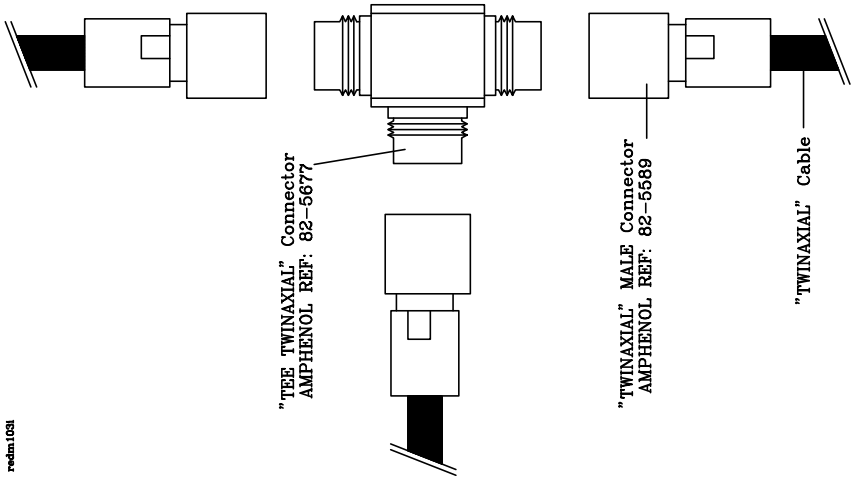
* The shield must be connected only to one of the LAN nodes using pin 1 of the corresponding connector.

* A 120-Ohm-1/4w terminating resistor must be connected between pins 3 and 8 of the nodes most distant from each other.

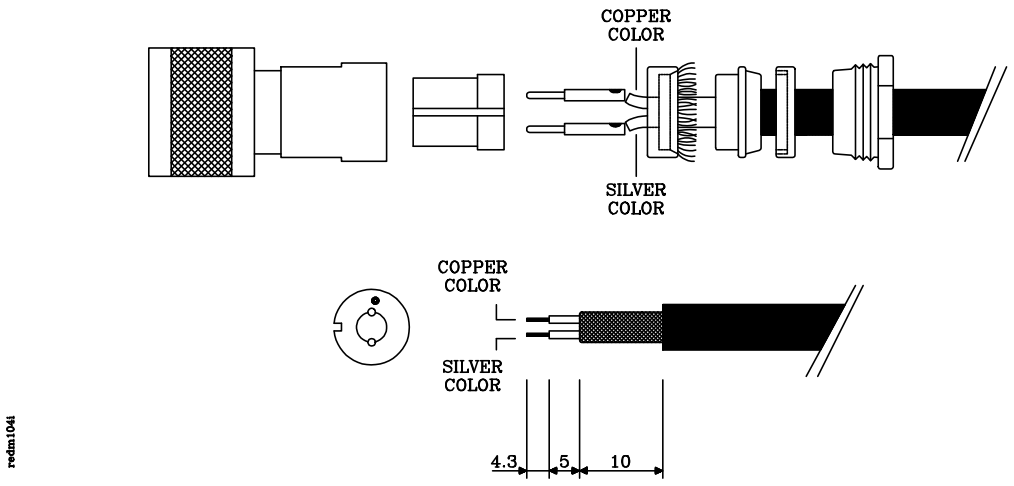


* When the FAGOR LAN consists of more than 2 nodes, "TEE TWINAXIAL" connectors must be used as indicated below.

  It must be borne in mind that the maximum cable length allowed between a "T Twinaxial" connector and the element is 80cm (31.49 inches).



MOUNTING DETAIL FOR THE "TWINAXIAL" CABLE



Before connecting or disconnecting the FAGOR LAN connectors, ALL the elements must be powered OFF.

Also, before connecting the FAGOR LAN, all the elements must be connected to ground and verified that between their 0V points:

- there is continuity (0 Ohms) (with the elements powered off).
- or there is no voltage difference (0 V) (with the elements powered on).

If this condition is not met, the internal circuitry of some elements may be damaged.

2. *THE 8025 CNC IN THE FAGOR LAN*

2.1 INTRODUCTION

Any PLC64 installed in the FAGOR Local Area Network can access the internal information of any CNC installed in this LAN. It can also:

* Know the number of the program being executed, the currently selected feedrate F, the axes currently in motion, etc.

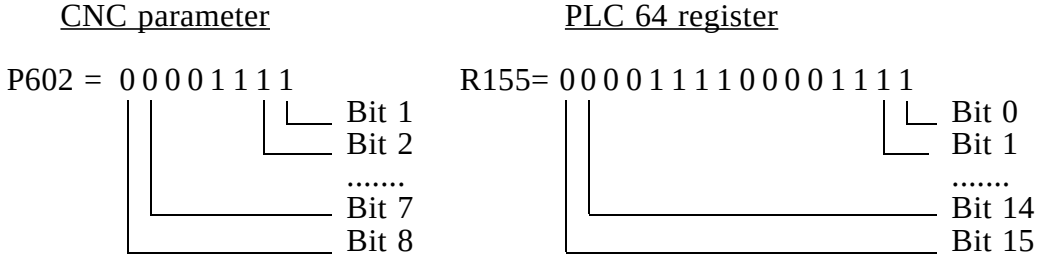
* Inhibit the axes, display messages at the CNC, vary the spindle speed override or the Feedrate override, simulate the CNC keyboard, etc.

Also, the 8025 CNC may intervene in the information exchange carried out by means of marks. This makes it possible, for example:

* To intercalate a PLC64 between the core of the CNC and its connectors in such a way that all the connector inputs are treated previously by the PLC64 so it "tells" the CNC what to do and it will also process the CNC output information deciding the activating or deactivating of the physical outputs at the connectors.

2.2 SETTING THE 8025 CNC IN THE FAGOR LAN

When this manual mentions a CNC parameter bit or a PLC64 register bit, it refers to the following nomenclature:



To install the 8025 CNC in the FAGOR LAN, the following machine parameters must be set:

P615(5) The CNC occupies the main node of the LAN

It indicates whether the CNC is NODE 0 (main node) or not.

P615(5) = 0 It is not the main node (0)
P615(5) = 1 It is the main node

P615(4, 3, 2, 1) Node number of the CNC or number of nodes in the FAGOR LAN

When the CNC occupies the main node, they indicate how many more nodes there are in the LAN. When the CNC does not occupy the main node, they indicate the node number it occupies.

P615(4)	P615(3)	P615(2)	P615(1)	P615(5)=1 CNC at Node 0		P615(5)=0
				Nodes besides Node 0	Total nodes in the LAN:	CNC at Node:
0	0	0	0	Wrong		Wrong
0	0	0	1	1	2	1
0	0	1	0	2	3	2
0	0	1	1	3	4	3
0	1	0	0	4	5	4
0	1	0	1	5	6	5
0	1	1	0	6	7	6
0	1	1	1	7	8	7
1	0	0	0	8	9	8
1	0	0	1	9	10	9
1	0	1	0	10	11	10
1	0	1	1	11	12	11
1	1	0	0	12	13	12
1	1	0	1	13	14	13
1	1	1	0	14	15	14
1	1	1	1	Wrong		Wrong

Examples:

The following elements are connected through the FAGOR LAN, 2 PLC64 (nodes 0 and 1) and one 8025M CNC (node 2).

8025M CNC	P615(5)	P615(4)	P615(3)	P615(2)	P615(1)
Node 2	0	0	0	1	0

The following elements are connected through the FAGOR LAN, one 8025M CNC (node 0) and 2 PLC64 (nodes 1 and 2).

CNC 8025M	P615(5)	P615(4)	P615(3)	P615(2)	P615(1)
Node 0	1	0	0	1	0

Once the CNC has been installed in the Fagor LAN, parameter P615(5, 4, 3, 2, 1), the following machine parameters related to the LAN must also be set:

P614 CNC's LAN identification parameter

Any element interconnected via LAN may read the contents of this parameter which is set arbitrarily by the manufacturer of the machine in order to "personalize", identify or distinguish it from other similar machines as he wishes.

Atention:

This parameter has absolutely no affect on the operation and performance of the CNC.

P620(3) Transfer-inhibit and M-done inputs independent from Feed-hold

If "P620(3) =0", the 8025 M CNC utilizes pin 15 of connector I/O1 as input for Feed-Hold, Transfer-Inhibit and M-done signals. This is described in full detail in chapter 1, section "Input of connector I/O 1", of the Installation manual.

If "P620(3)=1", the CNC acts as follows:

* The Feed-Hold input will be "taken" at pin 15 of connector I/O1.

* The Transfer-Inhibit input will be "taken" from the LAN. Its status may be set by any PLC64 installed in the LAN. To do this, use mark M1955 and bit "0" of register R155 at the PLC64.

* The M-done input is also taken from the LAN. Its status may be set by any PLC64 installed in the LAN. To do this, use mark M1955 and bit "1" of register R155 at the PLC64.

P622(7) The Transfer-Inhibit signal does not act upon the M, S, T functions

The CNC checks this parameter when "P620(3)=1".

If during the execution of a block, the Transfer Inhibit signal is set low (0V), the CNC checks the status of parameter P622(7) and acts as follows:

P622(7)=0 Once the current block is finished (in position), the program execution is interrupted and it will be resumed when this signal is set back high (24V).

P622(7)=1 Once the current block is finished (in position), the CNC checks the next program block:

* If it contains M, S, T type information, it executes it and analyzes the next block.

* If it contains other type of information, movement, etc. it interrupts the execution of the program and it will resume it when this signal is set back high (24V).

This makes it possible, for example, to make a tool change consisting in two blocks: "N100 T2.2" and "N110 M06" interrupting the execution of the program until the tool change is completed.

P613(3) There is a PLC64 in the LAN

This machine parameter indicates to the CNC whether there is or not a PLC64 installed in the FAGOR LAN.

Whenever the CNC executes an M, S or T function, it checks this parameter and if there is a PLC64 installed in the Fagor LAN it sends, via LAN, the code of the M, S or T function that has been executed.

This data is sent to the register specified by parameter "P731" of the node indicated by parameter "P730".

P730 Number of the node receiving the M, S, T function codes
P731 Number of the register receiving the M, S, T function codes at node P730

When there is a PLC64 installed in the LAN, "P613(3)=1", the CNC needs to know to which node it has to send the M, S, T function codes and in which registers of the PLC64 installed at that node it must deposit this code.

Parameter "P730" indicates the number of the node receiving the M, S, T function codes.

Atention:

If "P730=15", the CNC sends that information to all the nodes occupied by PLC64s.

Parameter "**P731**" indicates the number of the PLC64 register where the **M function** code is to be deposited. The codes corresponding to the S and T functions will be transferred to the next two registers.

Example: To send the M, S, T function codes to registers R20, R21 and R22 of the PLC64 connected to Node 3.

P730=3 Send this data to Node 3

P731=20 M function code will be deposited in register R20
S function code will be deposited in register R21
T function code will be deposited in register R22

P620(1) PLC messages received via marks

There are 3 ways to generate, from the PLC64, an error or error message at the CNC: Using the MSG instruction, marks M1801 through M1899 or the M1951-R151 combination.

The CNC needs to know the system being used and it will act differently as described below:

"P620(1)=1" Marks M1801 through M1899 are used to generate errors or messages at the CNC. Neither the MSG instruction nor the M1951-R151 combination can be used.

The CNC displays the associated text, defined in program P99998, corresponding to the mark with the highest priority and it allows access to the "Messages" option which lists all the messages and errors activated from the PLC.

"P620(1)=0" The MSG instruction as well as the M1951 - R151 combination may be used. Marks M1801 through M1899 cannot be used to generate errors or messages at the CNC.

When using an M1951-R151 combination, the message or error number must be selected at register R151 and activate mark M1951 (logic state "1")

In both cases, the CNC displays the associated text previously defined in program P99998.

In this case, only the number of the last message or error activated from the PLC is displayed.. To delete it, press [DELETE].

Atention:

It is recommended to use Marks M1801 through M1899 at the PLC64 to generate errors or messages at the CNC.

This method, besides being simpler, requires less execution time. Therefore, the PLC64 cycle scan is quicker.

2.3 DATA EXCHANGE BETWEEN AN 8025 CNC AND THE REST OF THE NODES OF THE FAGOR LAN

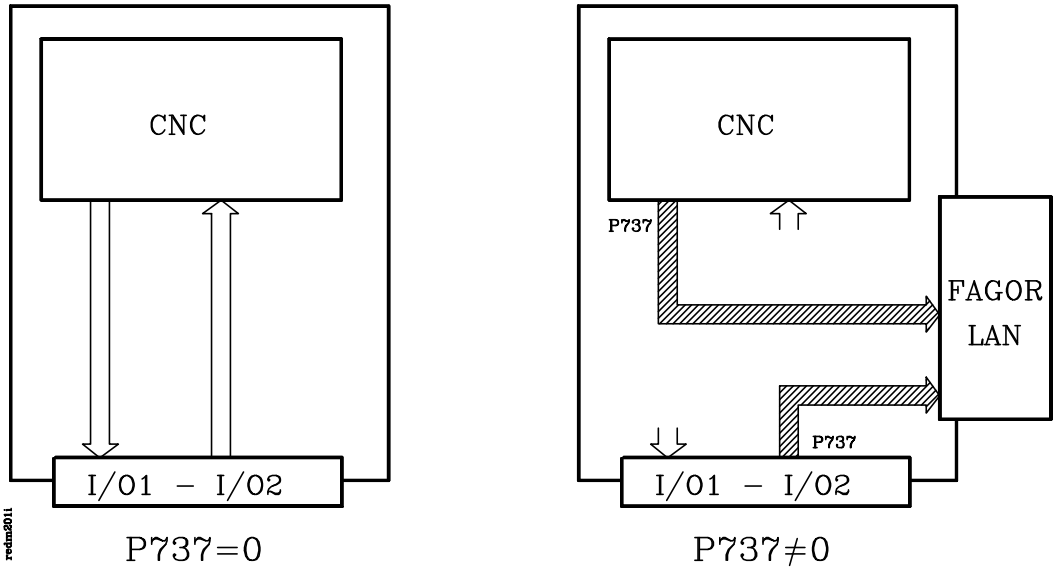
The CNC has 8 groups of 64 internal marks (relays) for data exchange with the rest of the elements of the FAGOR LAN.

Group 1	M1	through M64	Group 5	M257	through M320
Group 2	M65	through M128	Group 6	M321	through M384
Group 3	M129	through M192	Group 7	M385	through M448
Group 4	M193	through M256	Group 8	M449	through M512

It is also necessary to set the following machine parameters.

P737 Group of marks used by the CNC to send its internal data

It indicates the group of marks used by the CNC to send to the rest of the elements the status of the connectors (inputs) and of the outputs of the CNC itself.



The table below shows which connector inputs are sent by the CNC to the LAN and the marks where that information is deposited. It is assumed that P737=1, first group of marks. If P737=3, the "conditional Input" will be M(10+128) = M138.

INPUT	PIN	P737=1	P737=3
		MARK	MARK
Start / Rapid move / Enter in Play-back mode	17 (Connector I/O1)	M1	M129
Stop / Emergency Subroutine	16 (Connector I/O1)	M2	M130
Feed-Hold	15 (Connector I/O1)	M3	M131
Emergency Stop	14 (Connector I/O1)	M4	M132
W home switch	13 (Connector I/O1)	M5	M133
Z home switch	12 (Connector I/O1)	M6	M134
Y home switch	11 (Connector I/O1)	M7	M135
X home switch	10 (Connector I/O1)	M8	M136
DRO mode	19 (Connector I/O1)	M9	M137
Conditional Input (block skip)	18 (Connector I/O1)	M10	M138
V home switch	7 (Connector A5)	M11	M139

The table below shows the CNC outputs sent out to the LAN and the marks where that information is deposited. It has been assumed that P737=1. If P737=3, the "X axis Enable" signal will be M(20+128) = M148.

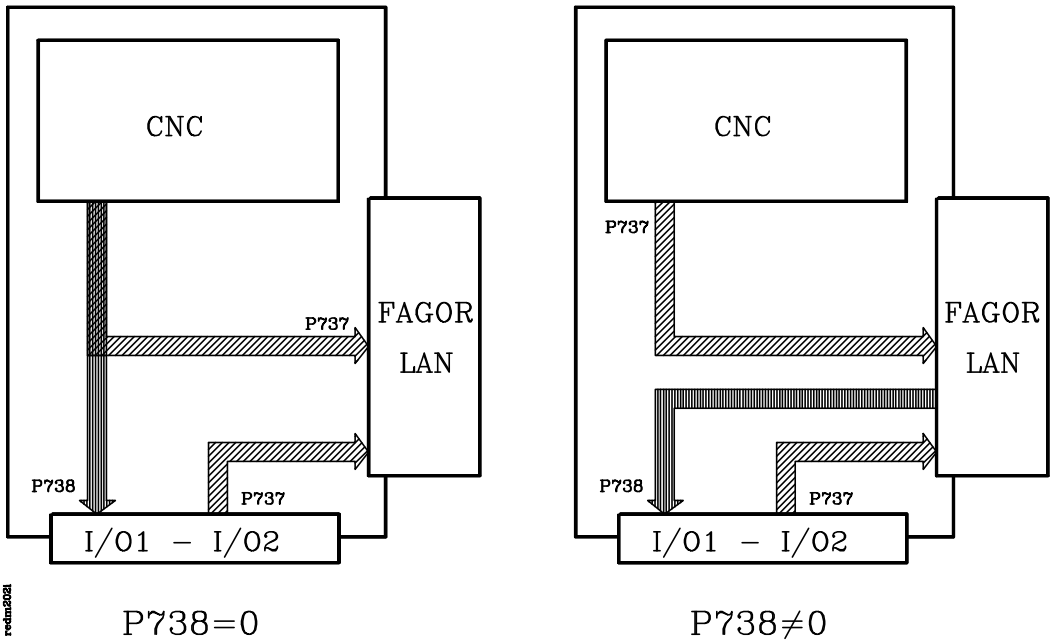
CNC Output	MARK	CNC Output	MARK
BCD code, weight 1	M12	Bit 9 of M function table	M36
BCD code, weight 2	M13	Bit 10 of M function table	M37
BCD code, weight 4	M14	Bit 11 of M function table	M38
BCD code, weight 8	M15	Bit 12 of M function table	M39
BCD code, weight 10	M16	Bit 13 of M function table	M40
BCD code, weight 20	M17	Bit 14 of M function table	M41
BCD code, weight 40	M18	Bit 15 of M function table	M42
BCD code, weight 80	M19	Jog mode	M43
X axis Enable	M20	V axis Enable	M44
Y axis Enable	M21	Special Tool Change	M45
Z axis Enable	M22	Direction of Vertical Axis	M46
W axis Enable	M23	RESET	M47
T Strobe.	M24	Tool Magazine rotating direction	M48
S Strobe	M25	W axis in motion	M49
M Strobe	M26	Z axis in motion	M50
Emergency Output	M27	Y axis in motion	M51
Bit 1 of M function table	M28	X axis in motion	M52
Bit 2 of M function table	M29	V axis in motion	M53
Bit 3 of M function table	M30	CNC in execution /CYCLE ON	M54
Bit 4 of M function table	M31	CNC interrupted	M55
Bit 5 of M function table	M32	Error	M56
Bit 6 of M function table	M33	Threading ON	M57
Bit 7 of M function table	M34	Automatic mode	M58
Bit 8 of M function table	M35	Rapid traverse (G00)	M59

Atention:

If the CNC intervenes in the data exchange carried out via marks, P737 must be set to "0". The CNC ignores parameters "P738", "P739" and "P740".

P738 Group of marks used by the CNC to update the status of the connectors

It indicates the group of marks from where the CNC takes the information to update the outputs corresponding to connectors I/O1 and I/O2. It must have a value different than "P737"; otherwise, there will be redundant data.



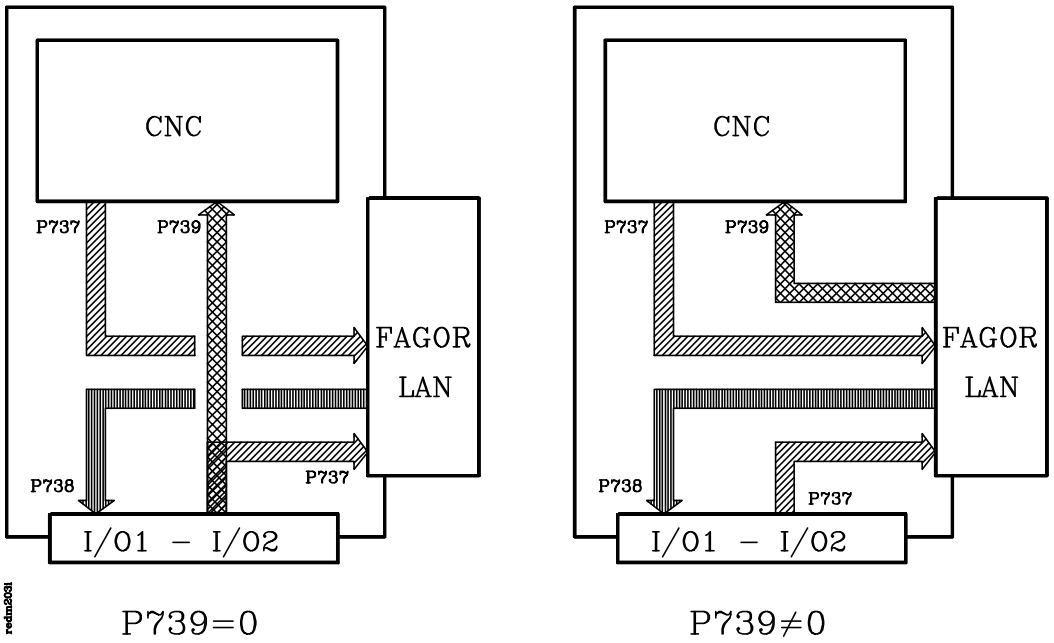
If "P738=0" The CNC will update the output status with the values of the CNC itself as if there were no LAN. It must be borne in mind that this very information is also sent out via LAN to the group indicated by "P737".

The table below shows the signal assignment to the outputs of connectors I/O1 and I/O2 and their corresponding marks. It has been assumed that "P738=1", first group of marks. If P738=3, the "T Strobe" signal will be M(29+128) = M157.

PIN	SIGNAL	MARK	PIN	SIGNAL	MARK
27 (I/O1)	BCD code, weight 1	M17	3 (I/O2)	Bit 1 of M function table	M33
26 (I/O1)	BCD code, weight 2	M18	4 (I/O2)	Bit 2 of M function table	M34
25 (I/O1)	BCD code, weight 4	M19	5 (I/O2)	Bit 3 of M function table	M35
24 (I/O1)	BCD code, weight 8	M20	6 (I/O2)	Bit 4 of M function table	M36
23 (I/O1)	BCD code, weight 10	M21	7 (I/O2)	Bit 5 of M function table	M37
22 (I/O1)	BCD code, weight 20	M22	8 (I/O2)	Bit 6 of M function table	M38
21 (I/O1)	BCD code, weight 40	M23	9 (I/O2)	Bit 7 of M function table	M39
20 (I/O1)	BCD code, weight 80	M24	10 (I/O2)	Bit 8 of M function table	M40
9 (I/O1)	X axis Enable	M25	11 (I/O2)	Bit 9 of M function table	M41
8 (I/O1)	Y axis Enable	M26	12 (I/O2)	Bit 10 of M function table	M42
7 (I/O1)	Z axis Enable	M27	13 (I/O2)	Bit 11 of M function table	M43
6 (I/O1)	W axis Enable	M28	25 (I/O2)	Bit 12 of M function table	M44
2 (I/O1)	T Strobe.	M29	24 (I/O2)	Bit 13 of M function table	M45
3 (I/O1)	S Strobe	M30	23 (I/O2)	Bit 14 of M function table	M46
4 (I/O1)	M Strobe	M31	22 (I/O2)	Bit 15 of M function table	M47
5 (I/O1)	Emergency	M32	21 (I/O1)	Jog mode	M48

P739 Group of marks used by the CNC to update its internal data

It indicates the group of marks used by the CNC to update its internal data. It must have a value different from that of "P737"; otherwise, there will be redundant data.



If "P739=0", the CNC updates its internal data with the values corresponding to the inputs at connectors I/O1 and I/O2 as if there were no LAN. It must be borne in mind that the same information is also sent out via LAN to the group indicated by "P737".

The following table shows the internal variables and their corresponding marks. It has been assumed that "P739=1", first group of marks. If "P739=3", the "Emergency Stop" signal will be M(4+128) = M132.

INTERNAL CNC VARIABLE	MARK
Start / Rapid move / Enter in Play-back mode	M1
Stop / Emergency Subroutine	M2
Feed-Hold	M3
Emergency Subroutine	M4
W axis home switch	M5
Z axis home switch	M6
Y axis home switch	M7
X axis home switch	M8
DRO mode	M9
Conditional Input (block skip)	M10
V axis home switch	M11

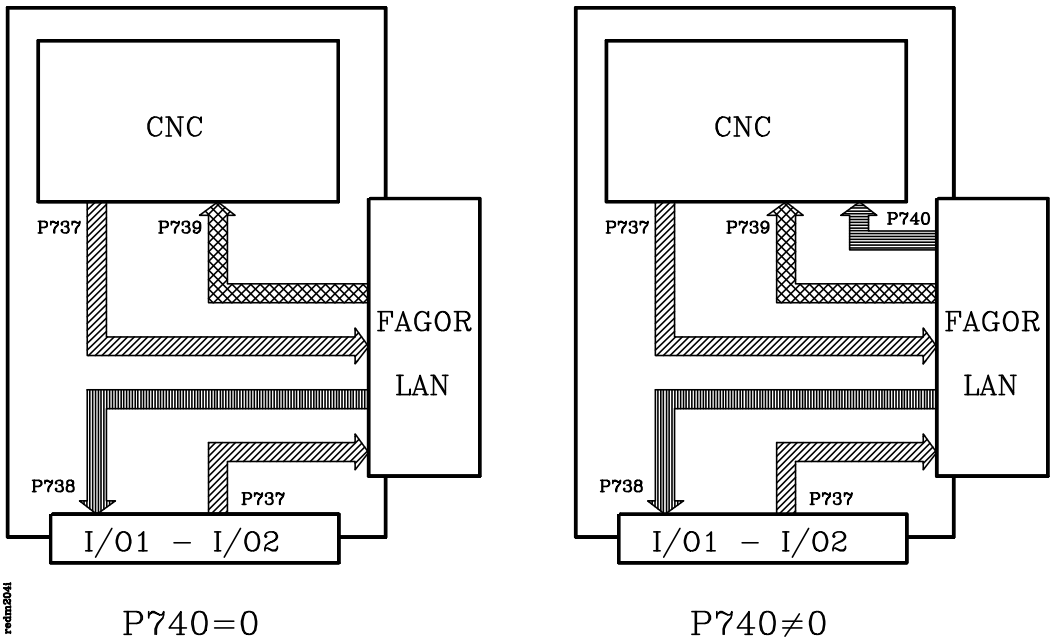
P740 Group of marks used by the CNC to update its additional internal data

Depending on the value assigned to P740, the CNC updates its additional internal data in one of the following ways:

* If "P740=0", the CNC does not use the groups of marks. A system call must be made while using mark M1955 and register R155 at the PLC64.

* If "P740<>0", the CNC uses the groups of marks.

  The number assigned to "P740" indicates the group of marks that the CNC uses to update its additional internal data. It must have a value different from that of "P737"; otherwise, there will be redundant data.



The next table shows the additional internal variables and the corresponding marks or register bits. As for the marks, it has been assumed that "P740=1", first group of marks. If "P740=3", the "M-done" signal will be M(50+128) = M178.

INTERNAL CNC VARIABLE	P740=0	P740=1
	PLC REGISTER	MARK
Transfer-Inhibit	R155 bit 0	M49
M-done	R155 bit 1	M50
Double cross-compensation	R155 bit 4	M53

Application example:

An 8025 CNC and PLC64 are being used in such a way that the information received by the CNC from the outside (inputs at connectors I/O1 and I/O2) will be previously handled by the PLC64.

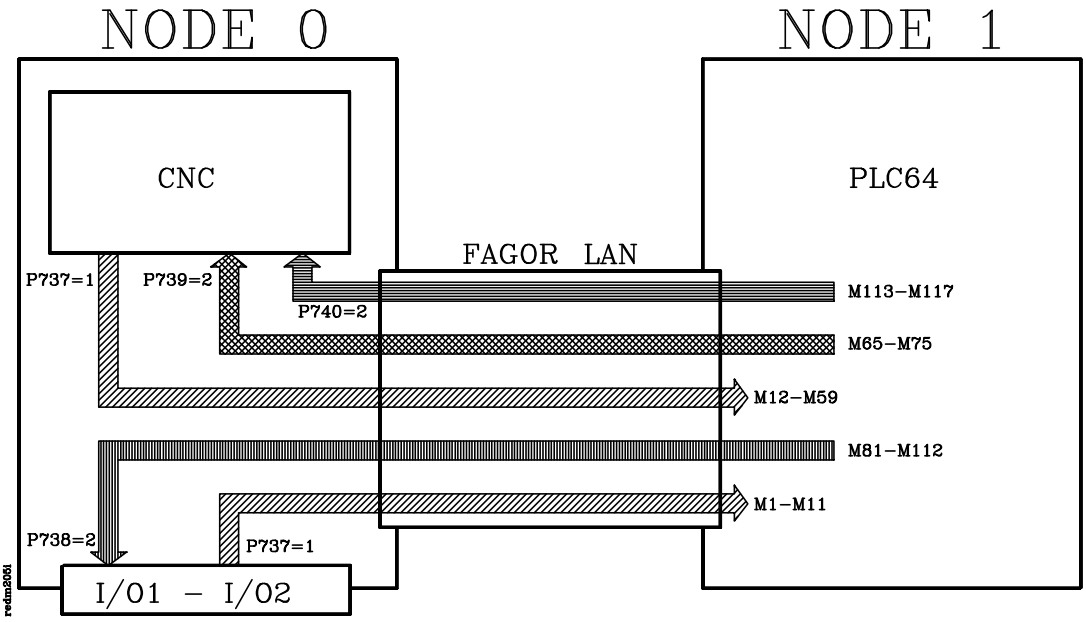
The CNC outputs are also to be handled by the PLC64 which will update the outputs at connectors I/O1 and I/O2.

To do this, both elements must be interconnected via RS485 and the LAN parameters set accordingly.

In this example, the CNC occupies Node 0 and the PLC64 Node 1.

CNC8025: P615(5)=1
P615(4)=0, P615(3)=0, P615(2)=0, P615(1)=1

PLC64: DS4=0
DS8=0, DS7=0, DS6=0, DS5=1



The first group of marks (M1 through M64) is being used to send internal CNC data to the PLC64 and the second group (M65 through M128) to update the internal CNC data as well as the status of connectors I/O1 and I/O2.

Parameters to be set **at the CNC**:

P737=1 The CNC sends to the PLC64, on marks M1 thru M11, the input status (connectors I/O1 and I/O2).

The CNC sends to the PLC64, on marks M12 thru M59, the status of the CNC outputs.

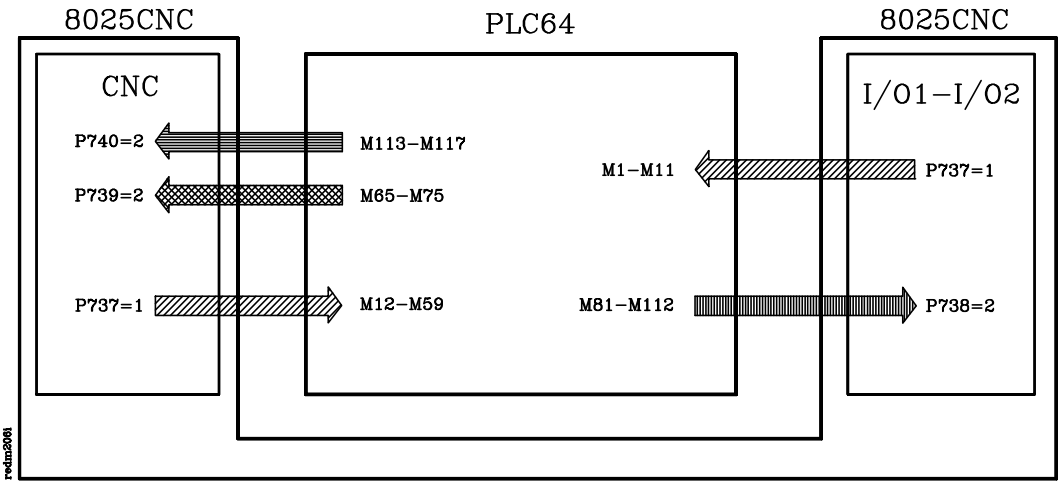
P738=2 The CNC updates the status of the outputs (connectors I/O1 and I/O2) with the value handed by the PLC64 on marks M87 thru M112.

P739=2 The CNC updates its internal data with the value handed by the PLC64 on marks M65 thru M75.

P740=2 The CNC updates its additional internal data with the value handed by the PLC64 on marks M113 thru M117.

Parameters to be set **at the PLC64**:

R240 bit 11 = 0
R240 bit 10 = 0
R240 bit 9 = 1
R240 bit 8 = 0 The PLC64 sends out to the CNC the contents of marks: M65 through M128



2.4 DATA EXCHANGE VIA MARKS BETWEEN AN 8025 CNC WITH PLCI AND THE REST OF THE LAN NODES

When the CNC has an integrated PLC, is the PLCI who intervenes in the data exchange via marks. The PLCI has 8 groups of 64 marks to exchange information with the rest of the elements of the Fagor LAN.

Group 1	M1 to M64	Group 5	M257 to M320
Group 2	M65 to M128	Group 6	M321 to M384
Group 3	M129 to M192	Group 7	M385 to M448
Group 4	M193 to M256	Group 8	M449 to M512

The PLCI will use one of these groups of marks to send its own data to the LAN and can access to the rest of the groups to know the information deposited on them by the other elements of the LAN.

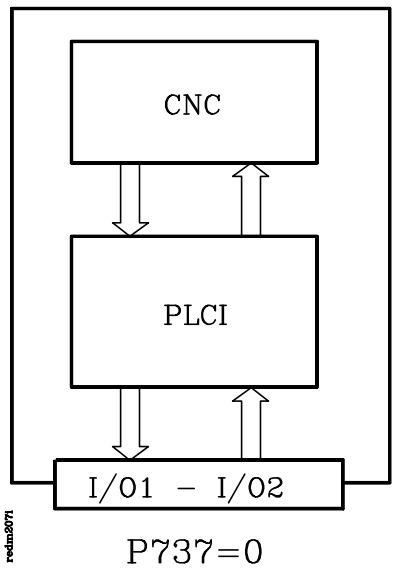
To do this, the following machine parameter must be set:

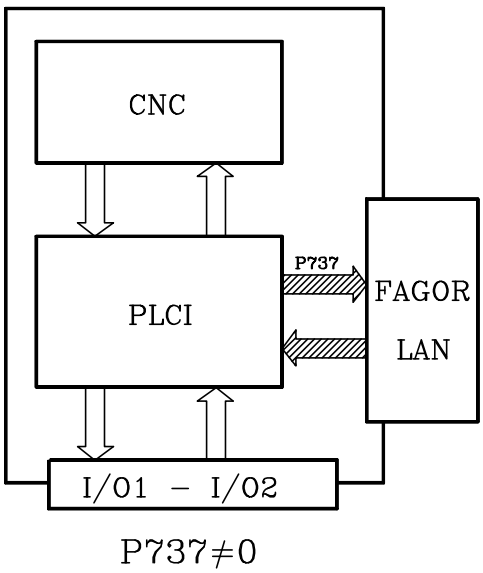
P737 Group of marks used by the PLCI to send its internal data

It indicates the group of marks used by the PLCI to send its internal data to the other elements of the LAN.

The meaning of each mark is set by the machine manufacturer when editing the PLCI program.

Parameters "P738", "P739" and "P740" are no longer used when the 8025 CNC has a PLCI.





Application example:

A PLC64 is used as I/O expansion for an 8025 CNC with PLCI.

Since the 8025 CNC with PLCI offers 40 inputs and 24 outputs and the PLC64 offers 64 inputs and 32 outputs, the PLCI will then control a total of:

Inputs: 40+64 = 104
Outputs: 24+32 = 56

In this example, the 8025 CNC with PLCI will occupy Node 0 and the PLC64 Node 1.

8025 CNC: P615(5)=1
P615(4)=0, P615(3)=0, P615(2)=0, P615(1)=1

PLC64: DS4=0
DS8=0, DS7=0, DS6=0, DS5=1

The PLC64 uses the first group of marks (M1 thru M64) to send the status of the inputs and the CNC uses the second group (M65 thru M128) to update the outputs of the PLC64.

8025 CNC: P737=2
PLC64: R240(8)=1, R240(9)=0, R240(10)=0, R240(11)=0

The input output nomenclature at the CNC and the PLC64 as well as in the PLCI program is as follows:

CALLED AT THE PLCI		I1 to I40	M1 to M64	O1 to O24	M65 to M96
CNC	Inputs	I1 to I40			
	Outputs			O1 to O24	
PLC64	Inputs		I1 to I64		
	Outputs				O1 to O32

The program of the PLC64 must update marks M1 through M64 with the status of its inputs, and outputs O1 thru O32 with the status of marks M65 through M96. Thus, the PLC64 program will be:

```
M2047 = MOV I1 M1 16
      = MOV I17 M17 16
      = MOV I33 M33 16
      = MOV I49 M49 16
      = MOV M65 O1 16
      = MOV M81 O17 16
```

If, for example, we would now like to have PLC64 output O32 active when inputs I10 of the PLCI and I64 of the PLC64 are active, the PLCI program will be:

```
I10 AND M64 = M96
```

2.5 INTERNAL 8025M CNC DATA ACCESSIBLE BY ANY PLC64 OF THE LAN

The PLC64 has a series of marks associated with registers that allow access to the various internal variables of the 8025M CNC. This information is available on all 8025 CNC models whether they have a PLCI or not.

The resources of the PLC64 for this application are:

M1901 - M1949 Each one of these marks is associated with a register (R101 - R149).

Whenever one of these marks is activated at the PLC64, the CNC puts the information corresponding to that mark into the associated register of the PLC64.

M1950 - M1964 Each one of these marks is associated to a register (R150 - R164).

Whenever one of these marks is activated, the PLC64 sends to the CNC the data stored at the associated register.

In order for the PLC64 to access the internal CNC variables, the following bits of PLC64 register R240 must be set accordingly:

Bits 0, 1, 2, 3 Indicate the number of the Node occupied by the CNC.

Node number occupied by the CNC	Bit 3	Bit 2	Bit 1	Bit 0
0	0	0	0	0
1	0	0	0	1
2	0	0	1	0
3	0	0	1	1
4	0	1	0	0
5	0	1	0	1
6	0	1	1	0
7	0	1	1	1
8	1	0	0	0
9	1	0	0	1
10	1	0	1	0
11	1	0	1	1
12	1	1	0	0
13	1	1	0	1
14	1	1	1	0
Incorrect	1	1	1	1

Bit 14 Must be set to "1" to access the internal data of the CNC which occupies the node number indicated by bits 0, 1, 2 and 3.

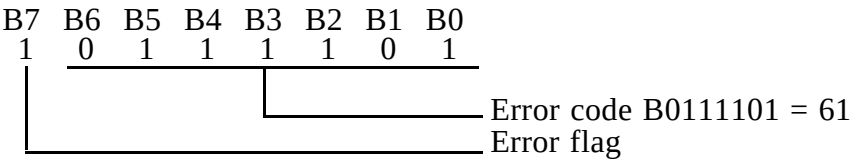
2.5.1 READING INTERNAL CNC VARIABLES

Whenever one of the marks M1901 thru M1949 is activated at the PLC64, the CNC deposits the information corresponding to that mark in its associated register (R101 thru R149).

INTERNAL CNC INFORMATION	ASSOCIATED REGISTER	MARK TO BE ACTIVATED
W axis in motion (0=No 1=Yes)	B0 R101	M1901
Z axis in motion (0=No 1=Yes)	B1 R101	M1901
Y axis in motion (0=No 1=Yes)	B2 R101	M1901
X axis in motion (0=No 1=Yes)	B3 R101	M1901
V axis in motion (0=No 1=Yes)	B4 R101	M1901
CNC in execution (0=No 1=Yes)	B5 R101	M1901
CNC interrupted (0=No 1=Yes)	B6 R101	M1901
Error	B7 R101	M1901
Lower half of X coordinate value (in microns)	R102	M1902
Upper half of X coordinate value (in microns)	R103	M1903
Lower half of Y coordinate value (in microns)	R104	M1904
Upper half of Y coordinate value (in microns)	R105	M1905
Lower half of Z coordinate value (in microns)	R106	M1906
Upper half of Z coordinate value (in microns)	R107	M1907
Lower half of W coordinate value (in microns)	R108	M1908
Upper half of W coordinate value (in microns)	R109	M1909
Lower half of V coordinate value (in microns)	R110	M1910
Upper half of V coordinate value (in microns)	R111	M1911
Program number	R112	M1912
Programmed F (in µm/min.)	R113	M1913
Programmed S in rpm	R114	M1914
Tool position in BCD	B0-7 R115	M1915
Tool size (0=normal)	B8-15 R115	M1915
% Spindle speed override	B0-7 R116	M1916
% Feedrate override	B8-15 R116	M1916
Parameter P614. CNC identification in LAN	B0-7 R117	M1917
Code corresponding to the last key pressed	B0-7 R118	M1918

Error:

When receiving the error indicator flag (B7 R101), the CNC will display the corresponding error code in bits B0 thru B6 of register R101.



Axis coordinates:

When requesting the X axis coordinate, the CNC shows the corresponding value in the double register R102-103. This value is always given **in microns** (never in inches), with respect to machine reference zero (home) and in hexadecimal format as shown below:

If X: 123.456 Value: H1E240 R103=0001 R102=E240
If X: -30.506 Value: HFFFF88D6 R103=FFFF R102=88D6

Program number:

The program number is given in BCD code. for example, if the program selected at the CNC is P12345, register R112 will show the value:

R112 = 0001 0010 0011 0100 0101

Feedrate F and Spindle rpm S:

Also, the **F** value, even when programmed in mm/min. or inches/min, is **always** expressed in **microns/min**. and in hexadecimal format
The programmed **S** value is given in rpm and in hexadecimal format.

G71 F 1.000	(1000 µm/min.)	R113=3E8 (H03E8)
G70 F 0.0394	(1000 µm/min.)	R113=3E8 (H03E8)
S 2500		R113=9C4 (H09C4)

Tool:

To find out the size and the position a tool occupies, proceed as follows:

* Indicate in register R153 the tool number to be consulted and activate mark M1953 (writing an internal CNC variable. See next section).

* Then, activate mark M1915 and the CNC will show at register R115 the tool position in BCD code (bits 0-7) and its size (bits 8-15).

  Example. If a tool has a normal size (not special) and it occupies position 12, register R115 will show the value: R115= 0000 0000 0001 00010.

Feedrate override and Spindle override:

The values corresponding to the spindle speed override and feedrate override currently selected at the CNC will appear according to the following code:

1000 0000 **100%**	0010 0000 **25%**	0000 0001 **1%**
0100 0000 **50%**	....	0000 0000 **0%**

LAN identification parameter:

When the CNC is reconnected to the LAN, it is possible to know the setting of machine parameter P614 by means of mark M1917.

This parameter value is given by the 8 least significant bits (0 through 7) of register R117 as shown by the table below. The other 8 bits (8 through 15) are not being used at this time.

P614(8)	P614(7)	P614(6)	P614(5)	P614(4)	P614(3)	P614(2)	P614(1)
R117(7)	R117(6)	R117(5)	R117(4)	R117(3)	R117(2)	R117(1)	R117(0)

KEY codes:

The key codes that the CNC can return in register R118 when mark M1918 is activated are described in the appendix at the end of this manual.

2.5.2 WRITING INTERNAL CNC VARIABLES

Whenever one of marks M1950 thru M1964 is activated, the PLCI sends to the CNC the information stored in its associated register (R150 thru R164).

INTERNAL CNC INFORMATION	ASSOCIATED REGISTER	Mark to be activated at the PLC64
Inhibit W axis (0=No 1=Yes)	B0 R150	M1950
Inhibit Z axis (0=No 1=Yes)	B1 R150	M1950
Inhibit Y axis (0=No 1=Yes)	B2 R150	M1950
Inhibit X axis (0=No 1=Yes)	B3 R150	M1950
Inhibit V axis (0=No 1=Yes)	B4 R150	M1950
Number of the message to be displayed	B0-7 R151	M1951
Number of the error to be displayed	B8-15 R151	M1951
Spindle Speed Override	B0-7 R152	M1952
Feedrate Override	B8-15 R152	M1952
Number of the tool whose position and size is to be read in R115	R153	M1953
Code of the key to be simulated	B0-7 R154	M1954
Enable or disable CNC keyboard	B8-15 R154	M1954
Transfer Inhibit (active low= 0V)	B0 R155	M1955
M done (active low = 0V)	B1 R155	M1955
Double Cross Compensation	B4 R155	M1955

Axis inhibit:

The PLC64 can inhibit an axis so it cannot be moved.

When the CNC attempts to execute a block involving the movement of an inhibited axis, it interrupts its execution until the axis inhibition is removed.

Messages:

There are three ways to generate an error or error message at the CNC from the PLC64: Using the MSG instruction, marks M1801 thru M1899 or the M1951 - R151 combination.

The CNC must know the system used, behaving differently as described below:

"P620(1)=1" Marks M1801 through M1899 are used to generate errors or messages at the CNC. Neither the MSG instruction nor the M1951-R151 combination can be used.

It is possible to use program P99998 to edit the texts associated with messages and errors. The CNC displays the text associated to the mark with the highest priority and it allows access to the "Messages" option which lists all the messages and errors activated from the PLC.

"P620(1)=0" The MSG instruction as well as the M1951 - R151 combination may be used. Marks M1801 through M1899 cannot be used to generate errors or messages at the CNC.

It is not possible to use program P99998 nor the "Messages" option. It only displays the number of the last message or error activated from the PLC. To delete it, press **[DELETE]**.

To send a message using the M1951-R151 combination, set "P620(1)=0", indicate the message number in Binary code at bits 0-7 of register R151 and activate mark M1951.

Error:

To send an error code to the CNC, indicate the error number in Binary code at bits 8-15 of register R151 and activate M1951.

Feedrate override and Spindle speed override:

These values must be indicated according to the codes below:

1000 0000	100%	0010 0000	25%	0000 0001	1%
0100 0000	50%	.....	.....	0000 0000	does not modify current value.

Tool:

To find out the size and the position a tool occupies, proceed as follows:

* Indicate in register R153 the tool number to be consulted and activate mark M1953.

* Then, activate mark M1915 and the CNC will show at register R115 the tool position in BCD code (bits 0-7) and its size (bits 8-15) (See previous section).

Kestroke simulation:

To manage the CNC from the PLC64 by simulating its keyboard (M1954 and bits 0-7 of R154), it may be convenient to disable the keyboard in order to prevent the operator from controlling the machine.

Enable the CNC keyboard	B8-15 R154 = 0000 0000
Disable the CNC keyboard	B8-15 R154 = 1111 1111

The keys will be simulated one by one, by indicating at bits 0-7 of register R154 the key code to be simulated and activating mark M1954. See key codes in the appendix at the end of this manual.

When done with the keyboard simulation from the PLC64, the control of the CNC should be returned to the operator by re-enabling the CNC keyboard (by setting bits 8-15 of R154 = 0000 0000 and activating mark M1954.)

Transfer-inhibit and M-done:

The 8025 M CNC utilizes pin 15 of connector I/O1 as input for Feed-Hold, Transfer-Inhibit and M-done signals.

With machine parameter P620(3), it is possible to separate such treatment. If "P620(3)=1", the CNC acts as follows:

* The Feed-Hold input will be "taken" at pin 15 of connector I/O1.

* The Transfer-Inhibit input will be "taken" from the LAN.

  To set its status, set bit 0 of register R155 accordingly and activate mark M1955.

  If during the execution of a block, the Transfer Inhibit signal is set low (0V), the CNC checks the status of parameter P622(7) and acts as follows:

  P622(7)=0 Once the current block is finished (in position), the program execution is interrupted and it will be resumed when this signal is set back high (24V).

  P622(7)=1 Once the current block is finished (in position), the CNC checks the next program block:

  * If it contains M, S, T type information, it executes it and analyzes the next block.

  * If it contains other type of information, movement, etc. it interrupts the execution of the program and it will resume it when this signal is set back high (24V).

  This makes it possible, for example, to make a tool change consisting in two blocks: "N100 T2.2" and "N110 M06" interrupting the execution of the program until the tool change is completed.

* The M-done input is also taken from the LAN. Bit 1 R155.

  To set its status, set bit 1 of register R155 accordingly and activate mark M1955. Value of "0" => low, value of "1" => high.

  The treatment of this signal is described in the section on "M,S,T function transfer" of the chapter on "Concepts" in the Installation Manual.

Double Cross Compensation:

This feature is to be used when having a cross compensation between two axes obtaining two different tables applicable according to two different external elements.

For example:

* Two different temperatures. One table for when the machine is cold and the other one when it is hot.

* Machine with two spindles of different weights. Thus, the beam sag error will be different when using one or the other.

When using this feature, 2P625(3) =1", the cross compensation table is divided into two 15-point tables.

These tables are selected from the PLC64 by setting bit 4 of register R155 to "0" or "1" and activating mark M1955.

To select the first table (parameters P180 through P209), set "B4R155 = 0".
To select the second table (parameters P210 through P239), set "B4R155 = 1".

With this CNC, it is possible to use either one cross compensation table or two; but not both at the same time.

2.6 ACCESS TO THE REGISTERS OF A PLC64 FROM AN 8025 CNC

With function G52, it is possible to read or modify the contents of a register of any PLC64 installed in the LAN or of the integrated PLC itself (PLCI). It is **not** possible to access the integrated PLC of another CNC.

Some of the applications for this feature may be:

* When the lubrication of the axes is controlled by the PLC64, it is possible to select, via CNC program, the lubrication time for the axes depending on the type of machining operation being performed.

* When the machine has several tool magazines controlled by a PLC64, it is possible to select via CNC program the specific tool magazine in each case.

* It is possible to select via CNC program the blow-time applied by the PLC64 during a tool change.

Assign a value to a Single Register

Programming format: G52 N2 R3 K5

N2 Indicates the node number of the PLC64. Possible values: N0 through N14.

R3 Indicates the number of the PLC64 register to be modified. Possible values R1 through R255.

K5 Value to be assigned to the selected register. Possible values: integers within ±32767. For example: K3000

Assign a value to a Double Register

Programming format: G52 N2 D3 H8

N2 Indicates the node number of the PLC64. Possible values: N0 through N14.

D3 Indicates the number of the PLC64 register to be modified. Only the first one must be defined. Possible values: R1 through R254. For example: R200 indicates that the double register consists of R200 and R201.

H8 Value to be assigned to the selected register. It is defined by a hexadecimal number between 0 and FFFFFFFF. For example: H1ABC.

Assign the value of an arithmetic parameter of the CNC to a Single Register

Programming format: G52 N2 R3 P3

N2 Indicates the node number of the PLC64. Possible values: N0 through N14.

R3 Indicates the number of the PLC64 register to be modified. Possible values: R1 through R255.

P3 Number of the arithmetic parameter. Possible values: P0 through P254.

Assign the value of an arithmetic parameter of the CNC to a Double Register

Programming format: G52 N2 D3 P3

N2 Indicates the node number of the PLC64. Possible values: N0 through N14.

D3 Indicates the number of the double register of the PLC64 to be modified. Only the first one must be defined. Possible values: R1 through R254. For example; R200 indicates that the double register consists of R200 and R201.

P3 Number of the arithmetic parameter. Possible values: P0 through P254.

Assign the value of a Single Register to an arithmetic parameter of the CNC

Programming format: G52 N2 P3 R3

N2 Indicates the node number of the PLC64. Possible values: N0 through N14.

P3 Number of the arithmetic parameter to be modified. Possible values: P0 through P254.

R3 Indicates the number of the PLC64 register. Possible values: R1 through R255.

Assign the value of a Double Register to an arithmetic parameter of the CNC

Programming format: G52 N2 P3 D3

N2 Indicates the node number of the PLC64. Possible values: N0 through N14.

P3 Number of the arithmetic parameter to be modified. Possible values: P0 through P254.

D3 Indicates the number of the double register of the PLC64. Only the first one must be defined. Possible values: R1 through R254. For example; R200 indicates that the double register consists of R200 and R201.

Atention:

When accessing a register of the integrated PLC itself, indicate the node occupied by the CNC+PLCI unit.

2.7 ACCESS TO AN 82, 101S, 102, 102S CNC FROM AN 8025 CNC

The 8025 CNC offers function G52 to get access to the internal variables of an 82, 101S, 102 or 102S CNC as well as for sending execution commands to any of these models.

2.7.1 ACCESS TO "READ" VARIABLES

The internal "read" variables of the 82, 101S, 102 and 102S CNCs have an associated register, at the CNC itself, which may be consulted by any 8025 CNC installed in the Local Area Network (LAN).

These registers may be single or double. They are described later on and the way to access them is as follows:

Assign the value of a Single Register to an arithmetic parameter of the 8025

Programming format: G52 N2 P3 R3

N2 Indicates the node number of the 82, 101S, 102 or 102S. Possible values: N0 through N14.

P3 Number of the arithmetic parameter to be modified at the 8025. Possible values: P0 through P254.

R3 Indicates the number of the 82, 101S, 102 or 102S register. Possible values: R1 through R11.

Assign the value of a Double Register to an arithmetic parameter of the 8025

Programming format: G52 N2 P3 D3

N2 Indicates the node number of the PLC64. Possible values: N0 through N14.

P3 Number of the arithmetic parameter to be modified at the 8025. Possible values: P0 through P254.

D3 Indicates the number of the double register of the 82, 101S, 102 or 102S. Only the first one must be defined. Possible values: R1 through R10. For example; R2 indicates that the double register consists of R2 and R3.

2.7.1.1 "READ" VARIABLES

The internal "read" variables of the 82, 101S, 102 and 102S CNCs are:

INTERNAL CNC VARIABLES	Register to be consulted at the 102 CNC
Y axis in motion (0=No 1=Yes)	B2 R1
X axis in motion (0=No 1=Yes)	B3 R1
CNC in execution (0=No 1=Yes)	B5 R1
CNC interrupted (0=No 1=Yes)	B6 R1
Error	B7 R1
Selected operating mode	B8,9,10 R1
Movement in G00 (0=No 1=Yes)	B11 R1
Lower half of X coordinate	R2
Upper half of X coordinate	R3
Lower half of Y coordinate	R4
Upper half of Y coordinate	R5
Number of the first program block	R6
Programmed F (in mm/minute)	R7
Programmed S in rpm.	R8
Active tool number	R9
Parameter P80. CNC Id in the LAN	R10
Code of the last key pressed	R11

Error

When receiving the error indicator (B7 R1 = 1), The CNC will show its corresponding binary code at bits B0 through B6 of register R1.

Example, error 26: R1= 0000 0000 1001 1010

Operating mode

Bits 8, 9 and 10 of the CNC register R1 indicate the operating mode currently selected.

Operating mode	R1		
	Bit 10	Bit 9	Bit 8
Peripherals	0	0	1
Aux-Mode	0	1	0
Jog	0	1	1

Operating mode	R1		
	Bit 10	Bit 9	Bit 8
Play-back	1	0	0
Editing	1	0	1
Single-Block	1	1	0
Automatic	1	1	1

Coordinates of the axes

When requesting the coordinate of the X axis, the CNC shows the corresponding value at the double register R2-3. This value is given **in microns**, with respect to Machine Reference Zero (home) and in hexadecimal format as shown below:

If X coordinate: 123.456 Value: H1E240 R3=0001 R2=E240
If X coordinate: -30.506 Value: HFFFF88D6 R3=FFFF R2=88D6

Number of the first program block

This number is shown in BCD code. For example: if the first block number is 278: The value of register R6 will be: 0000 0010 0111 1000

Axis feedrate "F" and spindle speed "S"

The F value is expressed in **mm/min** and the S value in rpm. both in hexadecimal format as shown below:

F 10000	Value: H2710	R7=2710
S 2500	Value: H9C4	R8=9C4

Active tool number

It is given in BCD code. For example, in the case of T12, register R9 will have the value of: 0000 0000 0001 0010

Fagor LAN

When this CNC is connected to the Fagor Local Area Network, this register shows the value allocated to the CNC machine parameter P80.

This value appears at the 8 least significant bits (0 through 7) of register R10. Bits 8 through 15 are not being used at this time.

The relationship between the parameter bits and register bits is the following:

P80(8)	P80(7)	P80(6)	P80(5)	P80(4)	P80(3)	P80(2)	P80(1)
R10(7)	R10(6)	R10(5)	R10(4)	R10(3)	R10(2)	R10(1)	R10(0)

Key codes

The key codes that the CNC can return at R11 are described in the appendix at the end of this manual.

2.7.2 ACCESS TO "WRITE" VARIABLES

The internal "write" variables of the 82, 101S, 102 and 102S CNCs have an associated register, at the CNC itself, which may be altered by any 8025 CNC installed in the Local Area Network (LAN).

These registers may be single or double. They are described later on and the way to access them is as follows:

Assign a value to a Single Register

Programming format: G52 N2 R3 K5

N2 Indicates the node number of the 82, 101S, 102 or 102S.
Possible values: N0 through N14.

R3 Indicates the number of the register to be modified.
Possible values R50 through R54.

K5 Value to be assigned to the selected register.
Possible values: integers within ±32767. For example: K30

Assign a value to a Double Register

Programming format: G52 N2 D3 H8

N2 Indicates the node number of the 82, 101S, 102 or 102S. Possible values: N0 through N14.

D3 Indicates the number of the double register to be modified. Only the first one must be defined. Possible values: R50 through R53. For example: R50 indicates that the double register consists of R50 and R51.

H8 Value to be assigned to the selected register. It is defined by a hexadecimal number between 0 and FFFFFFFF. For example: H1ABC.

Assign the value of an arithmetic parameter of the 8025 to a Single Register

Programming format: G52 N2 R3 P3

N2 Indicates the node number of the 82, 101S, 102 or 102S.
Possible values: N0 through N14.

R3 Indicates the number of the register to be modified.
Possible values: R50 through R54.

P3 Number of the arithmetic parameter of the 8025 CNC.
Possible values: P0 through P254.

Assign the value of an arithmetic parameter of the 8025 to a Double Register

Programming format: G52 N2 D3 P3

N2 Indicates the node number of the 82, 101S, 102 or 102S. Possible values: N0 through N14.

D3 Indicates the number of the double register to be modified. Only the first one must be defined. Possible values: R50 through R53. For example; R50 indicates that the double register consists of R50 and R51.

P3 Number of the arithmetic parameter. Possible values: P0 through P254.

2.7.2.1 "WRITE" VARIABLES

All internal variables of the 82, 101S, 102 or 102S CNC have an associated register at the CNC itself.

INTERNAL CNC INFORMATION	CNC register to be altered
Inhibit Y axis (0=No 1=Yes)	B2 R50
Inhibit X axis (0=No 1=Yes)	B3 R50
Error number to be displayed	B8-15 R51
Number of the block where the execution begins	R52
Number of the block to be executed	R53
Code of the key to be simulated	B0-7 R54

Axis inhibit

The axes may be inhibited so they cannot be moved.

When the CNC executes a block involving the movement of an inhibited axis, it interrupts the execution of the program (if it was running) until that inhibition is removed.

Error display

The most significant bits (8 through 15) of register R51 indicate the binary code of the error number to be displayed at the 82, 101S, 102 or 102S CNC.

For example, to make the 102 CNC display error 17, set R51 to:
0001 0001 0000 0000 .

The CNC interrupts the execution of the program (if it was running) and displays the following information:

LAn
Error 17

Number of the block where the execution begins

It is possible to set from an 8020, 8025 or 8030 CNC, the block number where this CNC will start executing the program.

To do this, set Register R52 to the BCD value of the desired initial block.

For example, to initiate the execution at block number 123, set:
R52= 0000 0001 0010 0011.

Number of the block to be executed

An 8020, 8025 or 8030 CNC can indicate to the 82, 101S, 102 or 102S CNC which program block to execute.

To do this, set register R53 to the binary code of the desired block number.

For example, to execute block number 456, set:
R53= 0000 0100 0101 0110.

Once executed, the 82, 101S, 102 or 102S will indicate its status in its register R1.

Code of the key to be simulated

Whenever a key code is sent to the 82, 101S, 102 or 102S, this behaves as if the corresponding key were pressed.

To do this, set R54 to the key code to be simulated.

When sending a keystroke sequence to the CNC, after each key code is sent and before sending the next one, it is a good idea to check whether that key has been accepted by the CNC or not by consulting register R11.

The key codes to be sent to the CNC are described in the appendix at the end of this manual.

2.7.3 GENERATE EXECUTION COMMANDS

Function G52 allows editing a command at an 8025 CNC which will be executed at a FAGOR 82 CNC.

For example: If a machine has two axes controlled by a FAGOR 82 CNC, the operator may control those two axes from the part program of an 8025 CNC.

Send a command to be executed by a FAGOR 82 CNC

Programming format: G52 N2 = (Command)

N2 Indicates the node number of the FAGOR 82 CNC. Possible values: N0 through N14.

() Command delimiters.

Command It must be written in the program editing format of the FAGOR 82, 101S, 102 or 102S CNC.

Function	Description	101S	102	102S
G00	Rapid positioning	*	*	*
G01	Linear interpolation	*	*	*
G02	Clockwise circular interpolation		*	*
G03	Counter-clockwise circular interpolation		*	*
G04	Dwell	*	*	*
G05	Round corner	*	*	*
G07	Square corner	*	*	*
G25	Unconditional jump	*	*	*
G26	Jump if zero	*	*	*
G27	Jump if not zero	*	*	*
G28	Jump if less than zero	*	*	*
G29	Jump if equal or greater than zero	*	*	*
G33	Synchronization	*		
G45	Increment parts counter	*	*	*
G47	Feedback pulse inhibit	*	*	*
G48	Cancel function G47	*	*	*
G51 to G60	Load zero offsets	*	*	*
G61	F not affected by "P18"	*	*	*
G62	Cancel function G61	*	*	*
G70	Inch programming	*	*	*
G71	Metric programming	*	*	*
G74	Home search	*	*	*
G75	Probing	*	*	*
G81	Batch programming	*		*
G84, G80	Rigid tapping			*
G90	Absolute programming	*	*	*
G91	Incremental programming	*	*	*
G92	Coordinate preset	*	*	*
G93	Modify acceleration ramp	*	*	*

Synchronized operation of an 8025 CNC with an 82, 101, 102 or 102S CNC

Programming format: G52 N2

N2 Indicates the node number of the FAGOR 82, 101, 102 or 102S CNC. Possible values: N0 through N14.

The 8025 CNC waits for the 82, 101S, 102 or 102S CNC to finish the execution of the current block before resuming the execution of its own program.

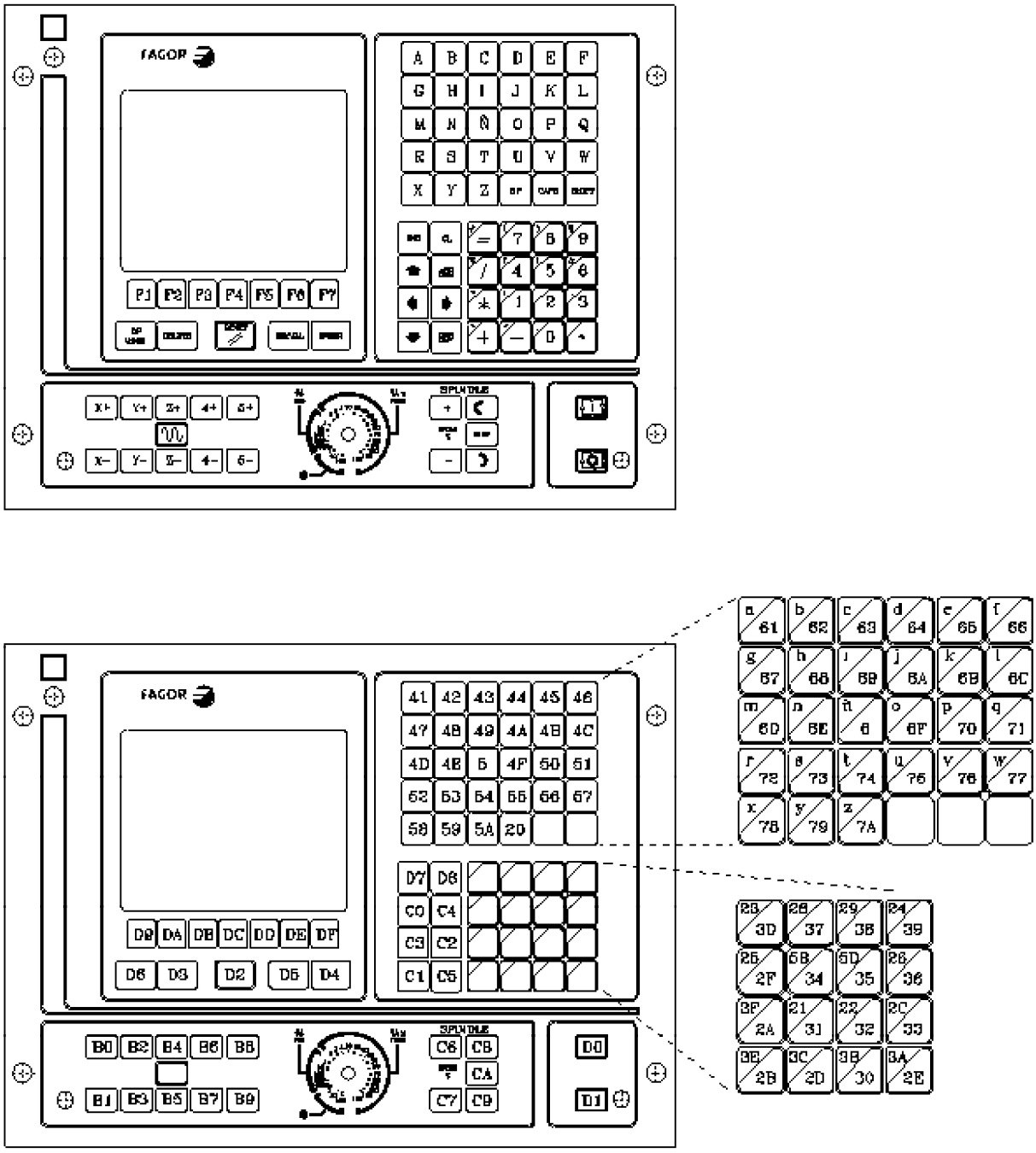
Example: The machine has two axes controlled by a 102 CNC and we would like to move them, one at a time, to X100 Y50.

Also, the 8025 CNC must wait for those axes to reach position X100 Y50 before resuming the execution of its own program.

```
N100 G52 N3 (G01 X100 F100) ; Move to X100
N110 G52 N3 (Y50)           ;The 8025 waits for 102's X axis to reach
                             position (N100 execution completed) and
                             then send the command for the 102's Y
                             axis to move to Y50.
N120 G52 N3                 ;The 8025 waits for block N110 to be
                             completed (102's Y axis in position Y50)
                             in order to go ahead with the next block
                             of its program.
```

APPENDIX A

KEY CODES OF THE 8025 CNC

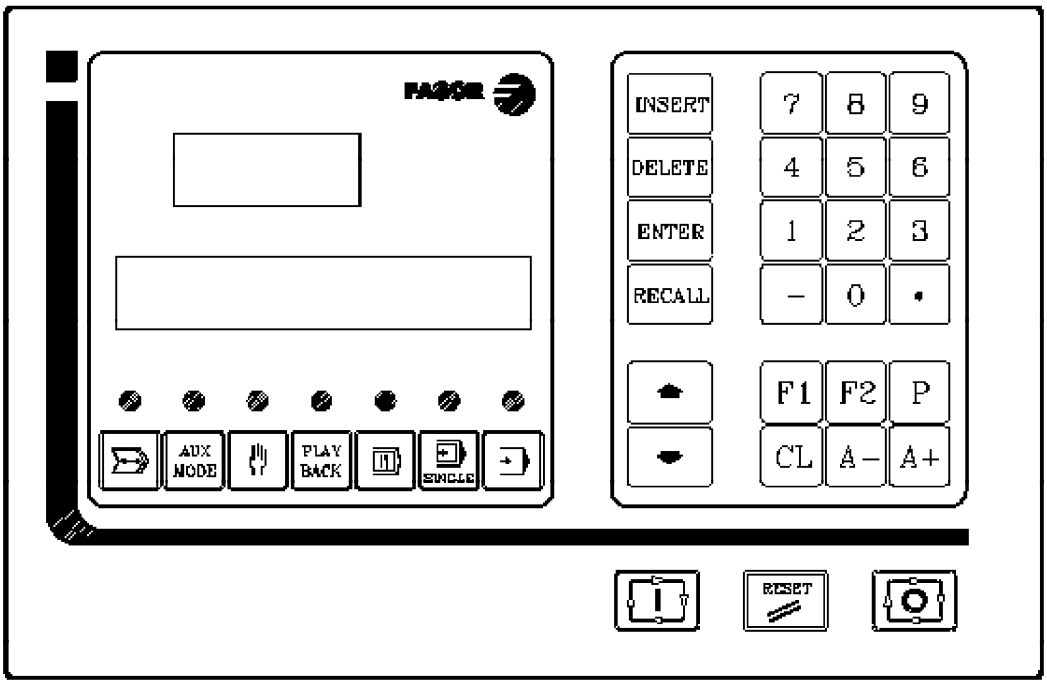


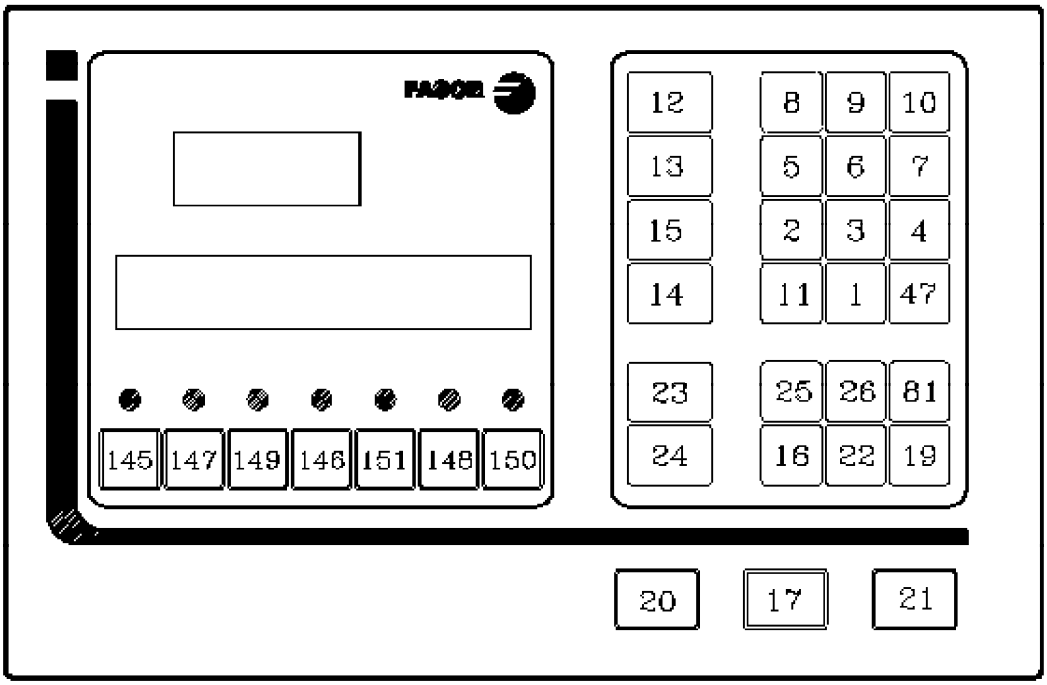
Atention:

The key can neither be read nor simulated from the PLC64.

APPENDIX B

KEY CODES OF THE 101S, 102, 102S CNC





The codes corresponding to the control keys available from an external operator panel are:

M3 (spindle clockwise) ..... Code: 27
M4 (spindle counter-clockwise) ..... Code: 28
M5 (spindle stop)..... Code: 29