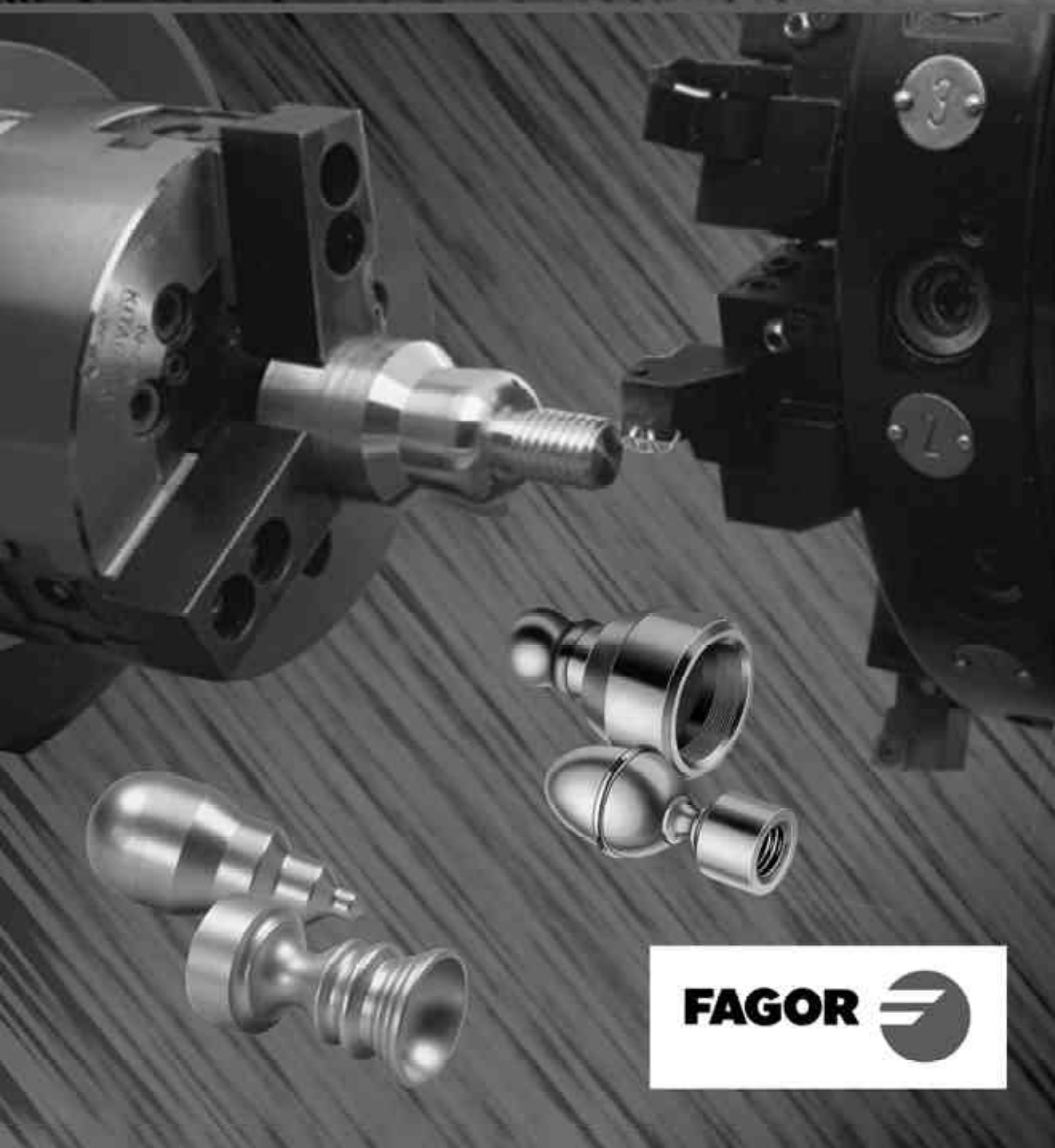
FAGOR AUTOMATION

CNC 8025 T, TS

New Features (Ref. 0107 in)



FAGOR

ERRORS FOUND IN THE PROGRAMMINGMANUAL (REF. 9701)

Page 64. Function G51.

When working in diameters, the "I" value in the table is in diameters and the value to be assigned to parameter "I" in the G51 function must always be given in radius.

Section 12.4. (Chapter 12 page 133) Nesting levels.

The figure reads M02 ó M30
It should read: M02 or M30

ERRORS FOUND IN THE OPERATING MANUAL (REF. 9701)

Page 46. Last paragraphs.

It should say:

The CNC asks which is the source program number and which is the new program number, after keying each one of them, press [ENTER].

If the number of the source program does not exist, or there is already a program in memory with the same number as the new one or if there is not enough when copying the new program , the CNC will issue a message indicating the cause.

ERRORS FOUNDS IN THE INSTALLATION MANUAL (REF. 9707)

Section 5.4 (chapter 5 page 7) Machine parameters for spindle control:

Parameter P606(3) is missing:

P606(3) Spindle counting direction
It sets the spindle counting direction. If correct, leave it as it is or change it if otherwise.
Possible values: "0" and "1".

MODIFICATIONS TO THE INSTALLATION MANUAL (REF. 9707)

Section 2.3.4 (chapter 2 page 9). Logic Outputs:

In the table, the following output is missing:
Output "C"
Row 1: (pin 3 I/O 1) M strobe
Row 2: (pin 5 I/O 2) output 3, decoded M function

Section 3.3.3 (chapter 3 page 11). P602(4). Another example:

Having a Fagor electronic handwheel (25 lines per turn) set as follows:
P602(1)=0 Millimeters P501=1 Resolution 0.001 mm. P602(4)=0 x4 Multiplication factor
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 LAN MANUAL (REF. 9701)

Section 2.2 (page 3). P616(7)

The first 2 paragraphs change. They should say:

If "P616(7)=0" the 8025 T CNC uses pin 15 of connector I/O1 as the input for the Feed-Hold, Transfer-Inhibit and M-done signals as described in the Installation manual, chapter 1 section "Inputs of connector I/O 1"

If "P616(7)=1" the CNC behaves as follows:
* The Feed-Hold input will be "taken" .....

1. EXPANSION OF THE INTEGRATED PLC RESOURCES

1.1 INPUTS

1.1.1 TYPE OF FEEDRATE (G94/G95)

PLCI input I86 will show at all times the type of feedrate (F) selected a the CNC.

I86 = 0 G94. Feedrate in millimeters (inches) per minute.
I86 = 1 G95. Feedrate in millimeters (inches) per revolution.

1.1.2 TYPE OF CUTTING SPEED (G96/G97)

PLCI input I87 will show at all times the type of cutter speed selected at the CNC.

I87 = 0 G97. Constant tool center speed.
I87 = 1 G96. Constant cutting-edge speed

1.1.3 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 3rd 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 4th axis is being homed (0=No / 1=Yes)
I104	Indicates whether the C axis is being homed (0=No / 1=Yes)

1.1.4 AXIS MOVING DIRECTION

Inputs I42, I43, I44 and I45 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 3rd 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 4th 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 "P621(7)" indicates whether this feature is available or not.

P621(7) = 0 This feature **is not** available.
P621(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 "P621(7)" indicates whether this feature is available or not.

P621(7) = 0 This feature **is not** available.
P621(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 3rd axis limits
O56 / O57	Positive / negative Z axis limits
O58 / O59	Positive / negative 4th 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 "P621(7)" indicates whether this feature is available or not.

P621(7) = 0 This feature **is not** available.
P621(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 "P621(7)" indicates whether this feature is available or not.

P621(7) = 0 This feature **is not** available
P621(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; b**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
M44	M43	M42	M41				M19	M1	M30			M4	M3	M2	M0

2. 4TH AXIS NOW AVAILABLE ON 8025T MODELS

From this version on, this feature is now available on all these models:

CNC-8025T (not available until now) CNC-8025TG CNC-8025TS
CNC-8025TI(not available until now) CNC-8025TGI CNC-8025TSI

3. SPINDLE SPEED DISPLAY UNITS

Until now, the spindle speed was always displayed in rpm. From now on, the display units may be selected by means of machine parameter "P621(6)".

P621(6) = 0 In rpm when operating in RPM and in m/min. (ft./min.) when at Constant Surface Speed.
P621(6) = 1 Always in rpm, even when operating at Constant Surface Speed.

4. SINGLE BLOCK TREATMENT

The CNC considers a "Single block" the group of blocks between a G47 and a G48.

After executing function G47, the CNC executes all the following blocks until executing a block containing function G48 **even when in Single Block mode**

If  is pressed while executing a "single block" in Automatic or Single-Block mode, the CNC keeps executing the rest of the blocks until it runs into a G48 and it, then, interrupts program execution.

While function G47 is active, the Manual Feedrate Override switch and the spindle speed override keys will be disabled, thus the program will be executed at 100% of the programmed F and S values.

Functions G47 and G48 are modal and incompatible with each other.

On power-up, after executing an M02/M30, after an EMERGENCY or a RESET, the CNC assumes G48.

5. TWO ELECTRONIC HANDWHEELS ARE NOW POSSIBLE

From this version on, up 2 electronic handwheels may be used one for the X axis and another one for the Z axis. The 4th axis and the Live Tool will no longer be available. The feedback inputs will be used as follows:

A1 - X axis; A2 - Z axis handwheel; A3 - Z axis; A4 - 3rd axis or "C" axis; A5 - Spindle; A6 - X axis handwheel

The handwheels will be operative when selecting the JOG mode. One of the handwheel positions must also be selected at the Manual Feedrate Override switch of the operator panel.

The possible positions are: 1, 10 and 100, which indicate the multiplying factor applied to the pulses coming from the electronic handwheel.

This way and after applying the multiplying factor, one obtains the axis moving units. These units correspond to the units used for the display format

Example: Handwheel Resolution : 250 lines per turn

MFO Switch position	Distance per turn
1	0.250 mm or 0.0250 inch
10	2.500 mm or 0.2500 inch
100	25.000 mm or 2.5000 inches

When attempting to "crank" an axis faster than its maximum feedrate (machine parameters "P110, P310"), the CNC will limit the actual axis feedrate to that parameter value ignoring the rest of the pulses supplied by the handwheel, thus preventing a Following Error message from being issued.

5.1 MACHINE PARAMETERS FOR THE HANDWHEELS:

P622(6) = 0 There **is no** electronic handwheel associated with the Z axis
P622(6) = 1 There **is** electronic handwheel associated with the Z axis

P609(1) = 0 The electronic handwheel being used **is not** a FAGOR 100P model.
P609(1) = 1 The electronic handwheel being used **is** a FAGOR 100P model.

This parameter makes sense when using a single handwheel associated with the X axis. It indicates whether or not it is a FAGOR 100P with axis selector button.

P500 Counting direction of the X axis handwheel (No / Yes)
P622(5) Counting direction of the Z axis handwheel (0 / 1)

P602(1) Feedback units of the X axis handwheel (0 = millimeters /1 = inches)
P622(3) Feedback units of the Z axis handwheel (0 = millimeters /1 = inches)

P501 Square-wave feedback resolution of the X axis handwheel.
P622(1,2) Square-wave feedback resolution of the Z axis handwheel.

P501	P622(2)	P622(1)	Resolution	
1	0	0	0.001 mm	0.0001"
2	0	1	0.002 mm	0.0002"
5	1	0	0.005 mm	0.0005"
10	1	1	0.010 mm	0.0010"

P602(4) Multiplying factor for X axis handwheel feedback pulses (0= x4 / 1= x2)
P622(4) Multiplying factor for Z axis handwheel feedback pulses (0= x4 / 1= x2)

P621(2) = 0 Handwheel disabled for Manual Feedrate Override (MFO) switch positions other than the handwheel positions.
P621(2) = 1 When the MFO is at a position other than those for the handwheel , the CNC takes it into account and applies a "x1" multiplying factor.

Example: Having a Fagor electronic handwheel (25 lines per turn) set as follows:
P602(1)=0 Millimeters; P501=1 Resolution 0.001 mm.; P602(4)=0 x4 Multiplication factor
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

5.2 USING ELECTRONIC HANDWHEELS

The machine uses one electronic handwheel

When using a single electronic handwheel, it must be connected to A6.

If the handwheel is a FAGOR 100P type, machine parameter "P609(1)" must be set to "1".

Once the desired handwheel position has been selected at the MFO switch, press one of the JOG keys of the axis to be jogged. The selected axis appears highlighted.

When using a FAGOR handwheel with an axis selector button, the desired axis can also be selected as follows:

* Press the push-button on the rear of the handwheel. The CNC selects the first axis and it highlights it.

* By pressing the button again, the next axis is selected and so on, rolling over from the last axis to the first one.

* By keeping the button pressed for more than 2 seconds, the CNC de-selects the currently selected axis.

The selected axis will be jogged as the handwheel is turned, reversing directions when reversing the turning direction of the handwheel.

When trying to move an axis faster than the maximum feedrate allowed (machine parameter "P110, P310"), the CNC will limit the actual feedrate to that parameter value ignoring the additional pulses, thus, avoiding following error messages.

The machine uses two electronic handwheels

Each axis will move as its associated handwheel is turned, reversing its direction as the handwheel turning direction is reversed and according to the selected MFO switch position.

When trying to move an axis faster than the maximum feedrate allowed (machine parameter "P110, P310"), the CNC will limit the actual feedrate to that parameter value ignoring the additional pulses, thus, avoiding following error messages.

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 "P619(5)" indicates whether this feature is to be used or not.

P619(5) = 0 This feature is not being used.
P619(5) = 1 This feature is being used.

2. JOGGING FEEDRATE

From this version on, machine parameter P812 sets the axis jogging feedrate selected by the CNC when accessing the JOG mode.

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.

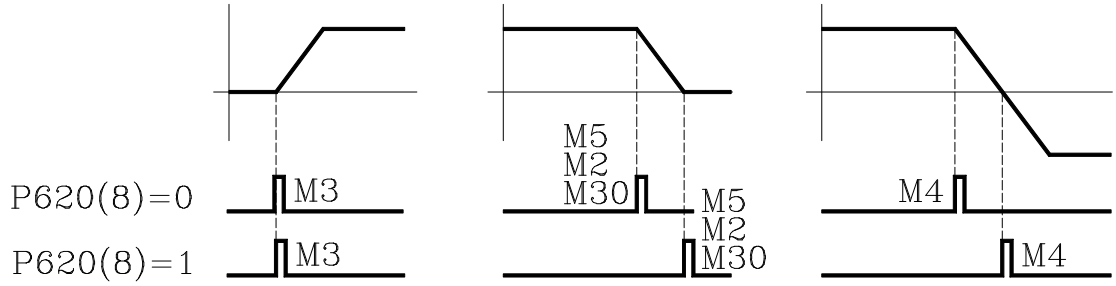
3. RAPID TRAVERSE KEY IN JOG MODE

Whenever the conditional input (block skip), pin 18 of connector I/O1, the CNC will ignore the rapid traverse key

Version 7.4 (May 1999)

1. NEW MACHINE PARAMETER ASSOCIATED WITH THE M FUNCTIONS

Machine parameter "P620(8)" 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 P620(5), P620(6), P613(8) and P613(7) are used together with P602(6), P602(5), P612(5) and P614(5) which indicate the multiplying factor to be applied to the feedback signals of the X, Z, 3rd and 4th axes respectively.

X axis	Z axis	3rd axis	4th axis
P602(6)	P602(5)	P612(5)	P614(5)
P620(5)	P620(6)	P613(8)	P613(7)

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

P620(5)=0, P620(6)=0, P613(8)=0 y P613(7)=0 They are not divided
P620(5)=1, P620(6)=1, P613(8)=1 y P613(7)=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 P602(6)=0 & P620(5)=0	x4 multiplying factor	Nr of pulses = 125
With P602(6)=1 & P620(5)=0	x2 multiplying factor	Nr of pulses = 250
With P602(6)=0 & P620(5)=1	x2 multiplying factor	Nr of pulses = 250
With P602(6)=1 & P620(5)=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.

P623(1) = 0Not affected. It is always at 100%, like in previous versions.
P623(1) = 1It 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 3rd axis
P821	Feedback factor for the Z axis	P822	Feedback factor for the 4th 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 TLI is now available.
It offers the same features as the TGI model and it is sold together with the motors and ACS drives..

FAGOR 8025/8030 CNC

Models: T, TG, TS

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
LATHE MODEL
FAGOR 8025/8030 CNCs

TECHNICAL DESCRIPTION

	T	TG	TS
INPUTS/OUTPUTS			
Feedback inputs.	**6**	**6**	**6**
Linear axes	**4**	**4**	**4**
Rotary axes	**2**	**2**	**2**
Spindle encoder	**1**	**1**	**1**
Electronic handwheel	**1**	**1**	**1**
Third axis as "C" axis			**x**
Synchronized tool			**x**
Probe input	**x**	**x**	**x**
Square-wave feedback signal multiplying factor, x2/x4	**x**	**x**	**x**
Sine-wave feedback signal multiplying factor, x2/x4/10/x20	**x**	**x**	**x**
Maximum counting resolution 0.001mm/0.001°/0.0001inch	**x**	**x**	**x**
Analog outputs (±10V) for axis servo drives	**4**	**4**	**4**
Spindle analog output (±10V)	**1**	**1**	**1**
Live tool	**1**	**1**	**1**
AXIS CONTROL			
Axes involved in linear interpolations	**3**	**3**	**3**
Axes involved in circular interpolations	**2**	**2**	**2**
Electronic threading	**x**	**x**	**x**
Spindle control	**x**	**x**	**x**
Software travel limits	**x**	**x**	**x**
Spindle orientation	**x**	**x**	**x**
PROGRAMMING			
Part Zero preset by user	**x**	**x**	**x**
Absolute/incremental programming	**x**	**x**	**x**
Programming in cartesian coordinates	**x**	**x**	**x**
Programming in polar coordinates	**x**	**x**	**x**
Programming by angle and cartesian coordinate	**x**	**x**	**x**
COMPENSATION			
Tool radius compensation	**x**	**x**	**x**
Tool length compensation	**x**	**x**	**x**
Leadscrew backlash compensation	**x**	**x**	**x**
Leadscrew error compensation	**x**	**x**	**x**
DISPLAY			
CNC text in Spanish, English, French, German and Italian	**x**	**x**	**x**
Display of execution time	**x**	**x**	**x**
Piece counter	**x**	**x**	**x**
Graphic movement display and part simulation		**x**	**x**
Tool tip position display	**x**	**x**	**x**
Geometric programming aide	**x**	**x**	**x**
COMMUNICATION WITH OTHER DEVICES			
Communication via RS232C	**x**	**x**	**x**
Communication via DNC	**x**	**x**	**x**
Communication via RS485 (FAGOR LAN)	**x**	**x**	**x**
ISO program loading from peripherals	**x**	**x**	**x**
OTHERS			
Parametric programming	**x**	**x**	**x**
Model digitizing			**x**
Possibility of an integrated PLC	**x**	**x**	**x**

PREPARATORY FUNCTIONS

	T	TG	TS
AXES AND COORDINATES SYSTEMS			
Part measuring units. Millimeters or inches (G70,G71)	x	x	x
Absolute/incremental programming (G90,G91)	x	x	x
Independent axis (G65)	x	x	x
REFERENCE SYSTEMS			
Machine reference (home) search (G74)	x	x	x
Coordinate preset (G92)	x	x	x
Zero offsets (G53...G59)	x	x	x
Polar origin offset (G93)	x	x	x
Store current part zero (G31)	x	x	x
Recover stored part zero (G32)	x	x	x
PREPARATORY FUNCTIONS			
Feedrate F	x	x	x
Feedrate in mm/min. or inches/min. (G94)	x	x	x
feedrate in mm/revolution or inches/revolution (G95)	x	x	x
Programmable feed-rate override (G49)	x	x	x
Spindle speed (S)	x	x	x
Spindle speed in rpm (G97)	x	x	x
Constant Surface Speed (G96)	x	x	x
S value limit when working at constant surface speed (G92)	x	x	x
Tool and tool offset selection (T)	x	x	x
Activate "C" axis in degrees (G14)			x
Main plane C-Z (G15)			x
Main plane C-X (G16)			x
AUXILIARY FUNCTIONS			
Program stop (M00)	x	x	x
Conditional program stop (M01)	x	x	x
End of program (M02)	x	x	x
End of program with return to first block (M30)	x	x	x
Clockwise spindle start (M03)	x	x	x
Counter-clockwise spindle start (M04)	x	x	x
Spindle stop (M05)	x	x	x
Spindle orientation (M19)	x	x	x
Spindle speed range change (M41, M42, M43, M44)	x	x	x
Tool change with M06	x	x	x
Live tool (M45 S)	x	x	x
Synchronized tool (M45 K)			x
PATH CONTROL			
Rapid traverse (G00)	x	x	x
Linear interpolation (G01)	x	x	x
Circular interpolation (G02,G03)	x	x	x
Circular interpolation with absolute center coordinates (G06)	x	x	x
Circular path tangent to previous path (G08)	x	x	x
Arc defined by three points (G09)	x	x	x
Tangential entry (G37)	x	x	x
Tangential exit (G38)	x	x	x
Controlled radius blend (G36)	x	x	x
Chamfer (G39)	x	x	x
Electronic threading (G33)	x	x	x
ADDITIONAL PREPARATORY FUNCTIONS			
Dwell (G04 K)	x	x	x
Round and square corner (G05, G07)	x	x	x
Scaling factor (G72)	x	x	x
Single block treatment (G47, G48)	x	x	x
User error display (G30)	x	x	x
Automatic block generation (G76)			x
Communication with FAGOR Local Area Network (G52)	x	x	x

	T	TG	TS
COMPENSATION			
Tool radius compensation (G40,G41,G42)	**x**	**x**	**x**
Loading of tool dimensions into internal tool table (G50, G51)	**x**	**x**	**x**
CANNED CYCLES			
Pattern repeat (G66)	**x**	**x**	**x**
Roughing along X (G68)	**x**	**x**	**x**
Roughing along Z (G69)	**x**	**x**	**x**
Straight section turning (G81)	**x**	**x**	**x**
Straight section facing (G82)	**x**	**x**	**x**
Deep hole drilling (G83)	**x**	**x**	**x**
Circular section turning (G84)	**x**	**x**	**x**
Circular section facing (G85)	**x**	**x**	**x**
Longitudinal threadcutting (G86)	**x**	**x**	**x**
Face threadcutting (G87)	**x**	**x**	**x**
Grooving along X (G88)	**x**	**x**	**x**
Grooving along Z (G89)	**x**	**x**	**x**
PROBING			
Probing (G75)	**x**	**x**	**x**
Tool calibration canned cycle (G75N0)			**x**
Probe calibration canned cycle (G75N1)			**x**
Part measuring canned cycle along X (G75N2)			**x**
Part measuring canned cycle along Z (G75N3)			**x**
Part measuring canned cycle with tool compensation along X (G75N4)			**x**
Part measuring canned cycle with tool compensation along Z (G75N5)			**x**
SUBROUTINES			
Number of standard subroutines	99	99	99
Definition of a standard subroutine (G22)	**x**	**x**	**x**
Call to a standard subroutine (G20)	**x**	**x**	**x**
Number of parametric subroutines	99	99	99
Definition of a parametric subroutine (G23)	**x**	**x**	**x**
Call to a parametric subroutine (G21)	**x**	**x**	**x**
End of standard or parametric subroutine (G24)	**x**	**x**	**x**
JUMP OR CALL FUNCTIONS			
Unconditional jump/call (G25)	**x**	**x**	**x**
Jump or call if zero (G26)	**x**	**x**	**x**
Jump or call if not zero (G27)	**x**	**x**	**x**
Jump or call if smaller (G28)	**x**	**x**	**x**
Jump or call if greater (G29)	**x**	**x**	**x**

NEW FEATURES AND MODIFICATIONS

Date: **March 1991** ***Software Version:*** **2.1 and newer**

FEATURE	MODIFIED MANUAL & SECTION	
The home searching direction is set by machine parameter P618(5,6,7,8)	Installation Manual	Section 4.7
The 2nd home searching feedrate is set by machine parameter P807...P810	Installation Manual	Section 4.7
New resolution values 1, 2, 5 and 10 for sine-wave feedback signals P619(1,2,3,4)	Installation Manual	Section 4.1
Access to PLCI registers from the CNC	Programming Manual	G52

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

FEATURE	MODIFIED MANUAL AND SECTION	
New function: F36. It takes the value of the selected tool number	Programming Manual	Chapter 13
G68 and G69 canned cycles modified. if P9=0 it runs another final roughing pass	Programming Manual	Chapter 13

Date: **September 1991** ***Software Version:*** **3.2 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Subroutine associated with the T function	Installation Manual	Section 3.3.5
G68 and G69 canned cycles modified. P9 can now have a negative value	Programming Manual	Chapter 13

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

FEATURE	MODIFIED MANUAL AND SECTION	
Bell-shaped ACC./DEC.	Installation Manual	Section 4.8
It is now possible to enter the sign of the leadscrew backlash for each axis P620(1,2,3,4)	Installation Manual	Section 4.4
Independent axis movement execution	Programming Manual	G65
It is now possible to work at Constant Surface Speed in JOG mode P619(8)	Installation Manual	Section 3.3.9

Date: **July 1992** ***Software Version:*** **4.2 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Synchronisation with independent axis P621(4)	Installation Manual	Section 3.3.10

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

FEATURE	MODIFIED MANUAL AND SECTION	
Linear & Bell-shaped acc./dec. ramp combination	Installation Manual	Section 4.8
Spindle acc/dec control. P811	Installation Manual	Section 5.
The subroutine associated with the tool is executed before the T function. P617(2)	Installation Manual	Section 3.3.5
G68 and G69 cycles modified. If P10 <> 0, it runs a final roughing pass before the finishing pass	Programming Manual	Chapter 13
When having only one spindle range, if G96 is executed without any range being selected, the CNC will automatically select it.	Programming Manual	Chapter 6
8030 CNC with VGA Monitor	Installation Manual	Chapter 1

Date: **March1995** ***Software Version:*** **5.3 and newer**

FEATURE	MODIFIED MANUAL AND SECTION	
Management of semi-absolute feedback devices (with coded Io)	Installation Manual	Sections 4.7 & 6.5.
Spindle inhibit by PLC	Installation Manual	Section 3.3.10
Handwheel managed by PLC	Installation Manual	Section 3.3.3
Simulation of the "rapid JOG" key from PLC	PLCI Manual	
Initialization of machine parameters in case of memory loss.		

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 T 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.

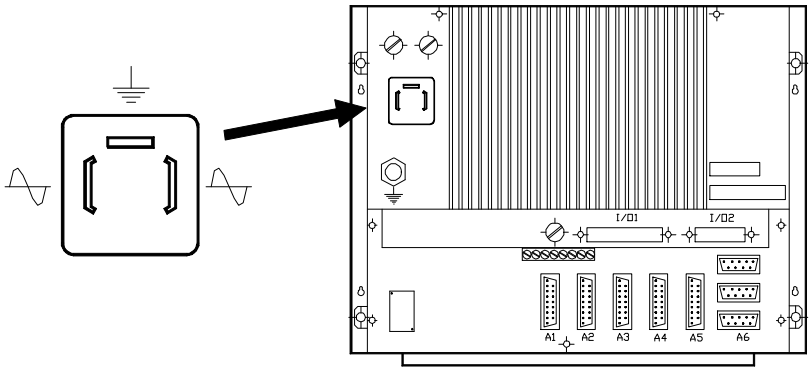
5.- Seal the cardboard box with packing tape or industrial staples.

ADDITIONAL REMARKS

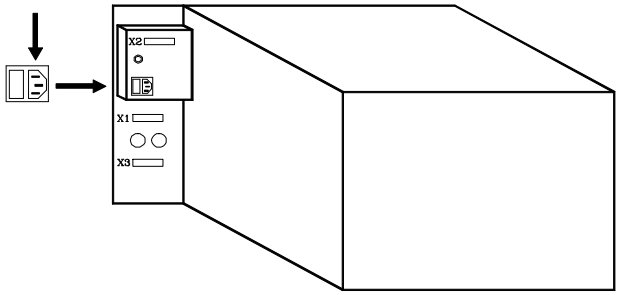
* Mount the CNC away from coolants, chemical products, blows, etc. which could damage it.

* 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)



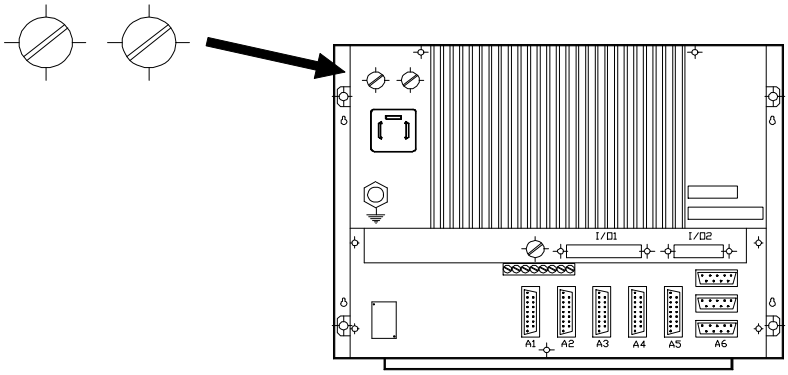
* To prevent electrical shock at the Monitor, use the proper mains AC connector at the Power Supply Module. Use 3-wire power cables (one for ground connection)



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

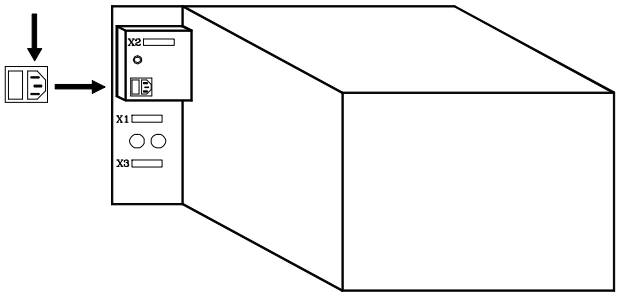
  Central Unit

  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 T CNC

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

It contains 2 manuals:

Operating Manual	describing how to operate the CNC.
Programming Manual	describing how to program the CNC.

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:

1. CONFIGURATION OF THE CNC

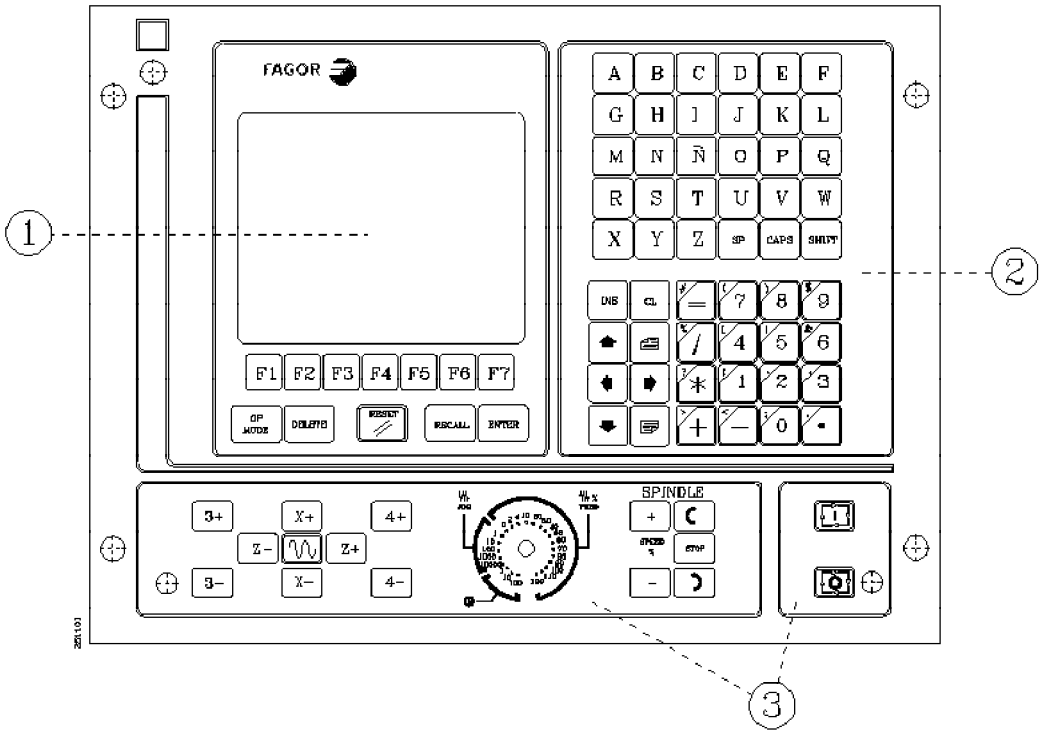
Atention:

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:



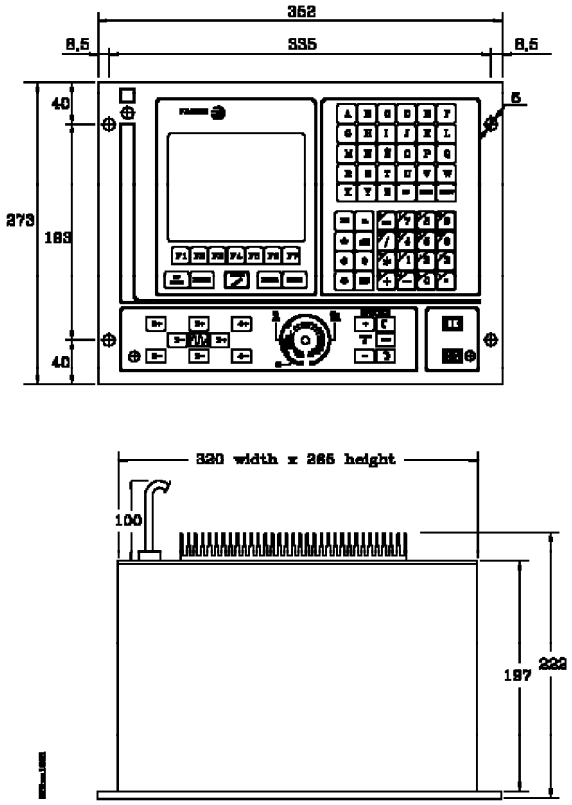
1. An 8" monochrome amber monitor or CRT screen used to display the required system information.

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.

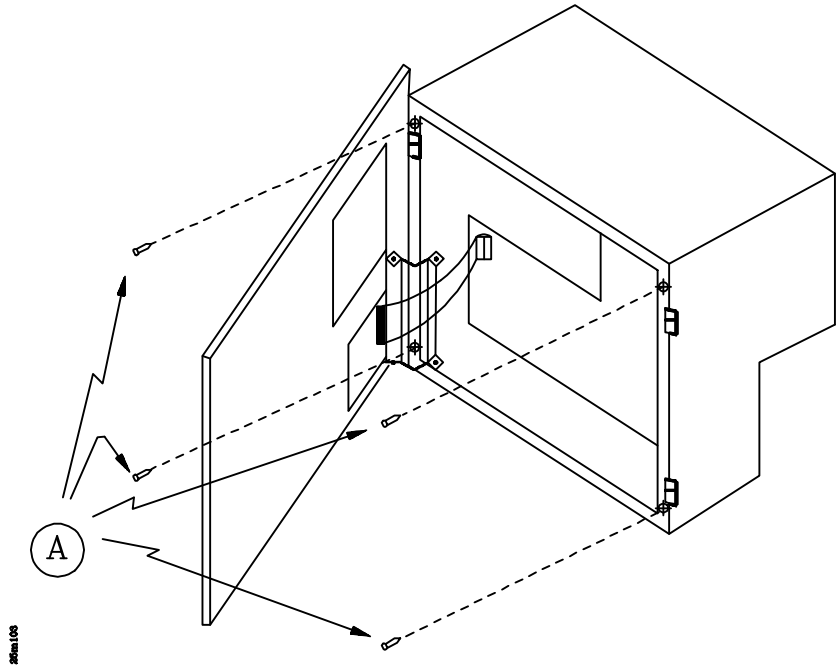
1.2 DIMENSIONS AND INSTALLATION OF THE 8025 CNC

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

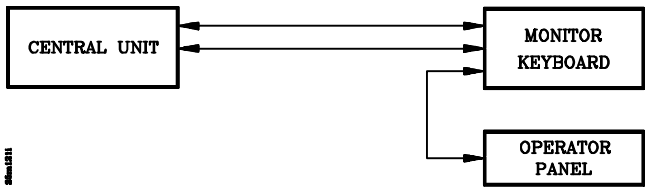
This model CNC consists of 3 independent interconnected modules. These modules can be mounted on different locations and they are:

- CENTRAL UNIT
- MONITOR/KEYBOARD
- OPERATOR PANEL

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

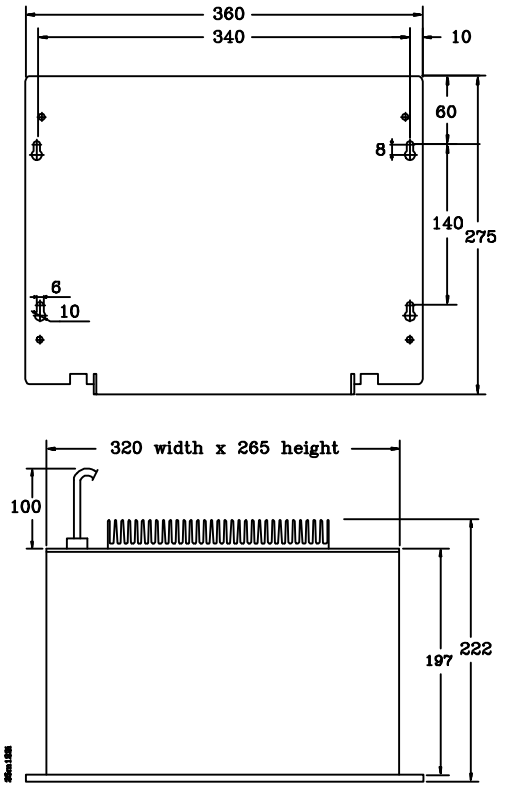
These two modules will be placed next to each other and must be connected with the CENTRAL UNIT module which could be located somewhere else. The two cables used to connect them together are also supplied with these modules. Their maximum length is 25 meters (82 feet) and they are referred to as:

- Video cable.
- Keyboard cable.



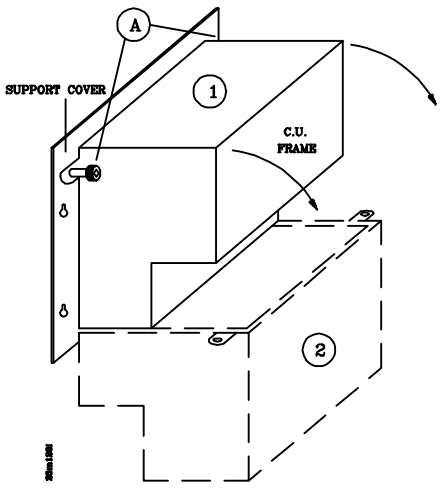
1.2.1 CENTRAL UNIT OF THE 8030 CNC

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.

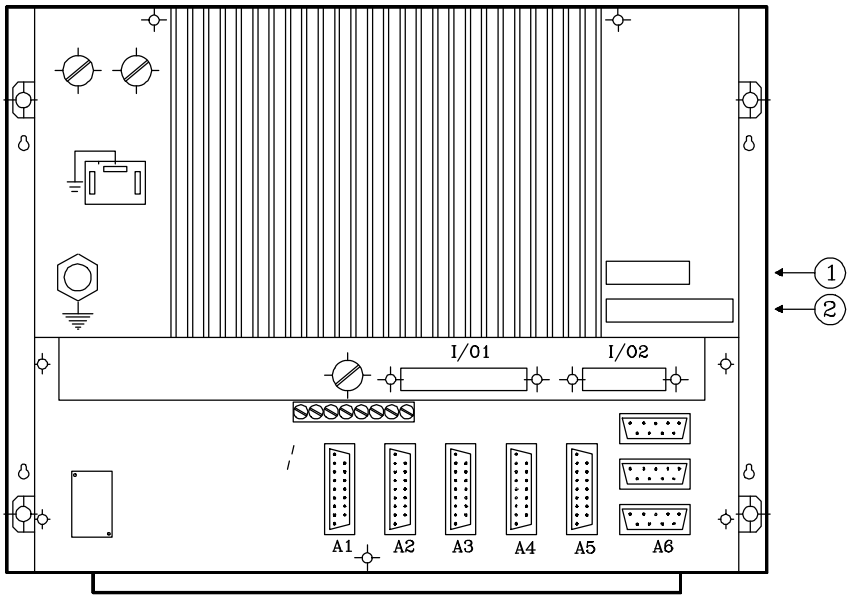


When installing it, observe enough clearance to swing the CENTRAL UNIT open in case of future inside manipulation.

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.

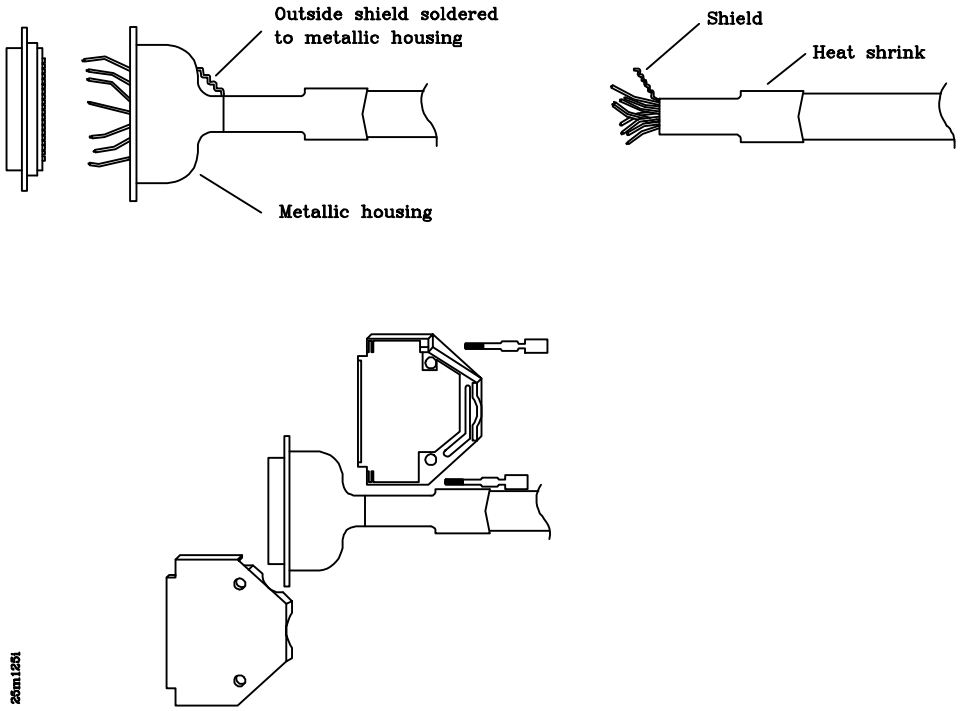
FAGOR AUTOMATION provides the cable required for this connection. It comes with a 25-pin SUB-D type male connector at each end.

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.

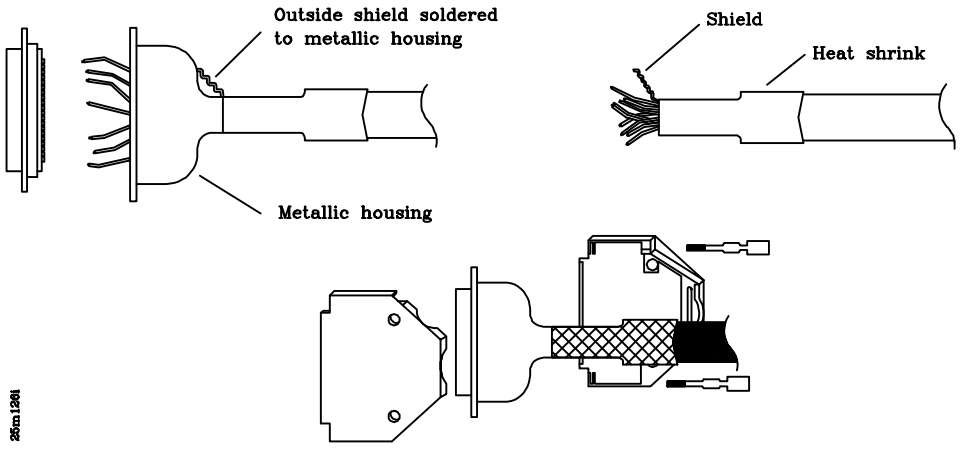
FAGOR AUTOMATION provides the cable required for this connection. It comes with a 15-pin SUB-D type male connector at one end and a 15-pin SUB-D type female connector at the other.

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.

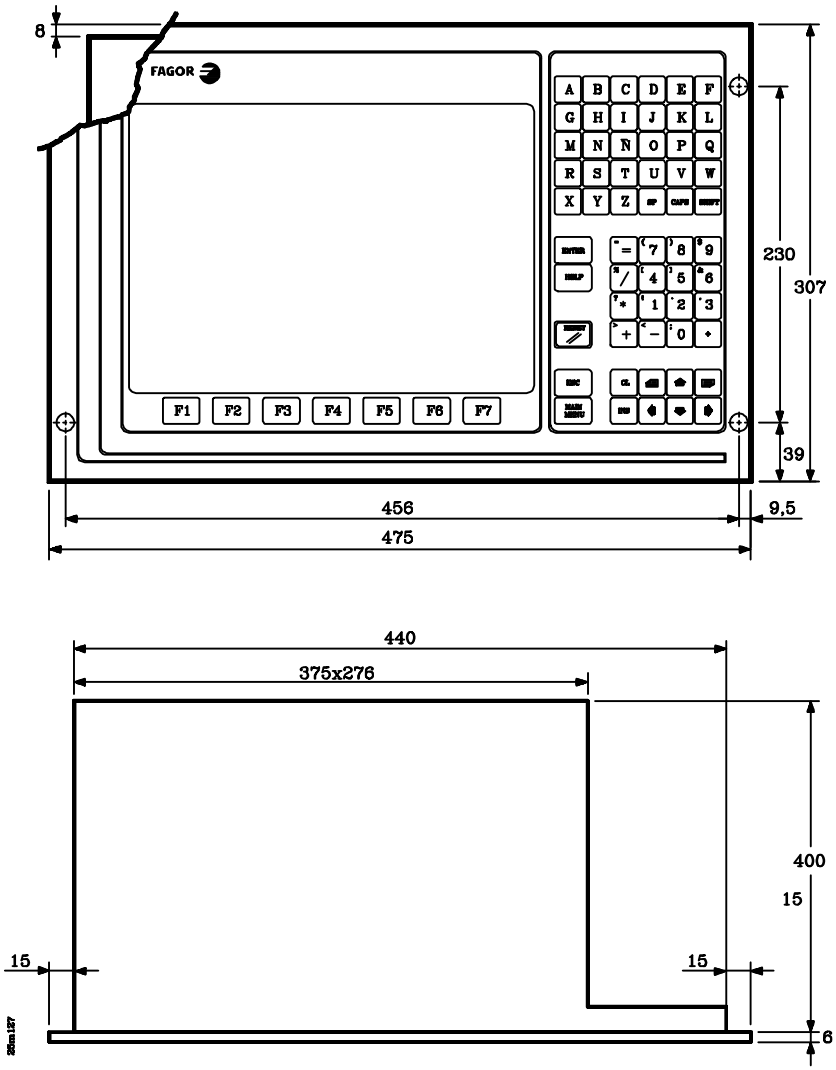


1.2.2 MONITOR/KEYBOARD OF THE 8030 CNC

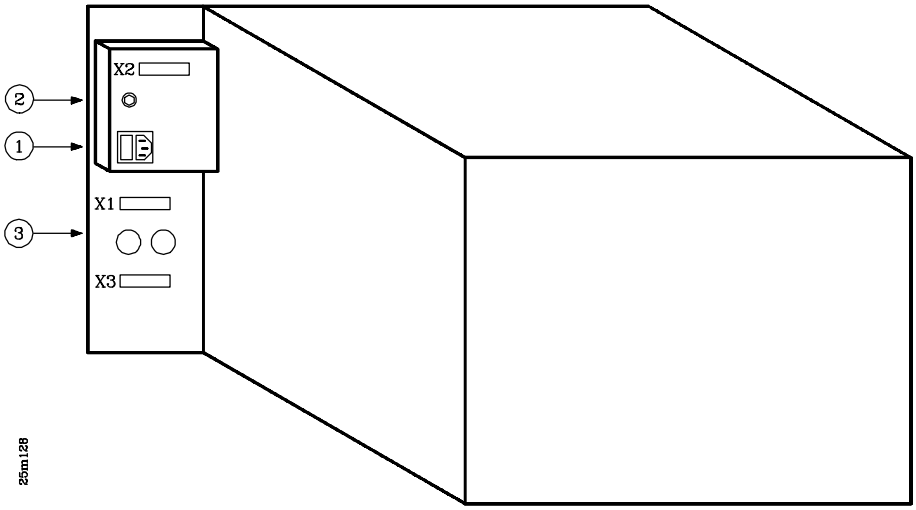
This module can be mounted on the machine pendant and it lets the operator get the necessary information at the MONITOR as well as operate the CNC by means of its KEYBOARD and OPERATOR PANEL.

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.

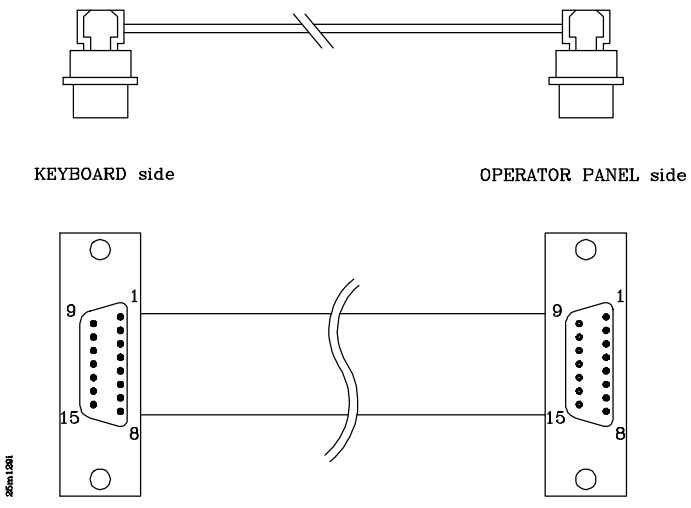
Connector X3

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.

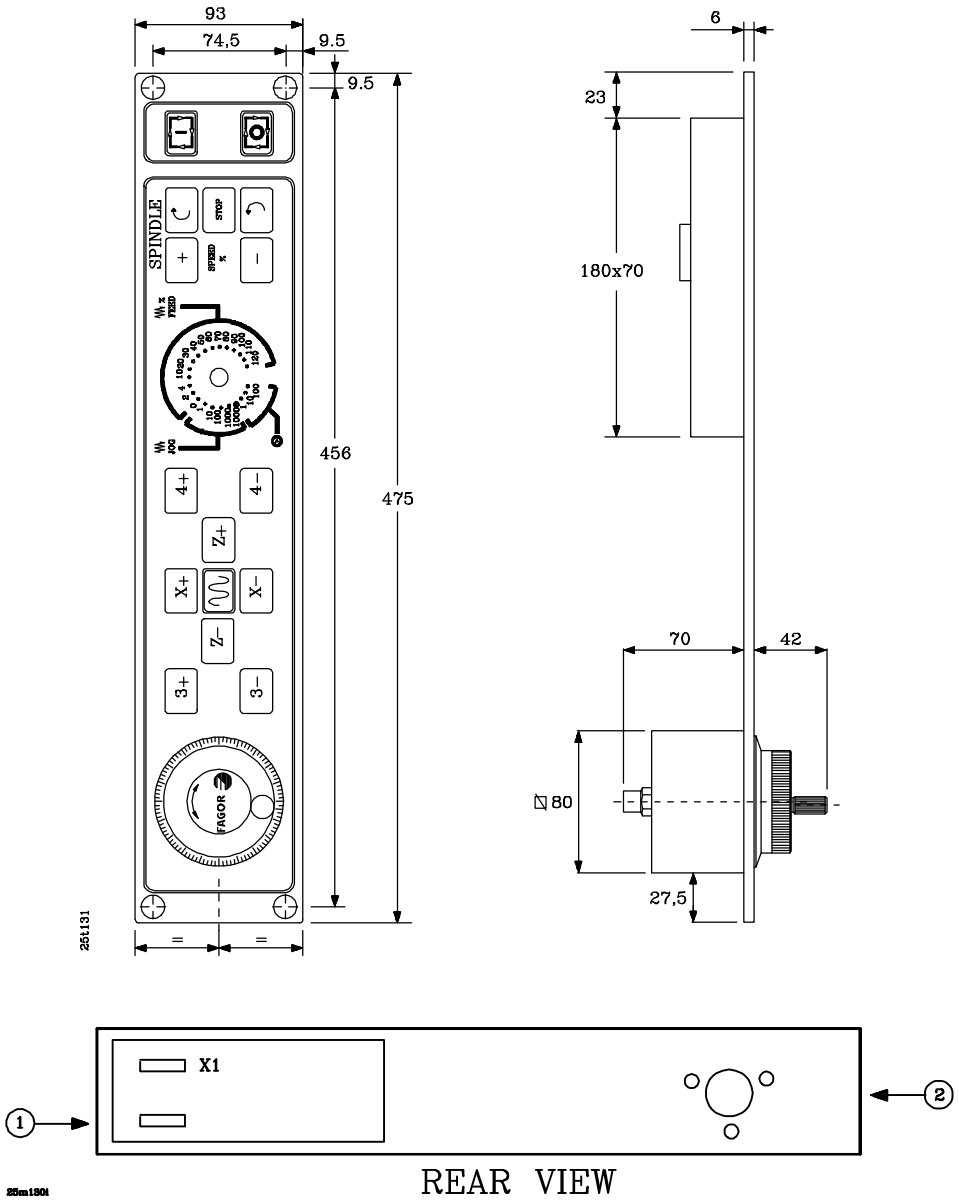
To obtain a greater distance between the Monitor/Keyboard and the Operator Panel, replace this cable with a round 15-conductor cable (15 x 0.14 mm²) with overall shield and acrylic rubber cover. The length of this cable plus the length of the one used between the Central Unit and the Keyboard (X1) must 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.



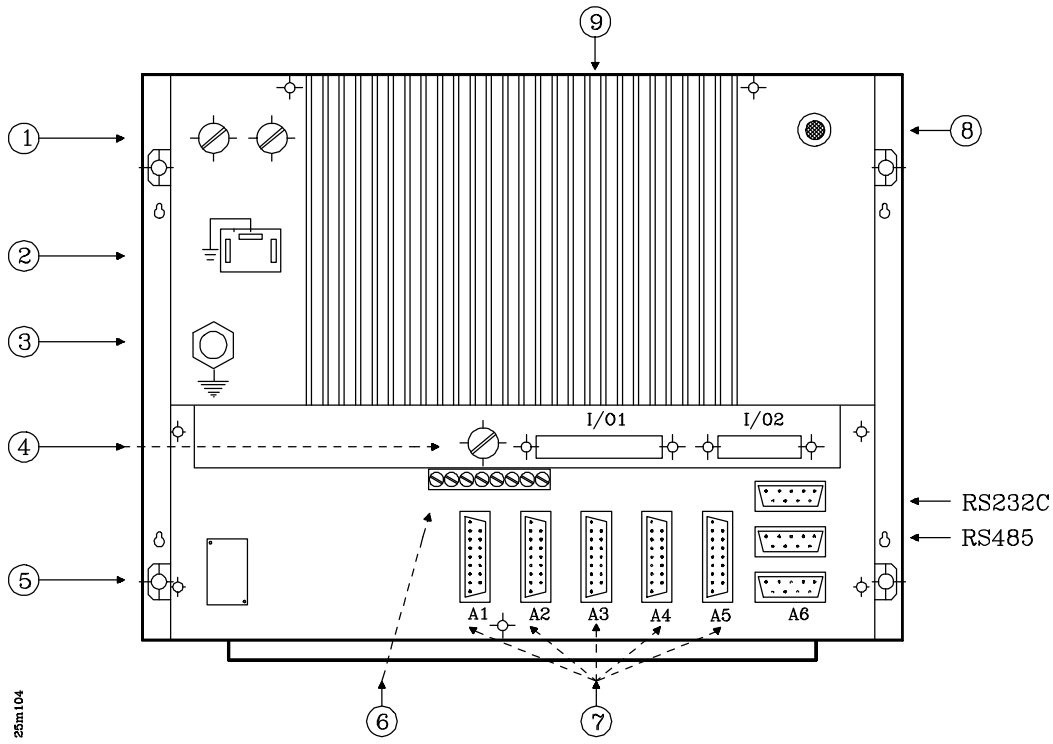
X1 15-pin SUB-D type female connector to connect the MONITOR/KEYBOARD module to the OPERATOR PANEL module.

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 feedback system for the synchronized tool or the 4th axis. 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 feedback system for the C or 3rd axis. It accepts sine-wave signal.

A5 **15-pin SUB-D type female connector** to connect the spindle 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 the feedback signals from the live or synchronized tool.

  When using a 4th axis, machine parameter "P614(1)" must be set to "1" using feedback input A6 for the live or synchronized tool.

  When not using a 4th axis, "P614(1)" must be set to "0" using this connector for the live or synchronized tool.

  Also, when using a live tool, machine parameter "P802" must be assigned a value other than "0" and when using a synchronized tool, "P802" and "P803" must be set to a value other than "0".

* Connector A3 for Z axis feedback signals.

* Connector A4 for the "C" or 3rd axis feedback signals.

  In both cases, machine parameter "P612(1)" must be set to "1" indicating that the machine has a 3rd axis.

  When the 3rd axis is a "C" axis, "P613(5)" must be set to "1" indicating that it is a "C" axis

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	$\overline{A}$ A $\overline{B}$ 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).

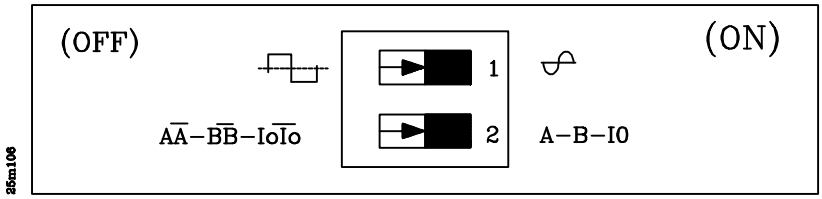
The possible types of feedback signals to be used at connectors A1 thru A4 are:

* 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 spindle feedback signal.
It does not accept 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.

PIN	SIGNAL AND FUNCTION	
1 2 3 4	$\overline{A}$ A $\overline{B}$ B	Double-ended square-wave signal.
5 6	$\overline{Io}$ Io	Machine Reference signals (marker pulse)
7 8	Micro Io 0V.	Spindle home switch signal input. Spindle 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).

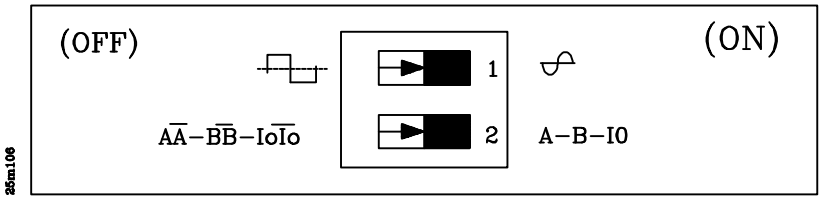
The possible types of feedback signals to be used at connector A5 are:

* 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 synchronized tool 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	A	Square-wave signals
2	B	Square-wave signals
3	Io	Home marker pulse (Machine Reference)
4	+5V.	Power to feedback system
5	0V.	Power to feedback system
6	PROB 5	Probe input: 5 V. TTL
7	PROB 24	Probe input: 24 Vcc
8	0 PROB	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 model 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.

1.3.4 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.

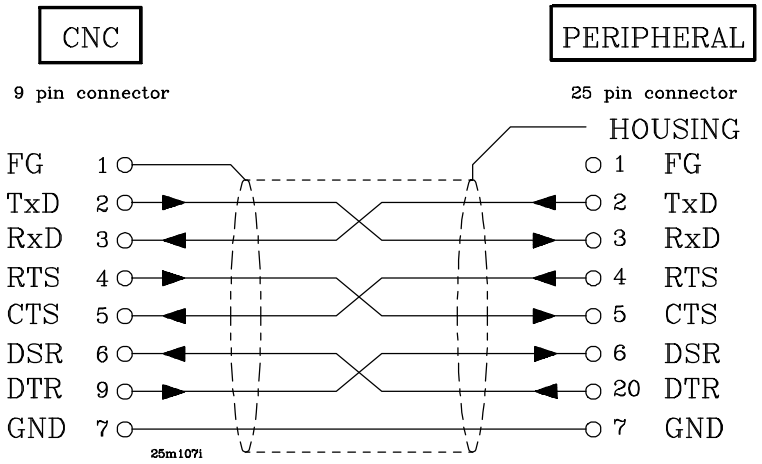
* **Transmission speed (baudrate)**. The baudrate normally used with peripherals is 9600 baud.

  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

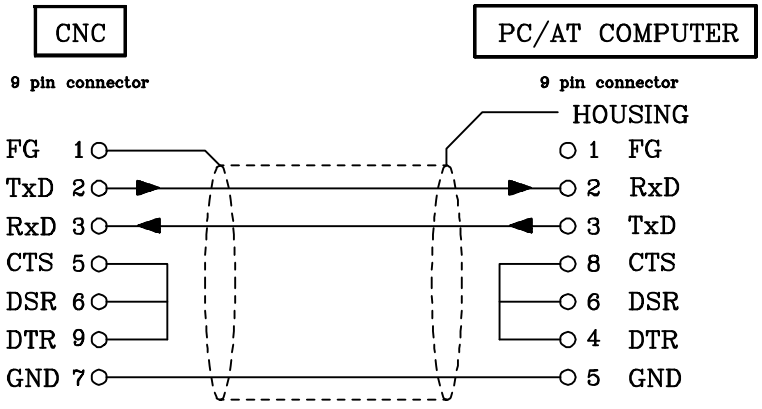
* Complete connection

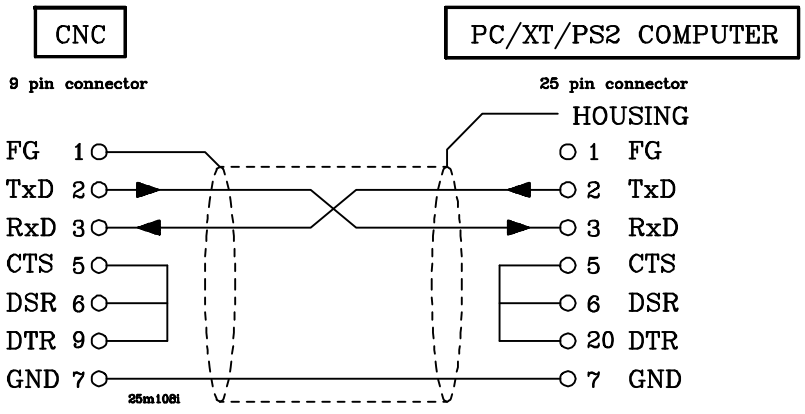


* Simplified connection

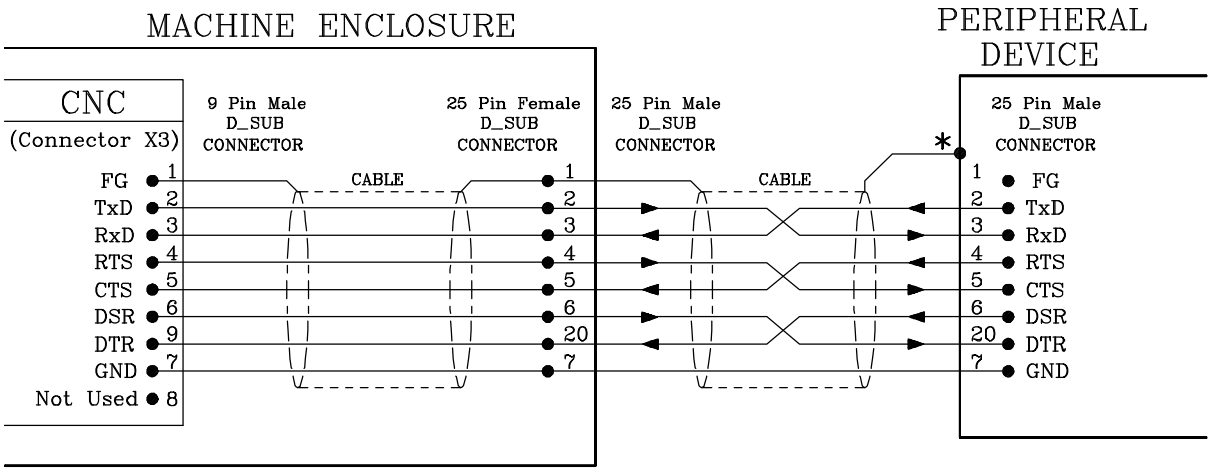
To be used when the peripheral or the computer meets one of the following requirements:

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

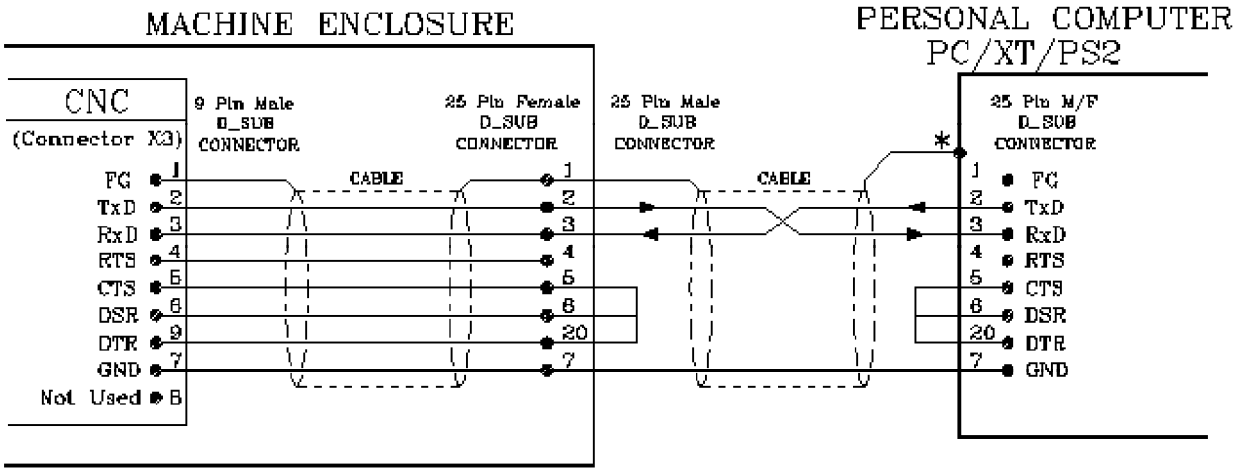




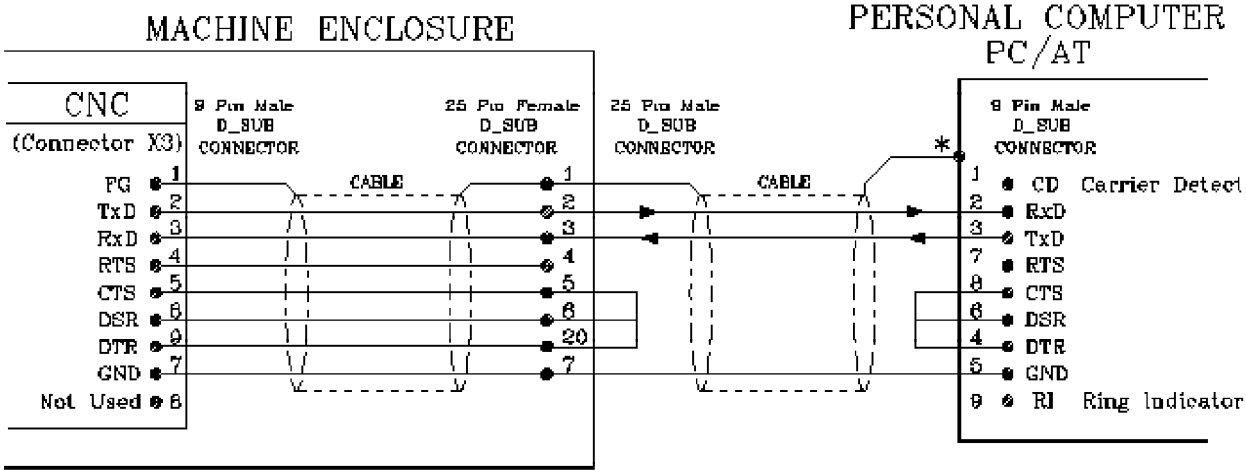
Nevertheless, it is suggested to refer to the technical manuals of the peripheral equipment in case there should be any discrepancy.



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	Threading ON	Output.
	Cycle ON	
7	Z Enable	Output.
8	Reset	Output.
9	X Enable	Output.
10	X home switch	Input from machine reference switch.
11	3rd axis home switch	Input from machine reference switch.
12	Z home switch	Input from machine reference switch.
13	4th axis home switch	Input from machine reference switch.
	Emerg. Subroutine	Activate the emergency subroutine.
14	Emergency Stop	Input.
15	Feed Hold	Input.
	Transfer inhibit	
	M-done	
16	Stop	Input.
	Emergency subrout.	Activate the emergency subroutine.
17	Start	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 live tool.
33	0V.	Analog output for live tool.
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

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.

"C" OR 3rd AXIS HOME SWITCH Pin 11

This INPUT must be high (24V) as long as the machine reference switch for the "C" or 3rd 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.

4th AXIS HOME SWITCH / $\overline{\text{Activate the emergency subroutine}}$ Pin 13

When the machine has a 4th axis, P614(1)=1, this input corresponds to the home switch for this axis. This input must be high (24V) as long as the machine reference switch for the 4th axis is pressed.

When the machine does not have a 4th axis, P614(1)=0, and an emergency subroutine has been programmed, P716<>0, the CNC will activate such subroutine every time this input is set low (0V).

$\overline{\text{EMERGENCY STOP}}$ 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 "P602(7)" 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/ Activate the emergency subroutine Pin 16

This INPUT must be normally high (24V) and its meaning depends on the the system configuration.

* If the machine has a 4th axis, "P614(1)=1", and there is an emergency subroutine (P716<>0), the CNC will activate the emergency subroutine whose number is indicated by "P716" every time this input is set low.

* If the machine has neither a 4th axis, "P614(1)=0", nor an emergency subroutine (P716=0), the CNC considers this input as (Cycle Stop) and acts as follows:

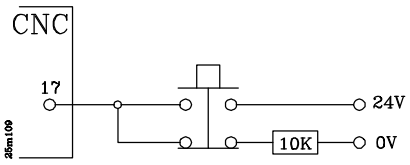
  When this input is set low (0V), the CNC interrupts the program execution just as if the  key were pressed at the OPERATOR PANEL.

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

START Pin 17

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

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



When an up-flank (leading edge or low-to-high transition) of this signal (START) is detected, 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 P601(5) to "1".

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).

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 P604(4).

THREADING ON / CYCLE ON Pin 6

This output is set normally low and its meaning depends on the setting of machine parameter "P605(4)".

"P605(4)=0" THREADING ON.

The CNC activates this output, setting it high, when a threading operation is being performed.

"P605(4)=1" CYCLE ON.

The CNC activated this output, setting it high, when executing a program block.

Z AXIS ENABLE Pin 7

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

RESET Pin 8

The CNC sets this output high (24V) when resetting the CNC by means of the [RESET] key.

The CNC keeps this signal active for 80 milliseconds.

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 live tool ±10V. Pin 32
Analog voltage for live tool 0V. Pin 33

These outputs provide the analog voltage for the the live tool. 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 to govern the spindle when in open loop (S) and when working as "C" axis. 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	Value of bit 10 of the decoded M function table.
13	Output M11 4th axis Enable	Value of bit 11 of the decoded M function table.
14	0V	Analog voltage output for 4th axis servo drive.
15	±10V.	Analog voltage output for 4th axis servo drive.
16	CHASSIS	Connect all cable shields to this pin.
17	0V	Analog voltage output for 3rd axis servo drive.
18	±10V.	Analog voltage output for 3rd 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 Spindle lock "C" axis Enable	Value of bit 15 of the decoded M function table.
23	Output M14 G00	Value of bit 14 of the decoded M function table.
24	Output M13 Turret Rotation	Value of bit 13 of the decoded M function table.
25	Output M12 3rd axis Enable	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 M11 / 4th axis Enable Pin 13

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

When the 4th axis is being used, "P614(1)=1", this output will be utilized as Enable signal for this axis.

Therefore, When having a 4th axis, be careful not to set the bit of the decoded M table which corresponds to this M11 output 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 / Spindle lock / "C" axis Enable Pin 22

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

When the spindle is working in closed loop (M19), this output is used as the spindle lock. That is, it will be set to "0" when the spindle has to move and it will return to "1" when the spindle has reached its programmed position (within the in-position or dead-band zone).

When the machine has a 3rd axis, "P612(1)=1" working as a "C" axis, "P613(5)=1", this output will be used as "C" axis Enable.

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 / G00 Pin 23

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

If machine parameter P604(3) 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 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 / Direction of turret rotation Pin 24

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

If machine parameter "P609(6)=1", this OUTPUT is set high whenever the turret turns in the positive direction.

For example, if a 12-tool turret is used and the current tool is number 2, the value of this output will depend on the next tool being selected:

* When selecting tool 4 (positive direction 2, 3, 4), this output will be set high.

* When selecting tool 10 (negative direction: 2, 1, 12, 11, 10).

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 / 3rd axis Enable Pin 25

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

When the machine has a 3rd axis "P612(1)=1", this OUTPUT is used as the 3rd axis Enable.

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%.

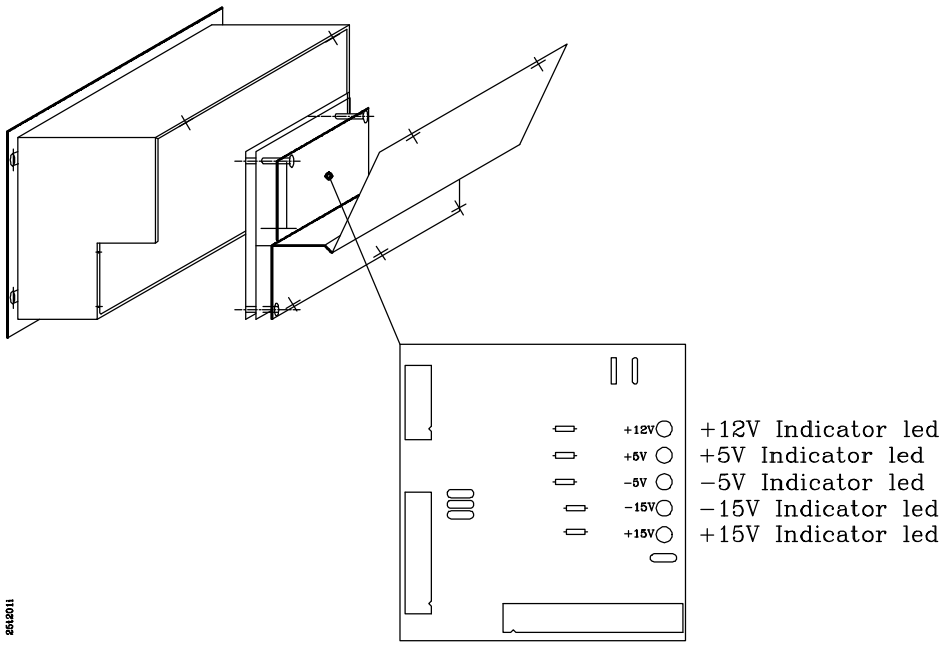
The power outlet to connect the equipment must be near it and easily accessible.

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.

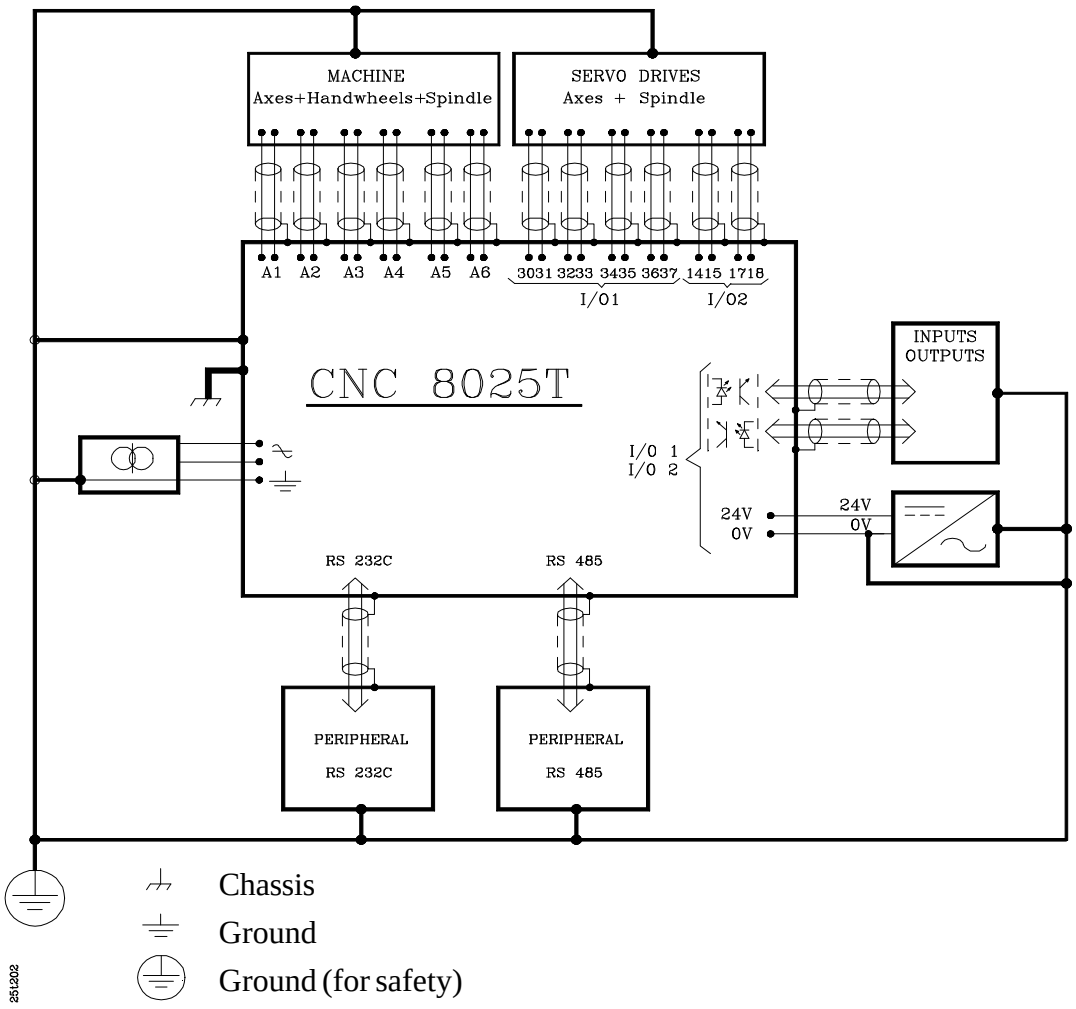
Even when a proper **ground** connection reduces the effects of electrical interference (noise), the signal cables require additional protection.
This is generally achieved by using twisted-pair cables which are also covered with anti-static shielding mesh-wire. This shield must be connected to a specific 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 feedback signals from the 4th axis or the synchronized tool 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 feedback signals from the 3rd or "C" axis and it accepts sine-wave and double-ended (differential) square-wave signals.

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

Connector A6 is used for the feedback signals from the probe and the electronic handwheel or the synchronized tool. 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-TS).

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)	4th axis home switch
F	12 (I/O 1)	Z axis home switch
G	11 (I/O 1)	3rd 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) Threading ON	(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) Reset	(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:

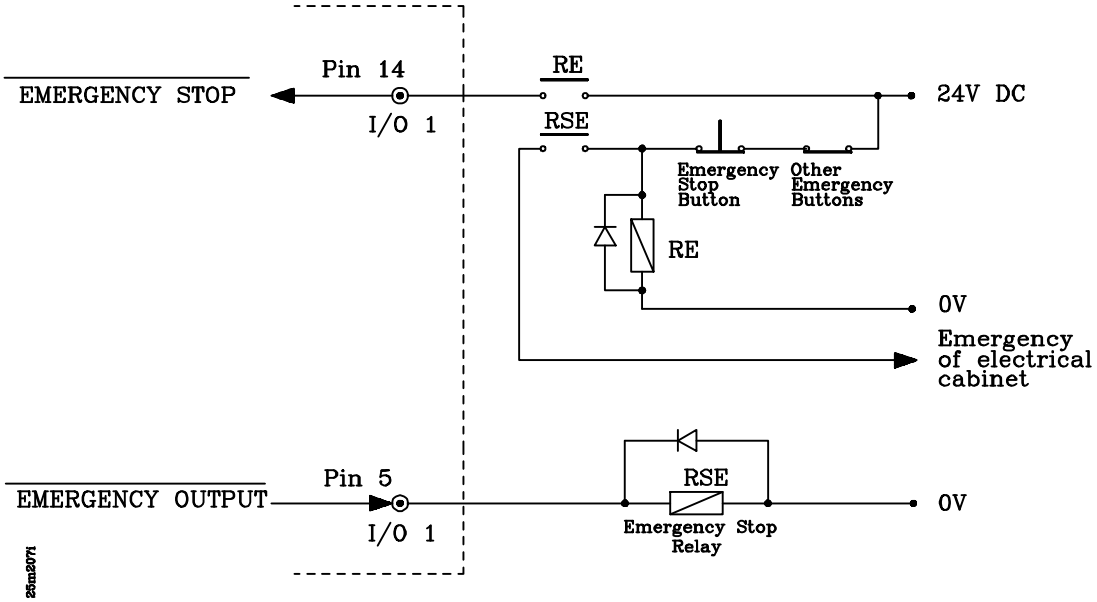
* An excessive axis following error has occurred.

* An axis feedback error has occurred.

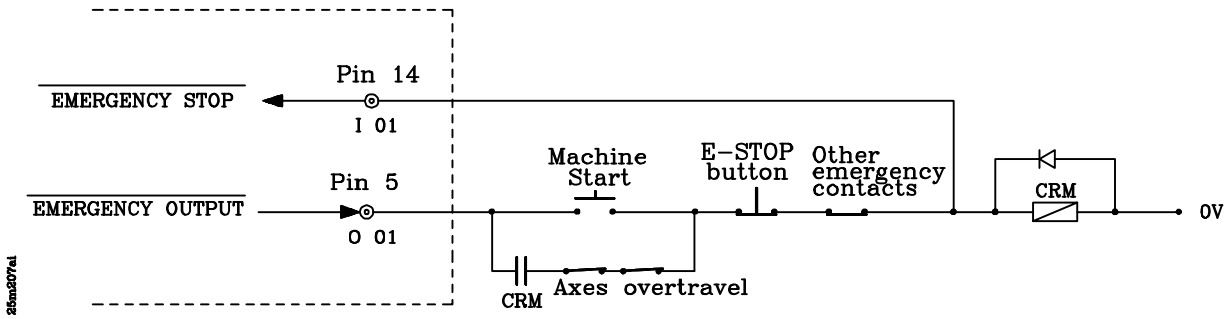
* There is erroneous data on the machine parameter table.

The recommended connection when P604(4)= 1 (output normally HIGH) is:

European Style:

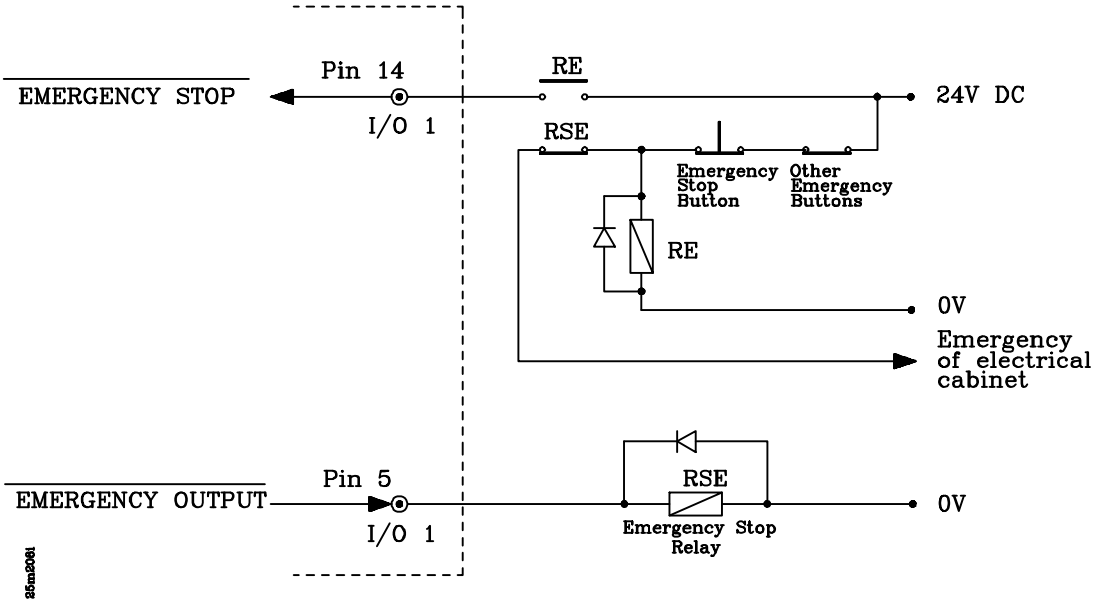


USA Style:

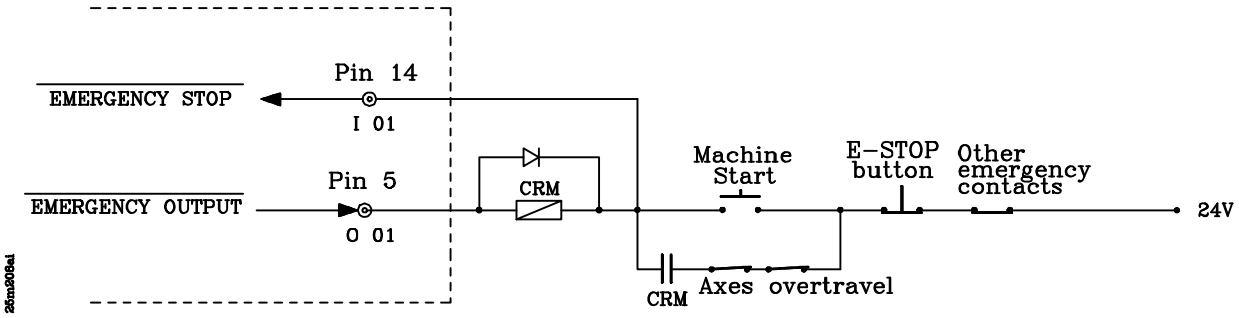


The recommended connection when P604(4)= 0 (output normally LOW) is:

European Style:



USA Style:



3. MACHINE PARAMETERS

Atention:

All unused machine parameters must be set to "0" to guarantee the proper functioning of the 8025T 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 "**PKAI1**" and press **[ENTER]** to lock the access or key in "**PKAI0**" 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 enter the machine parameter values via the keyboard, press the following keystroke sequence:

[OP MODE]	(SPECIAL FUNCTIONS)
[9]	(SPECIAL MODES)
[1]	(MACHINE PARAMETERS)

3.2 OPERATION 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:

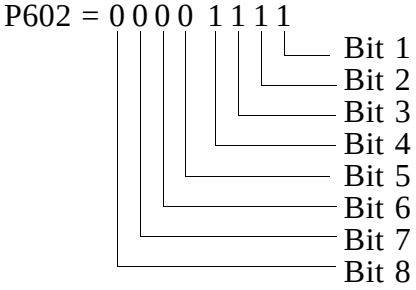
*	A number	P111 = 30000
*	A group of 8 bits	P602 = 00001111
*	A character	P105 = Y

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 parameters are locked, therefore protected against modifications.

Once all the desired parameters have been set, either press [RESET] or power the CNC off and back on so the CNC assumes the new values.

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



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)

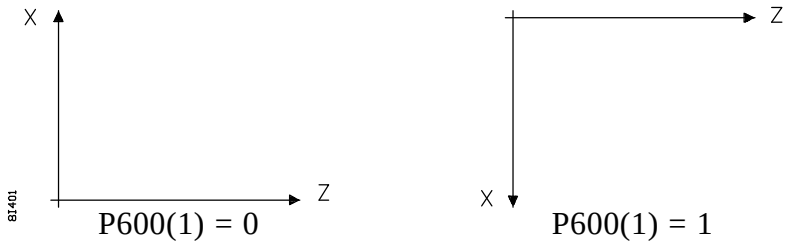
It determines the measuring units assumed by the CNC for machine parameters, tool tables and work units at power-up and after emergency or RESET.

0 = Millimeters.
1 = Inches.

P11 X axis display in radius or diameter

0 = Radius
1 = Diameter

P600(1) Orientation of the axes



P801 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 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????”.

3.3.1 PARAMETERS RELATED TO AXES CONFIGURATION

This CNC has 6 feedback inputs for the axes, A1 through A6, and the parameters below make it possible to adapt the CNC to the type of machine available.

The possible axis combinations are:

Without Synchronized Tool						With Synchronized Tool					
A1	A2	A3	A4	A5	A6	A1	A2	A3	A4	A5	A6
X	-	Z	-	S	Handwheel	X	Sync. Tool	Z	-	S	Handwheel
X	-	Z	3rd	S	Handwheel	X	Sync. Tool	Z	3rd	S	Handwheel
X	-	Z	C	S	Handwheel	X	Sync. Tool	Z	C	S	Handwheel
X	4th	Z	3rd	S	Handwheel	X	4th	Z	3rd	S	Handwheel Sync. Tool
X	4th	Z	C	S	Handwheel	X	4th	Z	C	S	Handwheel Sync. Tool

P612(1), P614(1) The machine has a 3rd, 4th axis

0 = The machine does **not** have a 3rd or a 4th axis.
1 = The machine has a 3rd or a 4th axis.

P613(5) The 3rd axis is the "C" axis

0 = It is **not** the C axis.
1 = It is the C axis.

P613(4), P615(4) the 3rd, 4th axis is called Y/W

0 = The 3rd, 4th axis is called Y.
1 = The 3rd, 4th axis is called W.

Obviously, when using both axes, one will be referred to as W and the other one as Y. The W axis usually corresponds to the tail-stock.

P613(3), P615(3) The 3rd, 4th axis as a DRO axis

Indicates whether the CNC treats the corresponding axis as a normal axis or as DRO axis.

0 = Normal axis.
1 = DRO axis.

P613(1), P615(1) The 3rd, 4th axis is rotary

It indicates whether the axis is linear or rotary.

The position of a rotary axis is shown in degrees. Therefore, it is not affected by the mm/inch unit change. **It does not admit tool radius compensation or circular interpolation.**

0 = Linear axis.
1 = Rotary axis.

P613(2), P615(2) The 3rd, 4th axis is rotary ROLLOVER

This parameter is used when the axis is set as rotary, “P613(1)=1, P615(1)=1” and the count is to roll over to 0° when reaching 360°.

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

P613(6), P615(5) The 3rd, 4th axis is rotary rollover via shortest path

This parameter is used when the axis is set as rotary rollover and the programmed movements are to be carried out via the shortest path.

0 = It is **not** positioned via the shortest path.
1 = It is positioned via the shortest path.

P616(8) Connector A6 shared by the handwheel and the synchronized tool

This parameter must be set to "1" when the machine has a 4th axis, synchronized tool and electronic handwheel (all three).

A2 must be the feedback connector for the 4th axis, A6 must then be shared by the synchronized tool and the electronic handwheel.

Pin 21 of connector I/O1 (output indicating JOG mode selected) may be used to switch both feedbacks (synchronized tool and electronic handwheel) since the electronic handwheel can only be used in JOG mode.

0 = Connector A6 is **not** shared.
1 = Connector A6 is shared.

3.3.2 I/O PARAMETERS

P604(4) 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(6) Pin 24 of connector I/O 2 indicates turret rotating direction

It determines whether pin 24 of connector I/O 2 is used to indicate the turrret rotating direction or not.

0 = It is output 13 of the decoded M functions.
1 = It is the output indicating the turret rotating direction and output 13 of the decoded M functions.

If this machine parameter is set to "1", this OUTPUT is set high (24V) whenever the turret turns in the positive direction.

For example, if a 12-tool turret is used and the current tool is number 2, the value of this output will depend on the next tool being selected:

* When selecting tool 4 (positive direction 2, 3, 4), this output will be set high.

* When selecting tool 10 (negative direction: 2, 1, 12, 11, 10).

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.

P604(3) G00 mode indicating output at pin 23 of connector I/O 2

It determines whether pin 23 of connector I/O 2 is used to indicate the G00 mode or not.

0 = It is output 14 of the decoded M functions.
1 = It is the G00 output and output 14 of the decoded M functions.

This output will stay active (24V) while the CNC is performing a G00 move (rapid traverse).

It must be borne in mind that the CNC uses the same pin to indicate both concepts (G00 and M14 output). Therefore, if it is to be used as an indicator for G00, this bit must not be used when setting decoded M functions.

P605(4) Pin 6 of connector I/O 1 as THREADING-ON or CYCLE-ON indicator

0 = This output will be active (24V) when a THREADING cycle is being executed.

1 = This output will be active (24V) when a program block is being executed (CYCLE ON) is being executed or when a "BEGIN-START", "END-START" type command is being executed.

P606(7) M function not output in BCD

When executing an M function which has been decoded at the M function table, the CNC will activate and/or deactivate the corresponding outputs at connector I/O 2.

This parameter determines whether or not besides activating the outputs set on the table, the CNC also activates the BCD outputs: "MST01" thru "MST80" (pins 20 thru 27 of connector I/O 1) corresponding to that M function.

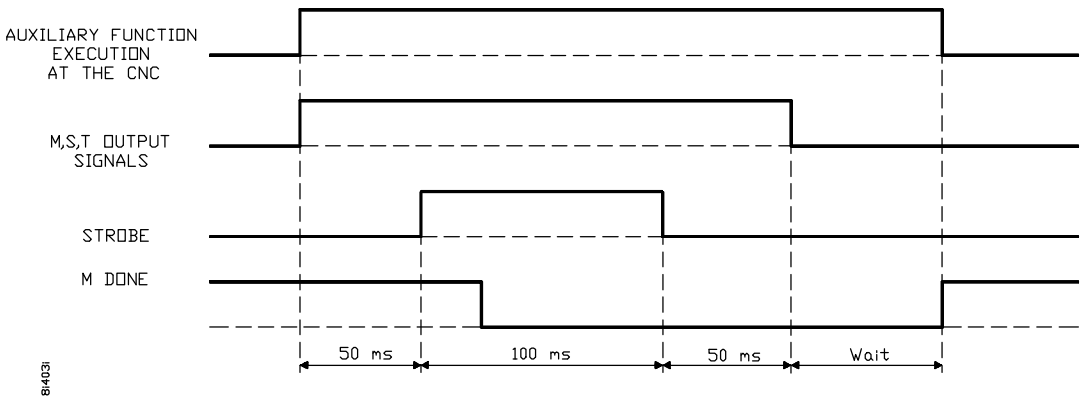
0 = The M function is **also** sent out in BCD.
1 = The M function is **not** sent out in BCD

P602(7) The CNC waits for a down flank (trailing edge) of the M-done signal.

It indicates whether it is necessary or not to wait for the down flank (change from 24V to 0V) 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.

"P602(7)=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.



“P602(7)=1”

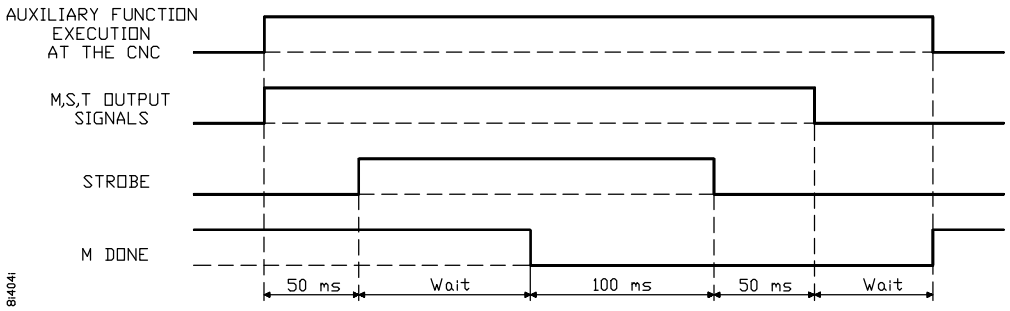
50 milliseconds after having sent the M, S or T BCD signals out to the electrical cabinet, it sends out the corresponding "Strobe" signal.

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

Once the "M-DONE" signal is set low, the CNC maintains the "Strobe" signal active for another 100 milliseconds.

After deactivating the Strobe signal, the M, S or T BCD 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.



P603(4), P603(3), P603(2), P603(1), P608(1)
Cancellation of feedback alarm for connectors: A1, A2, A3, A4 and A5

The CNC will issue a feedback alarm for an axis when its corresponding feedback signals are not received properly.

This parameter indicates whether this alarm is to be active or cancelled.

0 = Alarm active.
1 = Alarm cancelled.

This parameter must be set to "1" when the feedback system installed uses only three square-wave signals (A, B, Io).

3.3.3 HANDWHEEL PARAMETERS

P609(1) The electronic handwheel is the FAGOR 100P

Indicates whether the electronic handwheel is or not a FAGOR handwheel model 100P (with axis selector button).

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

P500 Counting direction of the handwheel

It sets the counting direction of the handwheel. If correct, leave it as is; otherwise, assign the other value.

0 = NO and 1 = YES.

P602(1) Feedback units for the handwheel

It indicates whether the CNC considers the handwheel pulses to be in mm or in inches.

0 = Millimeters.
1 = Inches.

P501 Feedback resolution of thehandwheel

It indicates the counting resolution of the handwheel.

Possible values with square-wave signals:

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

P602(4) Multiplying factor for the feedback signals from the handwheel

It indicates the x2 or x4 multiplying factor to be applied to the feedback signals supplied by the handwheel.

0 = x4.
1 = x2.

Example:

If the handwheel is set as follows:

P602(1) = 0 Millimeters
P501 = 1 0.001 mm resolution.
P602(4) = 0 x4

The feedrate override switch is positioned at x100.

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

P619(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 O44 and O45 or Marks M12, M13 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 O44 and O45 or Marks M12 and M13 of the PLC64.

O44 M12	**O45 M13**	
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

P606(6) 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.).

P710 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.

P806 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.

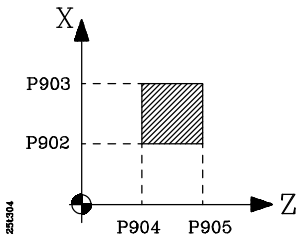
Possible values: 1 thru 65.535 mm./minute (degrees/minute).
1 thru 25.800 tenths-of-inch/minute.

P902 Minimum X coordinate of the touch probe
P903 Maximum X coordinate of the touch probe
P904 Minimum Z coordinate of the touch probe
P905 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.



3.3.5 TOOL PARAMETERS

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

P700 Number of tools

It is given by an integer between 0 and 32.

P730 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.

Atention:

When associating a subroutine to the T function, the T function must be the last programmed item on the block. Otherwise, the CNC will issue the corresponding error code.

P617(2) 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 the subroutine is executed before the T function, remember the following considerations:

* The T function must be programmed alone in the block.

* If the T function is executed in the JOG or TEACH-IN modes, the CNC will **not** execute the associated subroutine.

P604(5) The tool offset values are effective after executing M06

It indicates whether the tool offset is applied right after the T2.2 function is executed or after M06 is executed.

0 = Tool offset effective after T2.2
1 = Tool offset effective after M06

For example: If every time a new tool is selected, the machine must make some movements prior to the tool change; it is recommended to set this parameter to "1" and program an M06 right after the tool change in the subroutine associated to the T function.

P609(3) The theoretical position corresponds to the tool tip or tool base

It indicates whether the CNC ignores the tool dimensions or not when displaying the theoretical position. That is, whether the displayed position corresponds to the tool tip or to the tool base.

0 = The displayed theoretical position value corresponds to the tool tip.
1 = The displayed theoretical position value corresponds to the tool base.

When setting this parameter to "0", the CNC will update the displayed position after the tool change showing the new position corresponding to the tip of the new tool.

This way, if an incremental move (G91) is programmed after a tool change, the target position will be referred to the tip of the new tool.

On the other hand, when setting this parameter to "1", the CNC will not update the displayed position and if an incremental move (G91) is programmed after a tool change, the target position will be referred to the last programmed position.

3.3.6 PARAMETERS RELATED TO THE EMERGENCY SUBROUTINE

P716 EMERGENCY subroutine

It indicates the number of the standard subroutine (not parametric) that will be executed when activating the EMERGENCY SUBROUTINE input (pin 13 or16 of connector I/O1).

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

P616(2) 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.

P616(1) 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 P2=Z”, and after performing all the emergency operations we program, inside the emergency subroutine, a block with movement to point "XP0 ZP2", the tool will return to the point of program interruption or to the beginning point of the interrupted block.

3.3.7 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
1,200
2,400
4,800
9,600

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.

P605(5) DNC active

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

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

P605(6) Type of communication, FAGOR Floppy Disk Unit or Cassette

P605(6) =1 Communication with a FAGOR Floppy Disk Unit. The CNC uses the settings of machine parameters P0, P1, P2 and P3.

P605(6)=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

Atention:

In DNC communications as well as with peripherals, the CNC uses the settings of machine parameters: P0, P1, P2 and P3.

P605(7) 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.

P605(8) 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).

P606(8) 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.

3.3.8 DISPLAY RELATED PARAMETERS

P6 Theoretical or Real display

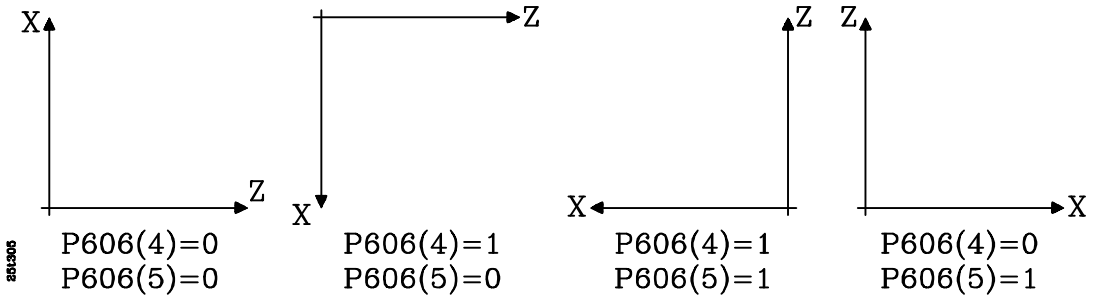
It determines whether the CNC will display the real axis position or the theoretical position.

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

It is recommended to set this parameter to "0" during the adjustment of the machine axes and then set it to "1" for normal operation.

P606(4,5) Axes orientation in graphic display

They determine the orientation of the axes in the graphic display so they match the orientation of the machine axes.



P612(8), P614(8) 3rd, 4th axis display

They indicate whether the corresponding axis is or not displayed by the CRT.

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

P611(7), P611(8) Monitor display color combination

They are used to choose the desired display color combination. The choices are:

P611(8)	P611(7)	Display color
0	0	Monochrome
0	1	Combination 1
1	0	Combination 2

3.3.9 JOG-MODE RELATED PARAMETERS

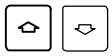
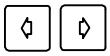
P12 Continuous or pulsating axis jog

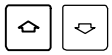
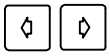
It determines whether the selected axis moves (jogs) while the corresponding jog key is pressed or it keeps moving until either the  key or another JOG key is pressed.

N = Continuous mode. The axis starts moving when its corresponding JOG key is pressed and it stops when the  key or any other JOG key is pressed. When pressing the jog key for another axis, this new axis will begin to move in the chosen direction until  or another JOG key is pressed.

Y = Pulsating mode. The axis will move while keeping the corresponding JOG key pressed.

P600(2) JOG key assignment to the X and Z axes.

0 = The  keys control the X axis and the  keys control the Z axis (horizontal lathe).

1 = The  keys control the Z axis and the  keys control the X axis (vertical lathe).

P603(5) Possibility to execute the "S" function in JOG mode

It indicates whether it is possible or not to execute the "S" function while in JOG mode.

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

P603(6) Possibility to execute the "T" function in JOG mode

It indicates whether it is possible or not to execute the "T" function while in JOG mode.

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

P603(7) Possibility to execute the "M" function in JOG mode

It indicates whether it is possible or not to execute the "M" function while in JOG mode.

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

P619(8) Possibility to work at Constant Surface Speed while in JOG mode

It indicates whether it is possible or not to work at Constant Surface Speed while in JOG mode.

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

P601(7) Recover initial conditions when returning to the standard work mode.

It determines whether or not the CNC must recover the initial conditions set by machine parameters (spindle status, feedrates, etc.) every time the standard work mode is accessed.

The standard work mode is accessed in the following cases:

* On CNC power-up, after pressing any key.

* When quitting the tool table.

* When quitting any of the auxiliary modes, general parameters, decoded M functions, leadscrew compensation table, peripherals or the lock/unlock option.

0 = No, it does not recover the initial conditions.
1 = Yes, it does recover the initial conditions.

If this parameter is set to "1", the CNC will also generate an M30 function.

3.3.10 OPERATING-MODE RELATED PARAMETERS

P601(5) Inhibiting the START key.

It indicates whether the  key from the front panel is ignored by the CNC or not.

0 = It is not ignored. Not inhibited.
1 = It is ignored. Inhibited.

P619(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 O43 signal of the PLCI or the M11 signal of the PLC64 to disable or re-enable the Spindle.

This machine parameter, P619(6), indicates whether or not O43 (at the PLCI) or M11 (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, (O43 =1) or (M11=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, (O43 =0) or (M11=0), the CNC outputs the corresponding spindle analog voltage again.

P600(3) Maximum % value of the Feedrate Override Switch applied by the CNC

It determines the maximum % value to be selected with The Feedrate Override Switch.

0 = 120% of the programmed feedrate as indicated by the switch.
1 = limited to 100% of the programmed feedrate even when the switch indicates 110% and 120%.

P4 Feedrate Override Switch active in rapid moves or not

It determines whether the Feedrate Override Switch is active during rapid moves or not

N = The switch is ignored and the rapid moves are carried out **at 100%**.
Y = The CNC applies the % override indicated by the switch (between **0%** and **100%** even when indicating 110% and 120%).

P607(8) G05 or G07 active on power-up

It determines whether the CNC assumes a G05 (round corner) or G07 (square corner) on power-up.

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

P607(3) Vectored G00 (interpolated)

It determines whether the G00 moves are vectored (interpolated) or not.

0 = Not vectored G00 moves
1 = Vectored G00 moves.

P616(4) G59 as additive zero offset

It indicates whether function G59 is treated as normal zero offset or as an additive zero offset.

0 = G59 as normal zero offset.
1 = G59 as additive zero offset.

When setting this parameter to "1", additive, every time a G54-G58 type function is executed, the CNC applies on to each axis a zero offset equal to the sum of the programmed offset value plus that of G59.

P609(5) Arithmetic parameters P150 through P254 as read-only

It indicates whether arithmetic parameters P150 through P254 are read/write or read-only when the machine parameters are locked (code: PKAI1).

0 = Always read/write.
1 = When the machine parameters are locked, these arithmetic parameters are read-only. When the machine parameters are not locked, these arithmetic parameters are read/write.

P611(6) Function P1=0X uses the current work units (millimeters or inches)

It indicates whether or not the CNC uses the current work units when executing a P1=0X type block.

0 = The current units are ignored. The axis position value with respect to machine reference zero (home) is always considered to be in millimeters.

1 = The current units are used. The axis position value with respect to home is taken in the currently active work units (millimeters or inches).

P621(4) Synchronization with independent axis

It indicates whether the independent axis (G65) is synchronized or not with the next block.

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

For example, When programming:
```
N100 G65 X32
N110 G01 Z100
N120 G01 Z20
```

Depending on the setting of this machine parameter, the CNC will behave as follows:

P621(4)=0 The CNC starts executing block N100 and, then, block N110. Once block N110 is in position, the CNC continues executing the program regardless of whether Block N100 has reached position or not.

P621(4)=1 The CNC starts executing block N100 and, then, block N110. Once block N110 is in position, the CNC waits until block N100 reaches position before going on to execute the rest of the program.

4. MACHINE PARAMETERS FOR THE AXES

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

P100, P300, P200, P400 Sign of the analog output for X, Z, 3rd and 4th axes.

They determine the sign of the analog output. If correct, leave them as they are; if not, change them.

Possible values: "N" and "Y".

IMPORTANT: When changing any of these parameters, also change the corresponding "P101", "P301", "P201" or "P401" parameter in order to prevent the axis from running away.

P101, P301, P201, P401 Counting direction of the X, Z, 3rd and 4th axis feedback devices.

If correct, leave them as they are; if not, change them.

Possible values: "N" and "Y".

IMPORTANT: When changing any of these parameters, also change the corresponding "P100", "P300", P200 or P400 parameter in order to prevent the axis from running away.

P102, P302, P202, P402 Jogging direction for the X ,Z, 3rd and 4th axes.

They determine the jogging direction by means of the JOG keys of the operator panel.

If correct, leave them as they are; if not, change them.

Possible values: "N" and "Y".

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, P303, P203, P403 X, Z, 3rd, 4th 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°.

P619(1), P619(2), P619(3), P619(4) Counting resolution for X, Z, 3rd, 4th axis with sine-wave feedback

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

Possible values for P103, P203, P303, and P403 with P619(1), P619(2), P619(3), P619(4) = 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, and P403 with P619(1), P619(2), P619(3), P619(4), = 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°.

P602(3), P602(2), P612(2), P614(2) X, Z, 3rd, 4th axis feedback units

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

0 = Millimeters or degrees
1 = Inches.

P106, P306, P206, P406 X, Z, 3rd, 4th 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.

P602(6), P602(5), P612(5), P614(5) **Multiplying factor for X, Z, 3rd, 4th 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	P602(6)	Feedback Signal period	FAGOR Linear transducer
P602(3)=0	P602(3)=1				
0.001 mm	0.0001 inch	1	x2 (=1)	0.002 mm	
			x4 (=0)	0.004 mm	CX, CVX, MX
0.002 mm	0.0002 inch	2	x2 (=1)	0.004 mm	CX, CVX, MX
			x4 (=0)	0.008 mm	
0.005 mm	0.0005 inch	5	x2 (=1)	0.010 mm	
			x4 (=0)	0.020 mm	CT, CVT, MT, MVT, FT
0.010 mm	0.0010 inch	10	x2 (=1)	0.020 mm	CT, CVT, MT, MVT, FT
			x4 (=0)	0.040 mm	

Using sine-wave linear transducers and P619(1)=1:

Besides the x2 or x4 selected by P602(6), 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 P619(1)=1, it is possible to obtain resolution of 1, 2, 5 and 10 microns or ten-thousandths of an inch.

Desired resolution		P103	P602(6)	Feedback signal period	FAGOR Linear transducer
P602(3)=0	P602(3)=1				
0.001 mm	0.0001 inch	1	x2 (=1)	0.010 mm	
			x4 (=0)	0.020 mm	CVS,MVS
0.002 mm	0.0002 inch	2	x2 (=1)	0.020 mm	CVS,MVS
			x4 (=0)	0.040 mm	
0.005 mm	0.0005 inch	5	x2 (=1)	0.050 mm	
			x4 (=0)	0.100 mm	FS
0.010 mm	0.0010 inch	10	x2 (=1)	0.100 mm	FS
			x4 (=0)	0.200 mm	

Using sine-wave linear transducers and P619(1)=0:

Besides the x2 or x4 selected by P602(6), 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 P619(1)=0, it is possible to obtain resolution of 1 and 2 microns or ten-thousandths of an inch.

Desired resolution		**P103**	**P602(6)**	**Feedback signal period**	**FAGOR Linear transducer**
P602(3)=0	P602(3)=1				
0.001 mm	0.0001 inch	5	x2 (=1)	0.010 mm	
			x4 (=0)	0.020 mm	**CVS,MVS**
0.002 mm	0.0002 inch	10	x2 (=1)	0.020 mm	**CVS,MVS**
			x4 (=0)	0.040 mm	

P604(2), P604(1), P612(3), P614(3) Binary encoder on X, Z, 3rd, 4th axis

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

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

P604(7), P604(6), P612(4), P614(4) Equivalence of the binary encoder used for the X, Z, 3rd and 4th axes.

These parameters are used when having binary rotary encoders (of 1024 or 2048 lines /rev.) and the desired resolution requires either 1000, 1250, 2000 or 2500 counts/rev.

These parameters are 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 MACHINE PARAMETERS FOR AXIS 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, P317, P217, P417 Minimum analog output for the X, Z, 3rd and 4th axes

It is given by an integer value 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)

P104, P304, P204, P404 Delay between Enable and Analog output for the X, Z, 3rd and 4th axes

They determine whether a 400 msec. delay must be applied from the time the Enable signal is activated to the instant the analog output is generated.

N = There is no delay between the two signals
Y = There is a 400 msec. delay between the two signals.

These parameters are to be used when there is no continuous control of the axes. therefore the 400 msec. delay could be used to deactivate possible axis holding devices (holding brake and so forth).

P118, P318, P218, P418 In-position zone for the X, Z, 3rd and 4th axes (dead band)

The In-position zone is the positioning tolerance area around the programmed position (coordinate) where the CNC considers an axis to be in position.

This area is expressed in microns regardless of the selected work units.

Value range: 0 thru 255 microns.

P105, P305, P205, P405 Continuous control of the X, Z, 3rd and 4th axes

They determine whether there is a continuous control of the axis or not once it has reached position. That is: whether the Enable signal remains on or not when the axis is in position.

N = The Enable signal disappears.
Y = The Enable signal is maintained (Continuous control).

The CNC keeps the axes in position when set as continuously controlled axis.

4.3 MACHINE PARAMETERS FOR THE TRAVEL LIMITS OF THE AXES

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

P107, P307, P207, P407 Positive travel limit for the X, Z, 3rd and 4th axes
P108, P308, P208, P408 Negative travel limit for the X, Z, 3rd and 4th axes

They determine the positive and negative travel limits for the axes. Each one must indicate the distance from Machine Reference Zero to each travel limit.

Value ranges: ± 8388.607 millimeters or degrees
± 330.2599 inches.

If both limits are set with the same value (for example "0"), the axis will not be able to move.

For safety reasons, it is only possible to move the axes up to 100 microns from the travel limits set by these parameters.

4.4 MACHINE PARAMETERS FOR THE LEADSCREWS

With this CNC, it is possible to compensate leadscrew errors due to the inaccuracy along the ballscrew or to its backlash when changing the direction of the axis movement.

4.4.1 LEADSCREW BACKLASH

P109, P309, P209, P409 Amount of leadscrew backlash for the X, Z, 3rd and 4th axes

When using linear scales, set this parameter to 0.

It is always expressed in microns regardless of the selected work units.

Value range: 0 thru 255 microns.

P620(1), P620(2), P620(3), Sign of the backlash for the X, Z, 3rd and 4th axes

Defines the sign of the backlash compensation value set in parameters P109, P209, P309 and P409.

0 = Positive sign.
1 = Negative sign.

P113, P313, P213, P413 Additional analog pulse for X, Z, 3rd and 4th axis backlash

Additional 40msec. analog pulse to recover the possible backlash of the leadscrew when reversing movement direction.

It is given by an integer between 0 and 255.

Value of	0	=	No additional pulse being applied.
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)

Every time the movement is reversed, the CNC will apply to this axis its corresponding analog voltage plus the additional pulse indicated by this parameter. This additional pulse will last for 40 milliseconds.

When using rotary encoders, set this parameter to "0".

4.4.2 LEADSCREW ERROR

There are 2 leadscrew compensation tables with 30 points each; one for the X axis and another one for the Z axis.

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%.

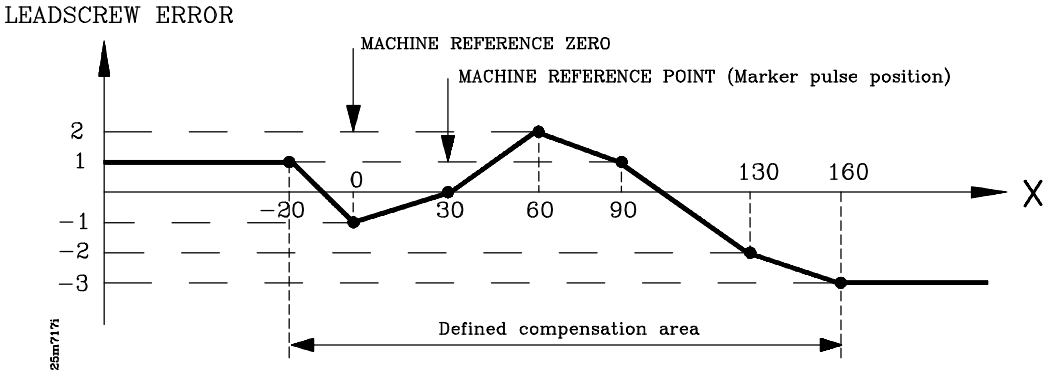
Examples: If the distance between two consecutive points is 3 mm. the maximum difference of their relevant error values can be 0.090 mm.
If the error difference between two consecutive points is the maximum (0.127mm), the distance between them cannot be smaller than 4.233mm.

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	

P605(2), P605(1) Leadscrew error compensation for the X and Z axes

They determine whether the CNC must apply or not leadscrew error compensation to the corresponding axis.

0 = No leadscrew error compensation is applied.
1 = Yes. Leadscrew error compensation is applied.

The CNC offers two leadscrew error compensation tables of up to 30 points each. one for the X axis and the other one for the Z axis.

4.5 MACHINE PARAMETERS FOR AXIS FEEDRATES

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:	Programming format	Programming unit	Minimum value	Maximum value
G94	F 4	F1= 1mm/min	F1 (1 mm/min)	F9999 (9999 mm/min)
G95	F3.4	F1= 1mm/rev.	F0.001 (0.001 mm/rev.)	F500.0000 (500 mm/rev.)

Inch programming:	Programming format	Programming unit	Minimum value	Maximum value
G94	F 4	F1= 0.1 inch/min	F1 (0.1"/min)	F3937 (393.7 inch/min)
G95	F 3.4	F1= 1 inch/rev.	F0.0001 (0,0001"/rev.)	F19.6850 (19,6850 inch/rev.)

When operating in inches and with rotary axes, we recommend setting machine parameter P618(2) to "1" so the programming units in G94 are in degrees/minute.

	P618(2)	Only rotary axis	Interpolation of rotary and linear axes
G94	P618(2)=0	F1= 2.54°/min	F1= 1 inch/min
	P618(2)=1	F1= 1°/min	F1= 1 inch/min

These and other feedrate related parameters are described next.

P618(2) 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, P310, P210, P410 Maximum programmable feedrate (F) for the X, Z, 3rd and 4th axes

Value range: 1 thru 65535 mm./minute
1 thru 25800 **inch/10 minutes**. (=2580 inch/min.)

P111, P311, P211, P411 G00 feedrate for rapid traverse of X, Z, 3rd and 4th axes

Value range: 1 thru 65535 mm./minute
1 thru 25800 **inch/10 minutes**. (=2580 inch/min.)

P717 Maximum feedrate F on arcs.

It determines the maximum feedrate allowed in a circular interpolation. This value depends on the radius of the arc and it is given by the following formula:

$$F\ max. = \frac{P717 \times Radius}{0.085}$$

It is expressed by an integer between 0 and 255 and if set to 0, it means that there is no feedrate limitation for circular interpolations.

Example:

Having set P717 to 17 in such a way that the maximum feedrate for arcs with a 15 mm radius is limited to 3000 mm/min.

When programming an arc with a 100 mm radius, the maximum feedrate allowed will be:

$$F\ max. = \frac{P717 \times Radius}{0.085} = \frac{17 \times 100}{0.085} = 20000\ mm/min.$$

P703 Feedrate/Override value when the analog voltage reaches 10V.

It indicates the Feedrate/override value (%) that the CNC will apply when the analog voltage of an axis reaches 10V.

It is given by an integer between 0 and 128.

Value of 0 = No override % is being applied.
Value of 32 = 25 %
Value of 64 = 50 %
Value of 128 = 100 %

This parameter makes the CNC "wait" for the axes on start-up by reducing the analog voltage for the axis and, therefore, its following error. Thus preventing the corresponding following error message from coming up.

P705 Error if the axis feedrate is not between 50% and 200% of the one programmed.

It indicates whether or not the CNC verifies that the actual axis feedrate is between 50% and 200% of the programmed feedrate (F).

It is defined by the time allowed for the feedrate to be out of this tolerance range. It is given by an integer between 0 and 255.

Value of 0 = This verification is not made.
Value of 1 = Error if out of tolerance range for more than 10 msec.
Value of 10 = Error if out of tolerance range for more than 100 msec.
Value of 255 = Error if out of tolerance range for more than 2550 msec.

4.6 MACHINE PARAMETERS FOR AXIS CONTROL

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

P114, P314, P214, P414 Proportional gain, K1 for X, Z, 3rd and 4th axes

They set 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.

$$\text{Analog (mV)} = K1 \times \text{Following error (microns)} \times \frac{2.5\text{mV.}}{64}$$

P115, P315, P215, P415 Gain break-point for the X, Z, 3rd and 4th 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. (P110, P210, P310, P410).

Value ranges: 1 thru 32766 microns
1 thru 12900 tenth-thousandths of an inch (=1.29inches)

P116, P316, P216, P416 Proportional gain K2 for the X, Z, 3rd and 4th axes

They determine the analog voltage corresponding to 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 speeds and rapid traverse (in G00).

P607(6) Apply only the proportional gain K1 or not during a threading operation.

During a threading operation, the CNC may apply either both "K1" and "K2" or just "K1".

0= It applies both proportional gains: K1 and K2. during a threading operation..
1= It applies only K1 during a threading operation.

P607(7) Apply only gain K2 in rapid positioning moves or both K1 and K2.

In rapid moves, it is possible to have the CNC apply either "K1" up to a set gain break-point of 256 microns and "K2" from that point on or just "K2" all the time.

0 = It applies both K1 and K2 gains with a set gain break-point of 256 microns.
1 = The whole rapid move is carried out with a gain of K2.

P715 Recovery of programmed position of the axes without continuous control.

It determines how the CNC behaves regarding the non-continuously controlled axes.

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 is no longer controlled by the CNC. However, depending on the value given to this parameter, it behaves as follows:

P715 = 0

If the axis drifts out of position a distance greater than 16 times the in-position value (P118, P218, P318 and P418). The CNC will issue the corresponding following error message.

P715 = other than 0.

If the axis drifts out of position a distance greater than P715/2 times the in-position value (P118, P218, P318 and P418), the CNC activates the corresponding enable signal in order to recover the drifted distance.

4.7 MACHINE PARAMETERS FOR MACHINE REFERENCE ZERO

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

P608(5), P608(8), P617(5), P615(8) Type of feedback marker pulse (reference mark "Io"). X, Z, 3rd, 4th 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.

P608(3), P608(6), P617(3), P615(6) Period of fixed Io signal (X, Z, 3rd,4th)

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

P608(4), P608(7), P617(4), P615(7) Variable Io signal period increasing in the positive or negative direction (X, Z, 3rd,4th axes)

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	P608 (5)	P608(3)	P608(4)
COS	1	0	1
COC	1	0	0
COX	1	0	0
COVS	1	0	1
COVC	1	0	0
COVX	1	0	0

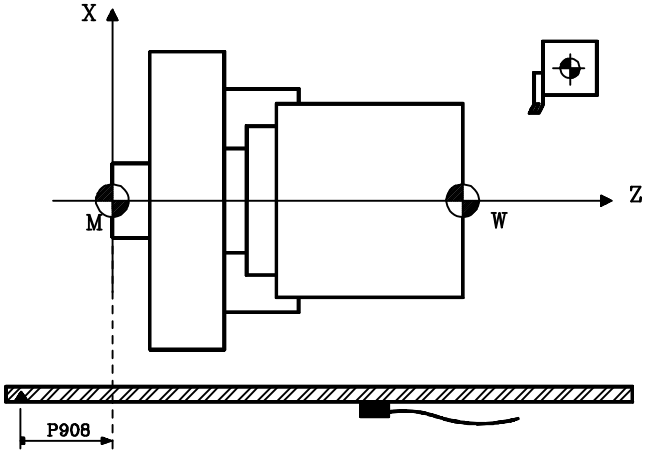
Scale	P608 (5)	P608(3)	P608(4)
MOVS	1	0	0
MOVC	1	0	0
MOVX	1	0	0
FOT	1	1	0
FOS	1	1	0
FOC	1	1	0

P908, P909, P910, P911 Offset of the semi-absolute linear scale. (X, Z, 3, 4)

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 P908, P909, P910 and P911 must be set with the Offset of the Machine home (M) with respect to the Absolute Reference Zero of the scale itself.



P600(5), P600(4), P612(6), P614(6) Home switch for the X, Z, 3rd, 4th axes

They indicate whether 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.

P618(8), P618(7), P618(6), P618(5) Direction of the home search along the X, Z, 3rd, 4th axes

They determine the direction of the axis move while searching the Machine reference point.

0 = Positive direction.
1 = Negative direction.

P600(7), P600(6), P612(7), P614(7) Type of machine reference pulse for X, Z, 3rd, 4th axes

They define the type of reference pulse (marker) (Io) of the feedback device being used.

Fagor scales have a negative marker pulse every 50mm and Fagor rotary encoders have one positive marker pulse per revolution.

1 = Positive pulse.

0 = Negative pulse.

FAGOR linear scales have a negative marker pulse (Io) every 50 mm (parameter=0) and FAGOR encoders supply one positive Io pulse per turn (parameter = 0).

P119, P319, P219, P419 Coordinate of Machine Reference Point for X,Z,3,4

They determine the distance from this point to the Machine Reference Zero.

Possible values: ± 8388.607 millimeters.
± 330.2599 inches.

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, P312, P212, P412 1st home searching feedrate for the X,Z,3rd,4th axes
P807, P808, P809, P810 2nd home searching feedrate for the X,Z,3rd,4th axes

They determine the feedrates used for the 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 65535 mm./minute.
1 thru 25800 inches/10 min. (= 2580 inches/min.)

When setting the 2nd feedrate to "0", the axis will move at 100 mm/min. (about 4 inches/min.).

P604(8) Home search on power-up

It determines whether it is mandatory or not to perform a home search on all the axes after powering the CNC up.

0 = No. It is **not** mandatory.
1 = Yes. It is mandatory.

Being this parameter set to "1", if the home search has not been carried out after powering the CNC up, the CNC will behave as follows:

* It allows jogging the axes by means of mechanical handwheels, electronic handwheels or by using the JOG keys.

* When attempting to execute an automatic operation or a “BEGIN [ENTER]” or “END [ENTER]” type command, the CNC will issue the corresponding error message.

P601(8) 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.

4.8 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.

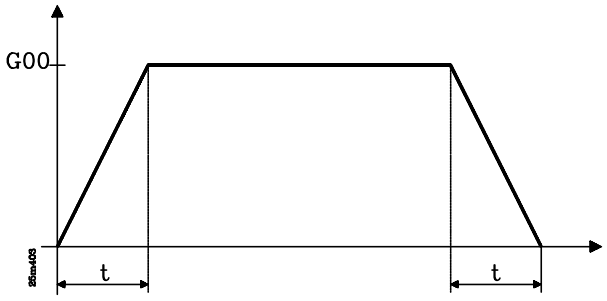
4.8.1 LINEAR ACCELERATION/DECELERATION

This type of acc./dec. is applied mainly on G00 and F00 moves although it is also possible to use it in G01 moves.

P712, P713, P714, P724 ACC./DEC. Control of the X, Z, 3rd, 4th 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.

P609(4) ACC./DEC. in all linear interpolations (G01).

It indicates whether the acc./dec. ramps (P712, P713, P714, P724) 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.

0 = Acc./Dec. applied only in linear interpolations at F0.
1 = Acc./Dec. applied in **all** linear interpolation (at any feedrate).

P616(6) 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.

4.8.2 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.

P621(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.

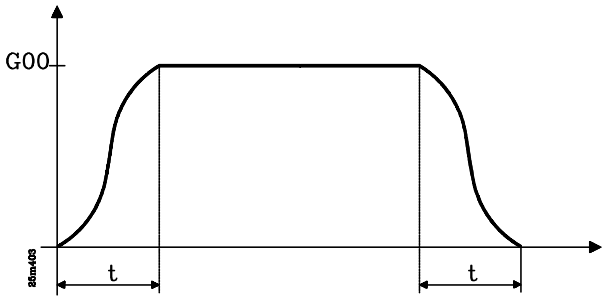
0 = No. This type of acc./dec. is not applied.
1 = Yes. This type of acc./dec. is applied.

Note that the acc./dec. ramps set by this parameter and P731 is common to all the axes.

P731 Duration of the Bell-shaped Acc./Dec. ramp

This parameter will be used when machine parameter "P621(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)

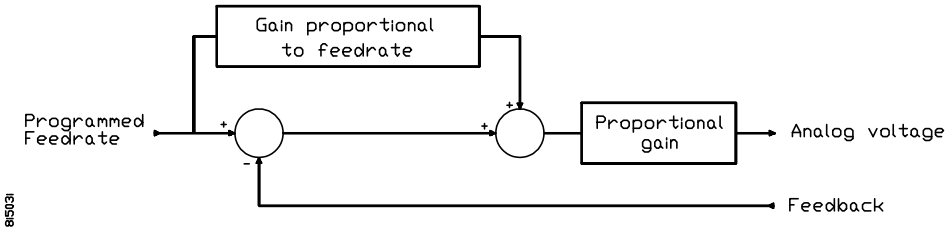
4.8.3 FEED-FORWARD GAIN

This gain may be used on both types of acc/dec.

P720, P721, P722, P723 FEED-FORWARD gain for X, Z, 3rd, 4th axes.

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 = P720"

* An eighth of the value assigned to this parameter in the case of bell-shaped acceleration/deceleration.
  For example, for the X axis: "Kf = P720/8"

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.

4.9 PARAMETERS FOR THE LIVE OR SYNCHRONIZED TOOL

The section on "Live/Synchronized Tool" in the chapter on "concepts" of this manual describes how these parameters may be used.

With this CNC, it is possible to control a live tool by means of function M45S. To do so, every time this function is programmed, indicate the spindle speed in rpm. for example, M45 S100, M45 S-125, etc.

The analog output corresponding to the live tool is provided by the CNC via pins 32 and 33 of connector I/O1. Also, the following machine parameters must be set:

P607(1) Sign of the analog output for the live or synchronized tool

It determines the sign of the analog output. If correct, leave it as is and change it if otherwise.

Possible values: 0 and 1

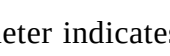
P802 Maximum programmable speed for the live or synchronized tool

It indicates the maximum programmable speed for the live or synchronized tool.

It is given in rpm by an integer between 0 and 9999.

A value of "0" assigned to this parameter will be interpreted as not having a live or synchronized tool.

P609(8) The turning speed of the live tool may be modified from the keyboard

This parameter indicates whether the spindle speed override keys  of the operator panel may also alter the turning speed of the live tool.

0 = No. The turning speed of the live tool **cannot** be modified.
1 = Yes. The turning speed of the live tool **can** be modified.

When setting this parameter to "1", the programmed live tool speed may be altered between 50% and 120% in steps of 5%.

Note that the CNC applies the selected % to both the spindle speed and the live tool.

When the machine has a synchronized tool, it is possible to govern it by means of function M45K. Every time this function is programmed, a K value must be given indicating the ratio between the speed of the live tool and that of the spindle. for example: M45K2.

The CNC uses the same analog output (pins 32 and 33 of connector I/O1) for the synchronized or for the live tool since only one of them can be used at a time.

Besides machine parameters P607(1) and P802 mentioned before, the following machine parameters must be set:

P803 Number of pulses (line count) of the encoder for the synchronized tool

It indicates the number of pulses per turn of the encoder for the synchronized tool.

It is given by an integer between 0 and 9999.

If set to "0", it will be interpreted by the CNC as not having a synchronized tool and that a live tool is being used if "P802 <> 0" (other than zero).

P607(2) Counting direction of the synchronized tool

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

Possible values: 0 and 1

It must be borne in mind that if this parameter is changed, it will also be necessary to change the parameter corresponding to the sign of the analog output "P607(1)".

P711 Proportional Gain K of the synchronized tool

It establishes the analog voltage corresponding to 1 micron of following error (lag).

It is given by an integer between 0 and 255. The value of 64 corresponds to an analog voltage of 2.5mV.

$$\text{Analog (mV)} = K1 \times \text{Following Error (microns)} \times \frac{2.5\text{mV.}}{64}$$

4.10 SPECIAL MACHINE PARAMETERS

P606(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 and P403 for axis resolution now acquire new units:

  1 = 0.01 mm or 0.001 inch resolution
  2 = 0.02 mm or 0.002 inch resolution
  5 = 0.05 mm or 0.005 inch resolution
  10 = 0.10 mm or 0.010 inch 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 P116, P216, P316, P416) must be given in (mV/0.01mm (mV/0.001 inches).

* Machine parameters P115, P215, P315 and P415 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 (leadscrew backlash) and P118, P218, P318, P418 (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).

* Machine parameters P112, P212, P312, P412, P807, P808, P809, P810 (homing feedrate) will also be expressed in 0.01mm/min. and 0.001 inches/min.

  P112 = 10000 Assigns a feedrate of 100m/min.

Examples to calculate resolution with P606(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:

$$\text{Number of pulses} = \frac{5 \text{ mm}}{4 \text{ x } 0.01\text{m}} = 125 \text{ pulses/rev.}$$

P103= 1 P602(3)=0 P106=N P602(6)=0

For a factor of x2:

$$\text{Number of pulses} = \frac{5\text{mm}}{2 \text{ x } 0.01 \text{ mm}} = 250 \text{ pulses/rev.}$$

P103= 1 P602(3)=0 P106=N P602(6)=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:

$$\text{Number of pulses} = \frac{0.25}{4 \text{ x } 0.001} = \textbf{62.5*} \text{ pulses/turn}$$

*** A gear reduction will be required to achieve this line count per turn.**

P103= 1 P602(3)=1 P106=N P602(6)=1

With a x2 factor:

$$\text{Number of pulses} = \frac{0.25}{2 \text{ x } 0.001} = 125 \text{ pulses/turn}$$

P103= 1 P602(3)=1 P106=N P602(6)=1

P609(7) 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, P303 and P403 for axis resolution now acquire new units:

  1 = 0.0001 mm or 0.00001 inch resolution
  2 = 0.0002 mm or 0.00002 inch resolution
  5 = 0.0005 mm or 0.00005 inch resolution
  10 = 0.0010 mm or 0.00010 inch resolution

* To calculate K1 and K2 and the Feed-forward gain, it must be borne in mind that 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 permissible is now: 3.20mm

  That is to say that K1 and K2 gains (parameters P114, P214, P314, P414 P116, P216, P316, P416) must be given in "mV/0.0001mm (mV/0.00001 inches).

* Machine parameters P115, P215, P315 and P415 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 (leadscrew backlash) and P118, P218, P318, P418 (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).

* Machine parameters P112, P212, P312, P412, P807, P808, P809, P810 (homing feedrate) will also be expressed in 0.0001mm/min. and 0.00001 inches/min.

* Machine parameters P119, P219, P319 and P419 indicate the value of the machine reference point and are also expressed in 0.0001mm and 0.00001 inch units.

* Machine parameters P112, P212, P312, P412, P807, P808, P809, P810 (homing feedrate) will also be expressed in 0.01mm/min. and 0.001 inches/min.

P112 = 10000 Assigns a feedrate of 100m/min.

Examples to calculate resolution with P609(7)=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:

$$\text{Number of pulses} = \frac{5\text{ mm}}{4 \text{ x } 0.0001\text{m}} = 12500 \text{ pulses/rev.}$$

P103= 1 P602(3)=0 P106=N P602(6)=0

For a factor of x2:

$$\text{Number of pulses} = \frac{5\text{mm}}{2 \text{ x } 0.0001 \text{ mm}} = 25000 \text{ pulses/rev.}$$

P103= 1 P602(3)=0 P106=N P602(6)=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:

$$\text{Number of pulses} = \frac{0.25}{4 \text{ x } 0.001} = 6250 \text{ pulses/turn}$$

P103= 1 P602(3)=1 P106=N P602(6)=1

With a x2 factor:

$$\text{Number of pulses} = \frac{0.25}{2 \text{ x } 0.00001} = 12500 \text{ pulses/turn}$$

P103= 1 P602(3)=1 P106=N P602(6)=1

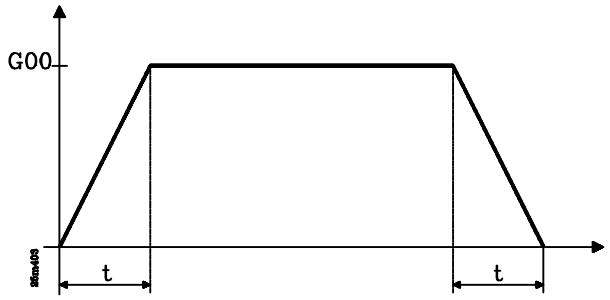
5. *SPINDLE MACHINE PARAMETERS*

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

P811 Acceleration/Deceleration control of the spindle

In order to avoid abrupt start-ups and brakes of the machine, it is possible to define some acceleration and deceleration ramps.

This parameter defines the time that the spindle needs to reach the indicated speed "S" while accelerating. This acceleration time will be the same as the deceleration time.



It is given by an integer between 0 and 65,535.

Value of 0 = There is no Acceleration/deceleration control.
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 MACHINE PARAMETERS FOR 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(1) The machine has an automatic spindle range changer.

Indicates whether the CNC must automatically generate the M function corresponding to the spindle range change (M41, M42, M43, M44) when a new speed has been selected or not.

M41 for 1st range
M42 for 2nd range
M43 for 3rd range
M44 for 4th range

Possible values:

0 = The machine does **not** have an automatic spindle range changer.
1 = The machine **has** an automatic spindle range changer.

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.

P701 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)

P702 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 MACHINE 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”.

P607(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  and a negative analog voltage (0 to -10V) to turn the spindle counter-clockwise  .

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.

5.3 MACHINE 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	S39	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 MACHINE PARAMETERS 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)

When working with spindle orientation (M19), 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 pulses of the spindle encoder.

It indicates the number of pulses per revolution (line count) of the rotary encoder for the spindle.

It is given by an integer between 0 and 9999.

If this parameter is set to "0", the CNC will assume that the spindle has no encoder.

P603(8) Active monitoring of actual spindle speed.

Besides simply displaying the real spindle speed, it is also possible to have a certain control over it as follows:

* When the real spindle speed is less than 50% of the programmed S speed, the CNC generates an internal Feed-Hold in order to provide more time for the spindle to reach that speed.

* When the real spindle speed is greater than 150% of the programmed S speed, the CNC activates the emergency output and it issues the corresponding error code.

0 = Active spindle speed monitoring ON.
1 = Active spindle speed monitoring OFF.

P704 Stabilizing time for spindle speed.

This parameter is used when having the active spindle speed monitoring ON, "P603(8) = 0" and it indicates the time allowed for the spindle to reach the programmed speed.

It is given by an integer between 1 and 255.

Value of	1	=	0.1 sec.
Value of	10	=	1.0 sec. (10 x 0,1)
Value of	255	=	25.5 sec. (255 x 0,1)

During this time the CNC does not actively monitor the actual (real) spindle speed but it displays it.

5.4.1 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.

P706 Spindle speed when working in M19

It is given in rpm with a value between 0 and 255.

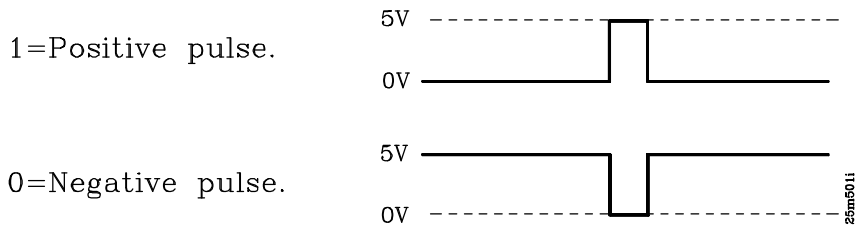
P606(2) Sign of the analog S output associated with M19

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”.

P600(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.



P709 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)

P707 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 P707 = 100, the in-position zone (dead band) will be:

$$\frac{360°}{1000 \times 4} \times 100 = \pm 9°$$

P708 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.

$$\text{Analog (mV.)} = K \times \text{Following Error (pulses)} \times \frac{2.5\text{mV.}}{64}$$

6. CONCEPTS

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, Z Main movements on the main work plane of the machine.

3rd, 4th Auxiliary axes which will be called W and Y depending on the values assigned to "P613(4)" and "P615(4)".

S Spindle.

All of them must be properly defined by means of their corresponding machine parameters.

With this CNC it is possible to control the following types of machines:

Without Synchronized Tool					
A1	A2	A3	A4	A5	A6
X	-	Z	-	S	Handwheel
X	-	Z	3rd	S	Handwheel
X	-	Z	C	S	Handwheel
X	4th	Z	3rd	S	Handwheel
X	4th	Z	C	S	Handwheel

With Synchronized Tool					
A1	A2	A3	A4	A5	A6
X	Sync. Tool	Z	-	S	Handwheel
X	Sync. Tool	Z	3rd	S	Handwheel
X	Sync. Tool	Z	C	S	Handwheel
X	4th	Z	3rd	S	Handwheel Sync. Tool
X	4th	Z	C	S	Handwheel Sync. Tool

6.2 FEEDBACK SYSTEMS

The feedback inputs of this CNC are:

A1 Used to connect the X axis feedback signals. It is a SUB-D type 15-pin female connector and it admits sine-wave signals.

A2 Used to connect the feedback signals from the 4th axis or the live/synchronized tool. It is a SUB-D type 15-pin female connector and it admits sine-wave signals.

When having a 4th axis, set machine parameter "P614(1)=1" and use feedback input A6 for the live tool or the synchronized tool.

When not having a 4th axis, set machine parameter "P614(1)=0".

When using a live tool, set "P802 <> 0" (other than "0") and when using a synchronized tool, set both P802 and P803 to a value other than "0".

A3 Used to connect the Z axis feedback signals. It is a SUB-D type 15-pin female connector and it admits sine-wave signals.

A4 Used to connect the feedback signals from the 3rd or "C" axis. It is a SUB-D type 15-pin female connector and it admits sine-wave signals.

When the machine has a 3rd axis, set machine parameter "P612(1)=1"; but if the 3rd axis is the "C" axis, set **also** "P613(5)=1".

A5 Used to connect the spindle encoder feedback signals. It is a SUB-D type 15-pin female connector and it admits differential (double ended) square-wave signals.

A6 Used to connect the feedback signals from the electronic handwheel and the live or synchronized tool (when having the 4th axis). It is a SUB-D type 9-pin female connector and it admits non-differential (single ended) square-wave signals.

When using this input for the live or synchronized tool, set machine parameter "P616(8)=1". Also, when having a live tool, set machine parameter P802 <> 0; and when having a synchronized tool, set both P802 and P803 to a value other than "0" (<>0).

The type of cable used on all of them must have overall shield. The rest of their characteristics, such as length, will depend on the feedback type and model being used.

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

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”) 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 1µm 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”) 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

They set the counting resolution for each axis.

P602(3), P612(2), P602(2), P614(1)

They set the measuring units for each axis feedback signal (mm or inches).

P106, P206, P306, P406

They set the type of feedback signal being used (square-wave or sine-wave) for each axis.

P602(6), P612(5), P602(5), P614(5)

They indicate the multiplying factor, x2 or x4 to be applied to the feedback signals of each axis.

P619(1), P619(2), P619(3), P619(4)

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 P602(3)=0 P106=N P602(6)=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 P602(3)=0 P106=N P602(6)=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:

P619(1)=0		
P602(6)	**Resolution**	**P103**
=1 (x2)	2 microns	10
=0 (x4)	2 microns	10

P619(1)=1		
P602(6)	**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 P602(6)=1 (x2)

$$\text{Nº pulses} = \frac{5000\mu\text{m/turn}}{5 \times 2 \times 2\mu\text{m/pulse}} = 250 \text{ pulses/turn}$$

Therefore:

If P619(1)=0 => P602(3)=0 P106=Y P602(6)=1 P103= 10
If P619(1)=1 => P602(3)=0 P106=Y P602(6)=1 P103= 2

For P602(6)=0 (x4)

$$\text{Nº pulses} = \frac{5000\mu\text{m/turn}}{5 \times 4 \times 2\mu\text{m/pulse}} = 125 \text{ pulses/turn}$$

Therefore:

If P619(1)=0 => P602(3)=0 P106=Y P602(6)=0 P103= 5
If P619(1)=1 => P602(3)=0 P106=Y P602(6)=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	P602(3)	P106	P602(6)
20µm	5	0	N	0
20µm	10	0	N	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:

P619(1)=0		
P602(6)	**Resolution**	**P103**
=0 (x4)	1 micron	5

P619(1)=1		
P602(6)	**Resolution**	**P103**
=0 (x4)	1 micron	1

Therefore:

If P619(1)=0 => P602(3)=0 P106=Y P602(6)=0 P103= 5
If P619(1)=1 => P602(3)=0 P106=Y P602(6)=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=0 P604(8)=0

With a x4 factor:

$$\text{Number of pulses} = \frac{0.25}{2 \times 0.0001} = 1250 \text{ pulses/turn}$$

P103= 1 P602(3)=1 P106=N P602(6)=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:

P619(1)=0		
P602(6)	**Resolution**	**P103**
=1 (x2)	0.0001 inch	5
=0 (x4)	0.0001 inch	5

P619(1)=1		
P602(6)	**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:

$$\text{Nº of pulses} = \frac{\text{Leadscrew pitch}}{5 \times \text{Multiplying factor} \times \text{Resolution}}$$

For P602(6)=1 (x2)

$$\text{Nº pulses} = \frac{0.25 \text{ inch/turn}}{5 \times 2 \times 0.0001 \text{ inch/pulse}} = 250 \text{ pulses/turn}$$

Therefore:

If P619(1)=0 => P602(3)=0 P106=Y P602(6)=1 P103= 5
If P619(1)=1 => P602(3)=0 P106=Y P602(6)=1 P103= 1

For P604(8)=0 (x4)

$$\text{Nº pulses} = \frac{0.25 \text{ inch/turn}}{5 \times 4 \times 0.0001 \text{ inch/pulse}} = 125 \text{ pulses/turn}$$

Therefore:

If P619(1)=0 => P602(3)=0 P106=Y P602(6)=0 P103= 5
If P619(1)=1 => P602(3)=0 P106=Y P602(6)=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:

$$\text{Max. Feed} = \frac{25{,}000 \text{ pul./sec.}}{125 \text{ pul./rev.}} \times 0.25 \text{ inch/rev.} = 50 \text{ inch/sec.} = 3000 \text{ inch/min.}$$

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 3rd axis.

Since the CNC applies a multiplying factor of x4 to obtain this resolution, the encoder must have the following line count (pulses):

$$\text{Nº of pulses} = \frac{\text{Degrees}}{\text{Mult. factor x Resolution}}$$

$$\text{Nº of pulses} = \frac{360°/\text{turn}}{4 \text{ x } 0.005°/\text{pulse}} = 18000 \text{ pulses/turn}$$

P203=5 P612(2)=0 P206=N P612(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:

P619(4)=1		
P612(5)	**Resolution**	**P203**
=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 and P405 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, and P404 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 and P400.

* 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 and P401) will have to be changed as well as that corresponding to the SIGN OF THE ANALOG VOLTAGE (P100, P200, P300 and P400).

* 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 and P402).

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 volt-meter.

* 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:

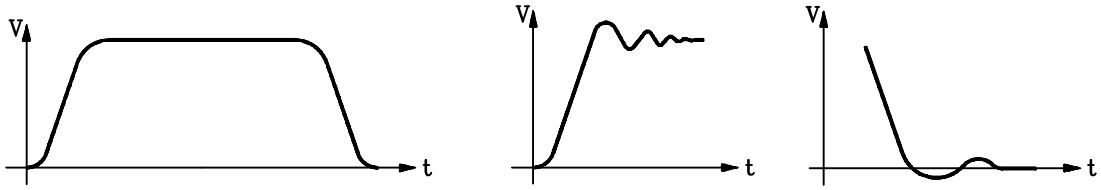
```
N10 G00 G90 X200
N20 X-200
N30 G25 N10
```

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

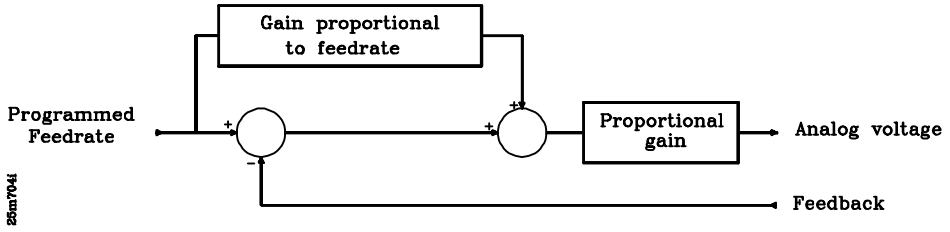
Defined by parameters: P115, P215, P315, P415, P515.

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.

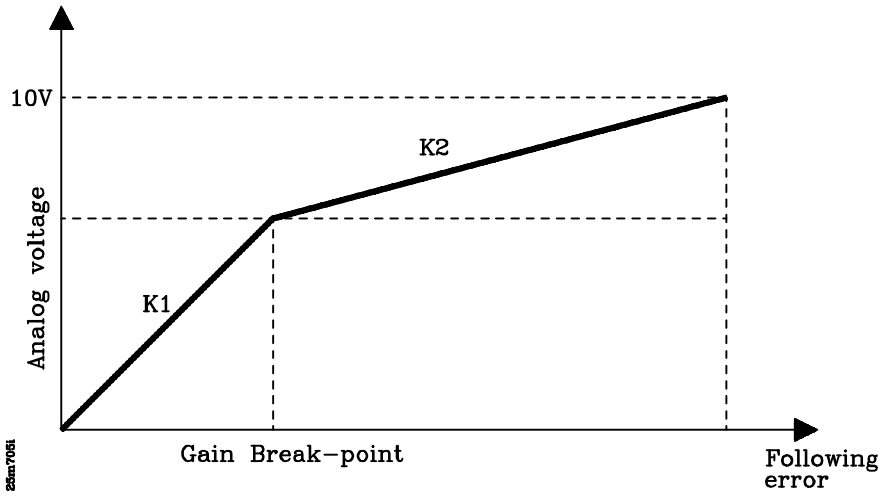
Analog output = Proportional gain “K” x Following Error

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 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

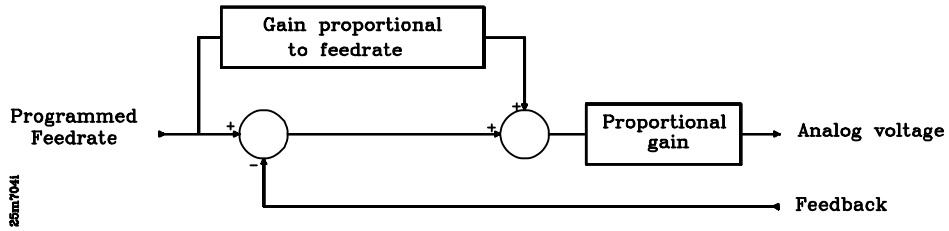
It is mainly applied onto G00 and F00 moves, although it may also be used in G01 moves.

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 P720, P722, P721, P723 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 2 leadscrew compensation tables with 30 points each: one for the X axis (P0 through P59) and another one for the Z axis P60 through P119).

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:"
* Key in "**PKAI1**" and press **[ENTER]** to lock the access or key in "**PKAI0**" 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 these leadscrew error compensation 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] [A] [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 and 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.

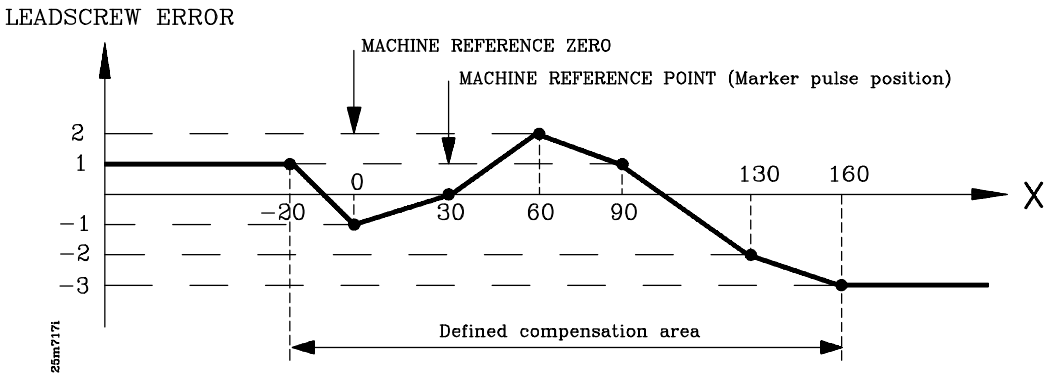
To EDIT a parameter, key in its number, press **[=]**, key in the desired value. Or, jog the corresponding axis to the desired value and press **[ENTER].**

Then, 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

6.5 REFERENCE SYSTEMS

6.5.1 REFERENCE POINTS

A CNC machine must have the following reference points established:

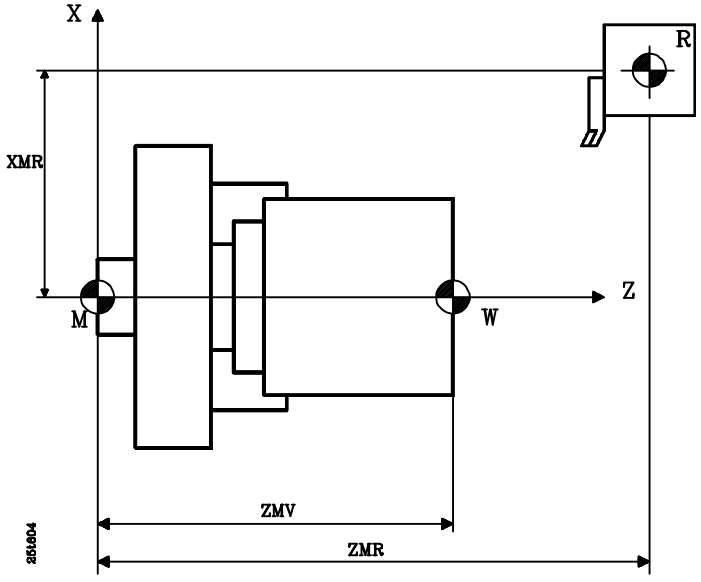
* **Machine Reference Zero** or origin point of the machine. It is set by the machine manufacturer as the origin of the coordinate system of the machine.

* **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 **uses 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 **does not 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, ZMW, etc.	Part Zero coordinates
XMR, 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 P618(8), P618(7), P618(6), P618(5)

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 until its corresponding home switch is pressed: P600(5), P612(6), P600(4), P614(6).

Then, it will look for the marker pulse (Io) at the feedrate indicated by machine parameters P807, P809, P808, P810 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 P807, P809, P808, P810 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, P319, P219, P419.

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 SYSTEMS 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:

* 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(7), P612(7), P600(6) and P614(7).

* Set the corresponding machine parameter to indicate the direction of the home search. Parameter P618 bits 8, 6, 7, 5.

* Set the machine parameters that determine the home searching feedrate until the home switch is pressed (P112, P212, P312, P412) and the one until the marker pulse is detected (P807, P809, P808, P810).

* The machine reference point will be assigned a value of "0". Parameter P119, P219, P319, P419.

* 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.

  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 P618(8), P618(6), P618(7), P618(5)] until releasing the home switch before starting the actual home search.

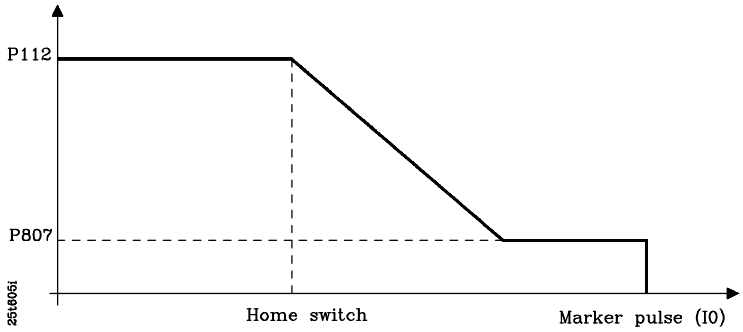
* If the axis is out of the soft travel limits (set by P107-P108, P207-P208, P307-P308, P407-P408), it has to be jogged into the work area (within limits) and, then, positioned at the correct side from home before starting the actual home search.

* Care must be taken when placing the home switch and when setting the home searching feedrates (P112, P212, P312, P412, P807, P809, P808, P810) to prevent any overshooting.

* If the selected axis does not have a home switch [P600(5), P612(6), P600(4), P614(6)], the CNC will consider it to be pressed and it will only make the marker pulse searching move at the feedrate set by P807, P809, P808, P810 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(7), P612(7), P600(6), P614(7)" = 0] and FAGOR rotary encoders output a positive marker pulse (Io) ["P600(7), P612(7), P600(6), P614(7)" = 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 P807, P809, P808, P810).



If there is no room for that, the first home searching feedrate (set by P112, P212, P312, P412) 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(7), P612(7), P600(6) and P614(7).

* Indicate, also, the homing direction for each axis. Parameters P618(8), P618(6), P618(7), P618(5).

* Indicate the homing feedrate for each axis. Parameters P807, P809, P808, P810..

* Set the machine parameters for amount of OFFSET to "0". Parameters P908, P909, P910, P911.

* 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 P908 = 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), 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), P612(7), P600(6), P614(7)" = 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.

* 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.

* 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 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.

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 P606(7) determines whether the CNC also outputs their corresponding BCD 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 shape and dimensions of each tool must be defined in order to obtain a better part finish.

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.

6.6.1 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:"
* Key in "**PKAI1**" and press **[ENTER]** to lock the access or key in "**PKAI0**" 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 decoded "M" function 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 determines the way the M function is transferred to the electrical cabinet. 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 M41 is executed.

	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	

6.6.2 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.

6.6.3 M, S, T FUNCTION TRANSFER USING THE M-DONE SIGNAL

When parameter P602(7) is set to “0”, the CNC maintains the BCD outputs and the corresponding Strobe signal (M, S, T) active for 100 milliseconds.

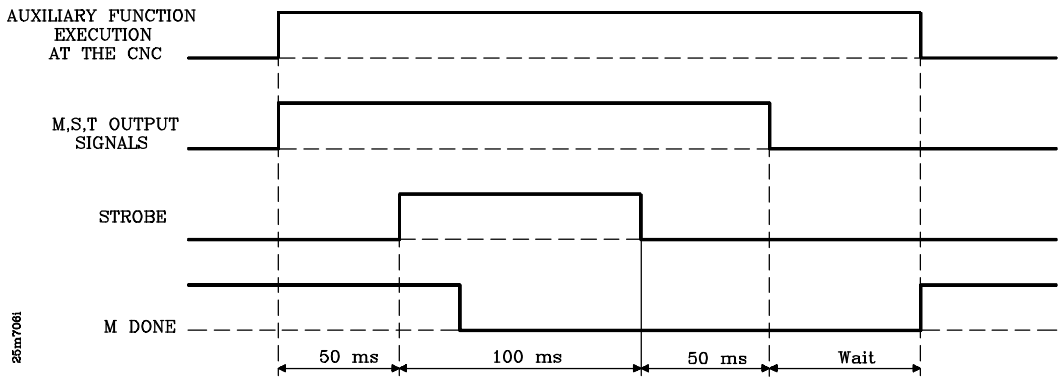
When the electrical cabinet has a device requiring the BCD signals to be active for a longer time, machine parameter P602(7) must be set to “1”.

In each case, the CNC acts as follows:

“P602(7)=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.

3.- The CNC will maintain the “M Strobe” signal for 100 milliseconds and the BCD signals for another 50 milliseconds.

After this time period, it will wait for the M-DONE signal provided by the electrical cabinet indicating to the CNC that the execution of the “M” function is completed.

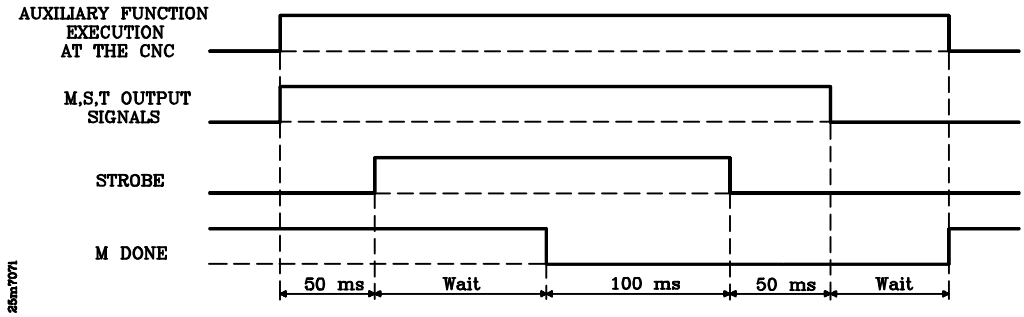
If the M-done signal was not deactivated in point 2, the CNC will consider the M transfer completed after the BCD signals disappear (there is no wait).

“P602(7)=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.

After this time period, it will wait for the electrical cabinet to reactivate the M-DONE input “telling” the CNC that the processing of the required function has concluded.

Atention:

An "M" function may be set to wait for the M-done signal to go high and back low.

To do this, set bit 17 of the bottom row of the corresponding decoded "M" function to "1" (bit 17 = 1).

When the CNC executes this "M" function, it will ignore the setting of machine parameter P602(7) and it will execute it as described above for "P602(7) =1".

The CNC takes into account the setting of P602(7) in the following cases:

* When executing "S" functions.
* When executing "T" functions.
* When executing "M" functions not defined in the decoded "M" table.
* When executing "M" functions defined in the decoded "M" table with its 17th bit of the bottom row set to "0" (bit17 = 0).

6.7 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 P607(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 output	P601(3)=1 and P601(2)=0
For 4-digit BCD output	P601(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:

S Programad	S BCD	S Programad	S BCD	S Programad	S BCD	S Programad	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	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

6.8 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.

To be able to automatically change the spindle speed range, machine parameter P601(1) must be set to “1”.

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).

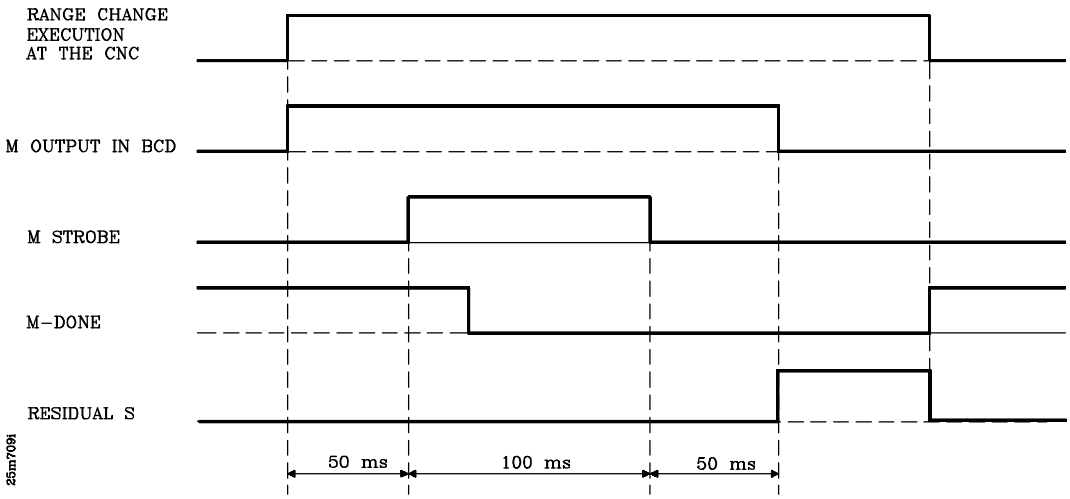
Also, in order to facilitate the range change, the CNC offers the possibility to use a residual analog output during a range change. Machine parameter for the spindle: P601(6).

The value of this residual analog voltage is defined by machine parameter P701 and the oscillation period for this residual analog voltage is set by machine parameter P702.

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 P701 if so established by parameter P601(6).

The oscillation period for this residual analog voltage is determined by machine parameter P702.

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 P607(2) must be set “1” (the CNC waits for the down flank of the M-Done signal).

6.9 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).

The machine parameters to be set are:

P800	Number of pulses (line count) of the spindle encoder.
P606(3)	Counting direction of the spindle.

Also, to work with "spindle orientation (M19)", the following parameters must be set:

P706	Spindle speed "S" when working in M19.
P606(2)	Sign of the analog output associated with M19.
P600(8)	Type of spindle encoder reference (marker) pulse.
P709	Minimum spindle analog output in M19.
P707	In position zone for the spindle in M19.
P708	Spindle proportional gain in M19.

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 P706. Then, it carries out a home search.

If "M19 S" is 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 P706.

The spindle orienting direction is determined by machine parameter P606(2). When programming function M19 by itself (without indicating a specific position), the spindle will orient to the position set by machine parameter P916.
Example:

M3 S1000	Spindle in open loop turning clockwise.
M19	Spindle in closed loop, home search
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°.

6.10 TOOLS

The machine parameters related to tools are the following:

P700	Number of tools
P730	Subroutine associated with the T function
P604(5)	Tool offset values active after M06 is executed
P609(3)	The CNC ignores the tool dimensions

It is advised to assign the highest possible tool number to "P701" in order to avoid errors during execution.

To change a tool, execute the command T2.2

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 P730), it will be executed after the T function has been transferred to the electrical cabinet.

Usually, the CNC assumes the new tool offset values right after T2.2 is executed.

However, if the machine must perform some moves before changing the tool, it might be necessary to keep the current tool offset values until the tool change is completed.

In that case, it is recommended to set “P604(5)=1” and program an M06 right after the tool change in the subroutine associated with the T function.

6.11 LIVE/SYNCHRONIZED TOOL

The machine parameters related to the live or synchronized tool are the following:

P607(1)	Sign of the analog output for the live or synchronized tool
P802	Maximum programmable speed for the live or synchronized tool
P609(8)	The speed of the live or synchronized tool may be changed from the operator panel
P803	Synchronized tool encoder line count
P607(2)	Counting direction of the synchronized tool
P711	Proportional gain K of the synchronized tool

6.11.1 LIVE TOOL

When using a live tool, machine parameters “P607(1)” and “P802” must be set.

With function M45 S it is possible to start or stop the live tool.

M45 Is a special auxiliary function to indicate to the CNC that either the status or the speed of the live tool is to be modified.

S±4 Indicates the live tool's turning direction (±) and speed in rpm.

Possible values: Within ±P802 rpm

To stop the live tool, just program "M45" or "M45 S0".

Programming example supposing that the live tool is stopped:

M45 S100	Start turning the live tool at 100 rpm
M45 S200	Keep its turning direction but do it at 200 rpm..
M45 S0	Stop the live tool
M45 S-50	Start the live tool turning at 50 rpm in the opposite direction
M45	Stop the live tool

If machine parameter “P609(8)=1”, the CNC acts as follows:

Every time one of the spindle speed override keys of the operator panel  is pressed, the CNC will also change the speed of the live tool.

6.11.2 SYNCHRONIZED TOOL

With this CNC it is possible to synchronize the rotation of the tool with that of the spindle. This requires both devices to have an encoder.

P800	Spindle encoder line count
P606(3)	Spindle counting direction
P803	Synchronized tool encoder line count
P607(2)	Counting direction of the synchronized tool

The following parameters must also be set:

P607(1)	Sign of the analog output for the synchronized tool
P711	Proportional gain K for the synchronized tool
P802	Maximum turning speed for the synchronized tool

With function M45 K it is possible to start or stop the synchronized tool

M45 Is a special auxiliary function to indicate to the CNC that either the status or the speed of the synchronized tool is to be modified.

K±3.4 Indicates the turning direction (±) and the ratio between the speed of the synchronized tool and that of the spindle.

Possible values: Within ±655.3509

For example a K2 value means that the synchronized tool rotates twice as fast as the spindle.

If the resulting speed is greater than the maximum set by machine parameter "P802", the CNC interrupts the execution issuing error 71.

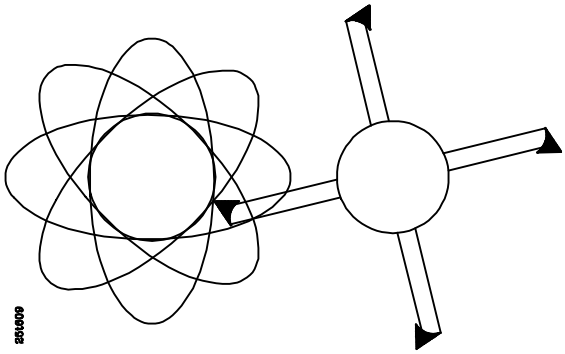
It must be borne in mind that the CNC permits assigning to K the value of an arithmetic parameter. For example, to indicate that the speed of the synchronized tool is one third of the spindle speed, the following may be programmed:

```
P1=K1F4K3     ;Assigns to P1 the value of 1/3
M45 KP1       ;Synchronizes the tool with the spindle at a 1/3 ratio
```

To stop the synchronized tool, program "M45" alone.

6.11.2.1 APPLICATION EXAMPLES FOR THE SYNCHRONIZED TOOL

***** Using a tool with several cutters to machine polygons on a part.



The formula to be used to calculate the K factor is the following:

$$\text{K factor} = \frac{\text{Number of sides of the polygon}}{\text{Number of cutters}}$$

Example: Having a 4-cutter tool, the factor to be applied in each case is:

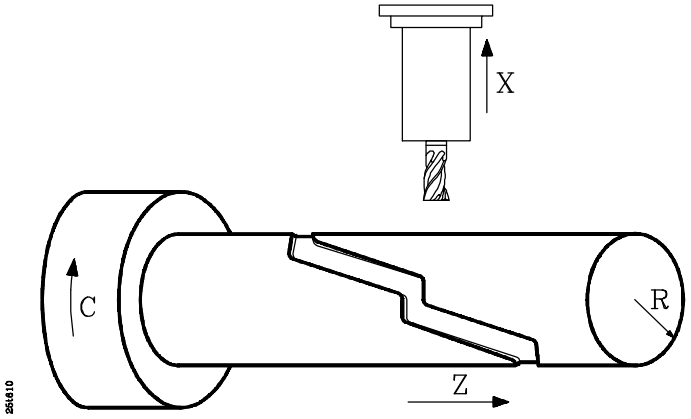
To machine a square	K= 4/4 = 1
To machine a hexagon	K= 6/4 = 1.5
To machine an octagon	K= 8/4 = 2

***** Synchronizing both spindles of a dual-spindle lathe

With this feature it is possible, for example, to transfer a part from one spindle to another.

6.12 "C" AXIS

With this CNC it is possible to interpolate the spindle with the X and Z axis in order to machine the cylindrical surface as well as the face of the part (cylinder).



To use this feature, it is necessary to install a rotary encoder on the spindle and to set the 3rd axis as "C" axis.

The "C" aixs is rotary and will be expressed in degrees. thus, the following machine parameters must be set accordingly:

P612(1)=1	The machine has a 3rd axis.
P613(5)=1	The 3rd axis is the "C" axis.
P612(2)=0	The measuring units are degrees.

6.12.1 ADJUSTMENT OF THE "C" AXIS

* The following machine parameters must also be set:

P203	Counting resolution of the "C" axis.
P619(3)	C axis counting resolution with sine-wave feedback signal.
P206	Type of feedback signal for the "C" axis.
P612(5)	Multiplying factor for the "C" axis feedback signals.

* It is recommended to use the following sections: "*Axis resolution*", "*Adjustment of the axes*" and "*Reference systems*" of this chapter in order to adjust the "C" axis.

  The "C" axis feedrate must be programmed in degrees/minute (P210, P212, P809, etc.). That is, the maximum programmable feedrate will be equivalent to:

$$65535 \text{ degrees/minute} = \frac{65535}{360} = 182 \text{ rpm}$$

* Then, execute function G14 in "Teach-in" mode. The CNC will home the spindle and activate the "C" axis.

The home search is carried out as described in the section on "*Reference systems*" of this very chapter and the machine parameters involved are:

P219	"C" axis home position value (coordinate)
P618(6)	"C" axis home searching direction
P612(7)	Type of "C" axis reference pulse (marker)
P612(6)	"C" axis home switch
P212, P809	"C" axis homing feedrates

It could happen that the spindle does not stop and keeps turning without finding home. This may happen if:

- The CNC does not receive the Io signal (marker pulse) . Check the rotary encoder, the extension cables, etc.

- The spindle is not properly adjusted and it is turning slower than 50% or faster than 150% of the parameter-set speed (P809).

  Check the actual spindle rpm.

  For example: if P809=10,000 degreees/minute, the corresponding rpm will be 10000/360=27,777 rpm. The actual spindle speed must be within: 13,888 rpm and 41,666 rpm.

On the other hand, if the home search is completed as soon as function G14 is executed, machine parameter "P612(7)" (type of "C" axis marker pulse) must be changed.

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 4 axes + spindle encoder + synchronized tool + 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.

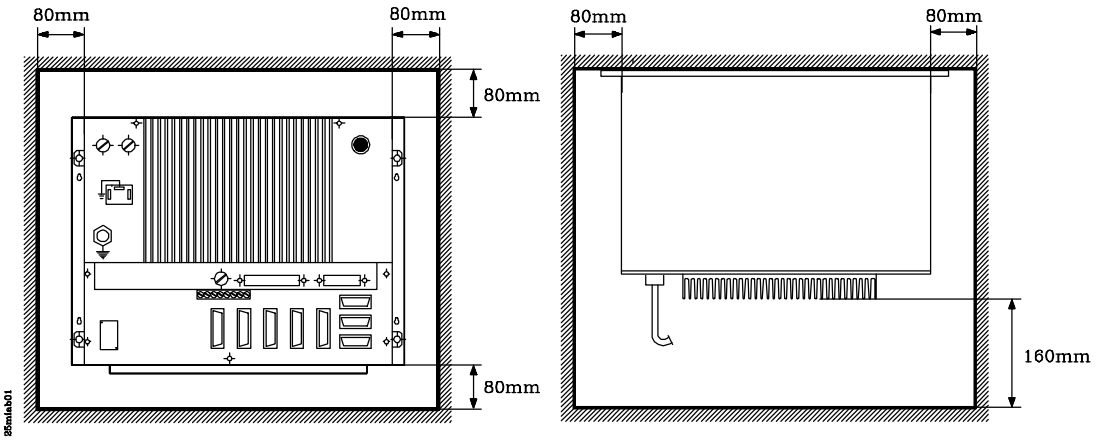
Atention:

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

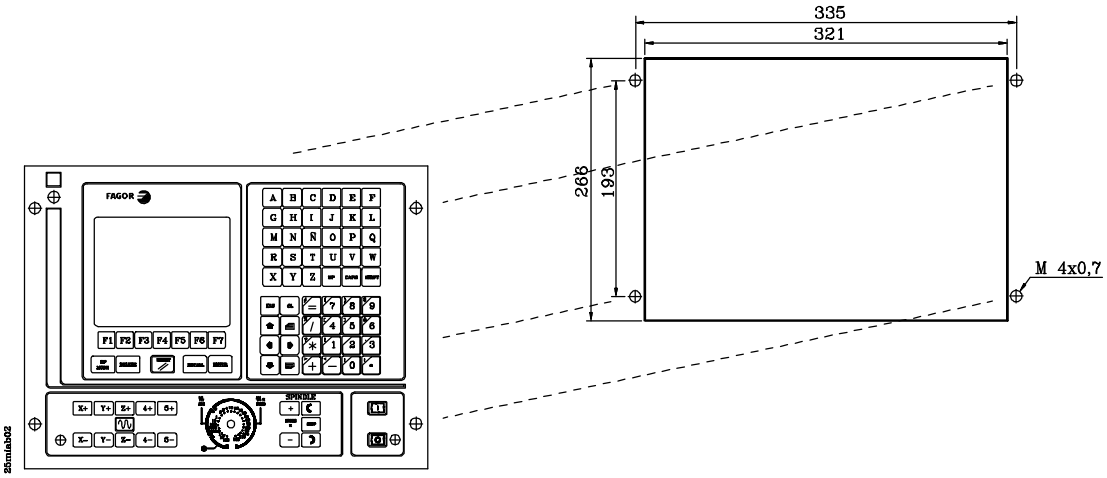
ENCLOSURES

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

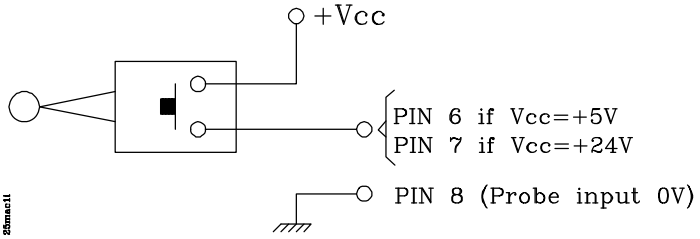
RECOMMENDED PROBE CONNECTION DIAGRAMS

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.

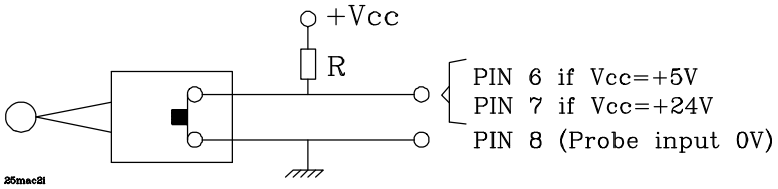
Depending on the type of connection used, machine parameter "P612(7)" must be set to indicate whether it is activated with an up-flank (leading edge) or down flank (trailing edge) of the probe signal.

DIRECT CONNECTION

- **Probe output as normally open contact:**



- **Probe output as normally closed contact:**

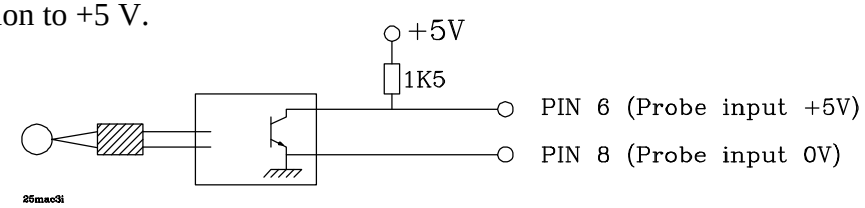


If Vcc=+5V R=10K
If Vcc=+24V R=50K

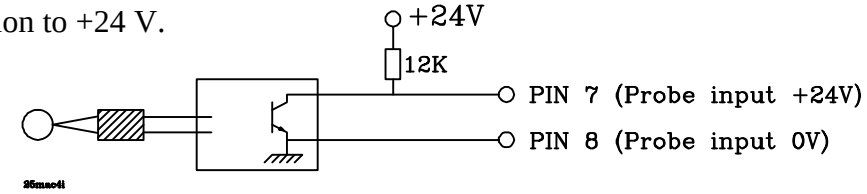
CONNECTION VIA INTERFACE

- **Interface with open-collector output:**

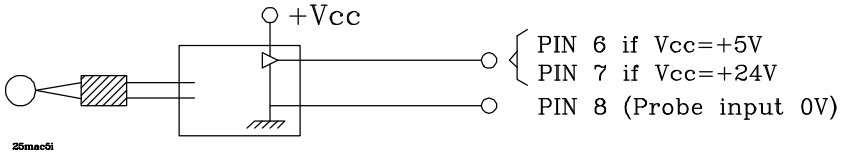
Connection to +5 V.



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	3rd axis Home switch
12	I/O 1	Z axis Home switch
13	I/O 1	4th 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
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	Threading on - Cycle ON
7	I/O 1	Z axis enable
8	I/O 1	Reset
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	Analog output for synchronized tool
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
13	I/O 2	Decoded M11 output - 4th axis enable
14, 15	I/O 2	4th axis analog output
17, 18	I/O 2	3rd axis analog output
21	I/O 2	"Jog mode selected" output
22	I/O 2	Decoded M15 output - Spindle lock - "C" axis enable
23	I/O 2	Decoded M14 output - G00
24	I/O 2	Decoded M13 output - Turret rotation
25	I/O 2	Decoded M12 output - 3rd axis enable

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.
P11	X axis in radius (0) or diameter (1)
P600(1)	Machine axes orientation
P801	Protected program

MACHINE PARAMETERS FOR AXIS CONFIGURATION

Section 3.3.1

P612(1), P614(1)	The machine has a 3rd, 4th axis. 0= No, 1= Yes.
P613(5)	The 3rd axis is the "C" axis. 0= No, 1= Yes.
P613(4), P615(4)	The 3rd, 4th axis is called Y/W. 0= Y, 1=W.
P613(3), P615(3)	The 3rd, 4th axis as DRO axis. 0= No, 1= Yes.
P613(1), P615(1)	The 3rd, 4th axis is rotary. 0= No, 1= Yes.
P613(2), P615(2)	The 3rd, 4th axis is rotary rollover. 0= No, 1= Yes.
P613(6), P615(5)	3rd, 4th axis rotary rollover via shortest path. 0= No, 1= Yes.
P616(8)	Connector A6 shared by handwheel and synchronized tool . 0= No, 1= Yes.

I/O RELATED MACHINE PARAMETERS

Section 3.3.2

P604(4)	Normal status of Emergency output (pin 5 connector I/O1). 0= 0V, 1=24V.
P609(6)	Pin 24 of connector I/O2 as turret rotating direction. 0= No, 1= Yes.
P604(3)	Pin 23 of connector I/O2 as G00. 0= No, 1= Yes.
P605(4)	Pin 6 of connector I/O1 as Threading ON (0) or Cycle ON (1)
P606(7)	Decoded M functions also output in BCD code. 1= No, 0= Yes.
P602(7)	CNC waits for down-flank at M-done signal input . 0= No, 1= Yes.
P603(4,3,2,1), P608(1)	A1, A2, A3, A4, A5 feedback alarm cancellation. 0= No, 1= Yes.

HANDWHEEL MACHINE PARAMETERS

Section 3.3.3

P609(1)	Electronic handwheel model FAGOR 100P. 0= No, 1= Yes.
P500	Counting direction of the electronic handwheel
P602(1)	Feedback units of the electronic handwheel. 0= mm, 1= inch.
P501	Counting resolution of the electronic handwheel
P602(4)	Multiplying factor for electronic handwheel feedback signals. 0= x4, 1= x2.
P619(7)	Handwheel managed by the PLC. 0= No, 1= Yes.

PROBE RELATED MACHINE PARAMETERS

Section 3.3.4

P606(6)	Type of probe pulse. 0= 0V, 1= 5V or 24V.
P710	M function associated with probing (G75)
P806	Probing feedrate in JOG mode
P902	Minimum X coordinate of the probe
P903	Maximum X coordinate of the probe
P904	Minimum Z coordinate of the probe
P905	Maximum Z coordinate of the probe

TOOL RELATED MACHINE PARAMETERS *Section 3.3.5*

P700	Number of tools (0..32)
P730	Subroutine associated with the T function
P617(2)	Associated subroutine executed before (1) or after (0) the T function
P604(5)	Tool offset value applied after executing T2.2 (0) or after executing M06 (1)
P609(3)	The CNC displays the tool tip position (0) or that of the tool base (1).

EMERGENCY SUBROUTINE RELATED MACHINE PARAMETERS *Section 3.3.6*

P716	Emergency subroutine
P616(2)	The emergency subroutine executes an M00 (interrupting the program). 1= No, 0= Yes.
P616(1)	Assignment of position values to arithmetic parameters in emergency subroutine. (0= beginning, 1= current).

MACHINE PARAMETERS RELATED WITH RS232C SERIAL LINE *Section 3.3.7*

P0	Communication speed (baudrate). 110, 150, 300, 600, 1200, 2400, 4800, 9600.
P1	Number of data bits per character (7/8)
P2	Parity. 0= No, 1= Odd, 2= Even.
P3	Stop bits. (1/2)
P605(5)	DNC active. 0= No, 1= Yes.
P605(6)	Communication with FAGOR cassette (0) or Floppy Disk Unit (1)
P605(7)	DNC Protocol active on power-up. 0= No, 1= Yes.
P605(8)	The CNC aborts DNC communications (program debugging). 1= No, 0= Yes.
P606(8)	Status report by interruption. 0= No, 1= Yes.

DISPLAY RELATED MACHINE PARAMETERS *Section 3.3.8*

P6	Theoretical (1) or Real (0) (actual) display
P606(4), P606(5)	Axis orientation in graphics representation
P612(8), P614(8)	The 3rd, 4th axis display. 1= No, 0= Yes.
P611(7), P611(8)	Monitor display color combination

JOG MODE RELATED MACHINE PARAMETERS *Section 3.3.9*

P12	Continuous (N) or pulsating (Y) axis jog
P600(2)	Invert the JOG keys for X and Z. 0= No, 1= Yes.
P603(5)	The S function can be executed in JOG mode. 1= No, 0= Yes.
P603(6)	The T function can be executed in JOG mode. 1= No, 0= Yes.
P603(7)	The M function can be executed in JOG mode. 1= No, 0= Yes.
P619(8)	Constant Surface Speed possible in JOG mode. 1= No, 0= Yes.
P601(7)	Recover initial conditions, issuing M30, when switching to JOG mode. 0= No, 1= Yes.

OPERATING MODE RELATED MACHINE PARAMETERS *Section 3.3.10*

P601(5)	CYCLE START key inhibit. 0= No, 1= Yes.
P619(6)	Spindle inhibit from PLC. 0= No, 1= Yes.
P600(3)	Maximum Feedrate Override Switch value applied by the CNC. 0= 120%, 1= 100%.
P4	Feedrate Override Switch active in G00. No/Yes.
P607(8)	G05 or G07 on power-up. 0= G07, 1= G05.
P607(3)	Vectored G00 (interpolated). 0= No, 1= Yes.
P616(4)	G59 as additive Zero offset. 0= No, 1= Yes.
P609(5)	Arithmetic parameters P150 through P254 read-only. 0= No, 1= Yes.
P611(6)	Function P1=0X assumes current work units. 0= No, 1= Yes.
P621(4)	Synchronization with independent axis. 0= No, 1= Yes.
P607(5)	Constant Surface Speed on multi-spindle lathe. 0= No, 1= Yes.

MACHINE PARAMETERS FOR THE AXES

P100, P300, P200, P400	Sign of the analog output for X, Z, 3rd, 4th axis	*Section 4.*
P101, P301, P201, P401	Counting direction for X, Z, 3rd, 4th axis	
P102, P302, P202, P402	Jogging direction for X, Z, 3rd, 4th axis	

MACHINE PARAMETERS FOR AXIS RESOLUTION *Section 4.1*

P103, P303, P203, P403	Feedback resolution for X, Z, 3rd, 4th axis
P602(3), P602(2), P612(2), P614(2)	Feedback units for X, Z, 3rd, 4th axis. 0= mm, 1= inches.
P106, P306, P206, P406	Type of feedback signal for X,Z,3rd,4th axis. Y= Sine, N= Square
P602(6), P602(5), P612(5), P614(5)	Multiplying factor for X, Z, 3rd, 4th axis feedback. 0= x4, 1= x2.
P619(1), P619(2), P619(3), P619(4)	1, 2, 5, 10 resolution for X, Z, 3rd, 4th axis sine-wave feedback
P604(2), P604(1), P612(3), P614(3)	Binary encoder for X, Z, 3rd, 4th axis. 0= No, 1= Yes.
P604(7), P604(6), P612(4), P614(4)	Equivalence of the binary encoder for X, Z, 3rd, 4th axis

MACHINE PARAMETERS FOR ANALOG OUTPUTS *Section 4.2*

P117, P317, P217, P417	Minimum analog output for X, Z, 3rd, 4th axis
P104, P304, P204, P404	Dwell Enable - Analog output for X, Z, 3rd, 4th axis. N=No, Y=Yes.
P118, P318, P218, P418	In-position zone (dead band) for X, Z, 3rd, 4th axis. (0..255µm)
P105, P305, P205, P405	Continuous control of X, Z, 3rd, 4th axis. 0= No, 1= Yes.

MACHINE PARAMETERS FOR AXIS TRAVEL LIMITS *Section 4.3*

P107, P307, P207, P407	Positive travel limit for X, Z, 3rd, 4th axis
P108, P308, P208, P408	Negative travel limit for X, Z, 3rd, 4th axis

LEADSCREW RELATED MACHINE PARAMETERS *Section 4.4*

P109, P309, P209, P409	Leadscrew backlash for X, Z, 3rd, 4th axis. (0..255µm)
P620(1), P620(2), P620(3), P620(4)	Sign of leadscrew backlash for X, Z, 3rd, 4th axis (0= +, 1= -).
P113, P313, P213, P413	Additional analog pulse for X, Z, 3rd, 4th axis. (1=2.5mV)
P605(2), P605(1)	Leadscrew error compensation for X, Z. 0= No, 1= Yes.

FEEDRATE RELATED MACHINE PARAMETERS *Section 4.5*

P618(2)	Rotary axis feedrate in degrees/minute (1) or 2.54º/min (0).
P110, P310, P210, P410	Maximum programmable feedrate for X, Z, 3rd, 4th axis
P111, P311, P211, P411	G00 Feedrate for X, Z, 3rd, 4th axis
P717	Maximum feedrate F for circular interpolations
P703	Feedrate/Override when analog output reaches 10V
P705	Error if actual feedrate not within 50% to 200% of programmed value

MACHINE PARAMETERS FOR AXIS CONTROL *Section 4.6*

P114, P314, P214, P414	Proportional gain K1 for X, Z, 3rd, 4th axis
P115, P315, P215, P415	Gain break-point for X, Z, 3rd, 4th axis
P116, P316, P216, P416	Proportional gain K2 for X, Z, 3rd, 4th axis
P607(6)	Apply only K1 when threading. 0= No, 1= Yes.
P607(7)	Apply only K2 in G00 moves. 0= No, 1= Yes.
P715	Recovery of programmed position on "non-continuously" controlled axes

HOME SEARCH RELATED MACHINE PARAMETERS *Section 4.7*

P608(5), P608(8), P617(5), P615(8)	Feedback "Io" type. X, Z, 3rd, 4th axis. 0= Regular, 1= Coded.
P608(3), P608(6), P617(3), P615(6)	Period of coded "Io". X, Z, 3, 4th axis. 0= 20mm, 1= 100mm.
P608(4), P608(7), P617(4), P615(7)	Variable "Io" increasing in "+" (0) or "-" (1) direction. X..4th
P908, P909, P910, P911	Offset of the semi-absolute scale (with coded "Io"). X..4th axis
P119, P319, P219, P419	Home coordinates for X, Z, 3rd, 4th axis
P618(8), P618(7), P618(6), P618(5)	Home search direction for X, Z, 3rd, 4th axis. 0= Pos., 1= Neg.
P600(7), P600(6), P612(7), P614(7)	Home pulse (marker) type for X..4th axis. 1= Pos., 0= Neg.
P600(5), P600(4), P612(6), P614(6)	Home switch for X, Z, 3rd, 4th axis. 1= No, 0= Yes.
P112, P312, P212, P412	1st homing feedrate for X, Z, 3rd, 4th axis
P807, P808, P809, P810	2nd homing feedrate for X, Z, 3rd, 4th axis
P604(8)	Mandatory home search on power-up. 0= No, 1= Yes.
P601(8)	Function G74 generates an M30. 0= No, 1= Yes.

ACCELERATION/DECELERATION RELATED MACHINE PARAMETERS *Section 4.8*

P712, P713, P714, P724	Apply ACC/DEC onto X, Z, 3rd, 4th axis. (1= 20ms)
P609(4)	Linear ACC/DEC on all linear interpolations (G01). 0= No, 1= Yes.
P616(6)	Linear ACC/DEC in G05 (round corner). 1= No, 0= Yes.
P621(8)	Bell-shaped ACC/DEC. 0= No, 1= Yes.
P731	Bell-shaped ACC/DEC ramp duration. (1=10ms)
P720, P721, P722, P723	FEED-FORWARD gain for X, Z, 3rd, 4th axis

MACHINE PARAMETERS FOR THE LIVE OR SYNCHRONIZED TOOL *Section 4.9*

P607(1)	Sign of the analog output for the live or synchronized tool
P802	Maximum programmable speed for the live or synchronized tool. (0..9999 rpm)
P609(8)	Speed of the live or synchronized tool variable from the operator panel. 0= No, 1= Yes.
P803	Synchronized tool encoder line count. (0..9999)
P607(2)	Counting direction of the synchronized tool
P711	Proportional gain K for the synchronized tool

SPECIAL MACHINE PARAMETERS *Section 4.10*

P606(1)	Machine with travels over 8 meters. 0= No, 1= Yes.
P609(7)	Resolution of 0.0001 millimeters (0.00001 inches).. 0= No, 1= Yes.

SPINDLE MACHINE PARAMETERS

Section 5.

P811 Acceleration/deceleration for the spindle. (1=10mm)

MACHINE PARAMETERS FOR SPINDLE RANGE CHANGE

Section 5.1

P7, P8, P9, P10	Maximum speed for RANGE 1, 2, 3 and 4. (0..9999 rpm)
P601(1)	The machine has an automatic range changer. 0= No, 1= Yes.
P601(6)	Residual analog "S" output during range change . 0= No, 1= Yes.
P701	Value of the residual analog "S" output. (1=2.5mV)
P702	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"
P607(4)	Unipolar (1) or bipolar (0) analog spindle speed output "S"

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	Spindle encoder line count. (0..9999)
P606(3)	Spindle encoder counting direction
P603(8)	The spindle speed is monitored. 1= No, 0= Yes.
P704	Stabilizing period for S. (1= 100ms)

MACHINE PARAMETERS FOR SPINDLE ORIENTATION (M19)

Section 5.4.1

P706	"S" spindle speed when in M19. (0..255 rpm)
P606(2)	Sign of the "S" spindle output associated with M19
P600(8)	Type of marker pulse (home) of the spindle encoder. 0= Negative, 1= Positive.
P709	Minimum spindle analog output in M19. (1=2.5mV)
P707	In-position zone (dead-band) for the spindle in M19
P708	Proportional gain K for the spindle in M19

APPENDIX G

SEQUENTIAL MACHINE PARAMETER LIST

P0	Communications baudrate (110, 150, 300, 600, 1200, 2400, 4800, 9600)	*Section 3.3.7*
P1	Number of communication data bits (7/8)	*Section 3.3.7*
P2	Parity . 0= No, 1= Odd, 2= Even).	*Section 3.3.7*
P3	Stop bits (1/2)	*Section 3.3.7*
P4	Feedrate Override active in G00. N= No, Y= Yes.	*Section 3.3.10*
P5	Mains (AC) frequency. (50/60)	*Section 3.3*
P6	Theoretical (1) or real (0) display	*Section 3.3.8*
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	X axis in radius (0) or diameter (1)	*Section 3.3*
P12	Continuous (N) or pulsating (Y) axis jog	*Section 3.3.9*
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 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. 0= No, 1= Yes	*Section 4.2*
P106	Type of feedback signal for the X axis. Y= Sine, N= Square	*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 4.4*
P110	X axis maximum programmable feedrate	*Section 4.5*
P111	X axis G00 feedrate	*Section 4.5*
P112	1st homing feedrate for the X axis	*Section 4.7*
P113	Additional analog pulse for the X axis. (1=2.5mV)	*Section 4.4*
P114	Proportional gain K1 for the X axis	*Section 4.6*
P115	Gain break-point for the X axis	*Section 4.6*
P116	Proportional gain K2 for the X axis	*Section 4.6*
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 4.7*
P200	Sign of the 3rd axis analog output	*Section 4.*
P201	Counting direction of the 3rd axis	*Section 4.*
P202	3rd axis jogging direction	*Section 4.*
P203	3rd axis feedback resolution	*Section 4.1*
P204	Dwell between Enable and analog output for the 3rd axis. N= No, Y= Yes	*Section 4.2*
P205	Continuous control of the 3rd axis. 0= No, 1= Yes	*Section 4.2*
P206	Type of feedback signal for the 3rd axis. Y= Sine, N= Square	*Section 4.1*
P207	3rd axis positive travel limit	*Section 4.3*
P208	3rd axis negative travel limit	*Section 4.3*
P209	3rd axis leadscrew backlash. (0..255µm)	*Section 4.4*
P210	3rd axis maximum programmable feedrate	*Section 4.5*
P211	3rd axis G00 feedrate	*Section 4.5*
P212	1st homing feedrate for the 3rd axis	*Section 4.7*
P213	Additional analog pulse for the 3rd axis. (1=2.5mV)	*Section 4.4*
P214	Proportional gain K1 for the 3rd axis	*Section 4.6*
P215	Gain break-point for the 3rd axis	*Section 4.6*
P216	Proportional gain K2 for the 3rd axis	*Section 4.6*
P217	Minimum 3rd axis analog. (1= 2.5mV)	*Section 4.2*
P218	In-position zone (dead-band) for the 3rd axis. (0..255µm)	*Section 4.2*
P219	3rd axis home coordinate	*Section 4.7*

P300 Sign of the Z axis analog output ..... *Section 4.*
P301 Counting direction of the Z axis ..... *Section 4.*
P302 Z axis jogging direction ..... *Section 4.*
P303 Z axis feedback resolution ..... *Section 4.1*
P304 Dwell between Enable and analog output for the Z axis. N= No, Y= Yes ..... *Section 4.2*
P305 Continuous control of the Z axis. 0= No, 1= Yes ..... *Section 4.2*
P306 Type of feedback signal for the Z axis. Y= Sine, N= Square ..... *Section 4.1*
P307 Z axis positive travel limit ..... *Section 4.3*
P308 Z axis negative travel limit ..... *Section 4.3*
P309 Z axis leadscrew backlash. (0..255µm) ..... *Section 4.4*
P310 Z axis maximum programmable feedrate ..... *Section 4.5*
P311 Z axis G00 feedrate ..... *Section 4.5*
P312 1st homing feedrate for the Z axis ..... *Section 4.7*
P313 Additional analog pulse for the Z axis. (1=2.5mV) ..... *Section 4.4*
P314 Proportional gain K1 for the Z axis ..... *Section 4.6*
P315 Gain break-point for the Z axis ..... *Section 4.6*
P316 Proportional gain K2 for the Z axis ..... *Section 4.6*
P317 Minimum Z axis analog. (1= 2.5mV) ..... *Section 4.2*
P318 In-position zone (dead-band) for the Z axis. (0..255µm) ..... *Section 4.2*
P319 Z axis home coordinate ..... *Section 4.7*

P400 Sign of the 4th axis analog output ..... *Section 4.*
P401 Counting direction of the 4th axis ..... *Section 4.*
P402 4th axis jogging direction ..... *Section 4.*
P403 4th axis feedback resolution ..... *Section 4.1*
P404 Dwell between Enable and analog output for the 4th axis. N= No, Y= Yes ..... *Section 4.2*
P405 Continuous control of the 4th axis. 0= No, 1= Yes ..... *Section 4.2*
P406 Type of feedback signal for the 4th axis. Y= Sine, N= Square ..... *Section 4.1*
P407 4th axis positive travel limit ..... *Section 4.3*
P408 4th axis negative travel limit ..... *Section 4.3*
P409 4th axis leadscrew backlash. (0..255µm) ..... *Section 4.4*
P410 4th axis maximum programmable feedrate ..... *Section 4.5*
P411 4th axis G00 feedrate ..... *Section 4.5*
P412 1st homing feedrate for the 4th axis ..... *Section 4.7*
P413 Additional analog pulse for the 4th axis. (1=2.5mV) ..... *Section 4.4*
P414 Proportional gain K1 for the 4th axis ..... *Section 4.6*
P415 Gain break-point for the 4th axis ..... *Section 4.6*
P416 Proportional gain K2 for the 4th axis ..... *Section 4.6*
P417 Minimum 4th axis analog. (1= 2.5mV) ..... *Section 4.2*
P418 In-position zone (dead-band) for the 4th axis. (0..255µm) ..... *Section 4.2*
P419 4th axis home coordinate ..... *Section 4.7*

P500 Handwheel counting direction ..... *Section 3.3.3*
P501 Handwheel feedback resolution ..... *Section 3.3.3*

P600(8) Type of spindle marker pulse (home). 0= Negative, 1= Positive ..... *Section 5.4.1*
(7) Type of X axis marker pulse (home). 0= Negative, 1= Positive ..... *Section 4.7*
(6) Type of Z axis marker pulse (home). 0= Negative, 1= Positive ..... *Section 4.7*
(5) X axis home switch. 0= Yes, 1= No ..... *Section 4.7*
(4) Z axis home switch. 0= Yes, 1= No ..... *Section 4.7*
(3) Maximum Feedrate Override value applied by CNC.0=120%, 1=100%. ..... *Section 3.3.10*
(2) Invert the JOG keys (X and Z). 0= No, 1= Yes ..... *Section 3.3.9*
(1) Machine axes orientation ..... *Section 3.3*

P601(8) Function G74 generates an M30. 0= No, 1= Yes ..... *Section 4.7*
(7) Recover initial conditions, issue M30, when switching to JOG. 0=No,1=Yes . *Section 3.3.9*
(6) Residual analog "S" during spindle range change. 0= No, 1= Yes ..... *Section 5.1*
(5) CYCLE START key inhibit. 0= No, 1= Yes ..... *Section 3.3.10*
(4) Sign of the spindle analog output "S" ..... *Section 5.2*
(3) 2-digit coded BCD "S" output. 0= No, 1= Yes ..... *Section 5.3*
(2) 4-digit coded BCD "S" output. 0= No, 1= Yes ..... *Section 5.3*
(1) The machine has an automatic spindle range changer. 0= No, 1= Yes ..... *Section 5.1*

P602(8) Not being used at this time (=0)
(7) The CNC waits for down flank of M-done signal. 0= No, 1= Yes ..... *Section 3.3.2*
(6) Multiplying factor for X axis feedback signals. 0= x4, 1= x2 ..... *Section 4.1*
(5) Multiplying factor for Z axis feedback signals. 0= x4, 1= x2 ..... *Section 4.1*
(4) Multiplying factor for handwheel feedback signals. 0= x4, 1= x2 ..... *Section 3.3.3*
(3) X axis feedback units. 0= mm, 1= inches ..... *Section 4.1*
(2) Z axis feedback units. 0= mm, 1= inches ..... *Section 4.1*
(1) Electronic handwheel feedback units. 0= mm, 1= inches ..... *Section 3.3.3*

P603(8) Spindle speed is monitored. 1= No, 0= Yes ..... *Section 5.4*
(7) The M function can be executed in JOG mode. 1= No, 0= Yes ..... *Section 3.3.9*
(6) The T function can be executed in JOG mode. 1= No, 0= Yes ..... *Section 3.3.9*
(5) The S function can be executed in JOG mode. 1= No, 0= Yes ..... *Section 3.3.9*
(4) A1 axis feedback alarm cancellation. 0= No, 1= Yes ..... *Section 3.3.2*
(3) A2 axis feedback alarm cancellation. 0= No, 1= Yes ..... *Section 3.3.2*
(2) A3 axis feedback alarm cancellation. 0= No, 1= Yes ..... *Section 3.3.2*
(1) A4 axis feedback alarm cancellation. 0= No, 1= Yes ..... *Section 3.3.2*

P604(8) Mandatory home search on power-up. 0= No, 1= Yes ..... *Section 4.7*
(7) Equivalence of the X axis binary encoder ..... *Section 4.1*
(6) Equivalence of the Z axis binary encoder ..... *Section 4.1*
(5) Tool offset values applied after T2.2 (0) or after M06 (1) ..... *Section 3.3.5*
(4) Normal status of emergency output, pin 5 of I/O1. 0= 0V, 1= 5V or 24V ..... *Section 3.3.2*
(3) Pin 23 of connector I/O2 as G00. 0= No, 1= Yes ..... *Section 3.3.2*
(2) Binary encoder on X axis. 0= No, 1= Yes ..... *Section 4.1*
(1) Binary encoder on Z axis. 0= No, 1= Yes ..... *Section 4.1*

P605(8) The CNC aborts DNC communications (debugging). 1= No, 0= Yes ..... *Section 3.3.7*
(7) DNC Protocol active on power-up. 0= No, 1= Yes ..... *Section 3.3.7*
(6) Communication with FAGOR Cassette (0) or Floppy Disk Unit (1) ..... *Section 3.3.7*
(5) DNC active. 0= No, 1= Yes ..... *Section 3.3.7*
(4) Pin 6 of connector I/O1 as Threading ON (0) or Cycle ON (1) ..... *Section 3.3.2*
(3) Not being used at this time (=0)
(2) X axis leadscrew error compensation. 0= No, 1= Yes ..... *Section 4.4*
(1) Z axis leadscrew error compensation. 0= No, 1= Yes ..... *Section 4.4*

P606(8) Status report by interruption. 0= No, 1= Yes ..... *Section 3.3.7*
(7) Decoded M also output in BCD code. 1= No, 0= Yes ..... *Section 3.3.2*
(6) Type of probe pulse. 0= 0V, 1= 5V, 24V ..... *Section 3.3.4*
(5) Axes orientation in graphic representation ..... *Section 3.3.8*
(4) Axes orientation in graphic representation ..... *Section 3.3.8*
(3) Spindle counting direction ..... *Section 5.4*
(2) Sign of the analog S output associated with M19 ..... *Section 5.4.1*
(1) Machine with travels over 8 meters. 0= No, 1= Yes ..... *Section 4.10*

P607(8) G05 (1) or G07 (0) on power-up ..... *Section 3.3.10*
(7) Only K2 applied on G00 moves. 0= No, 1= Yes ..... *Section 4.6*
(6) Only K1 applied when threading. 0= No, 1= Yes ..... *Section 4.6*
(5) Constant Surface Speed on multi-spindle lathes. 0= No, 1= Yes ..... *Section 3.3.10*
(4) Unipolar (1) or bipolar (0) "S" analog output ..... *Section 5.2*
(3) Vectored G00 (interpolated). 0= No, 1= Yes ..... *Section 3.3.10*
(2) Counting direction of the live or synchronized tool ..... *Section 4.9*
(1) Sign of the live or synchronized tool ..... *Section 4.9*

P608(8) Type of feedback Io for Z axis. 0= Regular, 1= Coded ..... *Section 4.7*
(7) Variable Z axis coded Io increasing in "+" (0) or "-" (1) direction. ..... *Section 4.7*
(6) Period of Z axis coded Io. 0= 20mm, 1= 100mm. ..... *Section 4.7*
(5) Type of feedback Io for X axis. 0= Regular, 1= Coded ..... *Section 4.7*
(4) Variable X axis coded Io increasing in "+" (0) or "-" (1) direction. ..... *Section 4.7*
(3) Period of X axis coded Io. 0= 20mm, 1= 100mm. ..... *Section 4.7*
(2) Not being used at this time (=0)
(1) A5 axis feedback alarm cancellation. 0= No, 1= Yes ..... *Section 3.3.2*

P609(8) The speed of the live tool variable from operator panel. 0= No, 1= Yes ..... *Section 4.9*
(7) Resolution of 0.0001 millimeters (0.00001 inch). 0= No, 1= Yes ..... *Section 4.10*
(6) Pin 24 of connector I/O2 as turret rotating direction. 0= No, 1= Yes. ..... *Section 3.3.2*
(5) Arithmetic parameters P150 through P254 read-only. 0= No, 1= Yes ..... *Section 3.3.10*
(4) Acceleration/deceleration on all linear interpolations. 0= No, 1= Yes ..... *Section 4.8*
(3) The CNC displays tool tip position (0) or that of the tool base (1) ..... *Section 3.3.5*
(2) There is a PLC64 in the FAGOR LAN. 0= No, 1= Yes ..... *Fagor LAN*
(1) The electronic handwheel is a FAGOR 100P. 0= No, 1= Yes ..... *Section 3.3.3*

P610 CNC identifaction in the FAGOR LAN ..... *Fagor LAN*

P611(8) Monitor display color combination ..... *Section 3.3.8*
(7) Monitor display color combination ..... *Section 3.3.8*
(6) Function P1=0X assumes current work units. 0= No, 1= Yes ..... *Section 3.3.10*
(5) The CNC occupies the main LAN node (0) ..... *Fagor LAN*
(4) Node occupied by the CNC or number of nodes in the LAN ..... *Fagor LAN*
(3) Node occupied by the CNC or number of nodes in the LAN ..... *Fagor LAN*
(2) Node occupied by the CNC or number of nodes in the LAN ..... *Fagor LAN*
(1) Node occupied by the CNC or number of nodes in the LAN ..... *Fagor LAN*

P612(8) 3rd axis display. 1= No, 0= Yes ..... *Section 3.3.8*
(7) Type of 3rd axis marker pulse (home). 0= Negative, 1= Positive ..... *Section 4.7*
(6) 3rd axis home switch. 1= No, 0= Yes ..... *Section 4.7*
(5) Multiplying factor for the 3rd axis feedback signals. 0= x4, 1= x2 ..... *Section 4.1*
(4) Equivalence of the 3rd axis binary encoder ..... *Section 4.1*
(3) 3rd axis with binary encoder. 0= No, 1= Yes ..... *Section 4.1*
(2) 3rd axis feedback units. 0= mm, 1= inches ..... *Section 4.1*
(1) The machine has a 3rd axis. 0= No, 1= Yes ..... *Section 3.3.1*

P613(8) Not being used at this time (=0)
(7) Not being used at this time (=0)
(6) Rotary axis rollover via shortest path. 0= No, 1= Yes ..... *Section 3.3.1*
(5) The 3rd axis is a "C" axis. 0= No, 1= Yes ..... *Section 3.3.1*
(4) The 3rd axis is called Y/W. (0= Y, 1= W) ..... *Section 3.3.1*
(3) The 3rd axis is a DRO axis. 0= No, 1= Yes ..... *Section 3.3.1*
(2) The 3rd axis is rotary and rollover. 0= No, 1= Yes ..... *Section 3.3.1*
(1) The 3rd axis is rotary. 0= No, 1= Yes ..... *Section 3.3.1*

P614(8) 4th axis display. 1= No, 0= Yes ..... *Section 3.3.8*
(7) Type of 4th axis marker pulse (home). 0= Negative, 1= Positive ..... *Section 4.7*
(6) 4th axis home switch. 1= No, 0= Yes ..... *Section 4.7*
(5) Multiplying factor for the 4th axis feedback signals. 0= x4, 1= x2 ..... *Section 4.1*
(4) Equivalence of the 4th axis binary encoder ..... *Section 4.1*
(3) 4th axis with binary encoder. 0= No, 1= Yes ..... *Section 4.1*
(2) 4th axis feedback units. 0= mm, 1= inches ..... *Section 4.1*
(1) The machine has a 4th axis. 0= No, 1= Yes ..... *Section 3.3.1*

P615(8) 4th axis feedback Io type. 0= Regular, 1= Coded ..... *Section 4.7*
(7) 4th axis variable Io increasing in "+" (0) or "-" (1) direction ..... *Section 4.7*
(6) Period of 4th axois coded Io. 0= 20mm, 1= 100mm ..... *Section 4.7*
(5) 4th axis rotary rollover via shortest path. 0= No, 1= Yes ..... *Section 3.3.1*
(4) The 4th axis is called Y/W. (0= Y, 1= W) ..... *Section 3.3.1*
(3) The 4th axis is a DRO axis. 0= No, 1= Yes ..... *Section 3.3.1*
(2) The 4th axis is rotary and rollover. 0= No, 1= Yes ..... *Section 3.3.1*
(1) The 4th axis is rotary. 0= No, 1= Yes ..... *Section 3.3.1*

P616(8) Connector A6 shared by handwheel and synchronized tool . 0= No, 1= Yes ..*Section 3.3.1*
(7) Transfer Inhibit and M-done inputs independent from Feed-hold .....*Fagor LAN*
(6) Acceleration/deceleration in G05 (round corner). 1= No, 0= Yes .....*Section 4.8*
(5) Marks M1801 through M1899 used to send messages to the CNC .....*LAN & PLCI*
(4) G59 as additive zero offset. 0= No, 1= Yes .....*Section 3.3.10*
(3) Not being used at this time (=0)
(2) The emergency subroutine executes an M00. 0= No, 1= Yes.....*Section 3.3.6*
(1) Assign coord. to arithm. param. in emergency subr. 0= Beg., 1= current .....*Section 3.3.6*

P617(8) For the exclusive use of the Service Department of Fagor Automation
(7) Not being used at this time (=0)
(6) Not being used at this time (=0)
(5) 3rd axis feedback Io type. 0= Regular, 1= Coded.....*Section 4.7*
(4) 3rd axis variable Io increasing in positive (0) or negative (1) direction. .....*Section 4.7*
(3) Period of the 3rd axis coded Io. 0= 20mm, 1= 100mm .....*Section 4.7*
(2) Associated subroutine executed before (1) or after (0) the T function .....*Section 3.3.5*
(1) The CNC has a PLCI. 0= No, 1= Yes .....*PLCI Manual*

P618(8) X axis homing direction. 0= Positive, 1= Negative .....*Section 4.7*
(7) Z axis homing direction. 0= Positive, 1= Negative .....*Section 4.7*
(6) 3rd axis homing direction. 0= Positive, 1= Negative .....*Section 4.7*
(5) 4th axis homing direction. 0= Positive, 1= Negative .....*Section 4.7*
(4) Not being used at this time (=0)
(3) The Transfer-inhibit signal has no effect on M, S, T functions .....*Fagor LAN*
(2) Rotary axis feedrate in degrees/minute (1) or in 2.54º/min (0) .....*Section 4.5*
(1) Not being used at this time (=0)

P619(8) It is possible to work at Constant Surface Speed in JOG mode.....*Section 3.3.9*
(7) Handwheel managed by the PLC.....*Section 3.3.3*
(6) Spindle inhibit from the PLC. 0= No, 1= Yes.....*Section 3.3.3 10*
(5) Not being used at this time (=0)
(4) Resolution values for 4th axis sine-wave feedback.....*Section 4.1*
(3) Resolution values for 3rd axis sine-wave feedback .....*Section 4.1*
(2) Resolution values for Z axis sine-wave feedback .....*Section 4.1*
(1) Resolution values for X axis sine-wave feedback .....*Section 4.1*

P620(8) Not being used at this time (=0)
(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) Sign of the 4th axis leadscrew backlash. 0= Positive, 1= Negative.....*Section 4.4*
(3) Sign of the 3rd axis leadscrew backlash. 0= Positive, 1= Negative .....*Section 4.4*
(2) Sign of the Z axis leadscrew backlash. 0= Positive, 1= Negative .....*Section 4.4*
(1) Sign of the X axis leadscrew backlash. 0= Positive, 1= Negative .....*Section 4.4*

P621(8) Bell-shaped acc/dec. 0= No, 1= Yes.....*Section 4.8*
(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) Synchronization with independent axis. 0= No, 1= Yes .....*Section 3.3.10*
(3) Not being used at this time (=0)
(2) Not being used at this time (=0)
(1) For the exclusive use of the Service Department of Fagor Automation

P622 For the exclusive use of the Service Department of Fagor Automation
P623 For the exclusive use of the Service Department of Fagor Automation

P700 Number of tools. (0..32) ..... *Section 3.3.5*
P701 Residual analog "S" value (1=2.5mV) ..... *Section 5.1*
P702 Oscillation period during spindle range change ..... *Section 5.1*
P703 Feedrate/Override when the analog output reaches 10 V ..... *Section 4.5*
P704 Stabilizing time for S (1=100ms) ..... *Section 5.4*
P705 Error if actual feedrate not within 50% and 200% of programmed ..... *Section 4.5*
P706 Spindle "S" speed in M19 (0=255 rpm) ..... *Section 5.4.1*
P707 Spindle in-position (dead-band) zone in M19 ..... *Section 5.4.1*
P708 Proportional gain K of the spindle in M19 ..... *Section 5.4.1*
P709 Minimum spindle analog output "S" in M19. (1=2.5mV) ..... *Section 5.4.1*
P710 M function associated with probing (G75) ..... *Section 3.3.4*
P711 Proportional gain K for synchronized tool ..... *Section 4.9*
P712 X axis Acceleration/Deceleration. (1=20ms) ..... *Section 4.8*
P713 Z axis Acceleration/Deceleration. (1=20ms) ..... *Section 4.8*
P714 3rd axis Acceleration/Deceleration. (1=20ms) ..... *Section 4.8*
P715 Recovery of programmed position on non-continuously controlled axes ..... *Section 4.6*
P716 Emergency subroutine ..... *Section 3.3.6*
P717 Maximum feedrate F for circular interpolations ..... *Section 4.5*
P718 Node receiving the M, S, T functions ..... *Fagor LAN*
P719 Nº of node register receiving the the M, S, T functions ..... *Fagor LAN*
P720 X axis Feed-Forward gain ..... *Section 4.8*
P721 Z axis Feed-Forward gain ..... *Section 4.8*
P722 3rd axis Feed-Forward gain ..... *Section 4.8*
P723 4th axis Feed-Forward gain ..... *Section 4.8*
P724 4th axis acceleration/deceleration. (1=20 ms) ..... *Section 4.8*
P725 Group of marks used by the CNC to send out its internal data ..... *Fagor LAN*
P726 Group of marks used by the CNC to update the status of its connectors ..... *Fagor LAN*
P727 Group of marks used by the CNC to update its internal data ..... *Fagor LAN*
P728 Group of marks used by the CNC to update its additional internal data ..... *Fagor LAN*
P729 How often is the PLCI cycle executed ..... *PLCI Manual*
P730 Subroutine associated with the T function ..... *Section 3.3.5*
P731 Bell-shaped acc/dec ramp duration (1= 10ms) ..... *Section 4.8*
P731 through P741 Not being used at this time (=0)

P800 Spindle encoder line count. (0..9999) ..... *Section 5.4*
P801 Protected program ..... *Section 3.3*
P802 Maximum programmable speed for live or synchronized tool. (0..9999) ..... *Section 4.9*
P803 Synchronized tool encoder line count. (0..9999) ..... *Section 4.9*
P804 Not being used at this time (=0)
P805 Not being used at this time (=0)
P806 Probing feedrate in JOG mode ..... *Section 3.3.4*
P807 X axis 2nd homing feedrate ..... *Section 4.7*
P808 3rd axis 2nd homing feedrate ..... *Section 4.7*
P809 Z axis 2nd homing feedrate ..... *Section 4.7*
P810 4th axis 2nd homing feedrate ..... *Section 4.7*
P811 Spindle acceleration/deceleration. (1=10ms) ..... *Section 5.*
P812 through P823 Not being used at this time (=0)

P900 Not being used at this time (=0)
P901 Not being used at this time (=0)
P902 Minimum X coordinate of the probe ..... *Section 3.3.4*
P903 Maximum X coordinate of the probe ..... *Section 3.3.4*
P904 Minimum Z coordinate of the probe ..... *Section 3.3.4*
P905 Maximum Z coordinate of the probe ..... *Section 3.3.4*
P906 Not being used at this time (=0)
P907 Not being used at this time (=0)
P908 Offset of X axis semi-absolute scale ..... *Section 4.7*
P909 Offset of Z axis semi-absolute scale ..... *Section 4.7*
P910 Offset of 3rd axis semi-absolute scale ..... *Section 4.7*
P911 Offset of 4th axis semi-absolute scale ..... *Section 4.7*
P912 through P923 Not being used at this time (=0)

APPENDIX H

MACHINE PARAMETER SETTING CHART

Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE
P0		P4		P8		P12	
P1		P5		P8		P13	
P2		P6		P10			
P3		P7		P11		P99	

Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE
P100		P200		P300		P400	
P101		P201		P301		P401	
P102		P202		P302		P402	
P103		P203		P303		P403	
P104		P204		P304		P404	
P105		P205		P305		P405	
P106		P206		P306		P4306	
P107		P207		P307		P407	
P108		P208		P308		P408	
P109		P209		P309		P409	
P110		P210		P310		P410	
P111		P211		P311		P411	
P112		P212		P312		P412	
P113		P213		P313		P413	
P114		P214		P314		P414	
P115		P215		P315		P415	
P116		P216		P316		P416	
P117		P217		P317		P417	
P118		P218		P318		P418	
P119		P219		P319		P419	

Parameter	VALUE	Parameter	VALUE
P500		P501	

Parameter	VALUE	Parameter	VALUE
P600		P612	
P601		P613	
P602		P614	
P603		P615	
P604		P616	
P605		P617	
P606		P618	
P607		P619	
P608		P620	
P609		P621	
P610		P622	
P611		P623	

Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE
P700		P711		P722		P733	
P701		P712		P723		P734	
P702		P713		P724		P735	
P703		P714		P725		P736	
P704		P715		P726		P737	
P705		P716		P727		P738	
P706		P717		P728		P739	
P707		P718		P729		P740	
P708		P719		P730		P741	
P709		P720		P731			
P710		P721		P732			

Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE
P800		P806		P812		P818	
P801		P807		P813		P819	
P802		P808		P814		P820	
P803		P809		P815		P821	
P804		P810		P816		P822	
P805		P811		P817		P823	

Parameter	VALUE	Parameter	VALUE	Parameter	VALUE	Parameter	VALUE
P900		P906		P912		P918	
P901		P907		P913		P919	
P902		P908		P914		P920	
P903		P909		P915		P921	
P904		P910		P916		P922	
P905		P911		P917		P923	

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	

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	

APPENDIX K

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 :

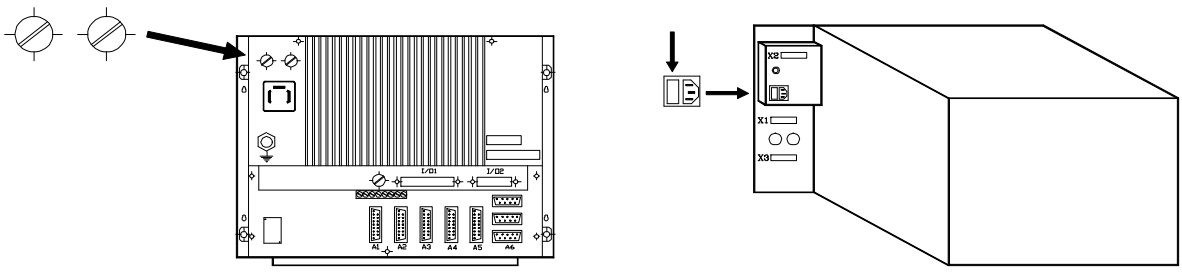
1.- Grease and mineral oils
2.- Bases and bleach
3.- Dissolved detergents
4.- Alcohol

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 TS 8030 TGI 8030 TSI	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	83750041 8G900062	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 * The range or the Constant Surface Speed has not 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 Canned cycle profile 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 G14, G15, G16, 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.

> Synchronization factor K of the synchronized tool too large.

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 More than one spindle range have been defined in the same block.

021 This error will be issued in the following cases:

> There is no block at the address defined by the parameter assigned to F18, F19, F20, F21, F22.
> The corresponding axis has not been defined in the addressed block

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//K has not been defined for a circular interpolation or thread.

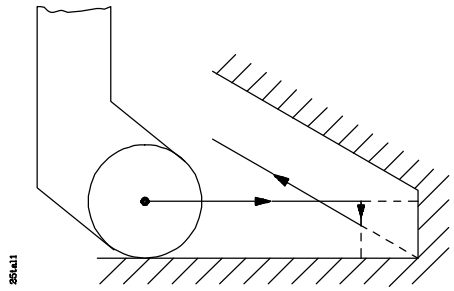
028 An attempt has been made to select a tool offset at the tool table or a non-existent external tool (the number of tools is set by machine parameter).

029 Too large a value assigned to a function.

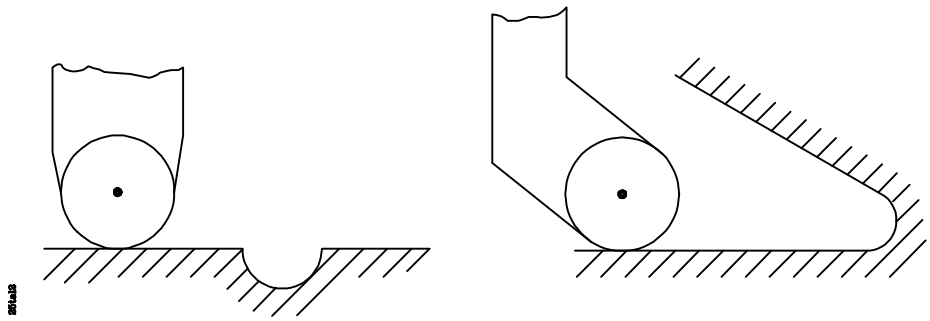
This error is often issued when programming an F value in mm/min (inch/min) and, then, switching to work in mm/rev (inch/rev) without changing the F value.

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 Z axis position Z-5000, if we want to move it to point Z5000, the CNC will issue error 33 when programming the block N10 Z5000 since the programmed move will be: Z5000 - Z-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 Z0          ; 5000 mm move
N10 Z5000       ; 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 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:

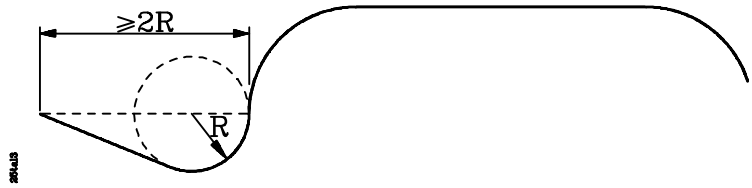
> More than 15 nesting levels when calling subroutines.

> 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:

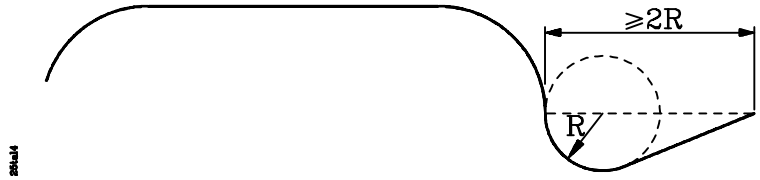
> There is no room to perform the tangential entry. A clearance of twice the rounding radius or greater is required.



> 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 Function M45 S programmed wrong (speed of the live tool).

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 Start or cancel tool radius compensation while in G02 or G03.

049 Chamfer programmed incorrectly.

050 G96 has been programmed while the S output is in BCD as set by machine parameter. (AC spindle).

051 * "C" axis programmed incorrectly

054 There is floppy disk in the FAGOR Floppy Disk Unit or no tape in the cassette reader or the reader head cover is open.

055 Parity error when reading or recording a cassette or a floppy disk.

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 tape.

058 Problems in floppy disk movement or sluggish tape movement.

059 Communication error between the CNC and the FAGOR Floppy Disk Unit or 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.

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.

070 ** X axis following error.

071 ** Synchronized tool following error

072 ** Z axis following error.

073 ** 4th axis following error.

074 ** This error is issued in the following cases:

> 3rd axis following error

>"C" axis following error

075 ** Feedback error at connector A1.

076 ** Feedback error at connector A2.

077 ** Feedback error at connector A3.

078 ** Feedback error at connector A4.

079 ** Feedback error at connector A5.

081 ** 3rd axis travel limit overrun.

082 ** Parity error in 4th axis parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

083 ** 4th axis travel limit overrun.

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 initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

095 ** Parity error in general parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

096 ** Parity error in Z axis parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

097 ** Parity error in 3rd or "C" axis parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

098 ** Parity error in X axis parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

099 ** Parity error in M table. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=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.

108 ** Error in Z axis leadscrew error compensation parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=1.

110 ** Error in X axis leadscrew error compensation parameters. The CNC initializes the RS232C serial line parameters: P0=9600, P1=8, P2=0, P3=1, P605(5)=1, P605(6)=1, P605(7)=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 "P729".

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.

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: T, TG, TS

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

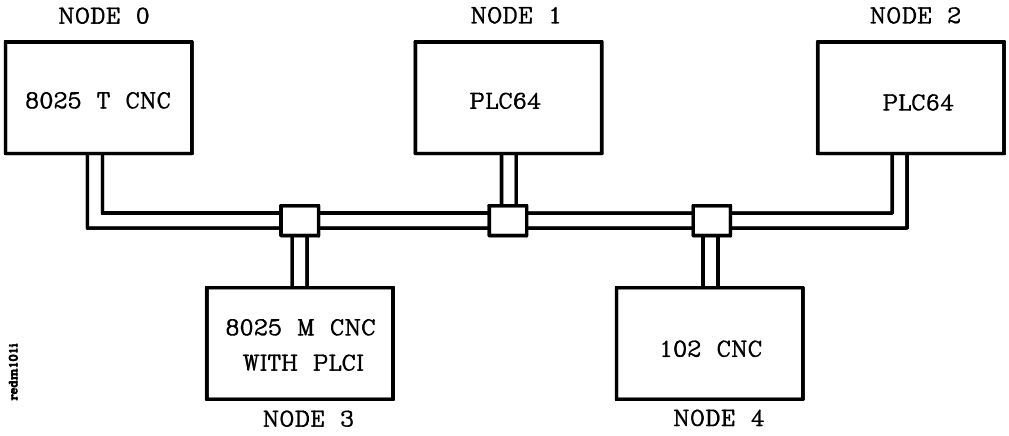
1. FAGOR LOCAL AREA NETWORK CONFIGURATION

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



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

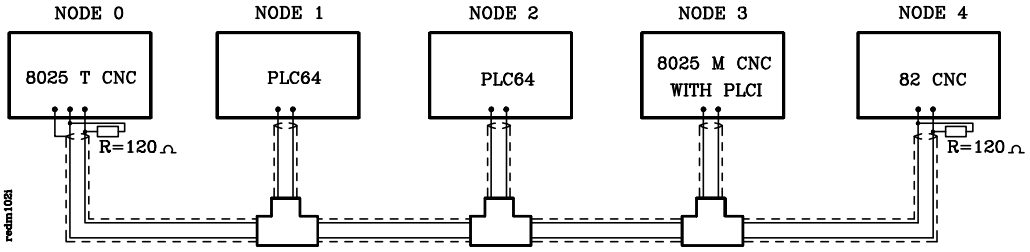
The nodes must be connected via a "TWINAXIAL" cable which must also meet the following requirements:

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:

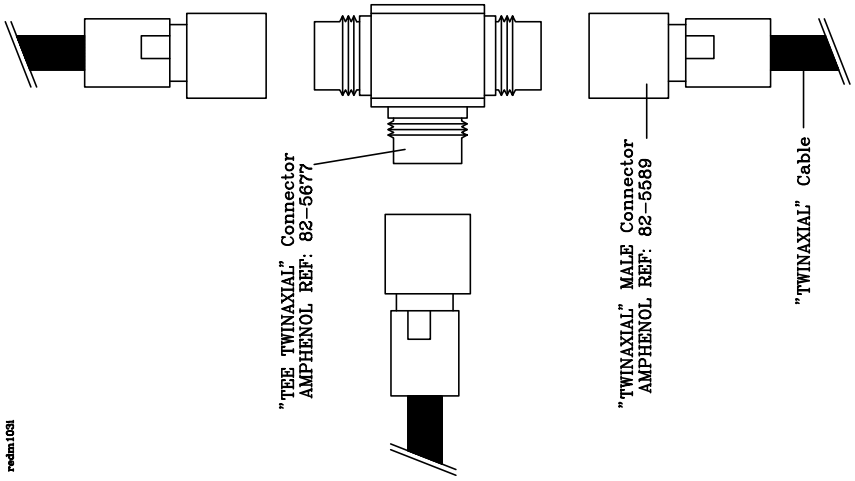
* The shield must be connected only to one of the LAN nodes using pin 1 of the corresponding connector.

* A 120-Ohm-1/4w terminating resistor must be connected between pins 3 and 8 of the nodes most distant from each other.

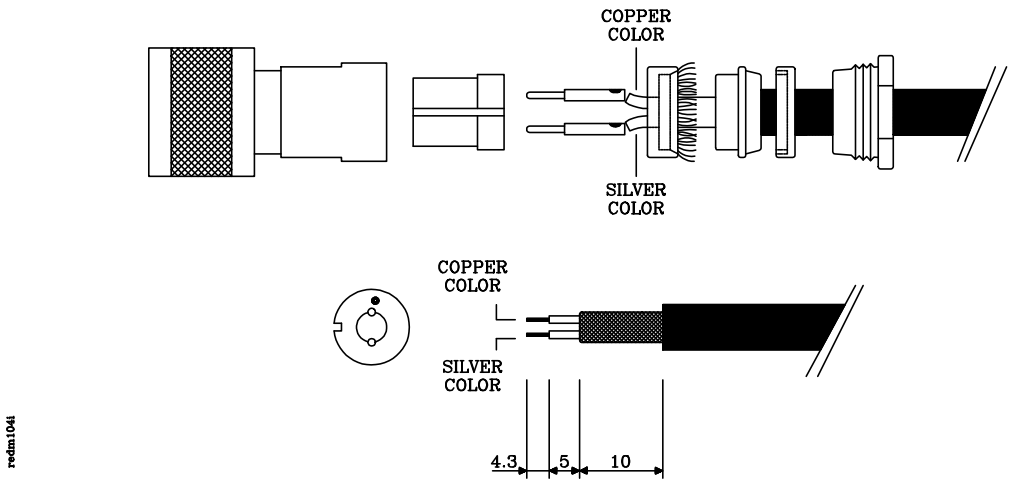


* When the FAGOR LAN consists of more than 2 nodes, "TEE TWINAXIAL" connectors must be used as indicated below.

  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.

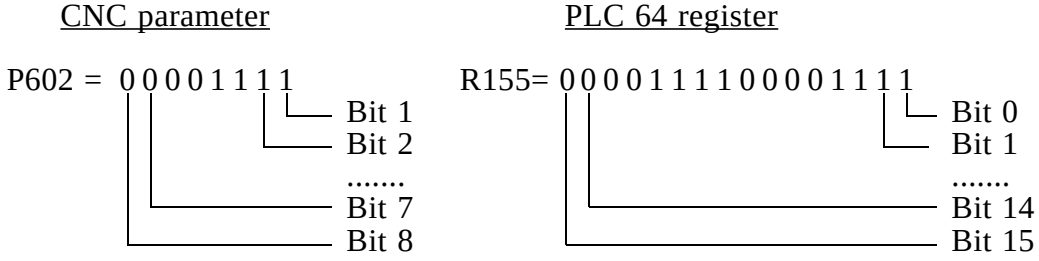
* Inhibit the axes, display messages at the CNC, vary the spindle speed override or the Feedrate override, simulate the CNC keyboard, etc.

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 T CNC in the FAGOR LAN, the following machine parameters must be set:

P611(5) The CNC occupies the main node of the LAN

It indicates whether the CNC is NODE 0 (main node) or not.

P611(5) = 0 It is not the main node (0)
P611(5) = 1 It is the main node

P611(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.

P611(4)	P611(3)	P611(2)	P611(1)	P611(5)=1 CNC at Node 0		P611(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 8025T CNC (node 2).

8025T CNC	P611(5)	P611(4)	P611(3)	P611(2)	P611(1)
Node 2	0	0	0	1	0

The following elements are connected through the FAGOR LAN, one 8025T CNC (node 0) and 2 PLC64 (nodes 1 and 2).

CNC 8025T	P611(5)	P611(4)	P611(3)	P611(2)	P611(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.

Note: This parameter has absolutely no affect on the operation and performance of the CNC.

P616(7) Transfer-inhibit and M-done inputs independent from Feed-hold

The 8025 T CNC utilizes pin 15 of connector I/O1 as input for Feed-Hold, Transfer-Inhibit and M-done signals.

With this parameter it is possible to separate such treatment. If "P617(7)=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, a system call must be made while using 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, a system call must be made while using mark M1955 and bit "1" of register R155 at the PLC64.

P618(3) The Transfer-Inhibit signal does not act upon the M, S, T functions

The CNC checks this parameter when "P616(7)=1".

If during the execution of a block, the Transfer Inhibit signal is set low (0V), the CNC checks the status of parameter P618(3) and acts as follows:

P618(3)=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).

P618(3)=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.

P609(2) 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 "P718" of the node indicated by parameter "P719".

P718 Number of the node receiving the M, S, T function codes
P719 Number of the register receiving the M, S, T function codes at node P730

When there is a PLC64 installed in the LAN, "P609(2)=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 "P718" indicates the number of the node receiving the M, S, T function codes.

Note: If "P718=15", the CNC sends that information to all the nodes occupied by PLC64s.

Parameter "**P719**" 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.

P718=3 Send this data to Node 3

P719=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

P616(5) 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:

"P616(5)=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.

"P616(5)=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].

Note: 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

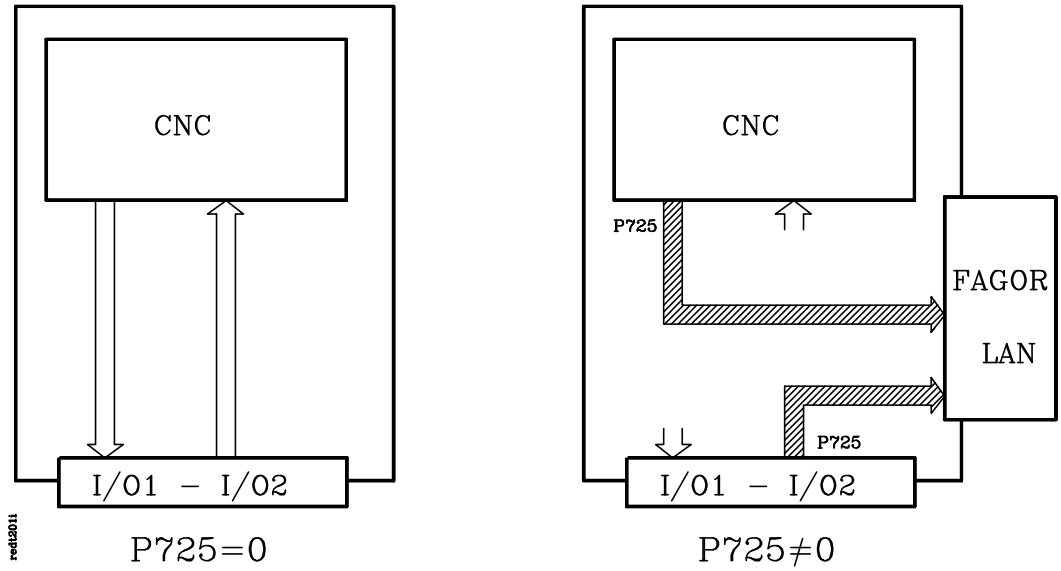
The CNC has 8 groups of 64 internal marks (relays) for data exchange with the rest of the elements of the FAGOR LAN.

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.

P725 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 P725=1, first group of marks. If P725=3, the "Conditional Input" will be M(10+128)= M138.

INPUT	PIN	P725=1 MARK	P725=3 MARK
Cycle Start	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
4th axis home switch/Emergency Subroutine	13 (Connector I/O1)	M5	M133
Z home switch	12 (Connector I/O1)	M6	M134
"C" or 3rd axis 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

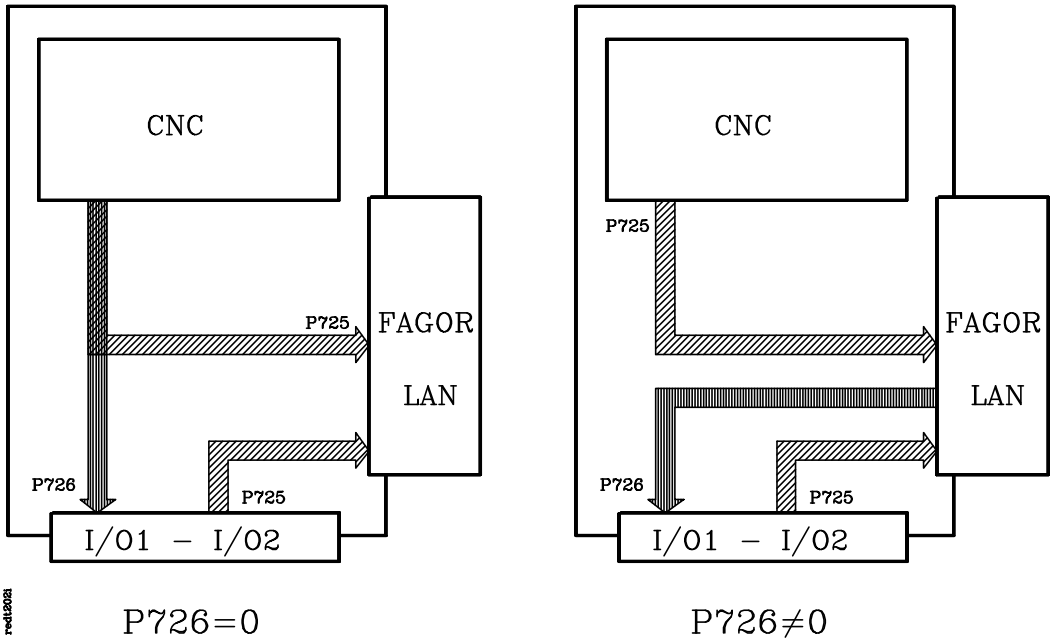
The table below shows the CNC outputs sent out to the LAN and the marks where that information is deposited. It has been assumed that P725=1. If P725=3, the "X axis Enable" signal will be M(20+128) = M148.

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	4th axis Enable	M44
RESET	M21	3rd axis Enable	M45
Z axis Enable	M22	"C" axis Enable	M46
Threading ON	M23	Spindle lock	M47
T Strobe.	M24	Turret rotating direction	M48
S Strobe	M25	4th 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	"C" or 3rd 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		M57
Bit 7 of M function table	M34	Automatic mode	M58
Bit 8 of M function table	M35	Rapid traverse (G00)	M59

Note: If the CNC intervenes in the data exchange carried out via marks, P725 must be set to "0". The CNC ignores parameters "P726", "P727" and "P728".

P726 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 "P725"; otherwise, there will be redundant data.



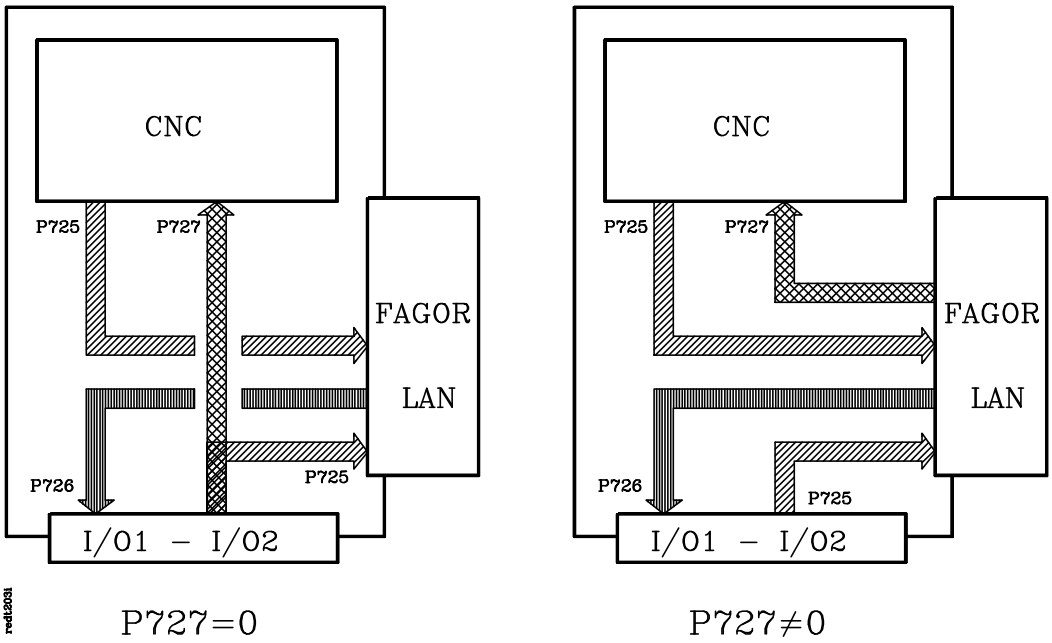
If "P726=0" The CNC will update the output status with the values of the CNC itself as if there were no LAN. It must be borne in mind that this very information is also sent out via LAN to the group indicated by "P725".

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 "P726=1", first group of marks. If P726=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

P727 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 "P725"; otherwise, there will be redundant data.



If "P727=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 "P727".

The following table shows the internal variables and their corresponding marks. It has been assumed that "P727=1", first group of marks. If "P727=3", the "Emergency Stop" signal will be M(4+128) = M132.

INTERNAL CNC VARIABLE	MARK
Cycle Start	M1
Stop / Emergency Subroutine	M2
Feed-Hold	M3
Emergency Subroutine	M4
4th axis home switch	M5
Z axis home switch	M6
"C" or 3rd axis home switch	M7
X axis home switch	M8
DRO mode	M9
Conditional Input (block skip)	M10

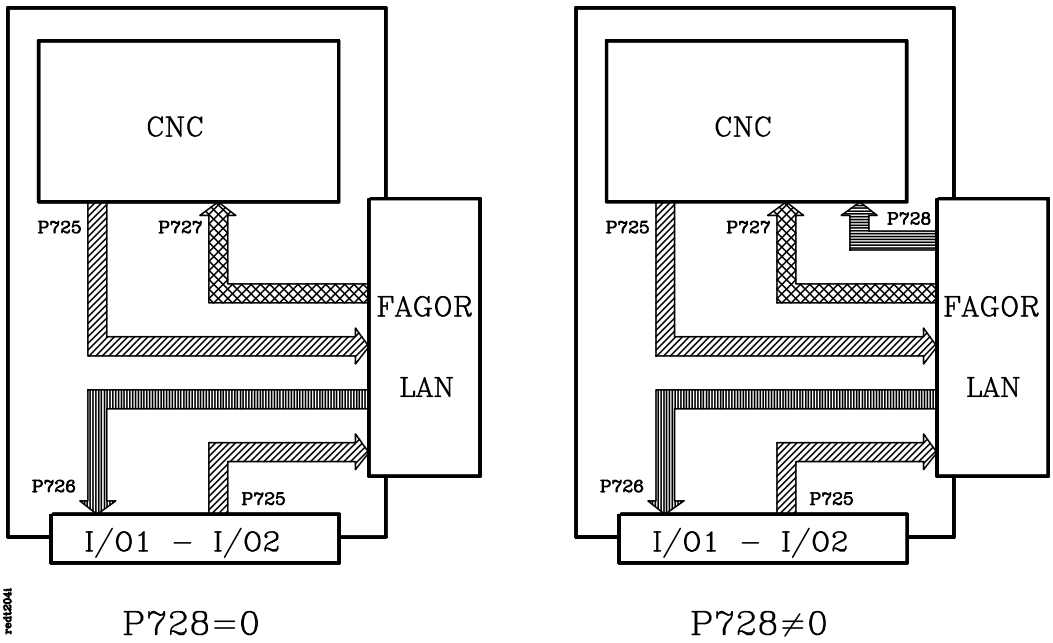
P728 Group of marks used by the CNC to update its additional internal data

Depending on the value assigned to P728, the CNC updates its additional internal data in one of the following ways:

* If "P728=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 "P728<>0", the CNC uses the groups of marks.

  The number assigned to "P728" indicates the group of marks that the CNC uses to update its additional internal data. It must have a value different from that of "P725"; 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 "P728=1", first group of marks. If "P728=3", the "M-done" signal will be M(50+128) = M178.

INTERNAL CNC VARIABLE	P728=0	P728=1
	PLC REGISTER	MARK
Transfer-Inhibit	R155 bit 0	M49
M-done	R155 bit 1	M50

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.

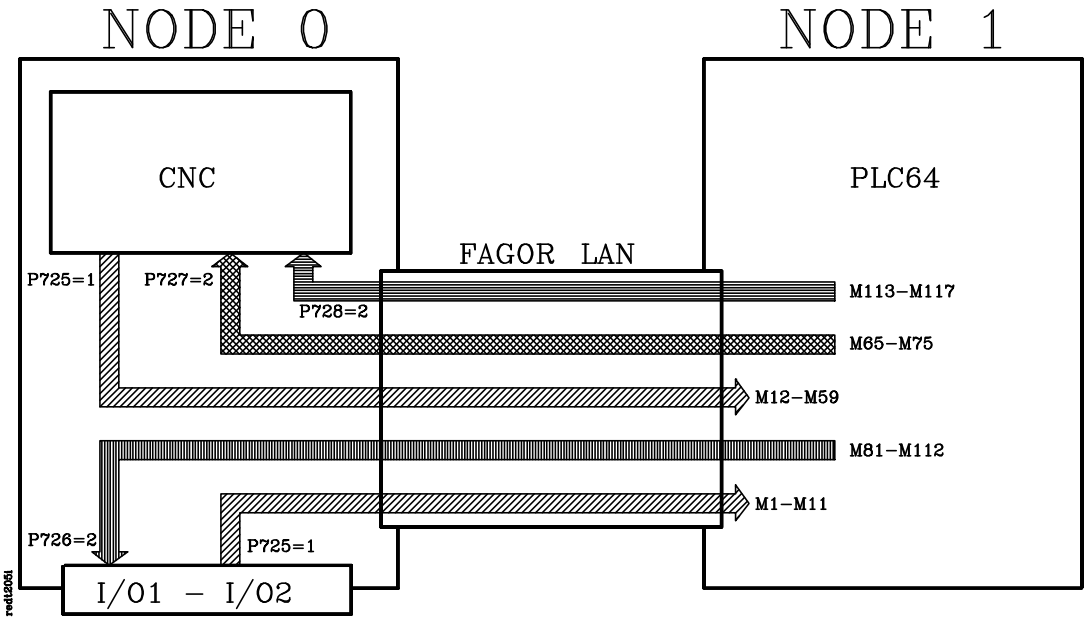
The CNC outputs are also to be handled by the PLC64 which will update the outputs at connectors I/O1 and I/O2.

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: P611(5)=1
P611(4)=0, P611(3)=0, P611(2)=0, P611(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:

P725=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.

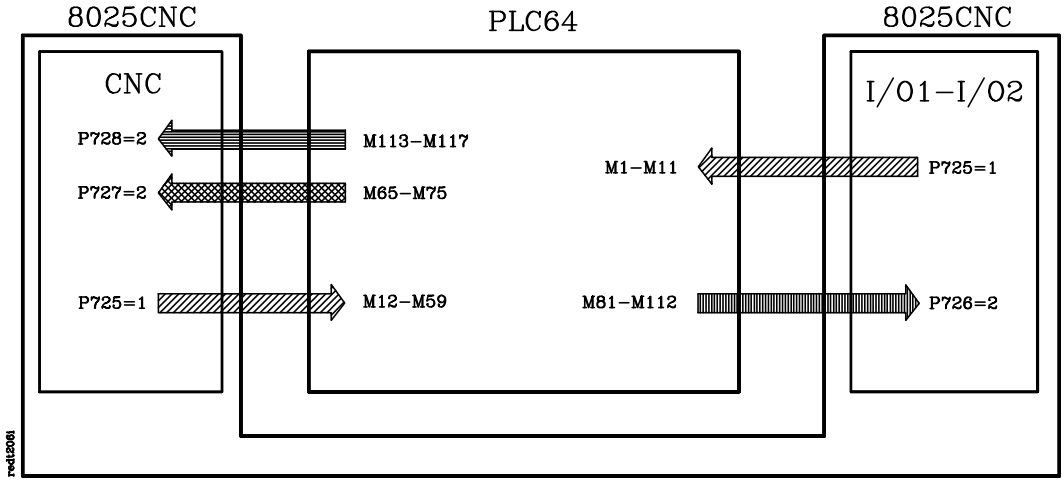
P726=2 The CNC updates the status of the outputs (connectors I/O1 and I/O2) with the value handed by the PLC64 on marks M87 thru M112.

P727=2 The CNC updates its internal data with the value handed by the PLC64 on marks M65 thru M75.

P728=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.

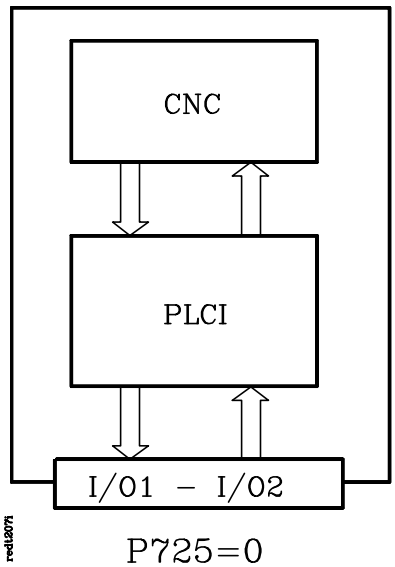
To do this, the following machine parameter must be set:

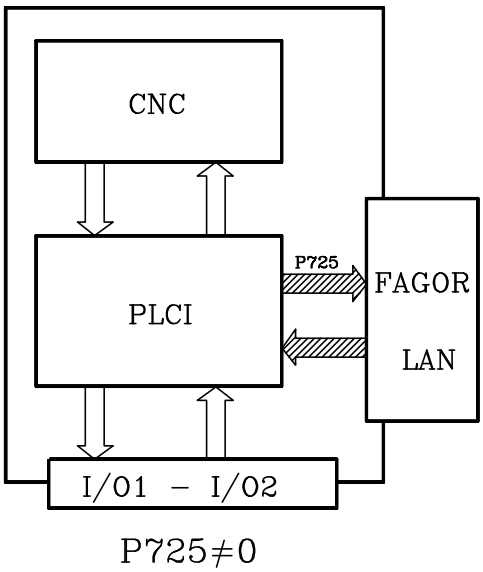
P725 Group of marks used by the PLCI to send its internal data

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 "P726", "P727" and "P728" 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: P611(5)=1
P611(4)=0, P611(3)=0, P611(2)=0, P611(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: P725=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 8025 T 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 8025T 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.

A PLC64, in order to access one of the internal CNC variables, must request it from the LAN indicating the node number it is requesting from and activating the corresponding mark.

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.

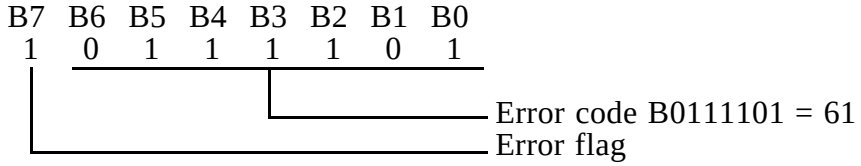
2.5.1 READING INTERNAL CNC VARIABLES

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
4th axis in motion (0=No 1=Yes)	B0 R101	M1901
Z axis in motion (0=No 1=Yes)	B1 R101	M1901
"C" or 3rd axis in motion (0=No 1=Yes)	B2 R101	M1901
X axis in motion (0=No 1=Yes)	B3 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 "C" or 3rd axis coordinate (in µm)	R104	M1904
Upper half of "C" or 3rd axis coordinate (in µm)	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 4th axis coordinate value (in µm)	R108	M1908
Upper half of 4th axis coordinate value (in µm)	R109	M1909
Program number	R112	M1912
Programmed F (in µm/min)	R113	M1913
Programmed S in rpm	R114	M1914
% Spindle speed override	B0-7 R116	M1916
% Feedrate override	B8-15 R116	M1916
Parameter P610. 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 P610 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.

P610(8)	P610(7)	P610(6)	P610(5)	P610(4)	P610(3)	P610(2)	P610(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
Inhibit Z axis (0=No 1=Yes)	B1 R150	M1950
Inhibit 3rd axis (0=No 1=Yes)	B2 R150	M1950
Inhibit X axis (0=No 1=Yes)	B3 R150	M1950
Inhibit 4th axis (0=No 1=Yes)	B4 R150	M1950
(#) Number of 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
Feed-Rate Override	B8-15 R152	M1952
Code of the key to be simulated	B0-7 R154	M1954
CNC keyboard enabled or disabled	B8-15 R154	M1954
Transfer Inhibit (active at 0V)	B0 R155	M1955
M-done (active at 0V)	B1 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:

"P616(5)=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.

"P616(5)=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 "P616(5)=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.

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 T CNC utilizes pin 15 of connector I/O1 as input for Feed-Hold, Transfer-Inhibit and M-done signals.

With machine parameter P616(7), it is possible to separate such treatment. If "P616(7)=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 P618(3) and acts as follows:

P618(3)=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).

P618(3)=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.

2.6 ACCESS TO THE REGISTERS OF A PLC64

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.

Note: 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.

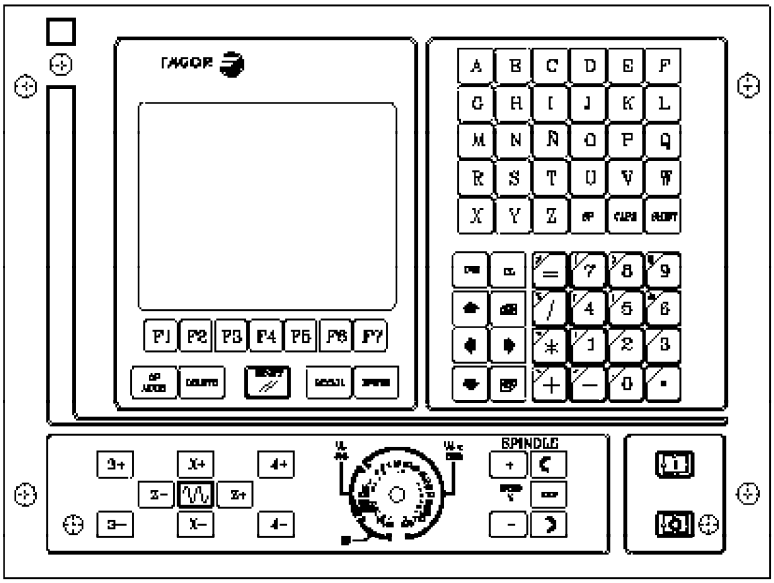
Example: The machine has two axes controlled by a 102 CNC and we would like to move them, one at a time, to X100 Y50.

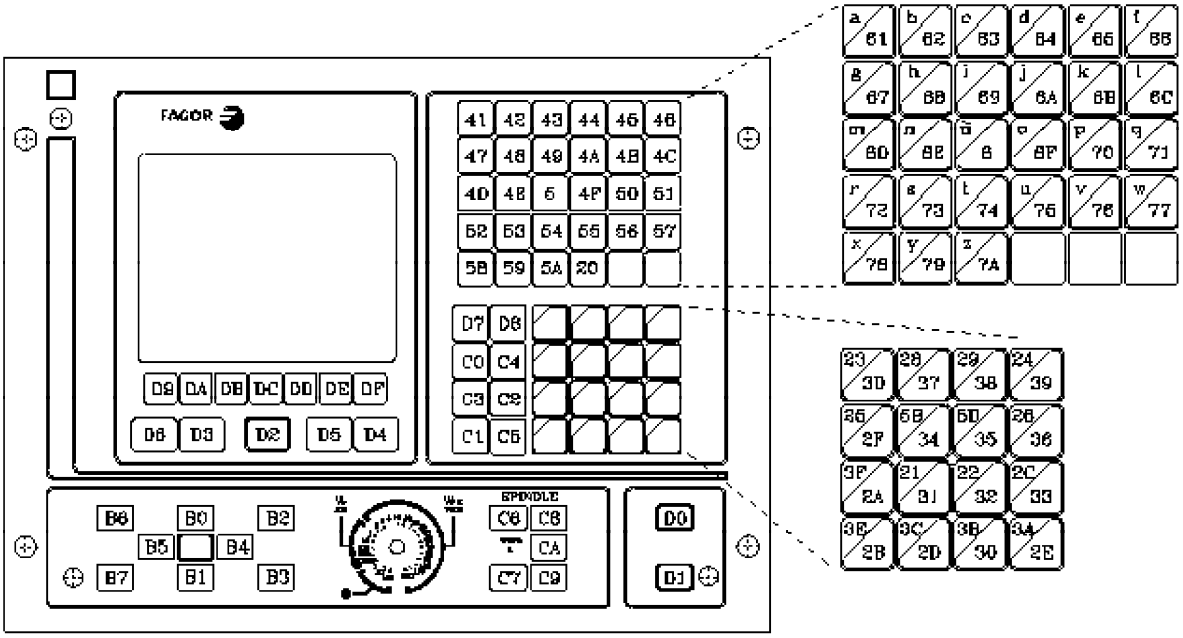
Also, the 8025 CNC must wait for those axes to reach position X100 Y50 before resuming the execution of its own program.

```
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

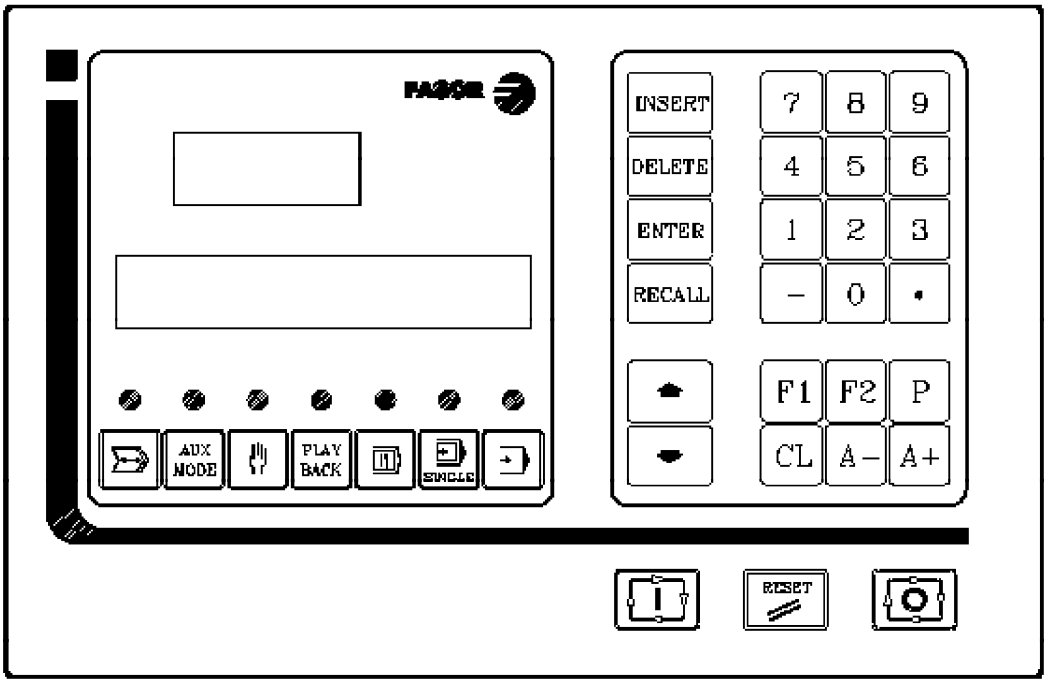


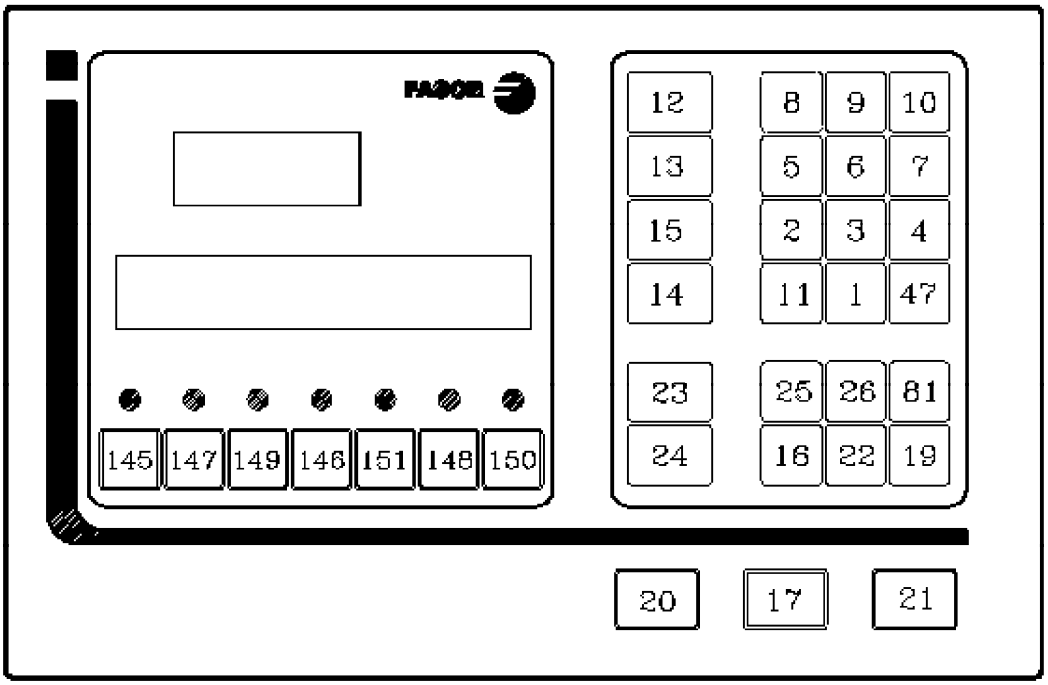


Note: The  key can be neither read nor simulated from the PLC64.

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