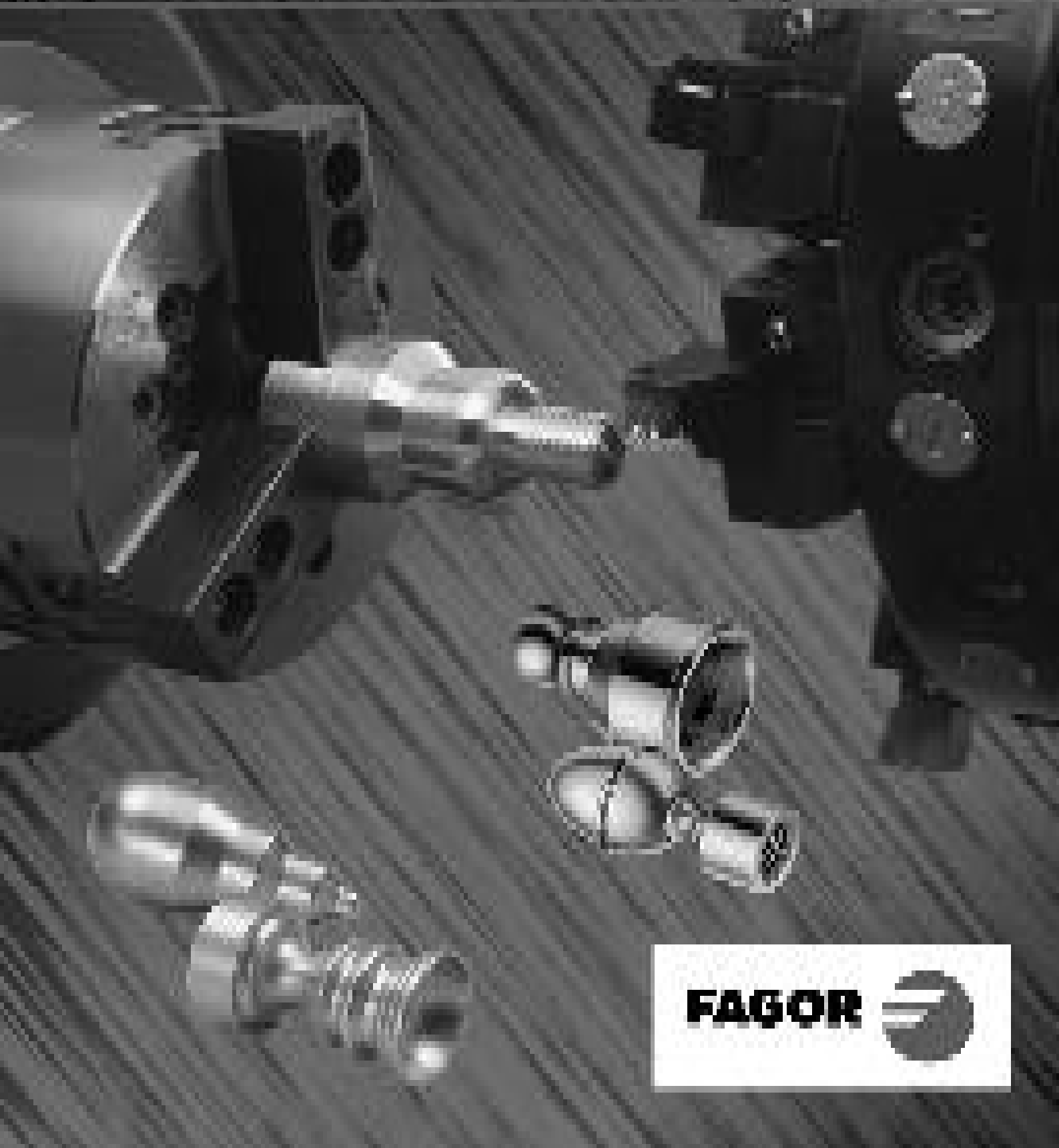
FAGOR AUTOMATION

CNC 800 T

New Features (Ref.0204in)



FAGOR

Version 5.2 (March 1995)

1. P621(4). DIVIDING FACTOR FOR ELECTRONIC HANDWHEEL FEEDBACK SIGNALS

Machine parameter P621(4) is used with P602(4) and P621(5) which indicate the multiplying factor for the electronic handwheel feedback signals for the 1st and 2nd axis respectively.

Machine parameter P627(1) indicates whether all handwheel feedback signals are to be divided or not.
P621(4)=0 They are not divided.
P621(4)=1 All handwheel feedback signals are divided by two.

Examples for the X axis so the CNC assumes 100 pulses/turn with 25, 50 and 100 line handwheels:

25 line Fagor handwheel:	P602(4)=0 and P621(4)=0	25 x 4 / 1 = 100 lines
50 line Fagor handwheel:	P602(4)=1 and P621(4)=0	50 x 2 / 1 = 100 lines
100 line Fagor handwheel:	P602(4)=1 and P621(4)=1	100 x 2 / 2 = 100 lines

Version 5.6 (June 1996)

1. JOG WITH MASTER HANDWHEEL

With this feature it is possible to jog the machine with the Master Handwheel once the path has been defined.

Requirements:

The control of the "Jog with Master Handwheel" is carried out with the Second Handwheel. Therefore, the machine must have two electronic handwheels and none mechanical.

Parameter setting:

Machine parameter "P622(6)" indicates whether this feature is being used or not.
P622(6) = 0 "Jog with Master handwheel" **is not** available.
P622(6) = 1 "Jog with Master handwheel" **is** available.

As stated above, the control of the "Jog with Master Handwheel" is carried out with the Second Handwheel. Therefore, the machine must have two electronic handwheels and none mechanical. This means that:
P621(7)=1 The machine does not use mechanical handwheels.
P622(3)=0 It uses two electronic handwheels.
P609(1)=0 The first handwheel is not a FAGOR 100P model.

The Master handwheel is connected via connector "A4". It admits both sine-wave and square-wave differential signals. This implies setting the following machine parameters as follows:
P621(6) Counting direction of the "Master Handwheel".
P621(3) Feedback units of the "Master Handwheel".
P621(1,2) Feedback resolution of the "Master Handwheel".
P621(5) Feedback multiplying factor for the "Master Handwheel".

Selection:

a) CNC Models: 800TI and 800TGI. From the PLCI.

Once all machine parameters have been set, PLCI output O39 must be used to enable or disable the "Jog with Master Handwheel" feature.

Parameter P622(6)	PLCI output O39	"Jog with Master Handwheel"
P622(6) = 0	-----	Feature not available
P622(6) = 1	O39 = 0	Feature disabled
P622(6) = 1	O39 = 1	Feature enabled

b) CNC Models: 800T and 800TG. Using pin 11 of connector "I/O 1".

Once all machine parameters have been set, the "Jog with Master Handwheel" input (pin 11 of I/O 1) must be used to enable or disable the "Jog with Master Handwheel" feature.

Parameter P622(6)	Pin 11 I/O1	"Jog with Master Handwheel"
P622(6) = 0	-----	Feature not available
P622(6) = 1	Pin 11 at 0Vdc	Feature disabled
P622(6) = 1	Pin 11 at 24Vdc	Feature enabled

Basic Operation. (P622(6)=1, O39=1)

a) When the machine is stopped.

Only the first handwheel is enabled, the second one (Master) is disabled.
Therefore, only the X axis may be jogged with the handwheels.

b) When the machine is running (CNC in Execution).

The axes do not start moving until the Master Handwheel is turned.

The axis feedrate depends on the turning speed of the Master Handwheel. If it stops, the axes also stop.

If the Master Handwheel is turned in the opposite direction, the axes also invert their moving direction (Return Function for one block only).

c) The "Jog with Master Handwheel" feature may be used with any type of execution, be it a cycle, an ISO-coded program, a Chamfer, etc.

Usually, with the CNC in execution, the first handwheel is disabled, except for the semi-automatic mode of the automatic operations: "Taper Turning" and "Rounding".

On both Semi-automatic operations, the Master Handwheel controls the feedrate of the tool path and the First Handwheel will move the X axis.

"Jog with Master Handwheel" feature disabled. (P622(6)=1, O39=0)

When this feature is disabled, PLC output O39 is set to "0" and the handwheels operate like until now (as on previous versions).

2. DYNAMIC GRAPHICS WHILE IN EXECUTION

Until now, with the 800T CNC, a part program could be simulated (verified) graphically before running it.

From now on, it is also possible to display dynamic graphics of the machining path while in execution.

Requirements:

This application requires a 800TG or 800TGI CNC model (G for graphics).

Operation:

When running an Automatic Operation, a Part program, the ISO-coded program in Automatic or Single Block mode, it is now possible to display the machining path dynamically in the execution stage.

To do this, once the execution has started, the following keys may be pressed:

[4] The CNC displays the graphics screen.
[3] The CNC shows the "Command, Actual and To-go" coordinates of the axes and, at the top of the screen, the values of the Arithmetic parameters.
[2] The CNC displays the Following Error (axis lag) in large characters.
[1] The CNC displays the actual axis position in large characters.
[0] The CNC returns to the standard display.

3. WORK ZONE / EXCLUSION ZONE

With this feature it is possible to select a predefined zone as work zone or exclusion zone from the PLCI.

Requirements:

This application requires an 800TI or 800TGI CNC model since one must use outputs O46 and O47 of the PLCI to set the zone as work zone or exclusion zone.

Parameter setting:

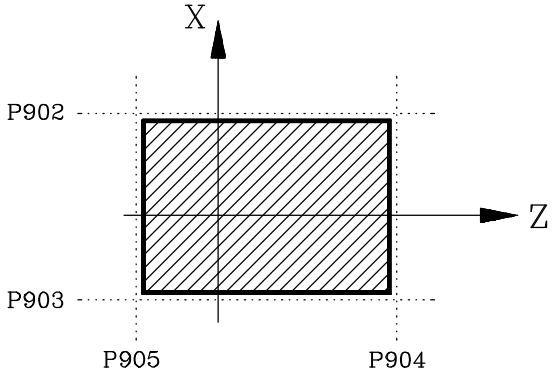
Machine parameter "P622(5)" indicates whether the CNC allows setting a work zone or an exclusion zone.

P622(5) = 0 This feature **is not** available.
P622(5) = 1 This feature **is** available.

When using this feature "P622(5)=1" the following machine parameters must also be set to define the zone to be considered either as work zone or exclusion zone.

P902 Most positive X coordinate
P903 Least positive X coordinate
P904 Most positive Z coordinate
P905 Least positive Z coordinate

The CNC must be turned off and back on in order for the new parameter values to be assumed.



Selection:

Once all machine parameters have been set, PLCI outputs O46 and O47 must be used to select the predefined zone either as a work zone or as an exclusion zone.

PLCI Output O46	PLCI Output O47	"Work Zone or Exclusion Zone" Feature
O46 = 0	O47 = 0	Feature disabled
O46 = 0	O47 = 1	Zone enabled as Work Zone (No-exit zone)
O46 = 1	O47 = 0	Zone enabled as Exclusion Zone (No-entry zone)
O46 = 1	O47 = 1	Feature disabled

Basic operation. "P622(5)=1"

On power-up, the CNC assumes the zone set by machine parameters "P902, P903, P904 and P905".

Nevertheless, the zone boundaries may be changed via part-program by allocating the new values to the following arithmetic parameters:

P206 Most positive X coordinate
P207 Least positive X coordinate
P208 Most positive Z coordinate
P209 Least positive Z coordinate

The CNC will then assume these new values; but it will not modify the actual settings of machine parameters "P902, P903, P904 and P905".

On the other hand, it must be kept in mind that, on power-up, the CNC will reset these zone boundaries to the values set by machine parameters.

As described earlier, this predefined zone may be enabled either as a work zone (no-exit) or as an exclusion zone (no-entry) from the PLCI by means of outputs O46 and O47.

When set as a work zone, the CNC acts as follows:

- The axes cannot be jogged **out of** this zone by using the jog keys or the handwheels.
- If attempted to do so during execution, the CNC will issue error 67: «X, Z Limit Error»

When set as an exclusion zone, the CNC acts as follows:

- The axes cannot be jogged **into** this zone by using the jog keys or the handwheels.
- If attempted to do so during execution, the CNC will issue error 67: «X, Z Limit Error»

4. MANUAL SPINDLE GEAR CHANGERS

Operation on previous versions

To manually change the spindle speed range (gear), machine parameter "P601(1)" had to be set to "0".

When the new selected spindle speed "S" involved a gear change, the CNC displayed a message indicating which range had to be selected.

The operator had to proceed as follows:

1st- Stop the spindle
2nd- Manually change gears
3rd- Restore spindle rotation
4th- Press **[ENTER]**

The CNC resumed program execution.

Operation on current and future versions

To manually change the spindle speed range (gear), machine parameter "P601(1)" must set to "0".

When the new selected spindle speed "S" involves a gear change, the CNC displays a message indicating which range has to be selected.

The operator must proceed as follows:

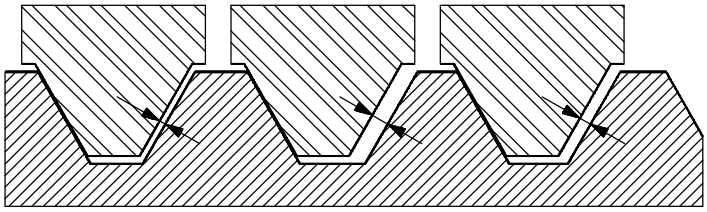
1st- Manually change gears
2nd- Press **[ENTER]**

The CNC restores spindle rotation and resumes program execution.

5. VARIABLE BACKLASH COMPENSATION

Until now, the 800T CNC allowed for a single leadscrew backlash compensation.

From now on, it is also possible to compensate for motion-reversal backlash depending on the particular backlash areas of the axes.



Requirements:

The leadscrew error compensation tables are now used for leadscrew error compensation and for this "Variable Backlash Compensation" (at the same time).

Parameter setting:

Machine parameters "P622(7)" and "P622(8)" indicate whether this feature is available or not.

P622(7)	=	0	Not available for the Z axis.
P622(7)	=	1	Available for the Z axis.
P622(8)	=	0	Not available for the X axis.
P622(8)	=	1	Available for the X axis.

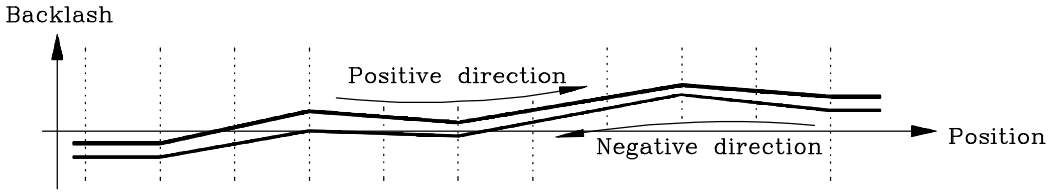
In order to use "Variable Backlash Compensation", regular leadscrew compensation must also be activated.

P605(2)=0	X axis Leadscrew error compensation (0= No, 1= Yes)
P605(1)=0	Z axis Leadscrew error compensation (0= No, 1= Yes)

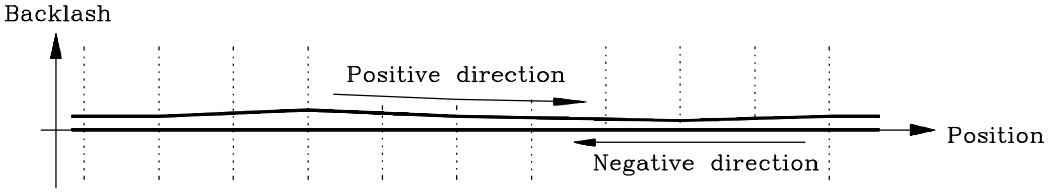
Operation:

The first 15 points of the table are for the positive direction and the other 15 for the negative direction.

When compensating for leadscrew error, the amount of backlash is the difference between both graphs.



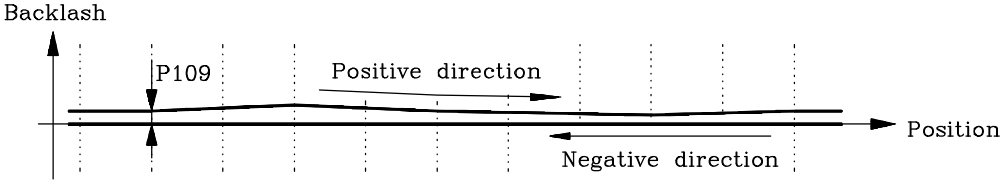
When leadscrew error compensation is not to be used, all the values of one of the tables must be set to "0"; thus, the other graph will correspond to the leadscrew backlash.

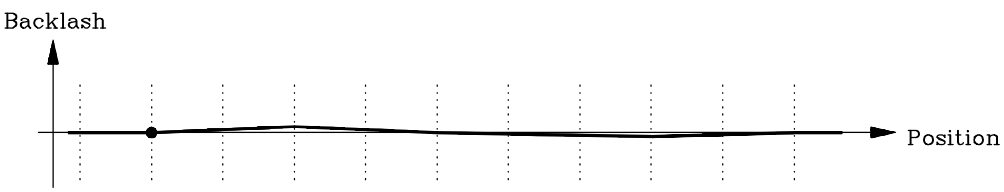


Notes:

- Both graphs must meet all the requirements of the leadscrew error compensation tables.
- One of these requirements states that the Machine Reference Zero (home) must be assigned an error of "0".
- If the leadscrew has some backlash at this Machine Reference Zero point, that amount of backlash must be allocated to machine parameter P109 or P309 (Backlash for the X axis or Z axis) and all the remaining points of the table must be offset by that amount.

  Example:





Operation:

When using "Variable Backlash Compensation", the CNC operates with leadscrew error compensation and, therefore, it applies at all times the backlash compensation set in the table for that point and in the machining direction.

When the axis movement is reversed, the CNC swaps graphs restoring or applying the amount of backlash compensation and direction corresponding to that point.

Version 5.7 (July 1996)

1. WORK ZONE / EXCLUSION ZONE DETECTION

When using this feature, the CNC prevents the axes from exiting or entering this zone while jogging them with either the jog keys or the electronic handwheel.

The operator might suspect a malfunction since the CNC does not issue any message. From this version on, in these cases, the CNC will behave as follows:

* When the zone has been set as a Work Zone, the CNC will set PLCI input I46 high when trying to exit the selected zone.

* When the zone has been set as a Work Zone, the CNC will set PLCI input I46 high when trying to enter the selected zone.

2. RESUME EXECUTION AT MID-PROGRAM

If while a part, the program is interrupted (due to a power failure, etc.), it is now possible to resume execution from the interrupted program on. This way, there is no longer need to repeat the whole program, thus saving considerable amount of time.

To resume program execution, follow these steps:

1st Select the DRO mode, the one appearing on CNC power-up after the "General Test Passed".
In this mode, no cycle appears selected.

2nd Press **[RECALL]** to open the part-programs window.

3rd Select the part that was running. Use the up and down arrow keys to position over the desired part program and press **[RECALL]**.

4th Use the up and down arrow keys to select the operation being interrupted and press 

The CNC will executed the selected operation and it will resume the part-program running it to the end.

Version 6.1 (January 1997)

1. NEW LANGUAGES (Taiwanese and Portuguese)

Machine parameter P99 P99 = 5 Portuguese P99 = 6 Taiwanese

2. MODIFICATIONS ON THE OPERATION WITH A MASTER HANDWHEEL

The operation with the master handwheel is now as follows:

a) When the machine is stopped.
Only the first handwheel is enabled, the second one (master) does not work.
Therefore, only the X axis can be jogged with a handwheel.

b) When the machine is running (CNC in Execution).
Only the Master handwheel is enabled, the first handwheel does not work.

The axes start moving when turning the Master Handwheel.

The feedrate of the axes depend on the turning speed of the Master Handwheel
When the handwheel stops, the machine also stops.

When the Master Handwheel is turned in the opposite direction, the CNC also reverses the moving direction (Retrace Function of a single block).

c) Semiautomatic Rounding Operation
The Semiautomatic Rounding operation starts when turning the Master Handwheel.

When stopping the Master Handwheel, the execution is interrupted.
When turning the Master handwheel again, execution is resumed. The turning direction of the handwheel cannot be changed.

When the operation is over, the CNC ignores the turning of the Master Handwheel for 1.4 seconds. Thus preventing another operation from being started.
After this time, when the Master Handwheel is turned, the CNC starts executing a new operation in the indicated direction.

d) Semiautomatic Taper Turning Operation

The Semiautomatic Taper Turning Operation starts when turning the Master Handwheel.

When stopping the Master Handwheel, the execution is interrupted.
When turning the Master handwheel again, execution is resumed.

When turning the Master Handwheel in the opposite direction, the operation is over. A new turn of the Master Handwheel in any direction implies the execution of a new operation in the indicated direction.

3. SOFTWARE VERSION OF THE CNC

From this version on, when accessing the EPROM checksum screen,
[Auxiliary Modes] [Special Modes] [8]

The CNC will show the checksum of each EPROM and the Software version of the CNC. For example: Version 6.1

Version 6.4 (May 1997)

1. TOOL CHANGE INDICATOR FOR THE PLC (I97)

On machines with a manual tool changer, when the CNC detects that the tool must be changed, it interrupts the execution and it displays a message for the operator to proceed with the tool change.

Certain precautions must be taken sometimes when changing tools. Those conditions must be handled by the PLC.

Therefore, from this version on, when the CNC displays the tool change warning message, it also activates the PLC input I97 and it cancels it when the message is removed.

Version 6.6 (November 1997)

1. HANDLING FEEDBACK SYSTEMS WITH CODED Io (semi-absolute)

Machine parameters

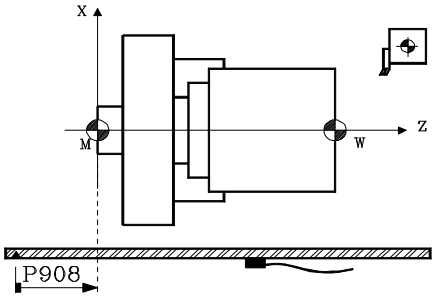
P608(5), P608(8) Type of Home marker signal of the feedback system. X, Z axes. (0 = normal "Io", 1 = coded "Io")
P608(3), P608(6) Period of the coded Io signal. X, Z signal. (0 = 20mm Period, 1 = 100 mm Period)
P608(4), P608(7) Increasing Io sequence with positive or negative count. X, Z axes
(0 = Increasing Io with positive count, 1 = increasing Io with negative count)

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 Scale offset or Home position (Machine Reference: M) with respect to the Scale "zero" point. X, Z axes.

Linear transducers (scales) with coded Io have a graduated scale with their own Scale "zero" point. Therefore, a 20 mm or 100 mm move is enough to know the axis position with respect to the Scale "zero" point.



Reference point.

When the feedback system has a coded Io, this point is only used when leadscrew error compensation is needed. The leadscrew error on this point must be "0".

Scale offset setting

The scale offset must be adjusted on one axis at a time. The following procedure is recommended:

* Indicate by parameters "P600(7) & P600(6)" the up or down flank of the marker pulse (Io) of the feedback system.
* Indicate by parameters "P618(8) & P618(7)" the home searching direction for the axes.
* Set parameters "P807 and P808" with the home searching feedrate for the axes.
* Set parameters " P908 and P909" to "0" (scale offset).
  Position the axis at the right position and execute the home search command for that axis..

  [X] or [Z], [up arrow] keys and 

When done with the home search, the CNC will display the axis position with respect to the scale's Zero point.

* After moving the axis to Machine Reference Zero point (home) or up to a known position with respect to home, Jot down the position reading of the CNC for that known position.

  The value to allocate to the machine parameter setting the scale's offset must be calculated with the following formula:
  Value = CNC reading at that point - Home coordinate of that point

  Example for the X axis: If the known position is physically located at 230 mm from Machine Reference Zero (home) and the CNC shows that its position is 423.5 mm, the scale offset will be:
  Machine parameter P908 = 423.5 - 230 = 193.5 mm.

* After setting the machine parameter with this value, press the [RESET] key so that value is assumed by the CNC.
* A new home search must be carried out for the proper values to be assumed for that axis.

2. THREADING WITH CONSTANT PENETRATION PASSES

From this version on, the penetration of each pass will depend on the sign assigned to parameter **D**

When **D** positive, the penetration of each pass depends on the corresponding pass (**D**$\sqrt{n}$)

With **D** negative, the penetration passes remain constant with the absolute value of parameter **D**

3. GENERATING AN ISO-CODED PROGRAM

With this CNC, the ISO code (low level) for an operation or a part-program may be generated.

To use this feature, machine parameter "P623(2)" must be set to "1".

This ISO program always has the number: 99996 and can be stored either at the CNC or at a PC.

Program 99996 is a special user program in ISO code and can be:
Generated from an operation or a part-program.
Edited at the CNC itself via menu option: "Auxiliary Modes - Edit program 99996"
Loaded into the CNC after being generated at a PC.

Generating the ISO program (99996) at the CNC.

This CNC has 7 K of memory space to store program 99996. If the generated program is larger than that, the CNC will issue the relevant error message.

To generate program 99996, proceed as follows:
* If it is an operation, select or define the desired operation.
* If it is a part-program, select the desired one in the part-program directory and place the cursor on its header ("PART 01435". A listing of the operations it consists of must appear).
* Press the keystroke sequence: [AUX] [7]. The CNC will show the graphic simulation screen.
* Press  . The CNC starts simulating the part and generating its ISO-coded program 99996.
* When done with the simulation, program 99996 stored in CNC memory will contain all simulated blocks in ISO code.

Generating the ISO program (99996) at a PC

Usually, the 99996 program generated from a part-program exceeds the available memory space of the CNC.

By using "DNC30", this program may be generated at a PC.

To do this, proceed as follows:
* Activate DNC communications and execute the DNC30 program at the PC.
* Select at the PC the menu option: "Program Management - Receive Digitizing".
* At the CNC, select the operation or place the cursor on the part-program header ("PART 01435"). A listing of the operations it consists of must appear).
* Press [AUX] [8]. The CNC will display the graphic simulation screen.
* Press  . The CNC starts simulating the part and generating program 99996.
* When done with the simulation, the 99996 program generated at the PC will contain all the blocks simulated by the CNC in ISO code.
  This program can be executed at the CNC through the menu option: "Execute infinite program" of the DNC30.

4. MACHINE SAFETY REGULATION

This CNC offers the following features to comply with machine safety regulations.

Enabling of the CYCLE START key  *from the PLC*

This feature is available when machine parameter "P619(7)=1"

PLC output O25 indicates whether the CYCLE START key is enabled (=1) or not (=0)

Axes movements affected by Feed-Hold. (It was already available)

Feed-Hold input, pin 15 of connector I/O 1, must be normally high.

If while moving the axes, the Feed-Hold input is brought low, the CNC keeps the spindle turning and stops the axes with 0V or velocity command (analog signal) and keeping their enables ON.

When this signal is brought back up, the CNC will resume the movement of the axes.

Axes jogging feedrate limited by PLC.

This feature is available when machine parameter "P619(7)=1"

When activating PLC output O26, the CNC assumes the feedrate set by machine parameter "P812"

Handwheel managed by the PLC.

Machine parameter "P623(3)" indicates whether the axes movements with handwheels are affected by Feed-Hold (=1) or not (=0)

Machine parameter "P622(1)" indicates whether the multiplying factor indicated by the MFO switch position is applied (=0) or the one indicated by the PLC outputs O44 and O45 (=1) (already available)

Spindle control from the PLC.

This feature is available when "P619(7)=1"

Output O27 =1 "tells" the CNC to apply the spindle analog voltage set by the PLC. The value of this analog signal is set at register R156 and sent to the CNC by mark M1956.

R156= 0000 1111 1111 1111 => +	10V.	R156= 0001 1111 1111 1111 => -	10V.
R156= 0000 0111 1111 1111 => +	5V.	R156= 0001 0111 1111 1111 => -	5V.
R156= 0000 0011 1111 1111 => +	2,5V.	R156= 0001 0011 1111 1111 => -	2,5V.
R156= 0000 0000 0000 0000 => +	0V.	R156= 0001 0000 0000 0000 => -	0V.

Also, PLC output O43, lets you control the rotation of the spindle. (Already available).
It must be normally low.
If it is brought up, the CNC stops the spindle.
When it is brought back up, the CNC restarts the spindle.

Information for the PLC on the status of the machine reference (home) search

I88 Home search in progress.
I100 X axis home search done.
I101 Z axis home search done.

Additional CNC information for the PLC

R120 The Lower half of this register indicates the code pressed.
This value is maintained for 200 milliseconds unless another key is pressed before then.
This register may be canceled from the PLC after being processed.

R121 bit 1 Indicates that the Turning operation is selected (=1)
bit 2 Indicates that the Facing operation is selected (=1)
bit 3 Indicates that the Taper Turning operation is selected (=1)
bit 4 Indicates that the Rounding operation is selected (=1)
bit 5 Indicates that the Threading operation is selected (=1)
bit 6 Indicates that the Grooving operation is selected (=1)
bit 7 Indicates that the Profiling operation is selected (=1)
bit 8 Indicates that the Auxiliary Modes option is selected (=1)
bit 9 Indicates that the Tool Calibration option is selected (=1)
bit 10 Indicates that the Multiple Drilling operation is selected (=1)
bit 11 Indicates that the Simple Drilling / Tapping operation is selected (=1)
bit 12 Indicates that the Slot milling (keyway) operation is selected (=1)
bit 13 Indicates that the Tool Inspection mode is selected (=1)
bit 14 Indicates that the Graphic Simulation mode is selected (=1)
bit 16 Indicates that the mode for the following cycle parameters: "Finishing pass, finishing feedrate, finishing tool and safety distances on X and Z " is selected (=1)

Version 6.8 (March 1998)

1. NEW LANGUAGES (SWEDISH AND NORWEGIAN)

The languages that can be selected with machine parameter P99 are:

Spanish	(P99=0)	German	(P99=1)	English	(P99=2)	French	(P99=3)	Italian ..... (P99=4)
Portuguese ..	(P99=5)	Taiwanese ...	(P99=6)	Swedish	(P99=7)	Norwegian ..	(P99=8)	

2. 1000 LINE ENCODER AS 1250 LINE AS ENCODER

This feature permits the CNC adapt a 1000 line encoder to be used as 1250 line encoder.

P623(7) Adapts the X axis feedback encoder (0=No, 1=Yes)
P623(8) Adapts the Z axis feedback encoder (0=No, 1=Yes)

A typical case: Having a 1000 line for a 5 mm pitch ballscrew .

The calculations necessary to set the axis resolution will be made with the selected pulses (1000 or 1250)

3. CROSS COMPENSATION

Cross compensation is used for compensating the measuring error suffered by the X axis when moving the Z axis.

P623(6) Cross compensation applied on to the X axis (0=No, 1=Yes)

When using cross compensation, no leadscrew compensation may be applied on the X axis (only on to the Z axis) since its corresponding table is being used for cross compensation with the following values:

P00 = X: ?????.??? P01 = DX: ????.???

in order to properly apply cross compensation, set: P605(2)=1 and P623(6)=1.

Note: The cross compensation table must meet the same requirements as those for the leadscrew error compensation. See section 3.8.4 of the installation manual.

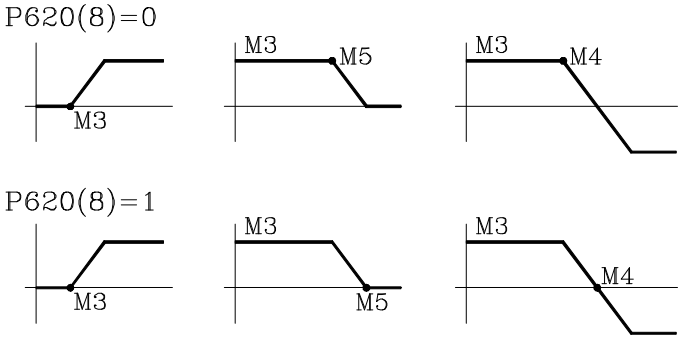
4. PLCI. INPUT I104

When the Feedrate Override Switch on the operator panel is set on one of the handwheel positions (x1, x10, x100), input I104 is set to "1".

Version 6.9 (February 1999)

1. NEW MACHINE PARAMETER ASSOCIATED WITH M FUNCTIONS

Machine parameter "P620(8)" indicates when the M functions M3, M4 and M5 are to be output during the acceleration and deceleration of the spindle.



2. CANCEL TOOL OFFSET DURING TOOL CHANGE

From this version on, it is possible to execute, within a subroutine associated with the tool, a "T.0" type block to cancel the tool offset. This allows to move to a specific position without having to do cumbersome calculations.

It is only possible to cancel (T.0) or change (T.xx) the tool offset. The tool cannot be changed (Txx.xx) within the subroutine associated with the tool.

3. X1 FACTOR FOR FEEDBACK PULSES

Machine parameters P620(5) and P620(6) are used together with P602(6) and P602(5) which indicate the multiplying factor of the feedback pulses for the X and Z axes respectively.

They indicate whether x1 factor is applied to the feedback pulses (=1) or not (=0).

P620(5)=0 and P620(6)=0 the x1 factor is NOT applied
P620(5)=1 and P620(6)=1 the x1 factor is applied

Example: We would like to obtain a 0.01 mm resolution with a square signal encoder mounted on the X axis having a leadscrew pitch of 5mm/turn.

Number of Encoder pulses = leadscrew pitch / (multiplying factor x Resolution)

Con P602(6)=0 and P620(5)=0	x4 multiplying factor	Number of pulses = 125
Con P602(6)=1 and P620(5)=0	x2 multiplying factor	Number of pulses = 250
Con P602(6)=0 and P620(5)=1	x2 multiplying factor	Number of pulses = 250
Con P602(6)=1 and P620(5)=1	x1 multiplying factor	Number of pulses = 500

Version 6.10 (March 2002)

1. FEEDBACK FACTOR.

The resolution of the axis is determined by the leadscrew pitch and the number of pulses of the encoder mounted on the motor. Sometimes, the resolution resulting from the available leadscrew / encoder combination does not match any of the resolution values allowed for the machine parameters (1, 2, 5, 10 microns or ten-thousandths of an inch).

Example: With a 6 mm/turn leadscrew pitch and a 2500 line encoder, the resulting resolution values are:
Resolution = Leadscrew pitch / (Nr of encoder pulses x multiplying factor).
With x1 multiplying factor 2.4 micron resolution
With x2 multiplying factor 1.2 micron resolution
With a x4 multiplying factor 0.6 micron resolution

A new axis machine parameter is now available to solve these cases and it is referred to as Feedback Factor in order to adapt the resulting resolution to the existing setup.
P819 Feedback factor for the X axis P820 Feedback factor for the Y axis P821 Feedback factor for the Z axis
Values between 0 and 65534, a "0" value means that this feature is not being used.

Use the following formula to calculate the "Feedback factor":
Feedback Factor = (Gear ratio x Leadscrew Pitch / Encoder pulses) x 8.192

Examples:						
	Gear ratio	1	1	2	1	
	Leadscrew Pitch	4000	6000	6000	8000	(microns)
	Encoder	2500	2500	2500	2500	(pulses/turn)
	Feedback factor	13,107.2	19,660.8	39,321.6	26,214.4	

The machine parameters only admit integers, but the Feedback Factor sometimes may have decimals. In those cases, set the machine parameter to the integer part of that value and use the leadscrew error compensation table to make up for the decimal part.

The values for this table are calculated using the following formula:
Leadscrew position = Leadscrew error (microns) x Integer portion of feedback factor / Decimal portion of feedback factor

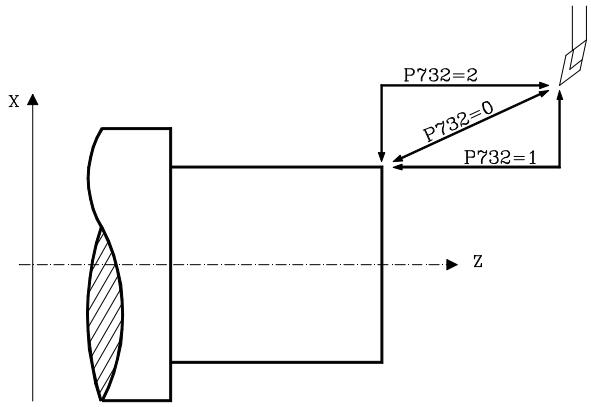
In this case: Gear ratio = 1 Pitch = 6000 Encoder = 2500
Feedback factor = 19,660.8 Machine parameter = 19660
For leadscrew error of 20 microns Leadscrew position = 20 x 19,660 / 0.8 = 491,520
The following table is obtained by using this calculation.

Leadscrew position		Amount of error at that position	
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

2. PART APPROACHING MOVEMENTS

From this version on, there is a new machine parameter to specify the part approaching and leaving movements.

P732=0 Like until now, interpolated movement
P732=1 Paraxial movements (one axis at a time).
Approach: X - Z Exit: Z - X
P732=2 Paraxial movements (one axis at a time).
Approach: Z - X Exit: X - Z



Headquarters (SPAIN): **Fagor Automation S. Coop.**
Bº San Andrés s/n, Apdo. 144
E-20500 Arrasate - Mondragón
Tel: +34-943-719200/039800
Fax: +34- 943-791712
+34-943-771118 (Service Dept.)
www.fagorautomation.com
E-mail: info@fagorautomation.es

FAGOR 800T CNC
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 800T CNC product.

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 800T 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.

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

Chapter 3 "Auxiliary Functions" shows how to access special operating modes.

To adapt the CNC to the machine, set the CNC machine parameters. We suggest that you consult chapters 4, 5, 6 and the appendices related to the machine parameters listed in numerical order.

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

When explaining each parameter in detail, chapters 4, 5 and 6, they sometimes refer to chapter 7 (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.

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 the manual without prior notice.

INDEX

Section Page

Chapter 2. POWER AND MACHINE INTERFACE

Chapter 3. AUXILIARY FUNCTIONS

Section Page

Chapter 4. MACHINE PARAMETERS

Chapter 5. MACHINE PARAMETERS FOR THE AXES

Chapter 6. SPINDLE MACHINE PARAMETERS

Section Page

Chapter 7. CONCEPTS

APPENDICES

ERROR CODES

COMPARISON TABLE FOR FAGOR 800T CNC MODELS

AVAILABLE 800T CNC MODELS

Compact model with 8" amber CRT.

Modular model with 9" amber Monitor.

Consisting of Central Unit, Monitor and Keyboard.

Modular model with 14" Color Monitor

Consisting of Central Unit, Monitor and Keyboard.

TECHNICAL DESCRIPTION

	800-T	800-TI	800-TG	800-TGI
X, Z axes control	I	I	I	I
Spindle control	I	I	I	I
Spindle in RPM	I	I	I	I
Constant Surface Speed (CSS)	I	I	I	I
Spindle Orientation	I	I	I	I
Tools	32	32	32	32
Tool Compensation	I	I	I	I
Live Tool	I	I	I	I
Electronic Handwheels	2	2	2	2
RS 232C Communications	I	I	I	I
Integrated PLC (PLCI)		I		I
ISO-coded program editing (P99996)	I	I	I	I
Execution of ISO-coded program (P99996)	I	I	I	I
Graphics			I	I

NEW FEATURES AND MODIFICATIONS

Date: **April 1993** ***Software Version:*** **2.1 and newer**

FEATURE	**AFFECTED MANUAL AND SECTION**	
Rapid jog  depending on position of Feedrate Override Switch	Operating Manual	Section 2.3.1
Tool for the finishing pass	Installation Manual Operating Manual	Section 3.5 Section 3.5
Handwheel movement limited to maximum allowed F	Operating Manual	Section 2.3.3
Control of software travel limits when using a handwheel		
Display format for S	Installation Manual	Section 6
Possibility to activate/deactivate outputs O1, O2, O3 after interrupting the program		
Automatic operation "Profile Rounding"	Operating Manual	Section 5.5.3
Profiles	Operating Manual	Chapter 6

Date: **October 1993**	***Software Version:*** **3.1 and newer**	
FEATURE	**AFFECTED MANUAL AND SECTION**	
Spindle acc./dec.	Operating Manual	Chapter 6
RPM Limitation when operating in CSS	Operating Manual	Section 4.3.1
Spindle orientation	Installation Manual Operating Manual	Section 6.4.1 Section 4.8
Live tool	Installation Manual Operating Manual	Section 5.9 Section 2.3
Automatic operation "Simple Drilling"	Operating Manual	Section 5.8
Automatic operation "Multiple Drilling"	Operating Manual	Section 5.9

Date: **December 1993** ***Software Version:*** **3.2 and newer**

FEATURES	**AFFECTED MANUAL AND SECTION**	
Assign a 5-digit number to the part program	Operating Manual	Chapter 7
Save part programs out to a peripheral	Operating Manual	Section 7.7
Automatic operation "Slot milling"	Operating Manual	Section 5.10
Delay before opening the positioning loop	Installation Manual	Section 4.3.2
"Special modes" accessing password	Installation Manual	Section 3.7
Handwheel inactive when Feedrate Override Switch out of handwheel positions	Installation Manual	Section 4.3.2

Date: **July 1994** ***Software Version:*** **4.1 and newer**

FEATURE	**AFFECTED MANUAL AND SECTION**	
Linear and Bell-shaped spindle acc./dec.	Installation Manual	Section 5.8
Profile with/without corner rounding.	Operating Manual	Section 6.2
Threading operation also with thread exit.	Operating Manual	Section 5.6.2
Rapid jog  at 200% or depending on the position of the Feedrate Override Switch.	Installation Manual Operating Manual	Section 4.3.3 Section 2.3.1
Tool inspection	Installation Manual Operating Manual Operating Manual	Section 3.4.3 Section 3.4.3 Section 5.1.3
Execution of program 99996	Installation Manual Operating Manual	Section 3.11 Section 3.10

Date: **January 1995** ***Software version:*** **5.1 and newer**

FEATURE	**AFFECTED MANUAL AND SECTION**	
M3/M44 confirmation by detecting feedback reversal	Installation Manual	Section 6.4
JOG movements also in mm/rev		
Handwheel governed by the PLCI	Installation Manual	Section 4.3.2
Spindle inhibit from PLCI	PLCI Manual	
Clear all arithmetic parameter contents setting them to "0".	Installation Manual Operating Manual	Section 3.10 Section 3.9 & 7.9
Automatic rounding operation (Cycle level) with angle other than 90°.	Operating Manual	Section 5.5.2
Automatic grooving operation on the face of the part and finishing pass.	Operating Manual	Section 5.7
Automatic profile rounding operation by pattern repeat of profile or roughing.	Operating Manual	Section 5.5.3
Approach point in profile rounding operation (modification).	Operating Manual	Section 5.5.3
Automatic Profile execution, Cycle Level, by pattern repeat or roughing.	Operating Manual	Section 6.2
Approach point in automatic Profile execution (modification).	Operating Manual	Section 6.2
Automatic tapping operation.	Operating Manual	Section 5.8
M20 at the end of part-program execution.	Installation Manual	Section 3.8.3.1
Graphic simulation	Operating Manual	Section 5.1.3
Execution / Simulation of program P99996 (ISO-coded user program)	Installation Manual Operating Manual	Section 3.11 Section 3.10
Automatic or Single-block execution of P99996	Operating Manual	Section 3.10
Editing of program P99996	Installation Manual Operating Manual Programming Manual	Section 3.12 Section 3.11
ISO-coded user program P99994 to store subroutines	Programming Manual	Chapter 9
Subroutine associated to the execution of a tool (only when executing program P99996)	Installation Manual Programming Manual	Section 4.3.4
ISO codes of the 800T CNC	Programming Manual	

Date: **March 1995** ***Software version:*** **5.2 and newer**

FEATURE	AFFECTED MANUAL AND SECTION	
Editing of program P99996 in all models.		
When interrupting the execution, the following keys are enabled: spindle, coolant, O1, O2, O3 and TOOL.	Installation Manual Operating Manual Operating Manual Operating Manual	Section 3.11 Section 3.10 Section 5.1.4 Section 7.5
Incremental JOG movements taking current work units (radius or diameter) into account.	Installation Manual	Section 4.3.3
ISO programming. New functions: G47, G48 (single block treatment).	Programming Manual	Section 6.7
ISO programming. New function: G86 (Longitudinal threadcutting canned cycle).	Programming Manual	Section 8.17
Request from the PLCI for real spindle rpm.	PLCI Manual	

Date: **November 1995** ***Software version:*** **5.5 and newer**

FEATURE	AFFECTED MANUAL AND SECTION	
Tool offset modification while in execution.	Operating Manual	Section 3.4.4
Operation with a single electronic handwheel.	Installation Manual Installation Manual	Section 4.3.2 Section 7.5
Actual "S" speed reading from the PLCI.	PLCI Manual	

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 800T 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

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.

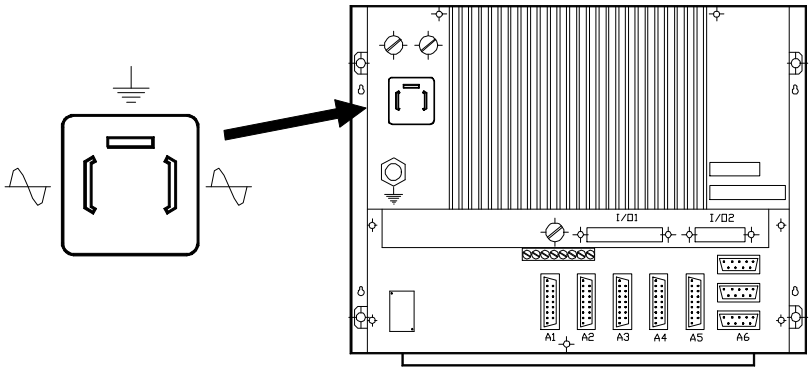
5.- Seal the cardboard box with packing tape or industrial staples.

ADDITIONAL REMARKS

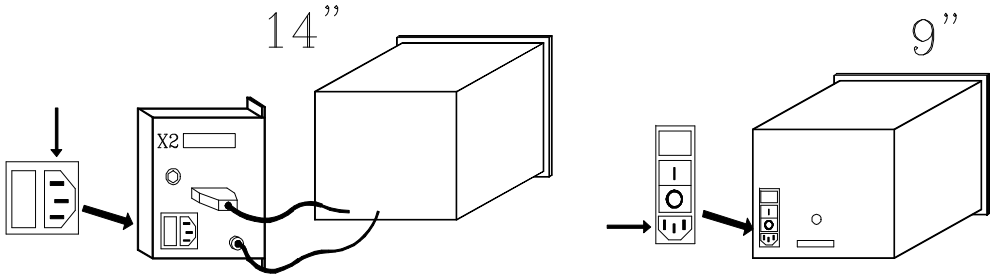
* Mount the CNC away from coolants, chemical products, blows, etc. which could damage it.

* 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)



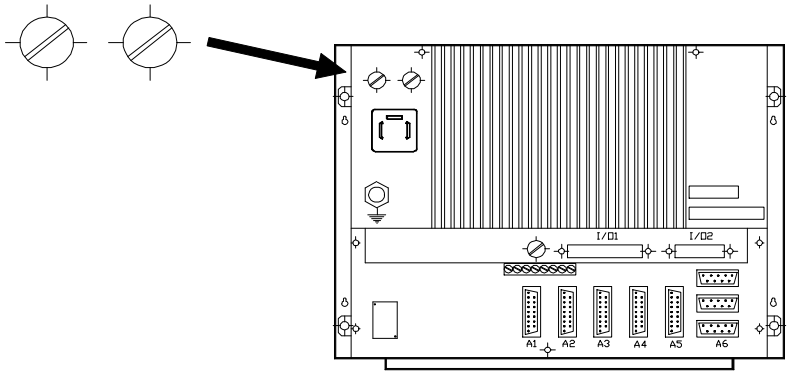
* To prevent electrical shock at the Monitor, use the proper mains AC connector at the Power Supply Module. Use 3-wire power cables (one for ground connection)



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

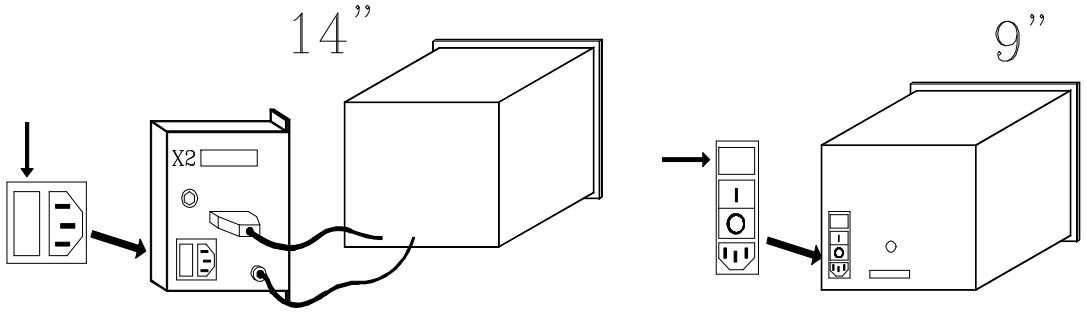
  Central Unit

  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 800T CNC

800T CNC OEM Manual Is directed to the machine builder or person in charge of installing and starting up the CNC.

It has the Installation manual inside. Sometimes, it may contain an additional manual describing New Software Features recently implemented.

800T 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 sections:

Chapter 7 Concepts.

- Feedback systems, resolution
- Adjustment of the axes and their gains.
- Reference Systems: Reference systems, search and setting
- Software travel limits for the axes.
- Acceleration / deceleration.
- Spindle: speed control and range change.
- Tools and tool magazine
- "Feed Hold" and "M-done" signal processing (treatment)
- Auxiliary M, S, T function transfer

Appendix

- A CNC technical characteristics.
- B Enclosures.
- C CNC inputs and outputs.
- D 2-digit BCD coded spindle "S" output
- E Machine parameter summary chart
- F Sequential machine parameter listing
- G Machine parameter setting chart
- H Maintenance

Error Codes

1. 800T CNC CONFIGURATION

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 INTRODUCTION

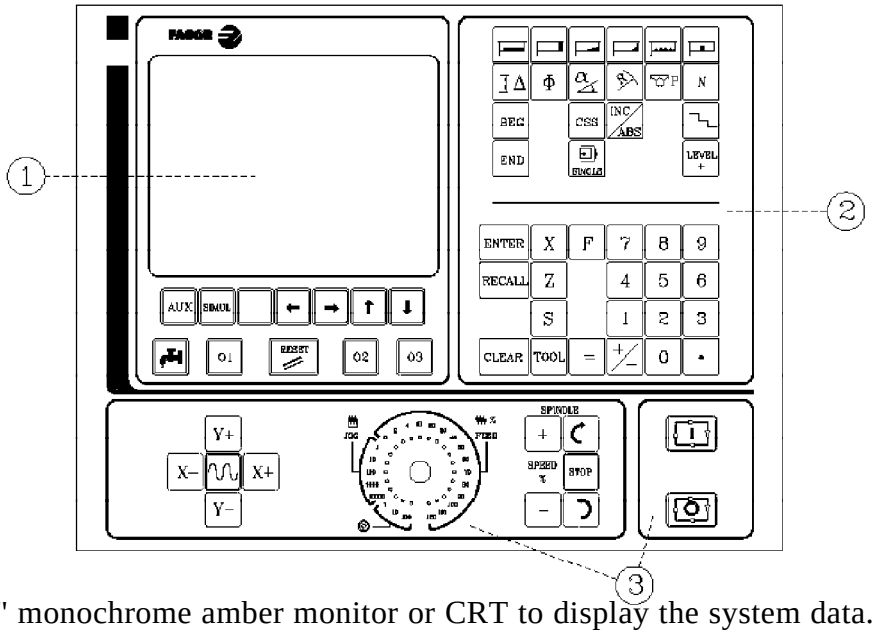
You have just received one of the models listed below:

Compact 800T CNC with 8" amber monitor
Modular 800T CNC with 9" amber monitor
Modular 800T CNC with 14" color monitor

This chapter describes the configuration of both compact and modular models as well as the dimensions of the 9" amber and 14" color monitors.

1.2 COMPACT 800T CNC

The compact 800T CNC is an enclosed module having the following elements on its front panel:



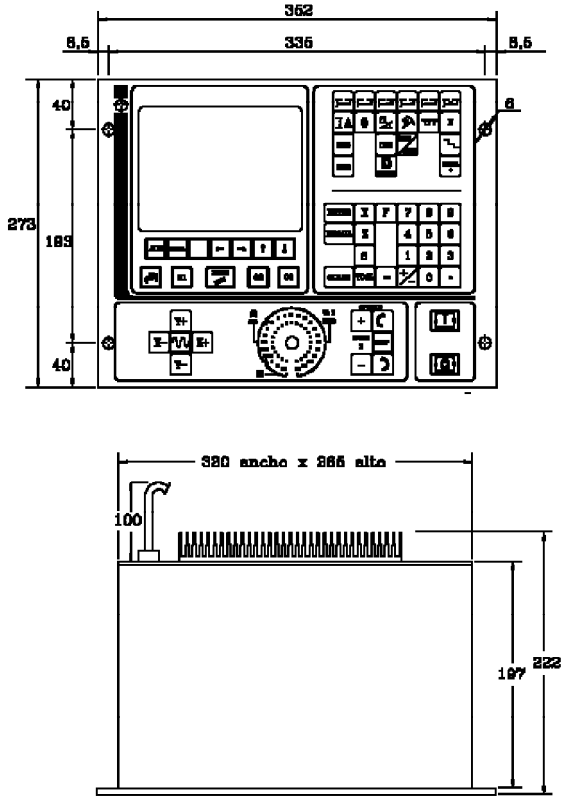
1. One 8" monochrome amber monitor or CRT to display the system data.

2. A keyboard to communicate with the CNC and be able to request or alter the status of the CNC by generating new instructions.

3. An operator panel with the necessary keys to operate in JOG mode and the Cycle Start and Cycle Stop keys.

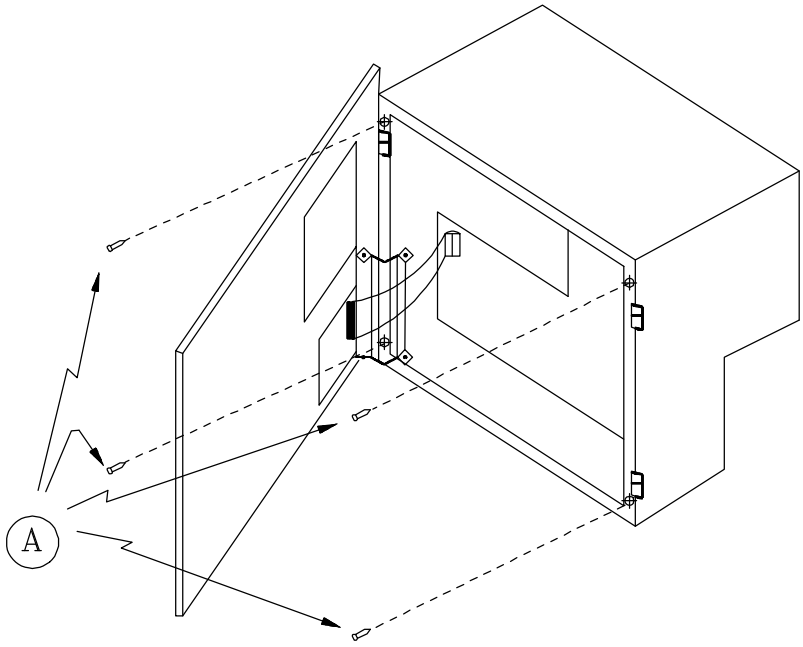
1.2.1 DIMENSIONS & INSTALLATION OF THE COMPACT 800T CNC

This CNC, usually located on the machine's operator panel, has 4 mounting holes.



When installing it, leave enough room to swing the cover open in order to be able to get to its interior in the future if necessary.

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



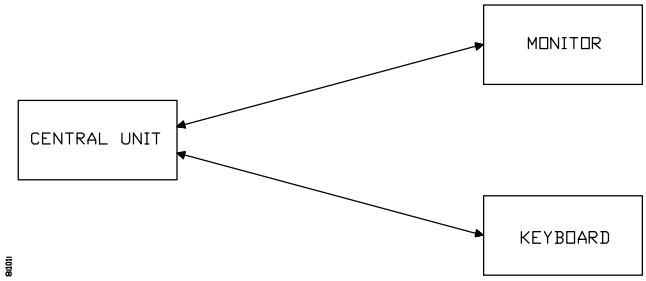
1.3 MODULAR 800T CNC

The 800T CNC consists of 3 independent interconnected modules. These modules may be mounted on different locations of the machine and they are:

- **CENTRAL UNIT**
- **MONITOR**
- **KEYBOARD**

The CENTRAL UNIT communicates with the MONITOR via the provided video signal cable which may be up to 25m long (82ft).

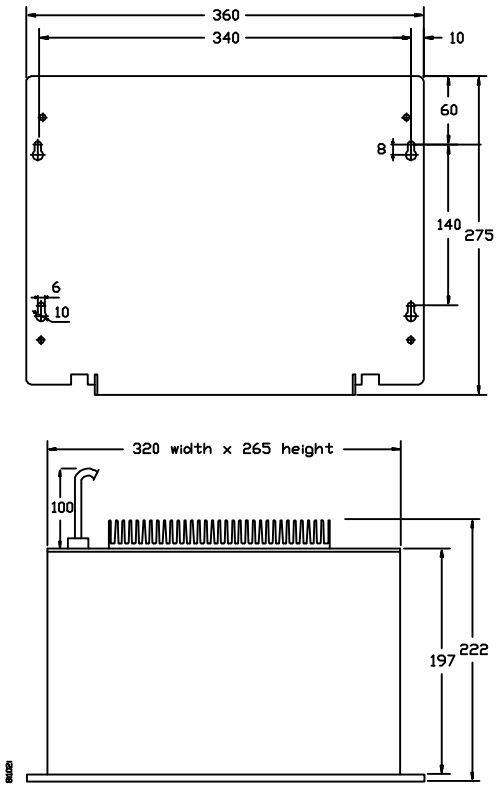
The CENTRAL UNIT communicates with the KEYBOARD via the provided keyboard signal cable which may be up to 25m long (82ft).



1.3.1 CENTRAL UNIT OF THE MODULAR 800T CNC

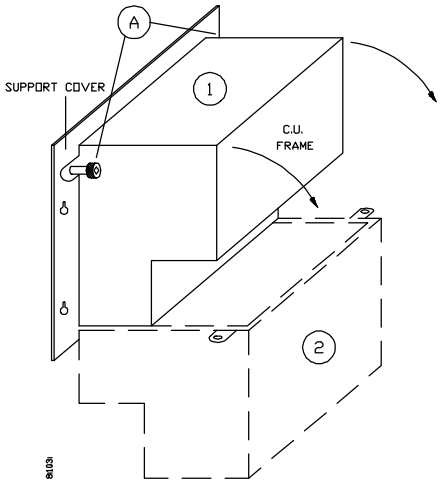
The CENTRAL UNIT is usually mounted in the electrical cabinet by means of the mounting holes located at the SUPPORT lid (cover).

All dimensions are in mm.



When installing it, it must be assigned enough room to swing it open in order to gain access to its interior.

To swing the central unit open, undo the two knurled nuts located on the back and swing it while supporting its body.



1.3.2 MONITOR OF THE MODULAR 800T CNC

It may be mounted anywhere on the machine, but it is recommended to be at operator's eye level. All dimensions are in mm.

Atention:

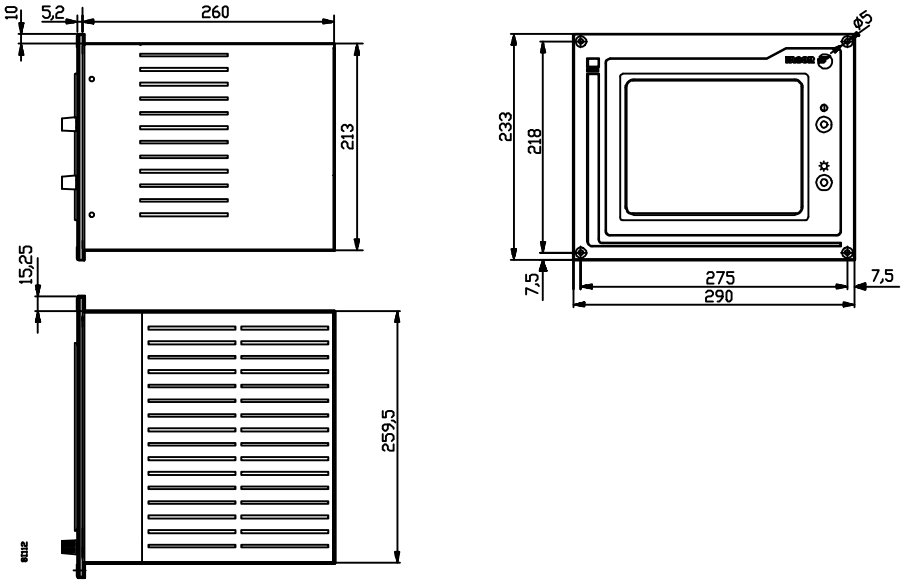
Do not manipulate inside this unit

Only personnel authorized by Fagor Automatin may manipulate inside this module.

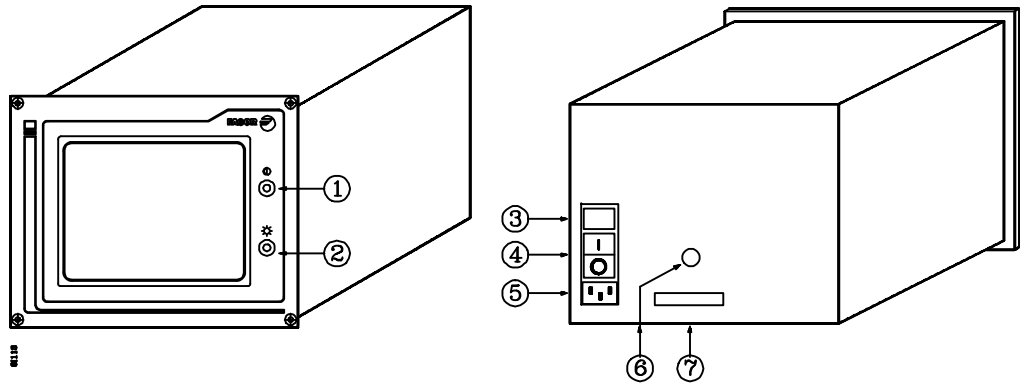
Do not manipulate the connectors with the unit connected to main AC power

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

9" Amber Monitor.



It has:



1.- Brightness adjustment knob for the MONITOR screen.

2.- Contrast adjustment knob for the MONITOR screen.

3.- Two AC power fuses. One per line for power input protection (3A. 250V.)

4.- Power ON/OFF switch.

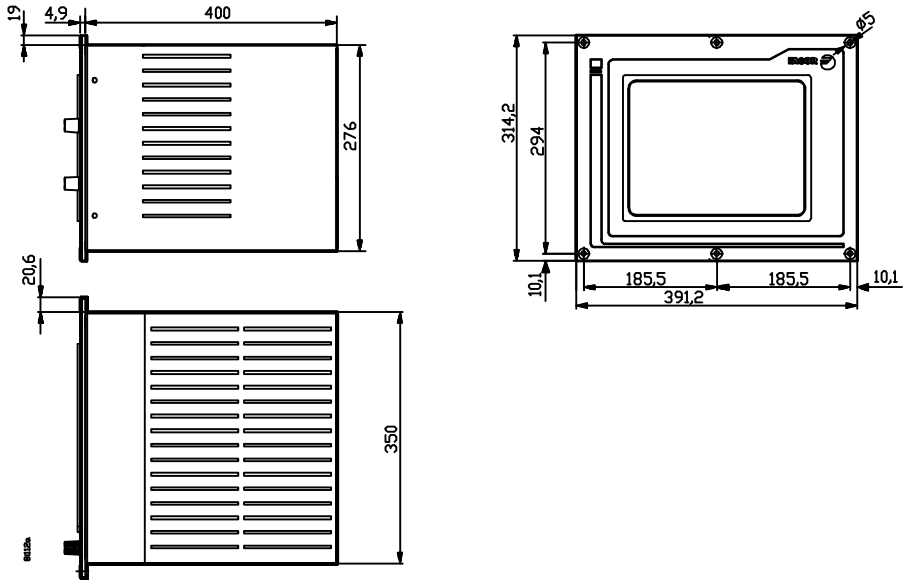
5.- AC power plug. Use the supplied plug to connect it to 220V AC and ground.

6.- Ground terminal. Used to connect the general machine ground. It is Metric 6mm.

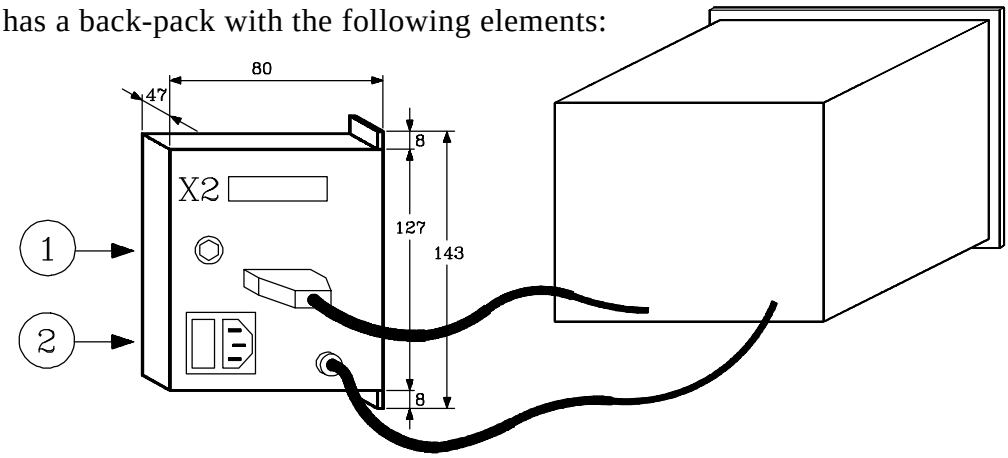
7.- 15-pin SUB-D type male connector to connect the MONITOR with the CENTRAL UNIT.

This connector is described in the section corresponding to the CENTRAL UNIT.

14" Color Monitor. (Dimensions in mm)



It has a back-pack with the following elements:



X2. 15-pin SUB-D type male connector to connect the MONITOR with the CENTRAL UNIT.

This connector is described in the section corresponding to the CENTRAL UNIT.

1.- Ground terminal. Used to connect the general machine ground. It is Metric 6mm.

2.- AC power plug. Use the supplied plug to connect it to 220V AC and ground.

1.3.2.1 MONITOR ENCLOSURE

Consult the appendix on "ENCLOSURES" at the end of this manual for proper dimensioning and ventilation.

1.3.2.2 MONITOR-CENTRAL UNIT CONNECTION CONNECTOR

It is a 15-pin SUB-D type female connector used to connect the MONITOR with the CENTRAL UNIT.

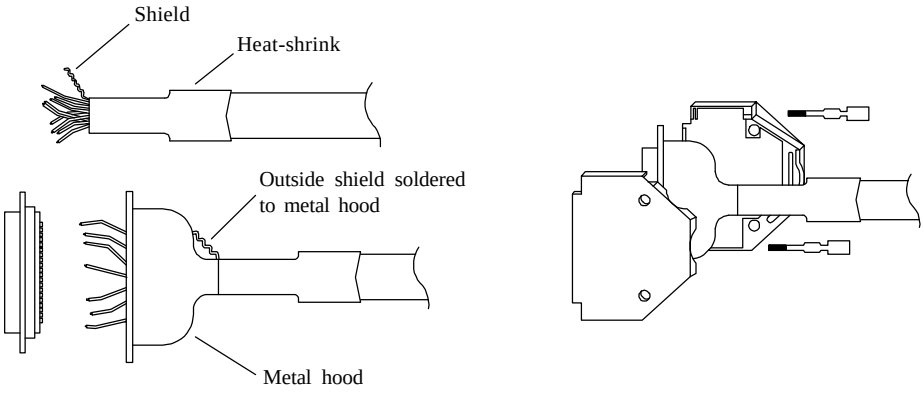
FAGOR AUTOMATION supplies the cable required for this connection and it consists of a cable with a 15-pin SUB-D type connectors at each end, one male and the other one female.

Both connectors have a latching system by means of two UNC4.40-type 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 housing	shield

The cable being used has 6 twisted pairs with a section of 0.34 mm² each (6 x 2 x 0.34mm²) with overall shielding and acrylic rubber covering. Its specific impedance is 120 Ohm and the maximum length permitted is 25m (82ft.).

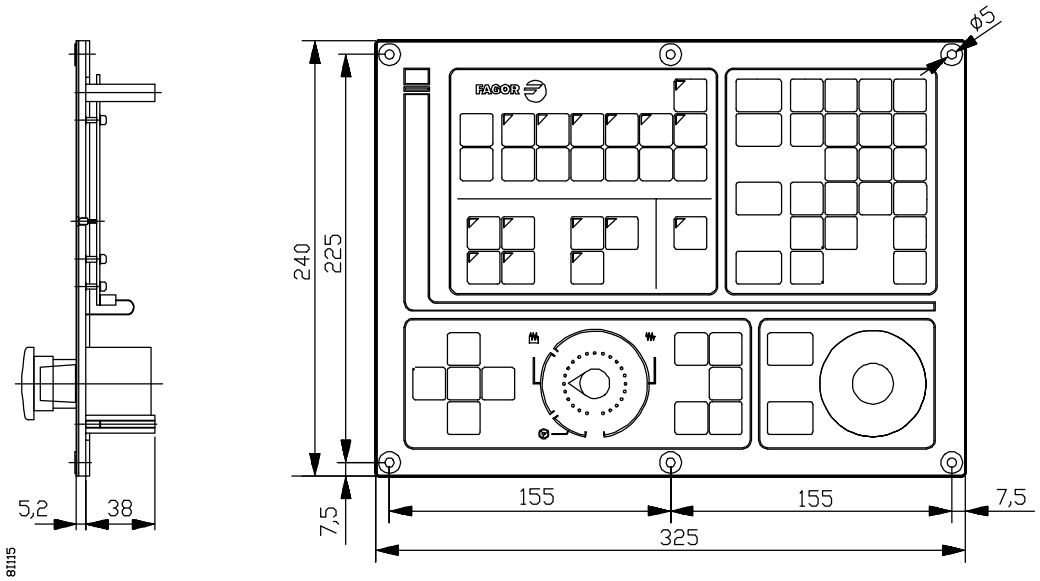
The cable shield is soldered to the metal housing of both cable connectors and it is connected to pin 1 of the female connector at the Central Unit as well as at the Monitor.



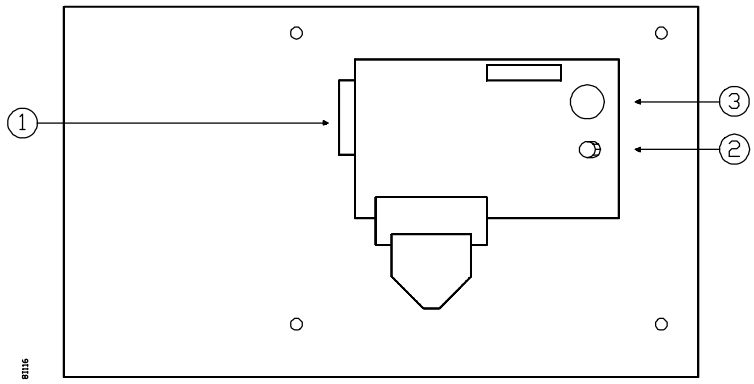
1.3.3 KEYBOARD OF THE MODULAR 800T CNC

The keyboard may be mounted anywhere on the machine.

All dimensions are in mm.



On its back it has:



1.- 25-pin SUB-D type female connector to connect the KEYBOARD to the CENTRAL UNIT.

This connector is described in the section corresponding to the CENTRAL UNIT.

2.- Potentiometer to adjust the volume of the buzzer.

3.- Buzzer.

1.3.3.1 KEYBOARD-CENTRAL UNIT CONNECTION CONNECTOR

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

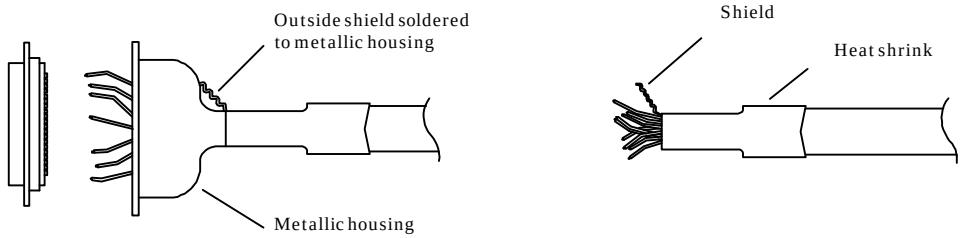
FAGOR AUTOMATION supplies the cable required for this connection which consists of a cable with a 25-pin SUB-D type male connector at each end.

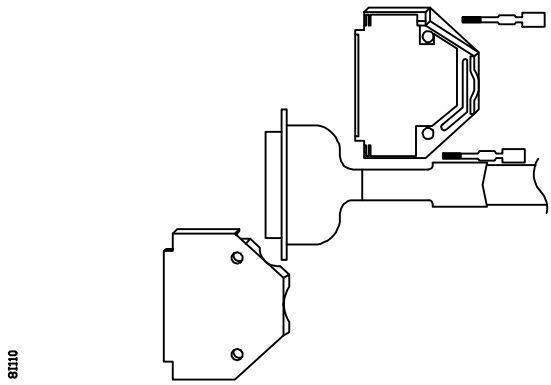
Both connectors have a latching system by means of two UNC4.40-type screws.

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 housing	shield

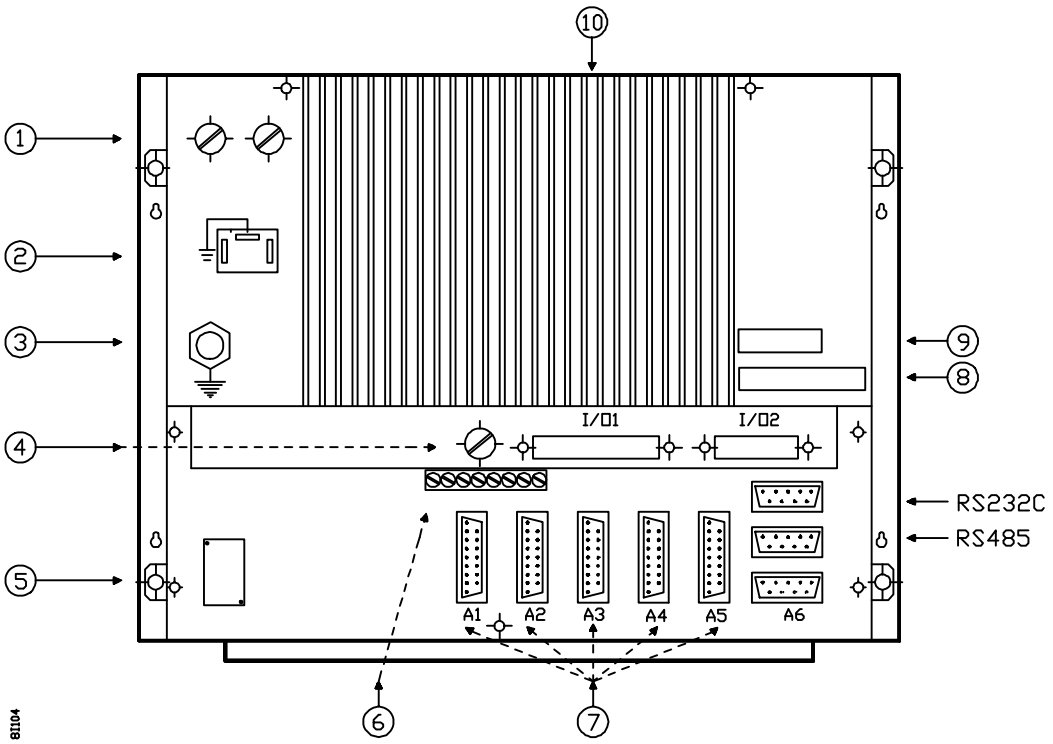
The cable has 25 wires with a section of 0.14 mm² each (25 x 0.14mm²) with overall shielding and acrylic rubber covering. The maximum permissible length is 25m (82ft.).

The cable shield is soldered to the metal housing of both cable connectors and it is connected to pin 1 of the female connector at the Central Unit as well as at the Keyboard.





1.4 CONNECTORS AND CONNECTIONS OF THE 800T CNC SYSTEM



A1 15-pin SUB-D type female connector for X axis feedback connection. It accepts sinusoidal signal.

A2 15-pin SUB-D type female connector. Not being used at this time.

A3 15-pin SUB-D type female connector for Z axis feedback connection. It accepts sinusoidal signal.

A4 15-pin SUB-D type female connector for 2nd electronic handwheel connection (for the Z axis). It accepts sinusoidal signal.

A5 15-pin SUB-D type female connector for spindle feedback connection. It does **not** accept sinusoidal signal.

A6 9-pin SUB-D type female connector for 1st electronic handwheel connection (for X axis only when having two handwheels). It does **not** accept sinusoidal signal.

RS485 9-pin SUB-D type female connector. Not being used at this time.

RS232C 9-pin SUB-D type female connector for serial port RS 232C connection.

I/O1 37-pin SUB-D type female connector for interface connection with the electrical cabinet.

I/O2 25-pin SUB-D type female connector for interface connection with the electrical cabinet.

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 plug. To supply AC voltage to the CNC by connecting it to the transformer and to ground.

3.- Ground terminal. Metric 6mm. Where the general machine ground must be connected.

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

5.- Lithium battery. It provides power back-up for data contained in RAM memory.

6.- Potentiometers for analog output adjustment. To be used **only** by the Technical Service Department at FAGOR AUTOMATION.

7.- 10 dip-switches. There are 2 dip-switches under each feedback input (A1 thru A5) to set the CNC according to the type of feedback signals being used in each case.

8.- 25-pin SUB-D type female connector to connect the KEYBOARD to the CENTRAL unit.

9.- 15-pin SUB-D type female connector to connect the MONITOR to the CENTRAL unit.

10.-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.

1.4.1 CONNECTORS A1, A3, A4

They are 15-pin SUB-D type connectors used to connect feedback devices.

* Connector A1 is used for the X axis feedback signals.
* Connector A3 is used for the Z axis feedback signals.
* Connector A4 is used for the 2nd handwheel (for the Z axis) feedback signals.

The type of cable used must have overall shield. The rest of its characteristics such as length will depend on the type and model of the feedback device being used.

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

PIN	MEANING AND FUNCTION	
1 2 3 4	$\overline{A}$ A $\overline{B}$ B	Differential square-wave feedback signals.
5 6	$\overline{Io}$ Io	Machine reference pulse signals (Marker).
7 8	Ac Bc	Sine-wave feedback signals.
9 10 11 12 13 14	+5V. 0V. -5V. 	Power to feedback device. Not connected. Power to feedback device. Not connected. Power to feedback device. Not connected.
15	FRAME	Shield.

Atention:

When using square-wave rotary encoders, their output signals must be TTL compatible and their outputs must **not** be Open Collector.

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.

1.4.1.1 DIP-SWITCHES FOR CONNECTORS A1, A3, A4

There are 2 dip-switches under each feedback input connector (A1, A3 and A4) to set the CNC according to the type of feedback signal being used in each case.

Dip-switch 1 indicates whether the feedback signal is sine-wave or square-wave and dip-switch 2 indicates whether the feedback signal is differential (double-ended) or not (single-ended).

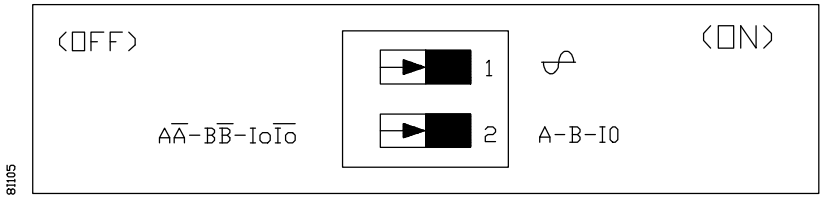
The feedback signals that can be used at connectors A1, A3 and A4 are:

* Sine-wave feedback signals: Ac, Bc, Io
* Square-wave feedback signals: A, B, Io
* Differential (double-ended) square-wave feedback signals: A, $\overline{A}$, B, $\overline{B}$, Io, $\overline{Io}$

The chart below shows the dip-switch combinations for the particular type of feedback signal used at each feedback input.

Dip-switch 1	Dip-switch 2	FUNCTION
ON	ON	Sine-wave signal (Ac, Bc, Io)
ON	OFF	Differential sine-wave signal **"not allowed"**
OFF	ON	Square-wave signal (A,B,Io)
OFF	OFF	Differential square-wave signal (A,$\overline{A}$,B,$\overline{B}$,Io,$\overline{Io}$)

There is a label next to each pair of switches indicating their meaning.



1.4.2 CONNECTOR A5

It is a 15-pin SUB-D type female connector used to connect the spindle feedback device. It does **not** take sine-wave signals.

The type of cable used must have overall shield. The rest of its characteristics such as length will depend on the type and model of the feedback device being used.

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

PIN	MEANING AND FUNCTION	
1 2 3 4	$\overline{A}$ A $\overline{B}$ B	Differential square-wave feedback signals.
5 6	$\overline{Io}$ Io	Machine Reference pulses (Marker).
7 8		Not being used at this time. Not being used at this time.
9 10 11 12 13 14	+5V. 0V. -5V. 	Power to feedback device. Not connected. Power to feedback device. Not connected. Power to feedback device. Not connected.
15	FRAME	Shield.

Atention:

When using square-wave rotary encoders, their output signals must be TTL compatible and their outputs must **not** be Open Collector.

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.

1.4.2.1 DIP-SWITCHES FOR CONNECTOR A5

There are 2 dip-switches under connector A5 to set the CNC according to the type of feedback signal being used in each case.

Dip-switch 1 indicates whether the feedback signal is sine-wave or square-wave and dip-switch 2 indicates whether the feedback signal is differential (double-ended) or not (single-ended).

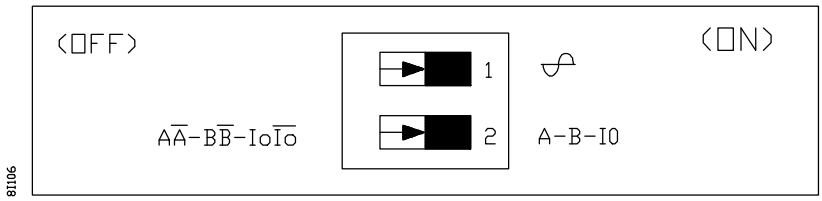
The feedback signals that can be used at connector A5 are:

* Square-wave feedback signals: A, B, Io
* Differential (double-ended) square-wave feedback signals: A, $\overline{A}$, B, $\overline{B}$, Io, $\overline{Io}$

The chart below shows the dip-switch combinations for the particular type of feedback signal used at this connector.

Dip-switch 1	Dip-switch 2	FUNCTION
ON	ON	Sine-wave signal **"not allowed"**
ON	OFF	Differential sine-wave signal **"not allowed"**
OFF	ON	Square-wave signal (A,B,Io)
OFF	OFF	Differential square-wave signal (A,$\overline{A}$,B,$\overline{B}$,Io,$\overline{Io}$)

There is a label next to these switches indicating their meaning.



1.4.3 CONNECTOR A6

It is a 9-pin SUB-D type female connector used to connect the feedback device for the first electronic handwheel. It does **not** take sine-wave signals.

The type of cable used must have overall shield. The rest of its characteristics such as length will depend on the type and model of the feedback device being used.

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

PIN	MEANING AND FUNCTION	
1 2	A B	Square-wave feedback signals from the electronic handwheel.
3	Io	Machine Reference pulse signal
4 5	+5V. 0V.	Power for the electronic handwheel. Power for the electronic handwheel.
6 7 8		Not being used at this time. Not being used at this time. Not being used at this time.
9	FRAME	Shield.

Atention:

When the handwheel supplies square-wave pulses they must be TTL compatible and the outputs must **not** be Open Collector.

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.

1.4.4 RS232C CONNECTOR

It is a 9-pin SUB-D type female connector used to connect the RS-232C serial line.

The shield of the cable being used must be connected to pin 1 of the connector at the CNC end and to the metal housing of the connector 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 RS232C INTERFACE

*** Connecting and disconnecting the peripheral device**

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

*** Cable length**

The EIA RS232C standards specify that the capacity of the cable must not exceed 2500pF, thus, since the cables usually have a capacity between 130 and 170 pF/m, their maximum length will be limited to 15 meters (50 feet).

It is suggested to use shielded cables and/or twisted-pair wires in order to minimize interference between cables thus avoiding faulty communications over lengthy cables.

It is recommended to use 7-conductor cables with a minimum section of 0.14 mm^2 and overall shielding.

*** Transmission speed (baudrate)**

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

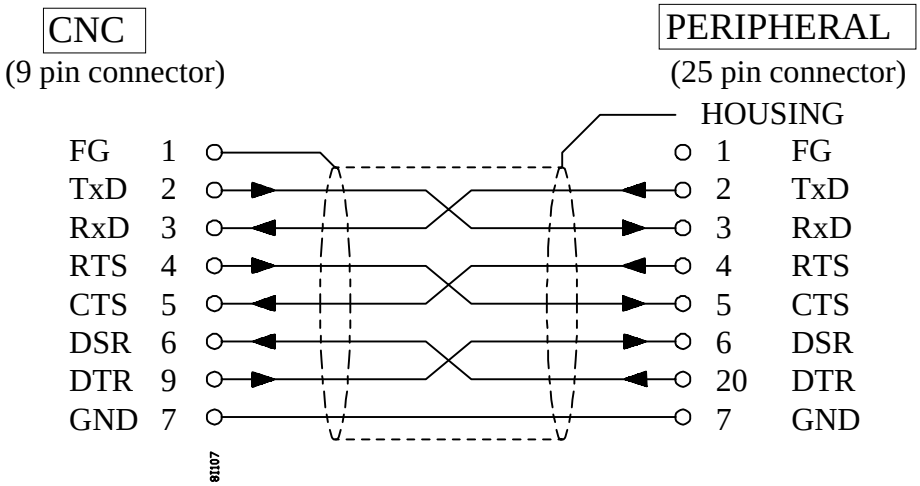
All unused wires should be grounded to avoid erroneous control and data signals.

*** Ground connection**

It is suggested to reference all control and data signals to the same ground wire (pin 7 -GND-) thus avoiding reference points at different voltage levels since there could be different voltages between both ends of long cables.

RECOMMENDED RS232C INTERFACE CONNECTIONS

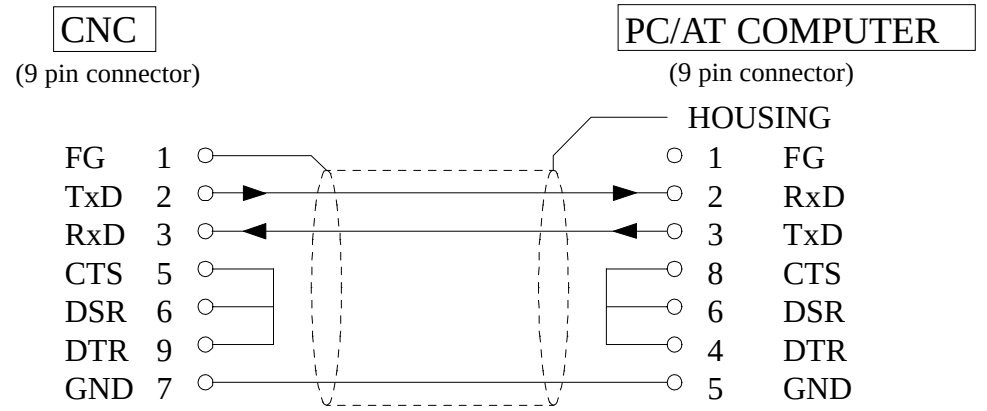
*** Complete connection**

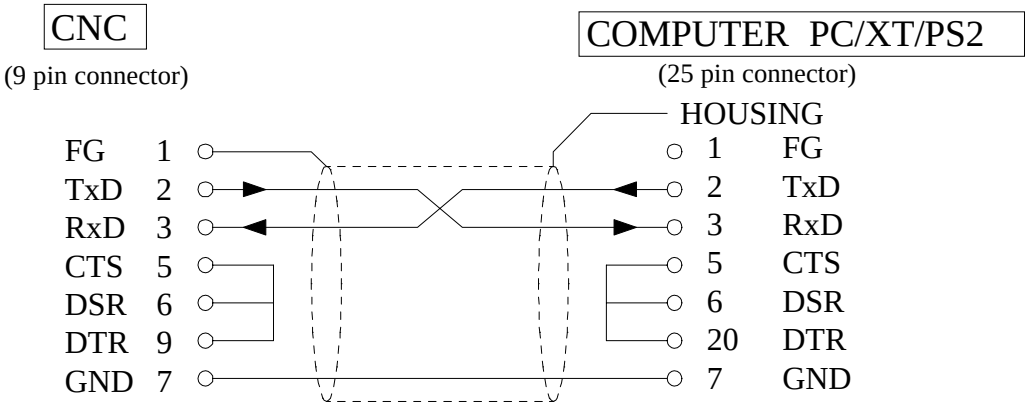


*** Simplified connection**

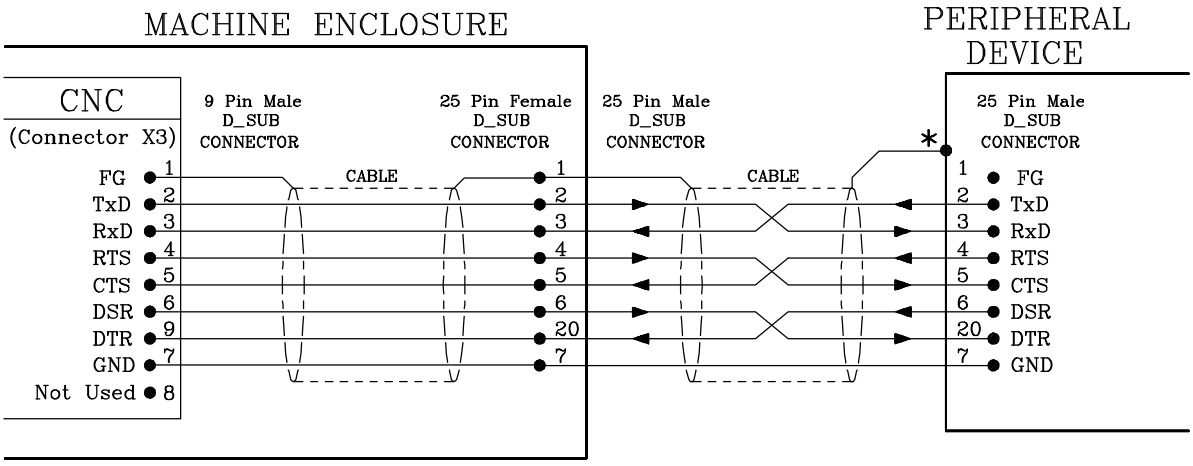
To be used when the computer or peripheral device meets one of the following requirements:

It does not have the RTS signal or
The receiving unit can receive data at the selected baudrate.

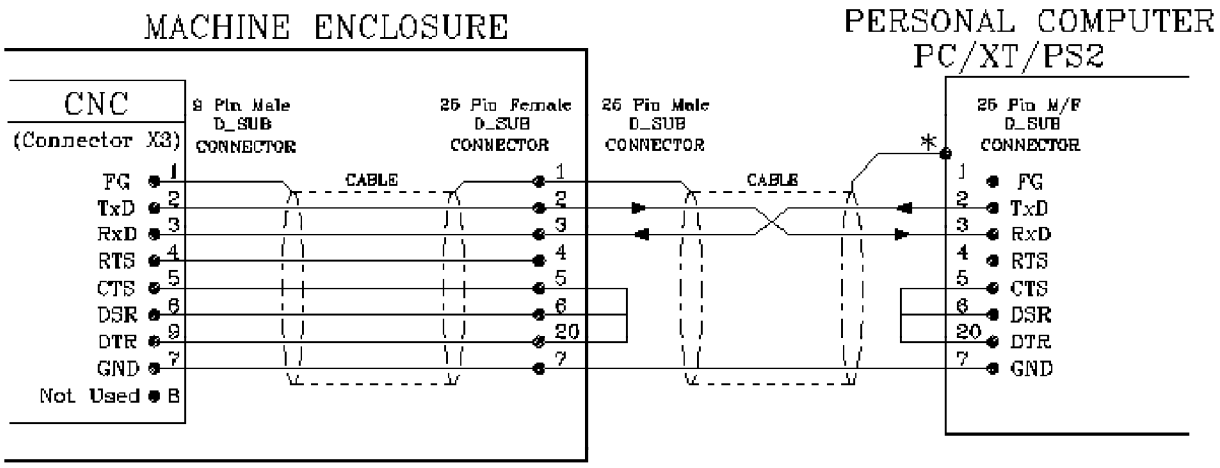




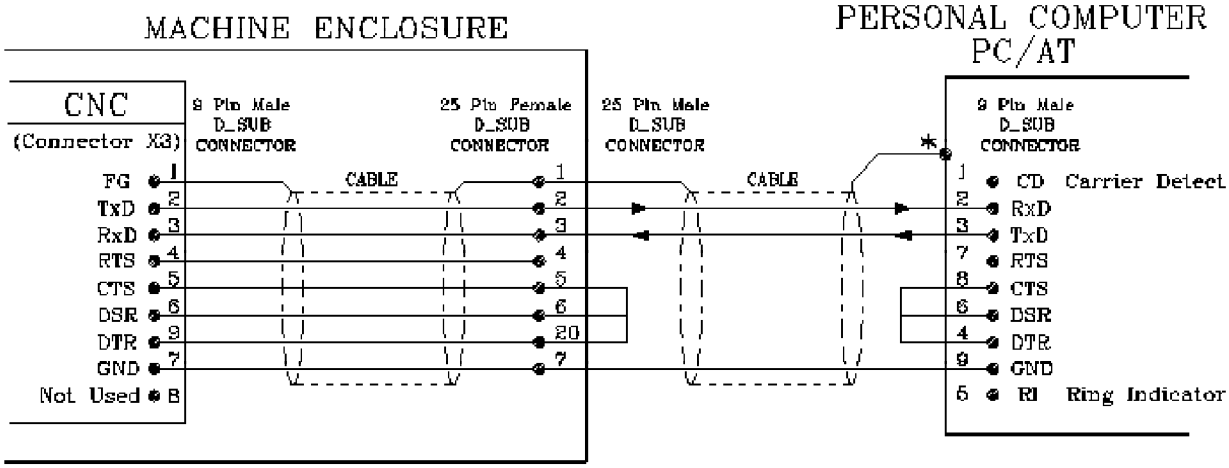
However, it is recommended to consult the technical manuals of the computer or peripheral device in case there are any discrepancies.



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

1.4.5 CONNECTOR I/O 1

It is a 37-pin SUB-D type female connector to connect the CENTRAL UNIT to the electrical cabinet.

Pin	SIGNAL AND FUNCTION	
1	0V.	External power supply input.
2	T Strobe	Output. The BCD code indicates tool number.
3	S Strobe	Output. The BCD code indicates S spindle speed.
4	M Strobe	Output. The BCD code indicates the M function number.
5	Emergency	Output.
6	Threading ON	Output.
7	Z Enable	Output.
8	Reset	Output.
9	X Enable	Output.
10	X home switch	Home switch input for the X axis.
11		Not being used at this time.
12	Z home switch	Home switch input for the Z axis.
13		Not being used at this time.
14	Emergency stop	Input.
15	Feed Hold	Input.
16	Stop	Input.
17	Start	Input.
18		Not being used at this time.
19	Manual	Input. The CNC works as a DRO.
20	MST80	Output. BCD code, significance: 80
21	MST40	Output. BCD code, significance: 40
22	MST20	Output. BCD code, significance: 20
23	MST10	Output. BCD code, significance: 10
24	MST08	Output. BCD code, significance: 8
25	MST04	Output. BCD code, significance: 4
26	MST02	Output. BCD code, significance: 2
27	MST01	Output. BCD code, significance: 1
28	FRAME	Connect all the shields used to this pin
29	24V.	External power supply.
30	±10V	Analog output for the X axis drive.
31	0V.	Analog output for the X axis drive.
32	±10V	Analog output for the live tool.
33	0V.	Analog output for the live tool.
34	±10V	Analog output for the Z axis drive.
35	0V.	Analog output for the Z axis 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.

1.4.5.1 LOGIC INPUTS OF CONNECTOR I/O1

X AXIS HOME SWITCH. Pin 10

This INPUT must be high when the home switch for the X axis is pressed.

Z AXIS HOME SWITCH. Pin 12

This INPUT must be high when the home switch for the Z axis is pressed.

EMERGENCY STOP Pin 14

This INPUT must be normally high (24V).

When set low (0V), the CNC deactivates the enables and cancels the analog outputs of all the axes; it interrupts the execution of the part program and it displays ERROR 64.

It does **not** activate the emergency output at 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.

* Work as FEED-HOLD. If this input is set low (0V) while moving the axes, the CNC keeps the spindle on and stops the movement of the axes by cancelling their analog outputs (0V) but keeping their enables activated.

  When returning this signal high, the CNC will resume the axes movement.

* Work as TRANSFER INHIBIT. If this input is set low while executing a motionless block, The CNC interrupts the execution of the program at the end of the current block.

  When returning this signal high, the CNC will resume the execution of the program.

* The "M-DONE" signal will be used when machine parameter "P602" bit 7 has been set to "1".

  The CNC waits for the electrical cabinet to execute the requested M function and return the "M-DONE" signal by setting this input high (24V).

$\overline{\text{STOP}}$ Pin 16

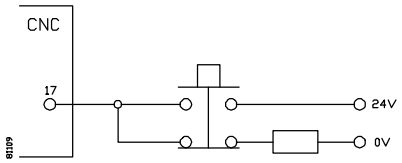
This INPUT must be normally high (24V).

When set low (0V), the CNC interrupts the program execution. It works just as the key on the operator panel.

To resume program execution it is necessary to bring this signal up (24V) and either press at the OPERATOR PANEL or activate the "START" input described below.

START Pin 17

This INPUT must be normally low (0V) and connected to 0V through a 10KOhm resistor.



When an up flank (leading edge) is detected (level change from 0V to 24V), the CNC acts as if the key of the OPERATOR PANEL were pressed.

To use **only this input instead of** the key of the OPERATOR PANEL, machine parameter **P601 bit 5** must be set to "**1**".

MANUAL (DRO mode) Pin 19

By setting this INPUT high (24V), the CNC behaves as a DRO.

1.4.5.2 LOGIC OUTPUTS OF CONNECTOR I/O 1

T Strobe Pin 2

The CNC activates this output (by setting it high-24V) whenever the BCD code sent out via pins 20 thru 27 corresponds to a tool number (T function).

S Strobe Pin 3

The CNC activates this output (by setting it high-24V) whenever the BCD code sent out via pins 20 thru 27 corresponds to a spindle speed value (S function).

M Strobe Pin 4

The CNC activates this output (by setting it high-24V) whenever the BCD code sent out via pins 20 thru 27 corresponds to a miscellaneous M function.

EMERGENCY Pin 5

The CNC activates this output whenever an **internal** emergency condition is detected.

This output is **normally high (24V) or low (0V)** depending on the setting of machine parameter **P604 bit 4.**

THREADING ON / CYCLE ON Pin 6

This output is normally low (0V) and its meaning depends on the setting of machine parameter **P605 bit 4.**

“P605 bit 4” = 0 THREADING ON.

The CNC activates this output (24V) during a threading operation.

“P605 bit 4” = 1 CYCLE ON.

The CNC activates this output (24V) whenever it is executing an automatic operation or a "BEGIN-START" , "END-START" type command.

Z ENABLE Pin 7

The CNC activates this output (24V) whenever the Z axis drive is to be enabled.

RESET Pin 8

This output is activated (24V), when the CNC is reset by pressing the [RESET] key.

The CNC maintains this signal active for 80 milliseconds.

X ENABLE Pin 9

The CNC activates this output (24V) whenever the X axis drive is to be enabled.

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 which M, S or T function has been selected.

This information is BCD coded and the significance or weight of each one of them is indicated by the corresponding mnemonic.

For example, to select the first spindle speed range, the CNC will send out to the electrical cabinet the code: M41

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

Besides these outputs, the "M-STROBE", "T-STROBE" or "S-STROBE" signal will also be activated depending on the type of function been selected.

X axis analog output ±10V. Pin 30
X axis analog output 0V. Pin 31

These outputs supply the analog voltage to move the X axis. They must be connected to the drive via a shielded cable.

Live tool analog output ±10V. Pin 32
Live tool analog output 0V. Pin 33

These outputs supply the analog voltage to move the live tool. They must be connected to the drive via a shielded cable.

Z axis analog output ±10V. Pin 34
Z axis analog output 0V. Pin 35

These outputs supply the analog voltage to move the Z axis. They must be connected to the drive via a shielded cable.

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

These outputs supply the analog voltage to move the spindle (S). They must be connected to the drive via a shielded cable.

1.4.6 CONNECTOR I/O 2

It is a 25-pin SUB-D type female connector used to connect the CENTRAL UNIT to the electrical cabinet.

PIN	SIGNAL AND FUNCTION	
1	0V.	Input from external power supply.
2	0V.	Input from external power supply.
3	Output M1	Bit 1 of M function table.
4	Output M2	Bit 2 of M function table.
5	Output M3	Bit 3 of M function table.
6	Output M4	Bit 4 of M function table.
7	Output M5	Bit 5 of M function table.
8	Output M6	Bit 6 of M function table.
9	Output M7	Bit 7 of M function table.
10	Output M8	Bit 8 of M function table.
11	Output M9	Bit 9 of M function table.
12	Output M10	Bit 10 of M function table.
13	Output M11	Bit 11 of M function table.
14		Not being used at this time
15		Not being used at this time
16		Not being used at this time
17		Not being used at this time
18		Not being used at this time
19	24V.	Input from external power supply.
20	24V.	Input from external power supply.
21	Work	Output. Selected work mode.
22	Output M15	Bit 15 of M function table.
23	Output M14	Bit 14 of M function table.
24	Output M13	Bit 13 of M function table.
25	Output M12	Bit 12 of 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.

1.4.6.1 LOGIC OUTPUTS OF CONNECTOR I/O 2

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

These outputs correspond to the bits selected at the decoded M function table.

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

M41 000100100100100 (outputs being activated)
00100100100100100 (outputs being deactivated)

The CNC will behave as follows every time the M41 function is executed (selection of the first spindle speed range):

	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		
at 0V			x			x			x			x			x
Does not modify	x	x			x			x			x			x	

Outputs M01 / Coolant Pin 3

This output besides providing the value of bit 1 of the selected decoded M function of the table, acts as output for the COOLANT.

When using this option, be careful not to use this bit for both functions since the CNC will activate it in both cases.

However, the CNC maintains this output active as long as the coolant is selected even when executing an M function which would deactivate this output.

WORK Pin 21

The CNC activates this OUTPUT on CNC power-up and it deactivates only when accessing the tool table (by the operator) or the auxiliary modes (with OEM password).

This output becomes active again when returning to the standard work mode.

M14 / G00 Output Pin 23

This OUTPUT, besides providing the status of bit 14 of the decoded M function table, it can also indicate that G00 has been selected.

To do this, set machine parameter **P604 bit 3 to "1".** This way, this output will be set high (24V) whenever the CNC is executing a rapid axis positioning move (in **G00)**.

Be careful not to use the bit corresponding to this output when setting the decoded M function table 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

On the back of the CENTRAL UNIT of the 800T CNC there is a three-prong connector to be connected to AC power and to ground..

The AC power must be supplied via an independent 110VA shielded transformer with an output voltage between 100V AC and 240V AC +10% and -15%.

The power outlet for the equipment must be near it and with easy access to it.

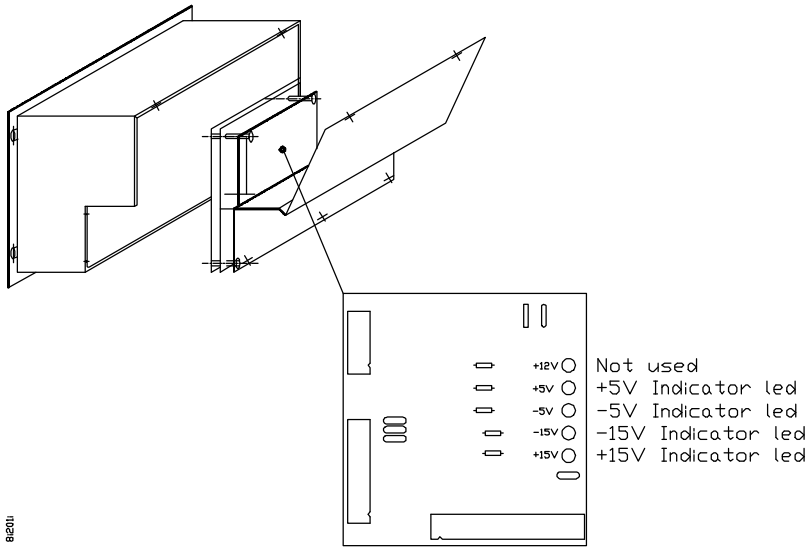
In the case of a power surge or voltage overload it is recommended to wait for about 3 minutes before connecting it back in order to avoid any damage to the power supply.

The MONITOR of the 800T CNC must be connected to 220V AC.

2.1.1 INTERNAL POWER SUPPLY

Inside the CENTRAL UNIT there is a power supply that provides the different required voltages.

Besides the two outside fuses for AC input protection (one per line), it has an inside 5A. fuse for protection against overvoltage.



2.2 MACHINE INTERFACE

2.2.1 GENERAL CONSIDERATIONS

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

* DC relay coils.

  1N4000 type diodes.

* AC relay coils.

  RC connected as close to the coils as possible with approximate values such as:

  R 220 Ohm/1W
  C 0,2 µF/600V

* A.C Motors.

  RC connected between phases with values:

  R 300 Ohm/6W
  C 0,47µF/600V

Ground connection.

A proper ground connection in an electrical installation is essential in order to achieve:

* The protection of people against electrical discharges caused by any malfunction.

* The protection of electronic equipment against interference generated at the machine itself or by other electronic equipment near by which could cause erratic malfunctioning of the equipment.

Thus, all metallic parts must be connected to the same point and this, in turn, to ground. It is essential to establish one or two main points in the installation where the elements mentioned above will be connected.

Cables with sufficient section must be used in order to obtain an impedance as low as possible and an effective interference suppression maintaining all installation components at the same voltage level with respect to ground.

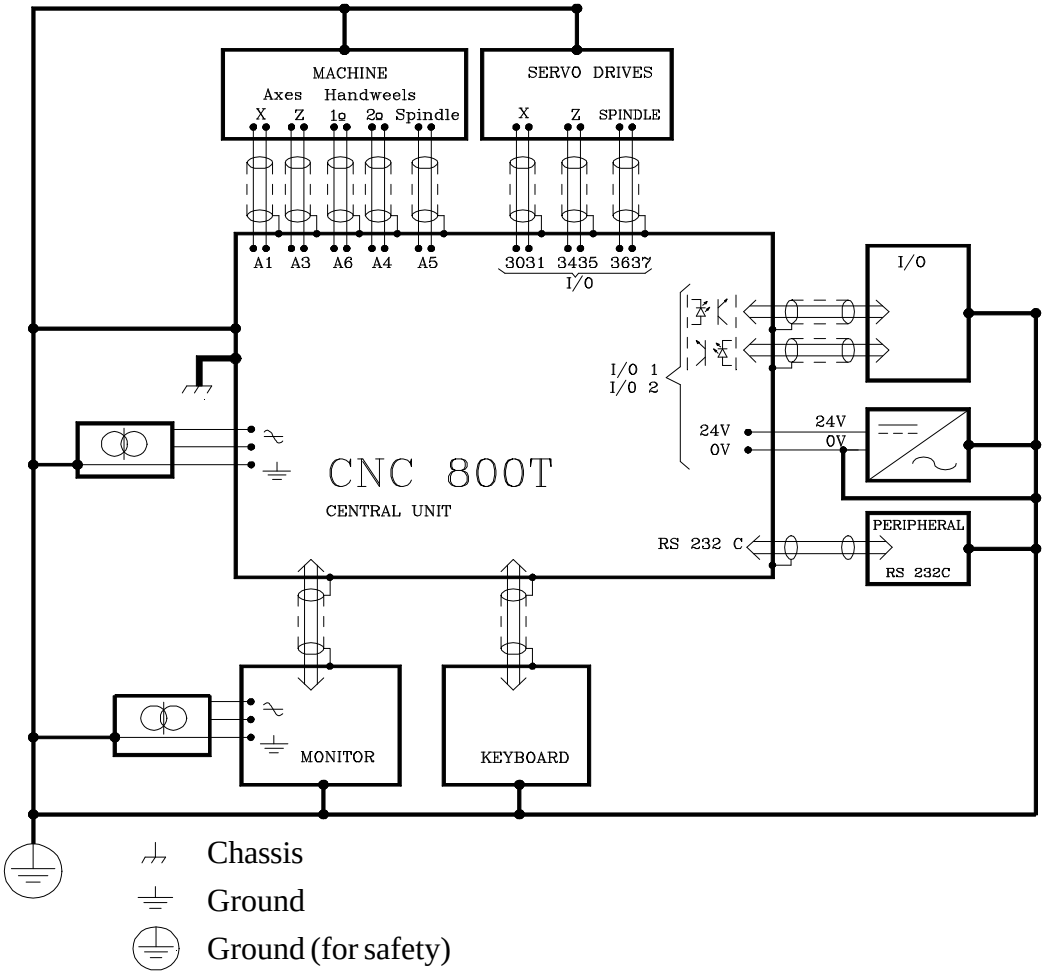
Besides the proper grounding of the installation, the signal cables must be shielded and twisted-pairs for greater necessary protection. The shield must be connected to a specific point avoiding ground loops which could cause undesirable effects. This grounding of the shield is done at the CNC's ground terminal.

Each element of the machine-tool/CNC installation must be connected to ground through the established main ground points. They will be conveniently set at a point close to the machine-tool itself and properly connected to the Main Ground Point.

When a second ground point is required, it is recommended to join both points by a cable with a section of 8 mm^2 or greater (or two 4 mm^2).

It must be verified that the resistance measured between each connector housing and ground is less than 1 Ohm.

Ground connection diagram:



2.2.2 DIGITAL OUTPUTS.

This CNC has a number of optocoupled digital outputs which can be used to activate relays, and so forth.

All the outputs are galvanically insulated by means of optocouplers from the outside world and they permit to commute a DC voltage supplied from the electrical cabinet of the machine.

The electrical characteristics of these outputs are:

Nominal voltage	+24 V DC.
Maximum voltage	+30 V DC.
Minimum voltage	+18 V DC.
Output voltage	2V < V DC.
Maximum output current	100 mA.

All the outputs are protected by:

Galvanic insulation by means of optocouplers.
External 3A fuse against output overloads greater than 125mA, overvoltage of the external power supply greater than 33V DC and for protection against reversed connection of the power supply.

2.2.3 DIGITAL INPUTS.

The digital inputs are used to read external devices and so forth.

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

The electrical characteristics of these inputs are:

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

All the inputs are protected by:

Galvanic insulation by means of optocouplers.
Against reversed connection of the power supply up to -30 V DC.

Atención:

The external 24 V DC power supply being used to feed the digital inputs and outputs must be regulated.

The 0V point of that power supply must be connected to the main ground point of the electrical cabinet.

2.2.4 ANALOG OUTPUTS.

There are several analog outputs to control the drives for the axes and for the spindle.

The electrical characteristics of these outputs are:

Analog voltage range	±10V DC.
Minimum impedance of the connected drive	10 KOhm.
Maximum unshielded cable length	75 mm.

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

Atention:

It is also recommended to adjust the axes drives in such a way that the desired maximum feedrate (P110, P111, P310, P311) corresponds to an analog voltage of ±9.5 V DC.

2.2.5 FEEDBACK INPUTS

These inputs are used to receive sine-wave as well as single-ended and double-ended (differential) square-wave signals supplied by linear (scales) and rotary transducers (encoders).

Connector A1 is used for X axis feedback connection and accepts both single ended sine-wave and differential square-wave signals.

Connector A3 is used for Z axis feedback connection and accepts both single ended sine-wave and differential square-wave signals.

Connector A4 is used for the second handwheel feedback connection (for the Z axis) and it accepts both single ended sine-wave and differential square-wave signals.

Connector A5 is used for spindle feedback connection and accepts differential square-wave signals.

Connector A6 is used for the first handwheel feedback connection (for the X axis only when having two handwheels) and it accepts only single-ended square-wave signals (not differential).

The electrical characteristics of these inputs are:

Sine-wave signals

Power supply voltage	±5V.±5%
Maximum counting frequency	25KHz.

Square-wave signals

Power supply voltage	±5V.±5%
Maximum counting frequency	200KHz.

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

2.3 SET-UP

2.3.1 GENERAL CONSIDERATIONS

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

This ground connection must be done at a single machine point (Main Ground Point) and all other ground points must be connected to this point.

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

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

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

Look for 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 dip-switches for each feedback device are set properly according to the type of feedback signals being used.

Verify that the E-STOP button is pressed.

2.3.3 CONNECTION

Verify that the A.C. 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 tach is operating properly.

Connect the A.C. power to the CNC. If there is any problem, the CNC will display the corresponding error; otherwise it will display the message: GENERAL TEST PASSED.

2.3.4 SYSTEM I/O TEST

This CNC offers a particular operating mode where it is possible to activate and deactivate the logic inputs and outputs of the CNC.

To access this mode, press the following keystroke sequence:

[AUX]	(SPECIAL FUNCTIONS)
[5]	(AUXILIARY MODES)
[0] [1] [0] [1]	(Password)
[1]	(SPECIAL MODES)
[0]	(TEST)

Once the CNC's self-test is done, press [7], the CNC will display the status of the logic inputs and outputs; and it will allow changing 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)	FEED-HOLD
D	14 (I/O 1)	EMERGENCY STOP
E	13 (I/O 1)	Not being used at this time
F	12 (I/O 1)	Z axis home switch (Io)
G	11 (I/O 1)	Not being used at this time
H	10 (I/O 1)	X axis home switch (Io)
I	19 (I/O 1)	MANUAL (DRO mode)
J	18 (I/O 1)	Not being used at this time
K		To be used only by the Service Department
L		To be used only by the Service Department
M		To be used only by the Service Department
N		To be used only by the Service Department

The CNC will show dynamically and at all times the status of all these inputs. To check any of them, act upon the external switches and observe the status change of the corresponding input on the screen.

The status of "1" indicates that the corresponding input is receiving 24V DC. If not, it will appear as "0".

Logic outputs

OUTPUT	1ST ROW PIN/FUNCTION	2ND ROW 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
C	(4 I/O 1) M Strobe	(5 I/O 2) Output 3, decoded M
D	(5 I/O 1) Emergency	(6 I/O 2) Output 4, decoded M
E	(6 I/O 1) CYCLE 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 Work mode

To check the outputs, select them by means of the up and down arrow keys.

Once the desired output is selected, turn it on (1) or off (0) by assigning to it its corresponding value.

It is possible to have several outputs activated at the same time and all of them will provide 24V DC at their corresponding pin when active.

Once the I/O test is completed, disconnect the electrical cabinet and, then, connect the I/O connectors as well as those of the feedback devices to the CNC.

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

To quit the system I/O test mode, press [END].

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 to all the axes and the spindle.
It does **NOT** imply the emergency output.

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/O 1.

This output will be normally high or low depending on the setting of machine parameter P604(4). High if P604(4) = 1, low if P604(4) = 0.

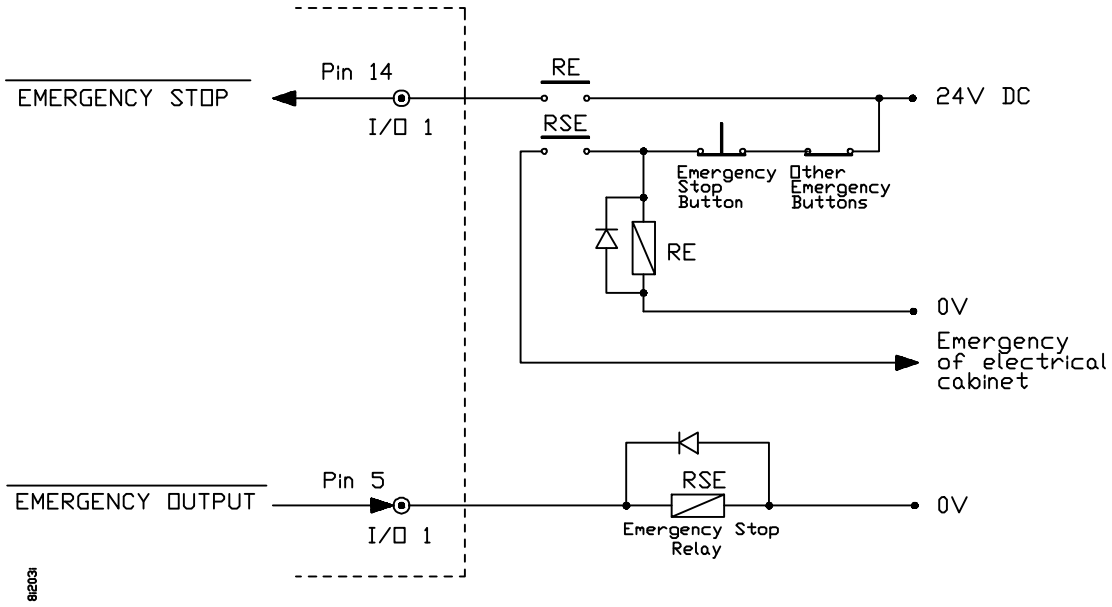
These are some of the internal causes that can activate this output:

* An excessive axis following error has occurred.

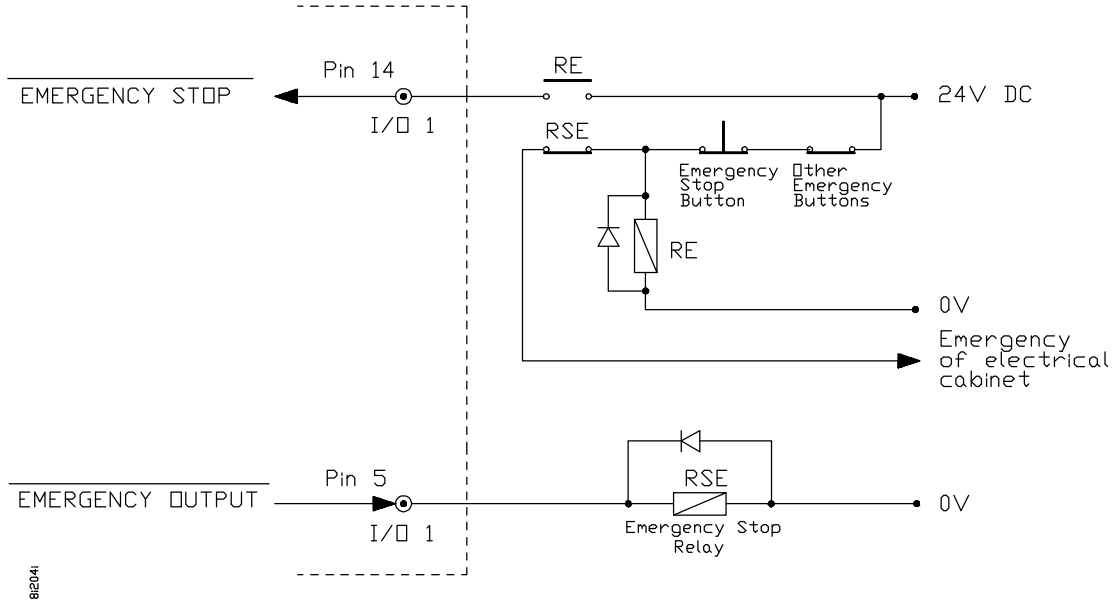
* An axis feedback error has occurred.

* There is erroneous data on the machine parameter table.

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

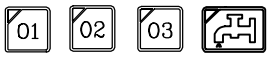


The recommended connection when P604(4) = 0 (output normally OFF) is:



2.5 ACTIVATION / DEACTIVATION OF EXTERNAL DEVICES

With this CNC it is possible to activate and deactivate up to 4 external devices including the coolant. The other devices depend on the type of machine available.

To do this, the following four keys are available: 

All these keys have a lamp to indicate whether the device is currently activated (lamp on) or deactivated (lamp off)

 If it is selected (lamp on) pin 3 of connector I/O2 stays high (24V).

When it is not selected, (lamp off) pin 3 of connector I/O2 stays low (0V)

 When this device (O1) is selected, the CNC sends the M10 function out to activate it and an M11 to deactivate it.

 When this device (O2) is selected, the CNC sends the M12 function out to activate it and an M13 to deactivate it.

 When this device (O3) is selected, the CNC sends the M14 function out to activate it and an M15 to deactivate it.

The coolant may be activated or deactivated at any time but the other devices (O1, O2 and O3) must be activated or deactivated only when the axes of the machine are in position.

3. AUXILIARY FUNCTIONS

Press [AUX] to access this option.

The CNC will show a series of options. To select one of them, simply press the number key indicated for the desired option.

The operator may access all the shown options except the last one referred to as "AUXILIARY MODES". When selecting this option, the CNC requests the password to access the various tables and operating modes exclusive for the OEM.

Press [END] to quit any of these options and return to the standard display mode.

3.1 MILLIMETERS <—> INCHES

When selecting this option, the CNC changes the display units from millimeters to inches and vice versa and it displays the X and Z axes coordinates in the selected units.

Also, The axes feedrates are also shown in the new selected units. These units appear to the right of the main window.

For example, if the position of the axes was displayed in millimeters and their feedrate was in mm/rev. the new units will be inches and inches/rev. respectively.

It must be borne in mind that the values stored in BEGIN, END as well as the data for special operations and the coordinates corresponding to the "point to point movements" have no units. Therefore, their values will remain unchanged when shifting from mm to inches and vice versa.

3.2 RADIUS <—> DIAMETER

When selecting this option, the CNC changes the displayed X axis value from radius to diameter and vice versa showing the new X axis position in the new selected units.

It will also change the text indicating the selected units and which is shown to the right of the X position value (coordinate).

It must be borne in mind that the values stored in BEGIN, END as well as the data for special operations and the coordinates corresponding to the "point to point movements" have no units. Therefore, their values will remain unchanged when shifting from radius to diameter and vice versa.

3.3 F MM(INCH)/MIN <—> F MM(INCH)/REV

When selecting this option, the CNC changes the axes feedrate units from mm/min to mm/rev and vice versa if the active display units are mm and from inch/min to inch/rev. and vice versa if the active display units are inches.

These units are shown to the right of the main window.

The value assigned to the axes feedrate "F" remains unchanged.

3.4 TOOL

When selecting this option, it is possible to access the tool table or to calibrate the tools.

3.4.1 TOOL TABLE

When selecting this option, the CNC shows the values assigned to each tool offset, that is, the dimensions of each tool being used to machine the parts.

Once the tool offset table has been selected, the operator will be able to move the cursor over the screen one line at a time by means of the up and down arrow keys.

Each tool offset has a series of fields defining the tool dimensions. These fields are:

* Tool length along the X axis.

  It will be given in radius and in the work units currently selected. Its value range is:

  X ±8,388.607 mm or X ±330.2599 inches.

* Tool length along the Z axis.

  It will be given in the work units currently selected. Its value range is:

  Z ±8,388.607 mm or Z ±330.2599 inches.

* Tool radius.

  It will be given in the work units currently selected. Its maximum value is:

  R 1000.000 mm or R 39.3700 inches.

  The CNC will use this "R" value and the tool location code "F" (shape code) when applying tool compensation on the finishing passes while machining the programmed profile.

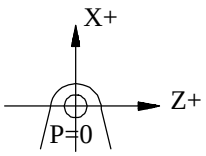
* Type of tool (location code "F").

  To indicate the type of tool being used, the CNC offers 10 different location codes (F0 thru F9).

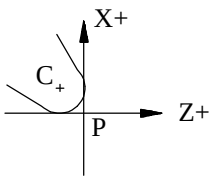
  This factor depends on the shape of the tool being used and on the sides of the cutter used to machine with.

  The following drawings illustrate the tool types commonly used on a lathe indicating for each one of them the center of the cutter (C) and its theoretical tip (point P).

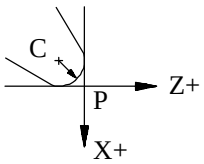
Code 0 and 9



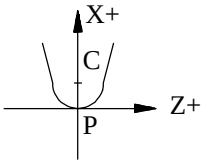
Code 1



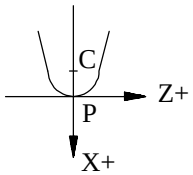
Code 7



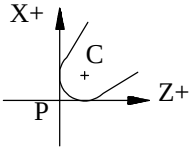
Code 2



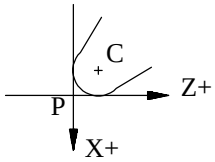
Code 6

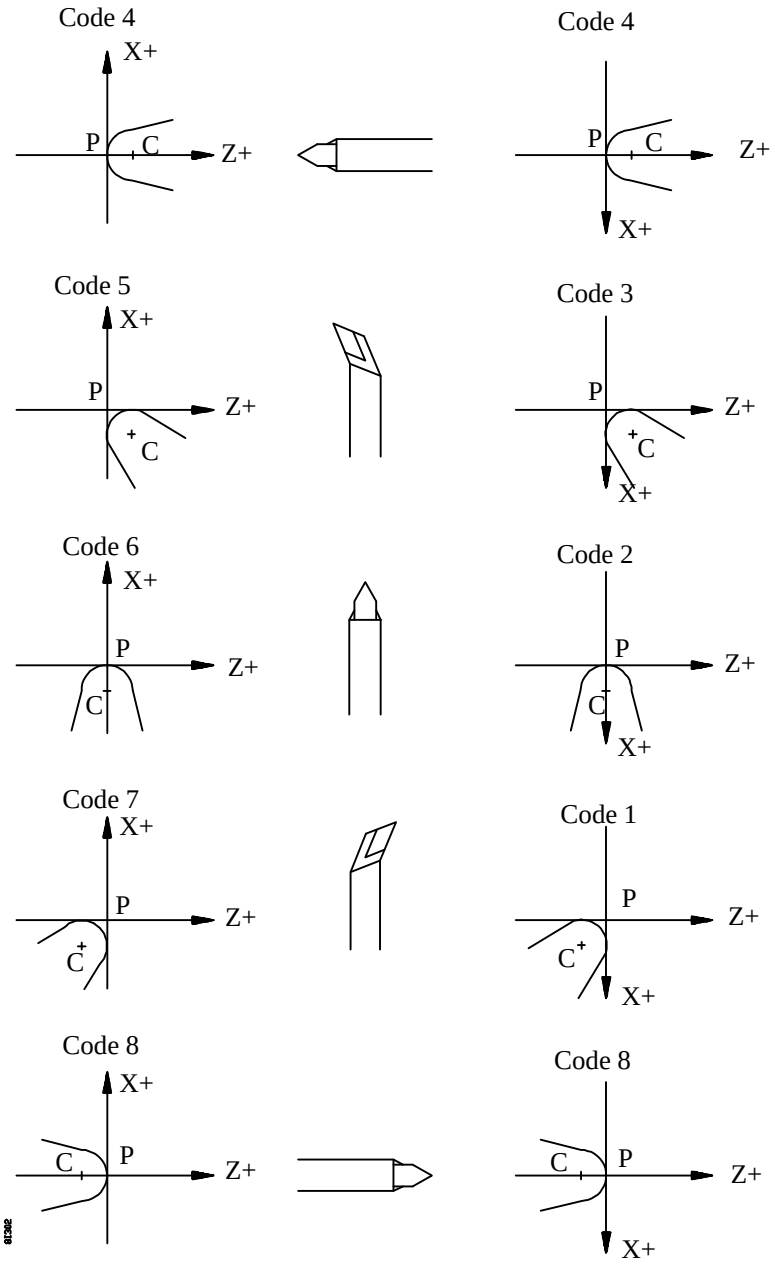


Code 3



Code 5




Code 4
X+
P
C
Z+
Code 4
P
C
Z+
X+
Code 5
X+
P
Z+
C
Code 3
P
Z+
C
X+
Code 6
X+
P
Z+
C
Code 2
P
Z+
C
X+
Code 7
X+
P
Z+
C
Code 1
P
Z+
C
X+
Code 8
X+
C
P
Z+
Code 8
C
P
Z+
X+

* Tool length wear along the X axis.

  It is given in diameters and in the work units currently selected. Its value range must be:

  I ±32.766 mm or I ±1.2900 inches

  The CNC will add this value to the nominal tool length along the X axis in order to calculate the real (total) tool length (X+I).

* Tool length wear along the Z axis.

  It is given in the work units currently selected. Its value range must be:

  K ±32.766 mm or K ±1.2900 inches

  The CNC will add this value to the nominal tool length along the Z axis in order to calculate the real (total) tool length (Z+K).

3.4.1.1 MODIFICATION OF TOOL DIMENSIONS

To clear the whole tool table by setting all its fields to 0, key in the following keystroke sequence: [R] [P] [N] [ENTER].

The 800T CNC has the "TOOL CALIBRATION" option described next. Once the tools have been calibrated, the CNC assigns to each tool offset the X and Z dimensions of the corresponding tool.

To complete the table values of a tool ("R" and "F") or to modify its dimensions; first, select, at the CNC, the corresponding tool offset by keying in the desired tool number and pressing [RECALL].

The CNC will show at the editing area the values currently assigned to that tool offset.

To modify these values, move the pointer with the up and down arrow keys until it is located on the current value. The new values must be keyed in over those currently assigned.

Once the new values have been keyed in, press [ENTER] so they are stored in memory.

To quit this mode, move the pointer to the right until it is out of the editing area and, then, press [END].

3.4.2 TOOL CALIBRATION

With this option it is possible to calibrate and load the dimensions of the tools onto the tool offset table of the CNC.

The CNC shows, at the lower right-hand side of the screen, a graphic aide to guide the operator during the tool calibration process and it will highlight the data requested at each moment

The tool calibration process consists of the following steps:

Once a work-piece of pre-established dimensions is set in the chuck of the lathe.

1.- The CNC requests the known dimension of the work-piece along the X axis.

Key in this value and press [ENTER]. It must be given in the work units of the machine (radius or diameter).

2.- The CNC requests the known dimension of the work-piece along the Z axis.

Key in this value and press [ENTER].

3.- The CNC requests the number of the tool to be calibrated.

Press [TOOL], then key in the desired tool number and then press  for the CNC to select it.

4.- Move the X axis of the machine with either the mechanical handwheels, the electronic handwheel or the jog keys until the tool tip touches the part along the X axis.

Then, press the key sequence: [X], [ENTER].

The CNC will display the work-piece dimension along the X axis and it will have calibrated the tool along this axis.

5.- Move the Z axis of the machine with either the mechanical handwheels, the electronic handwheel or the jog keys until the tool tip touches the part along the Z axis.

Then, press the key sequence: [Z], [ENTER].

The CNC will display the work-piece dimension along the Z axis and it will have calibrated the tool along this axis.

The CNC will then request a new tool to be calibrated. Repeat steps 3, 4 and 5 for each new tool to be added.

Press [END] to quit this mode and return to the standard display mode.

3.4.3 TOOL INSPECTION

With this option it is possible to interrupt the execution of program P99996 and inspect the tool to check its status and change it if necessary.

To do this, follow these steps:

a) Press  to interrupt the program.

b) Press [**TOOL**]

At this time, the CNC executes the miscellaneous function M05 to stop the spindle and it displays the following message on the screen:

JOG KEYS AVAILABLE
OUT

c) Move the tool to the desired position by using the JOG keys.

Once the tool is "out of the way", the spindle may be started and stopped again by its corresponding keys at the Operator Panel.

d) Once the tool inspection or replacement is completed, press [**END**].

The CNC will execute an M03 or M04 function to start the spindle in the direction it was turning when the program was interrupted.

the screen will display the following message:

RETURN
AXES OUT OF POSITION

"Axes out of position" means that they are not at the position where the program was interrupted.

e) Jog the axes to the program interruption position by means the corresponding jog keys. The CNC will not allow to move them passed (overtravel) this position.

When the axes are in position, the screen will display:

RETURN
AXES OUT OF POSITION
NONE

f) Press  to resume the execution of program P99996.

3.5 CYCLE FINISHING PASS AND SAFETY DISTANCE

When selecting this option, the CNC shows the values currently selected for the various parameters of the automatic operations.

These parameters are:

% △ Finishing pass = % of the roughing pass

It indicates the percentage (%) of the programmed roughing pass used as finishing pass.

It is an integer value. If assigned a value of "0", all machining passes (roughing and finishing) will be identical.

% F Finishing pass feedrate = % of the roughing pass feedrate

It indicates the percentage (%) of the programmed roughing pass feedrate used as finishing pass feedrate.

It is an integer value. If assigned a value of "0", the finishing feedrate will be the same as the one used for roughing.

T Finishing tool

With this CNC, it is possible to use one tool for the roughing operation and another one, indicated by this parameter, for the finishing operation. This parameter may be assigned an integer value between 0 and 32.

If this parameter is assigned a value of "0", the finishing operation will be carried out with the same tool used for the roughing operation.

Safety distance along the X axis during automatic operations.

It indicates the distance with respect to the "BEGIN" point where the tool will be positioned along the X axis during the approach.

Safety distance along the Z axis during automatic operations.

It indicates the distance with respect to the "BEGIN" point where the tool will be positioned along the Z axis during the approach.

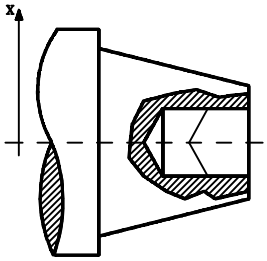
Once one of these parameters has been selected, the CNC highlights it and it requests, at the bottom of the screen, the new value to be assigned to this parameter.

After defining the new value, press [ENTER] in order to be assumed by the CNC.

3.6 OTHER AUTOMATIC OPERATIONS

When pressing AUX and selecting the option **[6]** corresponding to "OTHER CYCLES", or in DRO mode, +/- or LEVEL + (at the compact model) is pressed, the CNC will show the following machining cycles:

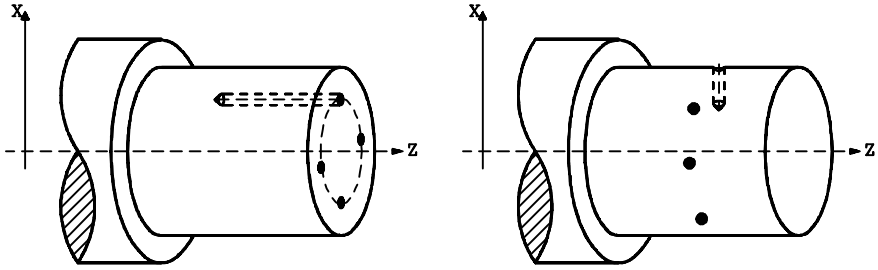
SIMPLE DRILLING. **TAPPING**



Consisting in drilling the face of the part only along its turning axis (center line).

MULTIPLE DRILLING.

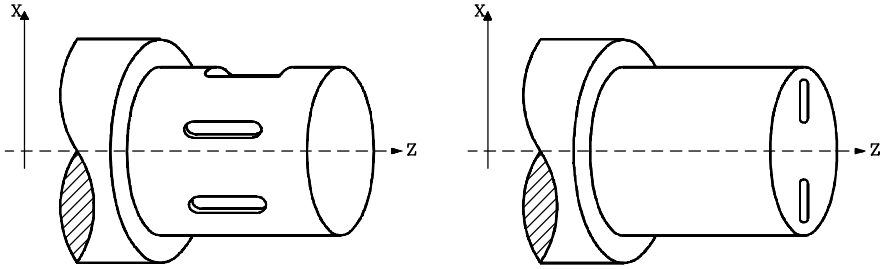
With this cycle it is possible to drill concentric holes on the face of the part (along the Z axis) as well as radial holes on its cylindrical side (along the X axis).



This feature requires spindle orientation and live tool. If the CNC lacks both features, it will not display this canned cycle.

SLOT MILLING.

With this cycle it is possible to mill radial slots on the face of a part (same Z, different X coordinates) or longitudinal slots parallel to the center line on the cylindrical side of the part (same X, different Z coordinates).



This feature requires spindle orientation and live tool. If the CNC lacks both features, it will not display this canned cycle.

A full description of these cycles can be found in the chapter on "Automatic Operations" of the "Operating Manual".

To quit the editing or execution of these cycles press any other operation key or:

* Press +/- or LEVEL + to return to the "Other automatic operations" menu.
* and then press +/- or LEVEL + again to return to the DRO mode.

3.7 AUXILIARY MODES

When selecting this option, the CNC shows the following menu:

1 - SPECIAL MODES
2 - PERIPHERALS
3 - LOCK / UNLOCK
4 - EXECUTION OF PROGRAM 99996
5 - EDITING PROGRAM 99996

After accessing one of these modes and operate with it, press **[END]** to quit. At this point, the CNC will show this menu again. Press **[END]** once more to return to the standard display mode.

3.8 SPECIAL MODES

When selecting this option, the CNC will request the password to access these auxiliary modes. This password is the following:

0101

Once this code has been entered, the CNC displays the following menu:

0 - TEST
1 - GENERAL PARAMETERS
2 - DECODED M FUNCTIONS
3 - LEADSCREW ERROR COMPENSATION

3.8.1 TEST

To select this option, press **[AUX]**, select the "Special Modes" option of the "Auxiliary Modes" menu, key in the password (**0101**) and press the key corresponding to "Test".

The CNC performs the General Test.

Once the test is completed, it is possible to test the logic inputs and outputs of the CNC, verify the checksum corresponding to the software version currently installed or perform the general test of the CNC again.

*** Testing the logic inputs and outputs of the CNC.**

Press **[7]** to access this option, the CNC will display the status of the logic INPUTS and it is possible to simulate the logic OUTPUTS of the CNC.

The inputs indicated by the letters "A" thru "M" have the meaning shown by the chart below and their status is indicated by either a "0" or a "1".

A value of "0" means that it receives 0V
A value of "1" means that it receives 24V

	CNC LOGIC INPUTS	
	Meaning	***Pin***
A	Start	17 (I/O 1)
B	Stop (It must be normally high)	16 (I/O 1)
C	Feed-Hold (It must be normally high)	15 (I/O 1)
D	Emergency Stop (It must be normally high)	14 (I/O 1)
E	Not being used at this time	
F	Z axis home switch	12 (I/O 1)
G	Not being used at this time	
H	X axis home switch	10 (I/O 1)
I	Manual (DRO mode)	19 (I/O 1)
J	Not being used at this time	
K	Not being used at this time	
L	Not being used at this time	
M	Not being used at this time	

The logic outputs are shown in two rows under the letters "A" thru "M" and with the meanings indicated by the chart below.

Each output may be assigned a "0" or "1" value which means:

If "0", the corresponding output will be at 0V (low).
If "1", the corresponding output will be at 24V (high).

Use the up and down arrow keys to move the cursor and select the desired outputs.

	TOP ROW		**BOTTOM ROW**	
	Meaning	***Pin***	***Meaning***	***Pin***
A	T Strobe	2 (I/O 1)	Output M01 (Decoded M)	3 (I/O 2)
B	S Strobe	3 (I/O 1)	Output M02 (Decoded M)	4 (I/O 2)
C	M Strobe	4 (I/O 1)	Output M03 (Decoded M)	5 (I/O 2)
D	Emergency	5 (I/O 1)	Output M04 (Decoded M)	6 (I/O 2)
E	Threading on / Cycle on	6 (I/O 1)	Output M05 (Decoded M)	7 (I/O 2)
F	Z axis Enable	7 (I/O 1)	Output M06 (Decoded M)	8 (I/O 2)
G	Reset	8 (I/O 1)	Output M07 (Decoded M)	9 (I/O 2)
H	X axis Enable	9 (I/O 1)	Output M08 (Decoded M)	10 (I/O 2)
I	MST01	27 (I/O 1)	Output M09 (Decoded M)	11 (I/O 2)
J	MST02	26 (I/O 1)	Output M10 (Decoded M)	12 (I/O 2)
K	MST04	25 (I/O 1)	Output M11 (Decoded M)	13 (I/O 2)
L	MST08	24 (I/O 1)	Output M12 (Decoded M)	25 (I/O 2)
M	MST10	23 (I/O 1)	Output M13 (Decoded M)	24 (I/O 2)
N	MST20	22 (I/O 1)	Output M14 (Decoded M)	23 (I/O 2)
O	MST40	21 (I/O 1)	Output M15 (Decoded M)	22 (I/O 2)
P	MST80	20 (I/O 1)	WORK mode Output	21 (I/O 2)

* **Checksum of the software version**

  Press **[8]** to access this option. The CNC will display the checksum of each EPROM memory corresponding to the software version currently installed at the CNC.

* **New general test of the CNC**

  Press **[9]** to access this mode. The CNC will perform the general self-test again.

After accessing one of these tests (inputs/outputs, checksum or general test), press [END] to return to the "AUXILIARY MODES" menu and press [END] again to return to the standard display mode.

3.8.2 GENERAL PARAMETERS

To select this option, press **[AUX]**, select the "Special Modes" option of the "Auxiliary Modes" menu, key in the password (**0101**) and press the key corresponding to "GENERAL PARAMETERS".

The CNC shows the machine parameter table.

The operator may view the following or previous pages by means of the up and down arrow keys.

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

To EDIT a parameter, key in the desired parameter number, press **[=]** and then, key in the desired value.

Depending on the type of machine parameter selected, the following types of values may be assigned:

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

Once the machine parameter has been set, press **[ENTER]** for that value to be entered on the table.

If when pressing **[=]**, the parameter being edited disappears from the screen, it means that the machine parameters are protected and cannot be modified.

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

* Press **[AUX]** and after selecting the "Lock/Unlock" option of the of the "Auxiliary Modes" menu...

* Key in: "P1111" and press **[ENTER]** to lock the access or: "P0000" **[ENTER]** to unlock it.

When the access to the machine parameter table is locked, only those parameters related to the RS 232 serial communications line may be edited.

Remember that once the desired parameters have been edited, **[RESET]** must be pressed or the CNC must be turned off and back on in order for the CNC to assume the new values.

The meaning of each parameter as well as the proper way to define them is described in another chapter of this manual.

3.8.3 DECODED "M" FUNCTIONS

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

* Press **[AUX]** and after selecting the "Lock/Unlock" option of the of the "Auxiliary Modes" menu...

* Key in: "P1111" and press **[ENTER]** to lock the access or: "P0000" **[ENTER]** to unlock it.

To select this option, press **[AUX]**, select the "Special Modes" option of the "Auxiliary Modes" menu, key in the password (**0101**) and press the key corresponding to "DECODED M FUNCTIONS".

The CNC shows the decoded "M" function table.

The operator may view the following or previous pages by means of the up and down arrow keys.

To view a particular M function, key in its number and press [RECALL]. The CNC will then show the page corresponding to that function.

To EDIT a parameter, key in the desired parameter number, press [=], key in the desired value and, then, press [ENTER] in order for this value to be entered on the table.

When executing an "M" function, outputs M1 thru M15 of connector I/O 2 will be modified depending on the setting of the corresponding function.

Two rows of "1s" and "0s" will appear to the right of each "M" function. The top row has 15 characters and the bottom one 17.

The **top-row** characters have the following meaning:

0 Indicates the outputs which do not change when the M function is executed. They keep the previous status.
1 Indicates the outputs which are **activated** (set to 24V) when the M function is executed.

The first 15 characters (from the left) of the **bottom row** have the following meaning:

0 Indicates the outputs which do not change when the M function is executed. They keep the previous status.
1 Indicates the outputs which are **deactivated** (set to 0V) when the M function is executed.

For example: If the M41 table (first spindle speed range selection) has been set as follows:

M41	000100100100100	(Outputs being activated)
	00100100100100100	(Outputs being deactivated)

The CNC will behave like this every time 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		
at 0V			x			x			x			x			x
Does not modify	x	x			x			x			x			x	

To activate the BCD outputs "MST01" thru "MST80" (pins 20 thru 27 of connector I/O 1) besides, the decoded ones, machine parameter "P606 bit 7" must be set to "0".

Bit 16 of the bottom row indicates whether the "M" function is executed at the beginning (0) of the block or at the end (1) once the programmed movements have been executed

Bit 17 of the bottom row determines whether the CNC must wait for confirmation from the electrical cabinet indicating that the execution of the M function has been completed (M-done) or not before resuming the execution of the program.

This confirmation is carried out by means of the "M-DONE" input at pin 15 of connector I/O 1. This bit may be set as follows:

0 The CNC **waits** for the "M-DONE" confirmation signal from the electrical cabinet.

1 The CNC **does not wait** for the "M-DONE" confirmation signal from the electrical cabinet.

Up to 32 M functions may be set. All those empty M-table positions are indicated as **M??**.

Whenever a previously defined M function is redefined, the new setting replaces the old one.

3.8.3.1 M FUNCTIONS SENT OUT IN BCD

The CNC generates a series of M functions to indicate to the electrical cabinet that a certain event has taken place.

The CNC activates the BCD outputs corresponding to the generated "M" function (pins 20 thru 27 of connector I/O 1).

If besides activating these BCD outputs, the decoded outputs are also to be activated (pins 3 thru 13 and 22 thru 25 of connector I/O 2), the corresponding M functions must be defined at the decoded M-function table.

The CNC generates the following M-functions in BCD:

M00 At the end of the execution of each step of the selected operation while in "SINGLE" mode.

M03 When pressing the key to start the spindle clockwise.

M04 When pressing the key to start the spindle counter-clockwise.

M05 When pressing the key to stop the spindle.

M10 When pressing the key to turn on the external device O1.

M11 When pressing the key to turn off the external device O1.

M12 When pressing the key to turn on the external device O2.

M13 When pressing the key to turn off the external device O2.

M14 When pressing the key to turn on the external device O3.

M15 When pressing the key to turn off the external device O3.

M20 To indicate that the part execution has ended.

For example, on a machine with a bar feeder, the PLC could control the machining of several parts in a row by using this function.

M30 When pressing the RESET key of the CNC.

M41 When selecting the first spindle speed range.

M42 When selecting the second spindle speed range.

M43 When selecting the third spindle speed range.

M44 When selecting the fourth spindle speed range.

3.8.4 LEADSCREW ERROR COMPENSATION

To select this option, press **[AUX]**, select the "Special Modes" option of the "Auxiliary Modes" menu, key in the password (**0101**) and press the key corresponding to "LEADSCREW ERROR COMPENSATION".

The CNC will display the Leadscrew Error Compensation table.

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: **[R] [P] [N] [ENTER]**.

There are up 30 parameter pairs for each axis. Parameters P0 thru P59 for the X axis and P60 thru P119 for the Z axis.

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%.

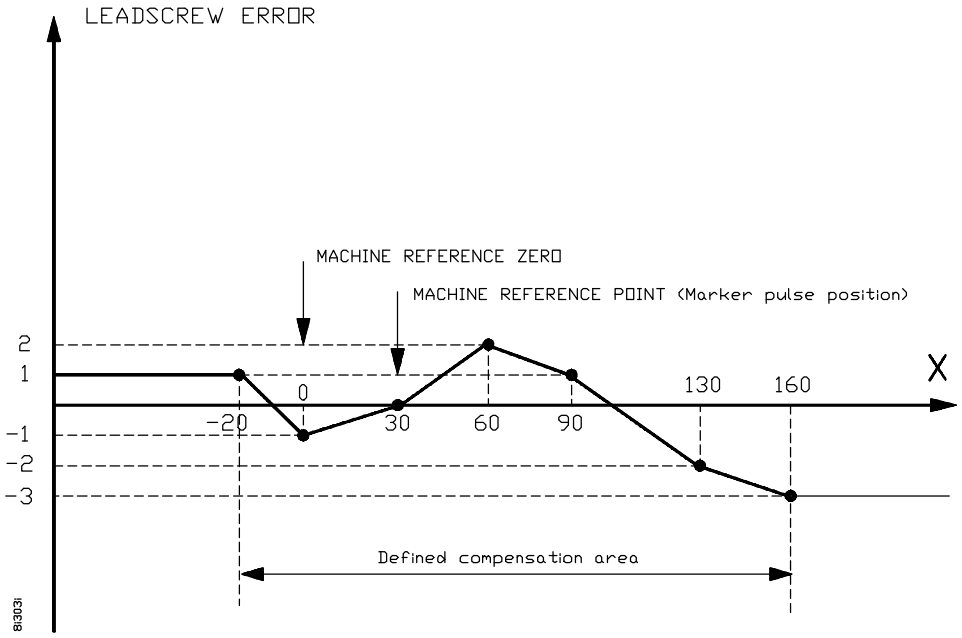
  Examples: If the distance between two consecutive points is 3 mm. the maximum difference of their relevant error values can be 0.090 mm.
  If the error difference between two consecutive points is the maximum (0.127mm), the distance between them cannot be smaller than 4.233mm.

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

3.9 PERIPHERALS

With this CNC it is possible to communicate with the FAGOR Floppy Disk Unit, with a general peripheral device or with a computer in order to transfer programs from and to one another. This communication may be managed either from the CNC when in the "**Peripheral mode**" or from the computer by means of FAGOR's DNC protocol in which case the CNC may be in any of its operating mode.

3.9.1 PERIPHERAL MODE

In this mode, the CNC may communicate with the FAGOR Floppy Disk Unit, with a general peripheral device or with a computer having a standard off-the-shelf communications program.

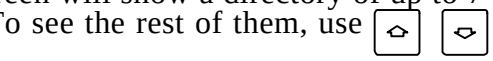
To access this mode, press **[AUX]** and after selecting "**Auxiliary modes**", press the key corresponding to the "**Peripherals**" option.

The CNC screen will show the following menu:

0 - RECEIVE FROM (Fagor) **FLOPPY DISK UNIT**
1 - SEND TO (Fagor) **FLOPPY DISK UNIT**
2 - RECEIVE FROM GENERAL DEVICE
3 - SEND TO GENERAL DEVICE
4 - (Fagor) **FLOPPY DISK UNIT DIRECTORY**
5 - (Fagor) **DELETE FLOPPY DISK UNIT PROGRAM**
6 - DNC ON/OFF

In order to use any of these options, the DNC mode must be **inactive.** If it is active (the upper right-hand side of the screen shows: **DNC**), press [**6**] (DNC ON/OFF) to deactivate it (the **DNC** letters disappear).

With options 0, 1, 2 and 3 it is possible to transfer machine parameters, the decoded M function table and the leadscrew error compensation table to a peripheral device.

The lower right-hand side of the CNC screen will show a directory of up to 7 part-programs of the 10 that may be stored. To see the rest of them, use 

To do this, key in the desired number when the CNC requests the number of the program to be transferred and press [**ENTER**].

P00000 to P99990	Corresponding to part-programs
P99994 and P99996	Special user program in ISO code
P99997	**For internal use and CANNOT be transmitted back and forth**
P99998	Used to associate texts to PLC messages
P99999	Machine parameters and tables

Atention:

The part-programs cannot be edited at the peripheral device or computer.

The CRT will show the message: "**RECEIVING**" or "**SENDING**" during the program transfer and the message: "**PROGRAM NUM. P23256** (for example) **RECEIVED**" or "**SENT**" when the transmission is completed.

When the transmission is not correct, it will display the message: "Transmission error" and when the data received by the CNC is not recognized (different format) by the CNC, it will issue the message: "Incorrect data received".

The CNC memory must be unlocked in order to perform any data transmission; if not so, the CNC will return to the menu of the peripheral mode.

When transmitting from a peripheral device other than a FAGOR Floppy Disk Unit, the following aspects must be considered:

* The program must begin with a "NULL" character (ASCII 00) followed by "%" "program number" (for example %23256) and a "LINE FEED" character (LF).

* Blank spaces, the carriage-return key and the "+" sign are ignored.

* The program must end with either 20 "NULL" characters (ASCII 00) or with one "ESCAPE" character or with one "EOT" character.

* Press [**CL**] to cancel the transmission. The CNC will issue the message: PROCESS ABORTED".

FLOPPY DISK UNIT DIRECTORY

This option displays the programs stored on the disk inserted in the FAGOR Floppy Disk Unit and the number of characters (size) of each one of them.

It also shows the number of free characters available (free memory space) on the tape.

DELETE FLOPPY DISK UNIT PROGRAM

With this option it is possible to delete a program contained at the FAGOR Floppy Disk Unit.

The CNC requests the number of the program to be deleted. After keying in the desired number, press [**ENTER**].

Once the program has been deleted, the CNC will display the message: "PROGRAM NUM: P____ DELETED".

It also shows the number of free characters on the disk (free memory space).

3.9.2 DNC COMMUNICATIONS

To be able to use this feature, the DNC communication must be active (the upper right-hand side of the screen shows: DNC). To do this the corresponding parameters [P605(5,6,7,8); P606(8)] must be set accordingly and option [**6**] of the "**Peripherals**" mode selected if it was not active.

Once active and by using the **FAGORDNC** application software supplied, upon request, in floppy disks it is possible to perform the following operations from the computer:

. Obtain the CNC's part-program directory.
. Transfer part-programs and tables from and to the CNC.
. Delete part-programs at the CNC.
. Certain remote control of the machine.

Atention:

At the CNC any operating mode may be selected.

3.10 LOCK/UNLOCK

With this option it is possible to lock/unlock the machine parameters and the part-program memory in order to protect them against accidental manipulation.

To access this mode, press **[AUX]** and after selecting "**Auxiliary modes**", press the key corresponding to the "**LOCK / UNLOCK**" option.

The codes used to do this are:

P0000 [ENTER] Unlocks the machine parameters.

P1111 [ENTER] Locks the machine parameters.

N0000 [ENTER] Unlocks part-program memory.

N1111 [ENTER] Locks part-program memory.

PF000 [ENTER] Erases the contents of all arithmetic parameters (data of the automatic operations) and sets them to "0".

3.11 EXECUTION / SIMULATION OF PROGRAM P99996

To select this option, press **[AUX]** and after selecting "Auxiliary Modes", press the key corresponding to "EXECUTION OF PROGRAM P99996".

Program P99996 is a special user program in ISO code. It may be edited (written) at the CNC or at a PC and, then, be sent to the CNC via the Peripherals option.

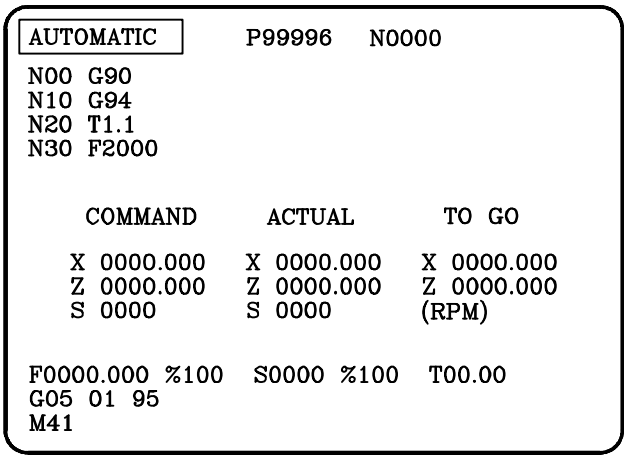
Once this option has been selected, it is possible to execute or simulate this program.

To simulate program P99996, press SIMUL at the compact CNC model and AUX at the modular CNC model.

The way to operate in either case is described next.

3.11.1 EXECUTION OF PROGRAM P99996

When selecting the option: "Execution of program P99996", the CNC displays the following information:



The top line shows the message "AUTOMATIC", the program number (P99996) and the number of the first block of the program or that of the block being in execution.

Then, the CRT shows the contents of the first program blocks. If the program is being executed, the first block of the list will be the one being executed at the time.

The position values along X and Z indicate the programmed values (COMMAND), the current position (ACTUAL) and the distance remaining (TO GO) for the axes to reach the "command" position.

It also shows the selected spindle speed, programmed value multiplied by the active %S override (COMMAND), and the real spindle speed (ACTUAL).

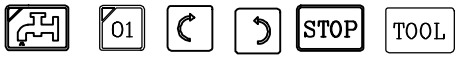
The bottom of the screen shows the machining conditions currently selected. The programmed feedrate F, the % F override, the programmed spindle speed S, the %S override, the programmed Tool as well as the active G and M functions.

To execute program P99996, proceed as follows:

* Select, if so desired, the first block to be executed indicated at the upper right-hand corner (by default: N0000), by keying in N**** [RECALL] and...
* press 

To interrupt the program, press 

Once interrupted, the following keys are enabled:



To resume execution, press 

3.11.1.1 TOOL INSPECTION

With this option it is possible to interrupt the execution of program P99996 and inspect the tool to check its status and change it if necessary.

To do this, follow these steps:

a) Press  to interrupt the program.

b) Press [**TOOL**]

At this time, the CNC executes the miscellaneous function M05 to stop the spindle and it displays the following message on the screen:

JOG KEYS AVAILABLE
OUT

c) Move the tool to the desired position by using the JOG keys.

Once the tool is "out of the way", the spindle may be started and stopped again by its corresponding keys at the Operator Panel.

d) Once the tool inspection or replacement is completed, press [**END**].

The CNC will execute an M03 or M04 function to start the spindle in the direction it was turning when the program was interrupted.

The screen will display the following message:

RETURN
AXES OUT OF POSITION

"Axes out of position" means that they are not at the position where the program was interrupted.

e) Jog the axes to the program interruption position by means the corresponding jog keys. The CNC will not allow to move them passed (overtravel) this position.

When the axes are in position, the screen will display:

RETURN
AXES OUT OF POSITION
NONE

f) Press  to resume the execution of program P99996.

3.11.1.2 EXECUTION MODES

With this CNC it is possible to execute program P99996 from beginning to end or one block at a time by pressing SINGLE

The top line of the screen shows the operating mode currently selected either "Automatic" or "Single Block".

To switch from one mode to the other, press SINGLE again

Once the execution mode has been selected, press [1]

3.11.1.3 CNC RESET

With this option it is possible to reset the CNC setting it to the initial conditions established by the machine parameters. When quitting this operating mode, the CNC displays the DRO mode.

To reset the CNC, simply interrupt the program execution, if running, and press RESET

The CNC will request confirmation of this function by blinking the message: **"RESET?"**.

To go ahead with reset, press RESET again; but to cancel it, press CLEAR

3.11.1.4 DISPLAYING PROGRAM BLOCKS

To display the previous or following blocks to those appearing on the screen, press:

- ⌃ Displays the previous blocks
- ⌄ Displays the following blocks

Atention:

Bear in mind that P99996 always starts executing from the currently selected starting block, regardless of the blocks currently displayed on the screen. By default, this starting block is N000.

To select another starting block, press N (block number) **[RECALL]**. For example: N110 RECALL.

3.11.1.5 DISPLAY MODES

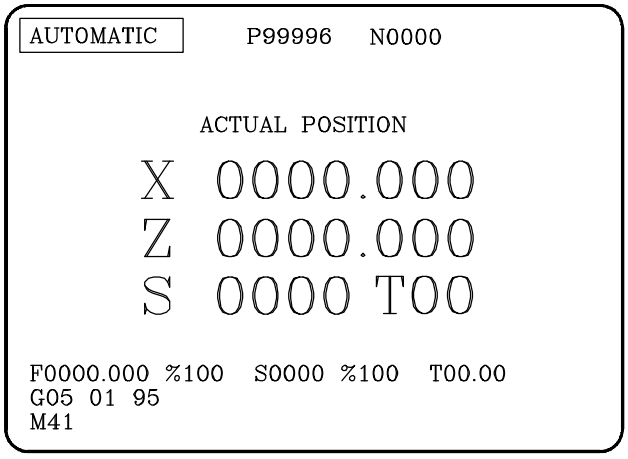
There are 4 display modes which can be selected by means of the following keys:

[0] STANDARD
[1] ACTUAL POSITION
[2] FOLLOWING ERROR
[3] ARITHMETIC PARAMETER

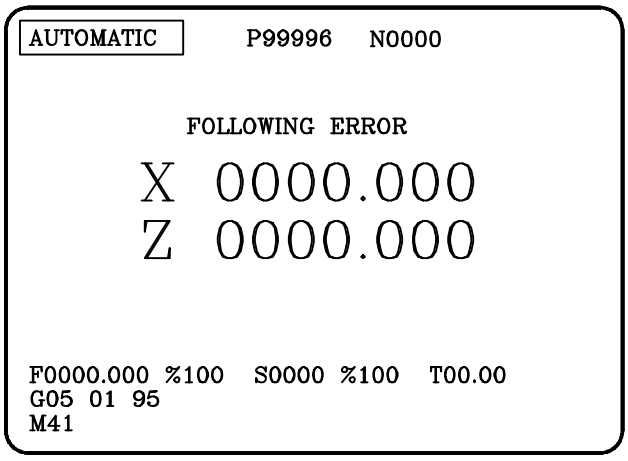
STANDARD display mode

It is the mode described before. When accessing the "Execution of program P99996" option, the CNC selects this display mode.

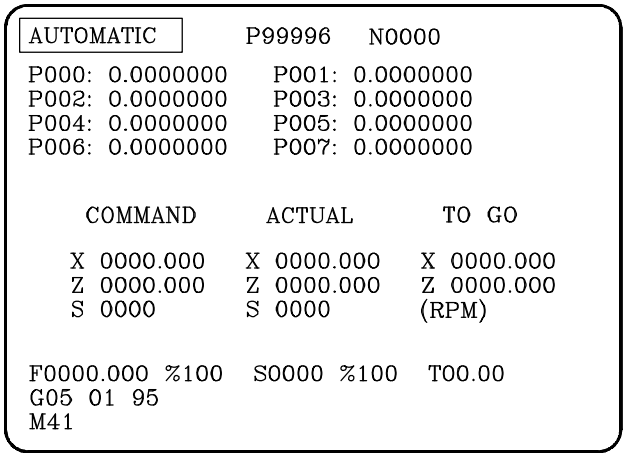
ACTUAL POSITION display mode



FOLLOWING ERROR display mode



ARITHMETIC PARAMETERS display mode



This mode shows a group of 8 arithmetic parameters. To view the previous and following ones, use these keys:

- Displays the previous parameters
- Displays the following ones

The value of each parameter may be expressed in one of the following formats:

P46 = -1724.9281	Decimal notation
P47 = -.10842021 E-2	Scientific notation

Where "E-2" means 10^{-2} (1/100). Therefore, the two types of notation for the same parameter below have the same value:

P47= -0.001234	P47= -0.1234 E-2
P48= 1234.5678	P48= 1.2345678 E3

3.11.2 SIMULATION OF PROGRAM P99996

With this CNC it is possible to check program P99996 in dry-run before executing it.

To do this, press [SIMUL] at the compact model and [AUX] at the modular model.

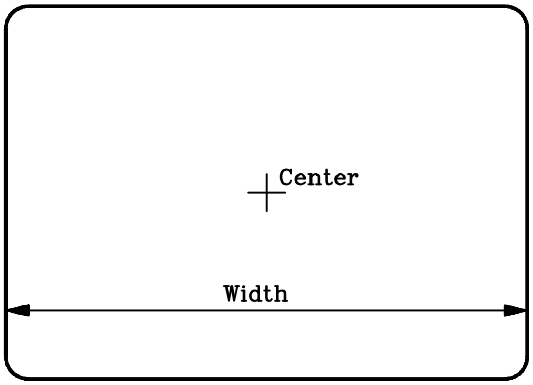
The screen shows a graphic page.

The lower left-hand side of the screen shows the axes of the plane.

To define the display area proceed as follows:

* Press [SIMUL] at the compact model and [AUX] at the modular model.
* Indicate the XZ coordinates of the position to be displayed at the center of the screen.
* Set the width of the display area.

After keying in each value, press **[ENTER]**.



To check the part, press [1] . This will start the corresponding graphic simulation.

Press **[CLEAR]** to clear the screen, and **[END]** to quit the simulation mode.

3.11.2.1 ZOOM FUNCTION

With this function, it is possible to enlarge or reduce the whole graphic-representation or part of it. To do this, the simulation of the program must be either interrupted or finished.

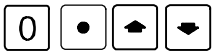
Press **[Z]**. The screen will show a rectangle over the original drawing. This rectangle represents the new display area to be enlarged or reduced.

To change the dimensions of the rectangle, use these keys:

Reduces the size of the rectangle (zoom in).

Increases the size of the rectangle (zoom out).

Use the following keys to move the zoom window around:

At the compact model 
At the modular model 

To set the area selected with the zoom window as he new display area, press **[ENTER]**.

To see he selected area enlarged or reduced while keeping the previous display area values, press 

at the compact model and 

at the modular model.

The area contained in the zoom window will now fill the whole screen.

To return to the previous display area (prior to the zoom), press **[END]**.

To use the zoom again, just press **[Z]** and proceed as before.

To quit the ZOOM function and return to the graphic representation, press **[END]**.

3.12 EDITING PROGRAM 99996

Program 99996 is a special ISO-coded user program. It can be edited either in this operating mode or at a PC and then sent out to this CNC.

To select this option, press **[AUX]** and after selecting "Auxiliary Modes", press the key corresponding to "EXECUTION OF PROGRAM P99996".

The CNC displays the editing page for this program.

If the program is currently being edited, the CNC shows a group of program blocks (lines).

Use the [▲] [▼] to display the display the previous and following lines.

To edit a new line, follow this procedure:

1.- If the program line number appearing at the bottom of the screen is not the desired one, clear it by pressing **[CL]** and key in the desired line number.

2.- Key in all the pertinent data for that line and press **[ENTER]**.

The programming format to be used is described in the programming manual. The keys on the front panel may be used: **[X]**, **[Z]**, **[S]**, **[F]**, **[N]** as well as: **[TOOL]** for T, [P] for P, [R] for R and [A] for A.

However, since some function keys are missing (G, M, I, K), an assisted editor is also available.

To access it, press **[AUX]**. After analyzing the syntax of what has been edited so far, the CNC will display, one by one, all the functions which can be edited at the time.

Press **[CL]** to delete characters.

To modify a previously edited line, proceed as follows:

1.- If the program line number appearing at the bottom of the screen is not the desired one, clear it by pressing **[CL]** and key in the desired line number.

2.- Press **[RECALL]**. The bottom of the screen of the CNC, editing area, will show the contents of that line.

3.- Use one of these methods to modify the contents:

a) Use the **[CL]** key to delete characters and edit it as described above.

b) Use the [▲] [▼] keys to position the cursor over the section to be modified and use the **[CL]** key to delete characters or **[INC/ABS]** to insert data.

While in the data inserting mode, the characters behind the cursor appear blinking. It is not possible to use assisted programming (the **[AUX]** key).

Key in all the desired data and press **[INC/ABS]**. If the syntax of the new line is correct, the CNC will display it without blinking and, if not, it will show it blinking until it is edited correctly.

4.- Once the line has been modified, press **[ENTER]**. The CNC will assume it replacing the previous one.

To delete a program line, proceed as follows:

1.- If the program line number appearing at the bottom of the screen is not the desired one, clear it by pressing **[CL]** and key in the desired line number.

2.- Press  and the CNC will delete it from memory.

4. MACHINE PARAMETERS

Atention:

All unused machine parameters must be set to "0" to guarantee the proper functioning of the 800T 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.

4.1 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 the access to the machine parameters, to the decoded M function table and to the leadscrew error compensation table, proceed as follows:

* Press **[AUX]** and after selecting the "Lock/Unlock" option of the of the "Auxiliary Modes" menu...

* Key in: "P1111" and press **[ENTER]** to lock the access or: "P0000" **[ENTER]** to unlock it.

When the access to the machine parameter table is locked, only those parameters related to the RS-232C serial communications line may be edited.

To enter the machine parameter values via the keyboard, press the following keystroke sequence:

[AUX]	(SPECIAL FUNCTIONS)
[5]	(AUXILIARY MODES)
[1]	(SPECIAL MODES)
[0] [1] [0] [1]	(Password)
[1]	(MACHINE PARAMETERS)

4.2 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:

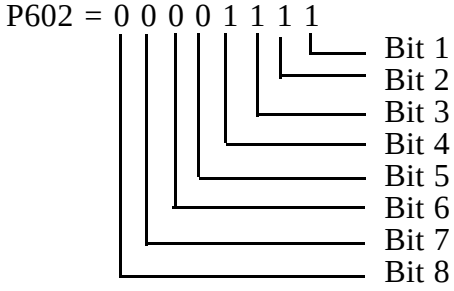
*	A number	P111 = 30000
*	A group of 8 bits	P602 = 00001111
*	A character	P105 = N

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:



4.3 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

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.

P617(2) Display of the following error.

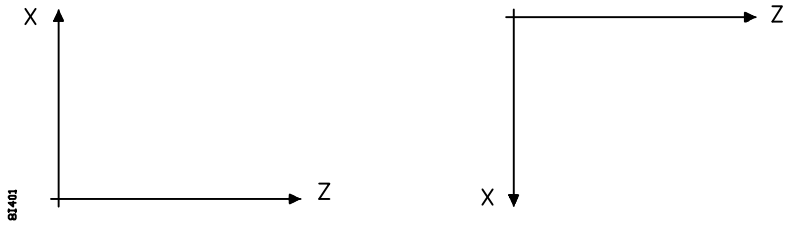
0 = The following error **is not** displayed
1 = The following error **is** displayed.

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

Atention:

The following error is displayed in the work mode and in point-to-point movements  . It is also necessary to set this parameter to "1", press **[R]** for the CNC to display this value.

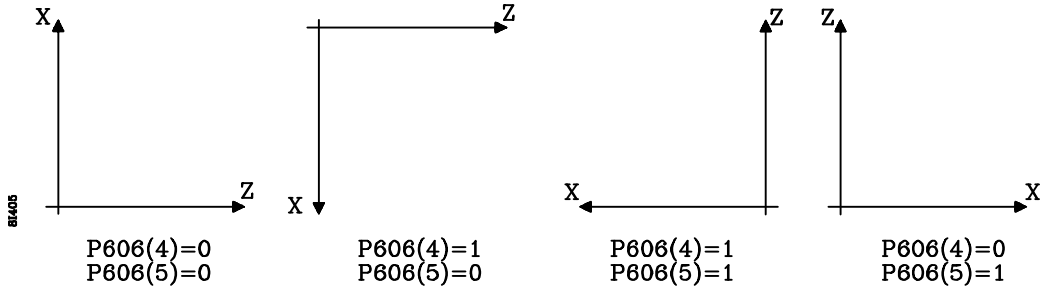
P600(1) Orientation of the axes



P600(1) = 0 P600(1) = 1

P606(4,5) Axis orientation in the graphic representation

By means of these parameters, it is possible to set the orientation of the axes in the graphic representation so they match the orientation of the axes on the machine.



P601(1) Availability of an automatic spindle speed range changer

It indicates whether or not the CNC automatically generates the M functions for the spindle range change (M41, M42, M43, M44) since the machine is using an automatic spindle range changer.

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

Enter the values:

0 = The machine **is not** using an automatic spindle range changer.
1 = The machine **is** using an automatic spindle range changer.

P617(3) Availability of an automatic tool changer

It indicates whether or not the machine uses an automatic tool changer managed by the CNC.

0 = The machine **is not** using an automatic tool changer.
1 = The machine **is** using an automatic tool changer.

4.3.1 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).

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 an automatic operation (CYCLE ON) is being executed or when a "BEGIN-START", "END-START" type command is being executed.

P606(7) M function output also 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.

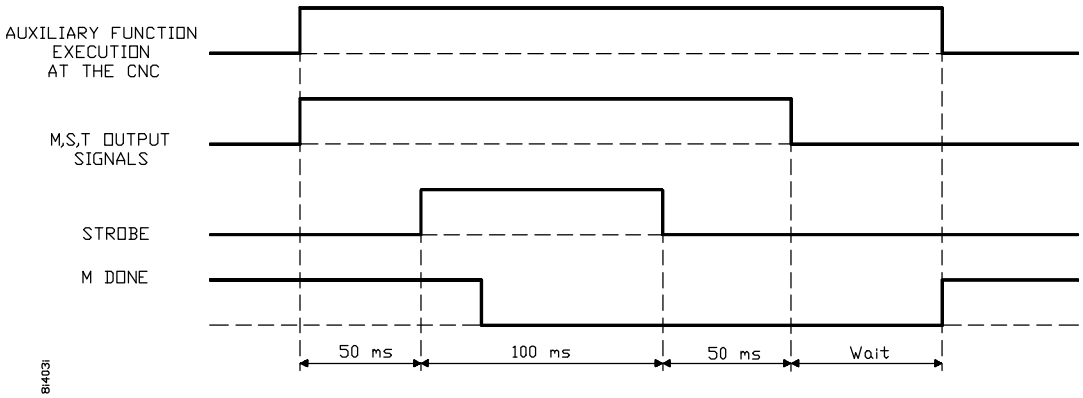
0 = The M function is **also** sent out in BCD.
1 = The M function is **not** sent out in BCD

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”

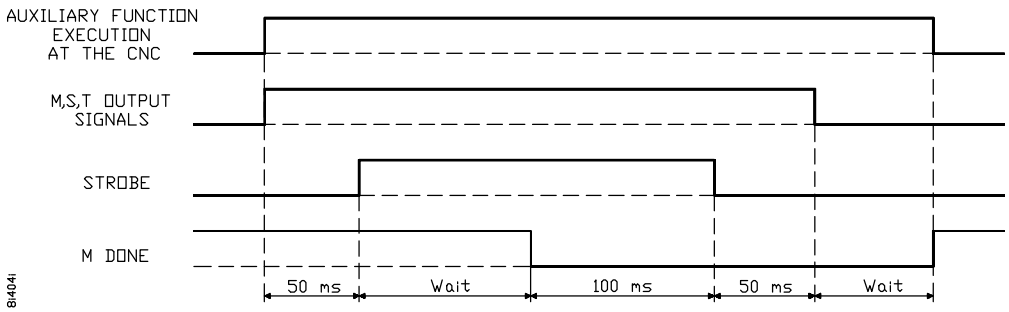
50 milliseconds after having sent the M, S or T BCD signals out to the electrical cabinet, it sends out the corresponding "Strobe" signal.

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).

4.3.2 HANDWHEEL PARAMETERS

P621(7) The machine has mechanical handwheels

Indicates whether the machine has or not mechanical handwheels. It must be borne in mind that when using electronic handwheels, no mechanical handwheels can be used.

0 = The machine has mechanical handwheels
1 = The machine has **no** mechanical handwheels

P823 Delay before opening the loop

When the machine has mechanical handwheels, "P621(7)=0", the axes cannot be continuously controlled "P105=N" and "P305=N".

P823 indicates the time delay from the moment the axes reach position to when the CNC opens the position loop of the axes.

It is expressed by an integer between 0 and 255 in units of 10 milliseconds.

Value of "0" = No delay is applied.
Value of 1 = 10 msec.
Value of 10 = 100 msec.
Value of 255 = 2550 msec.

P622(3) The machine has one single electronic handwheels

When the machine has no mechanical handwheels, "P621(7)=1", machine parameter "P622(3)" indicates whether 1 or 2 electronic handwheels are being used.

0 = 2 electronic handwheels are being used.
1 = 1 electronic handwheel is being used.

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

Indicates whether the electronic handwheel connected to A6 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, P621(6) Counting direction of the two handwheels (1st and 2nd respectively)

They set the counting direction of the handwheels. If correct, leave them as they are; otherwise, assign the other value. 0 = NO and 1 = YES.

P602(1), P621(3) Measuring units for the two handwheels (1st, 2nd)

They indicate whether the CNC considers the handwheel pulses to be in mm or in inches.

0 = Millimeters.
1 = Inches.

P501, P621(1,2) Feedback resolution of the two handwheels (1st, 2nd respectively)

They indicate the counting resolution of the two handwheels.

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

To set the first handwheel resolution, set P501 to the desired value and use the chart below to set it for the second handwheel.

P621(2)	P621(1)	Resolution
0	0	1
0	1	2
1	0	5
1	1	10

P602(4), P621(5) Multiplying factor for the feedback signals from the two handwheels (1st and 2nd respectively).

They indicate the x2 or x4 multiplying factor to be applied to the feedback signals supplied by the handwheels.

0 = x4.
1 = x2.

Example:

If the first 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.

P617(5) Handwheel inactive if Feedrate Override Switch not in handwheel position

Indicates whether it is possible or not to move the axes by means of the electronic handwheel when the FOS is not in a handwheel position.

0 = The axes **can** be moved by means of the handwheel when the FOS is not in a handwheel position just as if it were in the "**x1**" position.

1 = The axes **cannot** be moved with the handwheel if the FOS is not in a handwheel position. The handwheel is inactive.

P622(1) Handwheel settings established by PLC

Indicates whether the CNC assumes the handwheel positions of the manual feedrate override switch or the PLC outputs O44 and O45 when jogging the axes with the handwheel.

0 = Asssumes the Manual Feedrate Override Switch positions.

1 = Assumes the setting of PLC outputs O44 and O45.

O44	O45	
0	0	Assumes MFO switch positions
1	0	Equivalent to position x1 of MFO switch
0	1	Equivalent to position x10 of MFO switch
1	1	Equivalent to position x100 of MFO switch

4.3.3 OPERATING-MODE 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.

0 (NO) = 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 the  key or another JOG key is pressed.

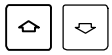
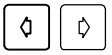
1 (YES) = Pulsating mode. The axis will move while keeping the corresponding JOG key pressed.

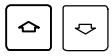
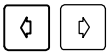
P601(5) Inhibiting the START key.

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.

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).

P622(2) Incremental JOG movements in radius or diameter

It indicates whether the selected work units (radius /diameter) are taken into account or not.

0 = The work units are ignored.
1 = The work units are taken into account.

Example: X axis in diameter. MFO switch position at 100.

	Initial position	Real movement	Final display
P622(2)=0	0.000	1.000	2.000
P622(2)=1	0.000	0.500	1.000

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

0 (NO) = The switch is ignored and the rapid moves are carried out **at 100%**.
1 (YES) = The CNC applies the % override indicated by the switch (between **0%** and **100%** even when indicating 110% and 120%).

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.

P617(6) The "rapid jog" key applies a feedrate override range over 100 %

This parameter goes into effect on software version 3.3 and up. It indicates the type of jogging feedrate override that will be applied while pressing this key 

0 = While keeping this key pressed, the CNC will apply a feedrate override amount according to the table below.

% selected	0	2	4	10	20	30	40	50	60	70	80	90	100	110	120
% applied	0	102	104	110	120	130	140	150	160	170	180	190	200	200	200

When this key is released, the amount of override will return to its face value (0 through 120%).

1 = While keeping this key pressed, the CNC will apply the maximum feedrates set by machine parameters: P111 and P311.

It may be interesting to set this parameter to "0" on short travel machines and to "1" on large ones.

CNCs with a software version prior to 3.3 behave as when setting "P617(6) = 0"

P617(8) Rounding possibility when defining a profile

0 = Rounding is not possible when defining a profile. The profile must be defined only by straight sections. Up to 12 points may be defined.

1 = Rounding is possible when defining a profile. Up to 9 points may be defined as well as roundings associated with points: P2, P3, P4, P5, P6 and P7.

4.3.4 TOOL PARAMETERS

P700 Number of tools

It is given by an integer between 0 and 32.

P900 X coordinate for the tool change position
P901 Z coordinate for the tool change position

When performing a tool change, it is recommended to use a position away from the part to do it, especially when running repetitive parts.

This tool change position may be established by machine parameter "P900" for the X axis coordinate and "P901" for the Z axis coordinate. Every time a tool has to be changed, the machine will move to this position automatically.

Range or values: ± 8388.607 millimeters.
± 330.2599 inches.

Also, when calibrating a tool, the CNC will ignore the "P900" and "P901" position values and the tool change will be done at the point where it is requested.

If parameters P900 and P901are set to "0", the CNC acts as follows:

* If while executing a part-program, a tool change is required, it will take place at the starting point of the part-program execution.

* If the tool change is required while being in any other mode, the CNC will not move the axes and the tool change will be carried out right where it was requested.

P617(3) The machine has an automatic tool changer

It indicates whether or not the CNC must manage the tool changer when a new tool is programmed.

0 = The machine **does not** use an automatic tool changer
1 = The machine **uses** an automatic tool changer

P730 Subroutine associated with the T function

The CNC takes this parameter into consideration when executing the ISO-coded user program 99996.

It indicates the number of the standard (non-parametric) subroutine to be executed by the CNC whenever a tool is selected in the execution of program 99996.

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

Atention:

If a subroutine is associated to the T function, nothing else must be programmed after the T. Otherwise, the CNC will issue the corresponding error message.

The subroutine is executed before the T function. That is, it executes the subroutine first and, then, it assumes the new T.

The associated subroutine to the T function must be defined in one of the special ISO-coded user programs, P99994 and P99996.

When setting "P730=0" and not using an automatic tool changer, "P617(3)=0", the CNC displays the message: "TOOL CHANGE" (in English for all languages) and interrupts program execution.

4.3.5 RS232C SERIAL LINE PARAMETERS

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 units of baud.

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.

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.

0 = None
1 = ODD.
2 = EVEN.

P3 Stop bits

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

1 = 1 Stop bit.
2 = 2 Stop bits.

P605(5) DNC active

It determines whether the CNC can work with the DNC protocol or not

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

P605(6) Communication settings for the FAGOR Floppy Disk Unit or Cassette.

P605(6)=1 For Floppy Disk Unit. The CNC uses the values set for machine parameters P0, P1, P2 and P3.

P605(6)=0 For Cassette Unit. The CNC does not modify the P0, P1, P2 and P3 settings but uses the values corresponding to the FAGOR Cassette Unit.

Baudrate: 13,714 Baud
Data bits: 7
Parity: Even
Stop bits: 1

Atention:

In DNC and Peripheral communications, use 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 acitve 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 trasmission 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.

5. *MACHINE PARAMETERS FOR THE AXES*

Atention:

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

P100, P300 Sign of the analog output for the X and Z axes.

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

Possible values: Press [0] for NO and [1] for YES.

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

P101, P301 Counting direction of the X and Z feedback devices.

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

Possible values: Press [0] for NO and [1] for YES.

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

P102, P302 Jogging direction for the X and Z 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: Press [0] for NO and [1] for YES.

5.1 MACHINE PARAMETERS FOR AXIS RESOLUTION

P103, P303 Feedback (counting) resolution for the X and Z axes.

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

P602(3), P602(2) Feedback units for the X and Z axes.

They determine whether the resolution values set by P103 and P303 are in millimeters or inches.

0 = Millimeters.
1 = Inches.

P106, P306 Type of feedback signals being used for the X and Z axes.

They determine whether square-wave or sine-wave feedback signals are being used.

0 (NO) = Square-wave signals being used.
1 (YES) = Sine-wave signals being used.

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

P602(6), P602(5) Multiplying factor for the feedback signals of the X and Z axes.

They determine the x2 or x4 multiplying factor to be applied to the feedback signals of the X and Z axes.

0 = x4.
1 = x2.

Remember that in the case of sine-wave signals, they will also be multiplied by 5.

When using FAGOR linear scales, set these parameters to "0".

Possible obtainable resolutions:

Signal type and pitch	P602(6)	Resolution	P103
Square-wave / 20µm	x4	5 micron	5
	x2	10 micron	10
Square-wave / 40µm	x4	10 micron	10
	x2	20 micron	—
Sine-wave / 20µm	x4	1 micron	5
	x2	2 micron	10
Sine-wave / 40µm	x4	2 micron	10
	x2	4 micron	—

P619(1), P619(2) Special x5 multiplying factor for sine-wave feedback signals of the X and Z axes.

They determine whether the CNC always applies a x5 multiplying factor to the sine-wave feedback signals for the X and Z axes or only when using 1µm or 2µm axis resolution.

0 = Only applied when using 1µm or 2µm axis resolution
1 = Applied always

P604(2), P604(1) Binary encoder for the X and Z axes.

They determine whether the X and/or the Z axes use a BINARY ROTARY ENCODER (1024/2048 pulses/rev.)

0 = No. It is not a binary encoder
1 = Yes. It is a binary encoder.

P604(7), P604(6) Equivalence of the binary encoder used for the X and Z 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.

0 = Obtain 1250 count with a 1024 encoder or 2500 count with 2048 encoder.
1 = Obtain 1000 count with a 1024 encoder or 2000 count with 2048 encoder.

The axis resolution (P103, P303) is calculated using the obtained pulse count (1000, 1250, 2000, 2500) and not the binary one (1024, 2048).

5.2 MACHINE PARAMETERS FOR AXIS ANALOG OUTPUTS

P117, P317 Minimum analog output for the X and Z 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 Delay between Enable and Analog output for the X and Z 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.

0 (NO) = There is no delay between the two signals
1 (YES) = 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 In-position zone for the X and Z 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 Continuous control of the X and Z 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.

0 (NO) = The Enable signal disappears.
1 (YES) = The Enable signal is maintained (Continuous control).

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

5.3 MACHINE PARAMETERS FOR THE TRAVEL LIMITS OF THE AXES

P107, P307 Positive travel limit for the X and Z axes
P108, P308 Negative travel limit for the X and Z 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
± 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.

Note: When using two electronic handwheels (no mechanical handwheels), the CNC ignores these travel limits.

5.4 MACHINE PARAMETERS FOR THE LEADSCREWS

P109, P309 Amount of leadscrew backlash for the X and Z 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) Sign of the backlash for the X and Z axes.

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

0 = Positive sign.
1 = Negative sign.

P113, P313 Additional analog pulse for X and Z 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".

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.

5.5 MACHINE PARAMETERS FOR AXIS FEEDRATES

P110, P310 Maximum programmable feedrate (F) for the X and Z axes.

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

P111, P311 G00 feedrate for rapid traverse of X and Z 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:

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

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:

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

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.

5.6 MACHINE PARAMETERS FOR AXIS CONTROL

P114, P314 Proportional gain, K1 for X and Z .

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.

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

Note: Refer to the chapter on "concepts" for calculation and adjustment of K1, K2 and gain break-point values.

P115, P315 Gain break-point for the X and Z 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, P310).

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

Note: Refer to the chapter on "concepts" for calculation and adjustment of K1, K2 and gain break-point values.

P116, P316 Proportional gain K2 for the X and Z 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.

$$\text{Analog} = (K1 \times Ep) + [K2 \times (\text{Following error} - Ep)]$$

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).

Note: Refer to the chapter on "concepts" for calculation and adjustment of K1 and K2 and gain break-point values.

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, and P318). 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 and P318), the CNC activates the corresponding enable signal in order to recover the drifted distance.

5.7 MACHINE PARAMETERS FOR MACHINE REFERENCE ZERO

P119, P319 Coordinate of Machine Reference Point for X and Z axes

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

Possible values: ± 8388.607 millimeters.
± 330.2599 inches.

Note: For further information, refer to the chapter on "concepts".

P618(8), P618(7) Direction of the home search along the X and Z axes.

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

0 = Positive direction.
1 = Negative direction.

Note: For further information, refer to the chapter on "concepts".

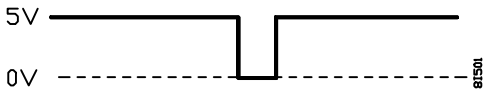
P600(7), P600(6) Type of machine reference pulse for X and Z.

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.



P600(5), P600(4) Home switch for the X and Z 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.

Note: For further information, refer to the chapter on "concepts".

P112, P312 1st home searching feedrate for the X and Z axes.
P807, P808 2nd home searching feedrate for the X and Z 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.).

Note: For further information, refer to the chapter on "concepts".

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.

5.8 MACHINE PARAMETERS FOR ACCELERATION/DECELERATION OF THE AXES

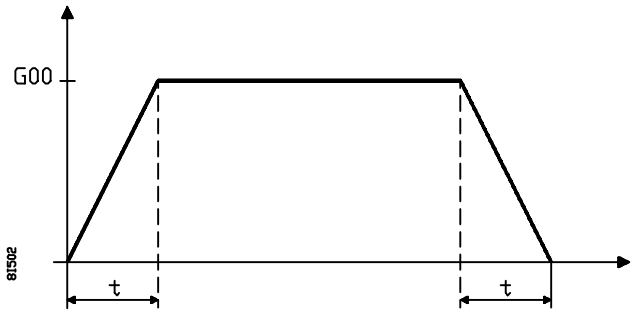
5.8.1. LINEAR ACCELERATION/DECELERATION

This type of ramp is applied mainly on movements carried out at the maximum feedrate (set by machine parameters P110 and P310), although it may also be used on linear interpolations.

P712, P713 ACC./DEC. Control of the X and Z 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, P311) 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.

It indicates whether the acc./dec. ramps are to be applied on all linear interpolations or only when they are carried out at the maximum feedrate set by machine parameter P110, P310).

0 = Acc./Dec. applied only in linear interpolations at maximum feedrate.
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.

5.8.2 BELL-SHAPED ACCELERATION/DECELERATION RAMP

This type of acc/dec ramp is used on all types of movements and with any kind of feedrates.

P621(8) Bell-shaped Acceleration/Deceleration ramp

To be used on high speed machines.

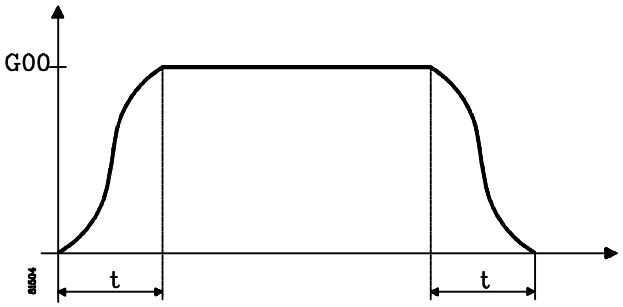
0 = No. This type of acc./dec. is not applied.
1 = Yes. This type of acc./dec. is applied.

Note that the acc./dec. ramp set by parameter 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.

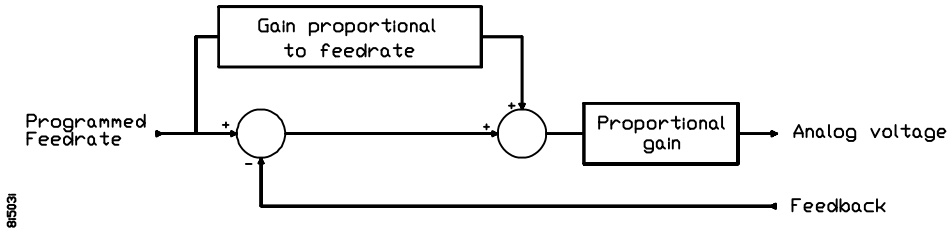
Value of 0 = There is no Acc./Dec. common to all the axes
Value of 1 = 0.010 seconds.
Value of 10 = 0.100 seconds. (10 x 0.01)
Value of 255 = 2.550 seconds. (255 x 0.01)

5.8.3 FEED-FORWARD GAIN

P720, P721 FEED-FORWARD gain for the X and Z 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 when using linear acc/dec.
  For example, for the X axis: "Kf = P720"

* 1/8 of the value assigned to this parameter when using bell-shaped acc/dec.
  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.

5.9 PARAMETERS FOR THE LIVE TOOL

When working with a live tool, it must be borne in mind that its analog output corresponding 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 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

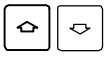
P802 Maximum programmable speed for the live tool

It indicates the maximum programmable speed for the live 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 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.

5.10 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 and P303 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

* The formulae given in the chapter on "concepts" to calculate K1 and K2 and the procedure to set the gain break-point remain valid. When it comes to calculating the Feed-forward gain, it must be borne in mind that 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

  In other words, the K1 and K2 gains (P114, P314, P116, P316) are now given in mV/0.01mm (mV/0.001inches).

* Machine parameters P115 and P315 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, P309 (leadscrew backlash) and P118, P318 (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, P312, P807, P808 (homing feedrate) will also be expressed in 0.01mm and 0.001 inch units.

  For example: P112 =10000 sets a homing feedrate of 100m/min. or 10 inches/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:

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

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

For a factor of x2:

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

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

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

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

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

With a x2 factor:

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

P103= 1 P602(3)=1 P106=0 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 and P303 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, 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

  In other words, K1 and K2 (P114, P314, P116, P316) are now given in: mV/0.0001mm (mV/0.00001 inches).

* Machine parameters P115 and P315 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, P309 (leadscrew backlash) and P118, P318 (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, P312, P807, P808 (homing feedrate) will also be expressed in 0.0001mm and 0.00001 inch units.

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

  For example: P112 =10000 sets a homing feedrate of 1m/min. or 0.1 inch/min.

Examples to calculate resolution with P609(7)=1:

Example 1: **Resolution in "mm" with square-wave encoder**

We want to obtain a 0.0001mm resolution with a square-wave encoder mounted on the X axis whose leadscrew has a 5mm/turn pitch.

Since the multiplying factor applied by the CNC may be either x2 or x4 (depending on machine parameter setting). The resulting encoder line count will be:

$$\text{Number of pulses} = \frac{\text{Leadscrew pitch}}{\text{Multiplying Factor x Resolution}}$$

For a factor of x4:

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

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

For a factor of x2:

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

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

$$\text{Number of pulses} = \frac{0.25}{4 \times 0.00001} = 6250 \text{ pulses/turn}$$

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

With a x2 factor:

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

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

6. *SPINDLE MACHINE PARAMETERS*

Atention:

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

P617(4) The actual (real) spindle speed is always displayed in rpm

It indicates the display units for the actual (real) spindle speed "S"

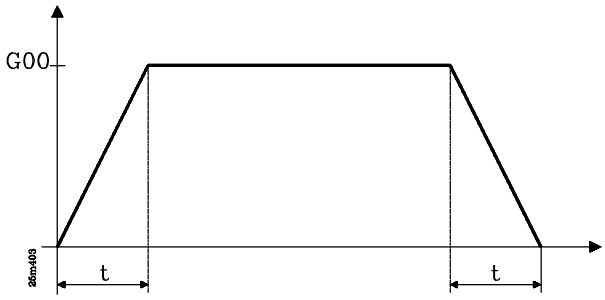
0 = The actual spindle speed is shown in RPM when working in the RPM mode and it is shown in m/min (feet/min) when working in the CSS mode.

1 = The actual spindle speed is always shown in RPM whether it is working in RPM mode or CSS mode.

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)

6.1 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.

6.2 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.

6.3 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

6.4 MACHINE PARAMETERS FOR SPINDLE CONTROL

It is necessary to have a spindle feedback encoder installed in order to perform the following operations:

* Automatic threading operation
* Spindle orientation

When working with spindle orientation, 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.

P617(7) M3 / M4 confirmation by detecting feedback direction reversal

It indicates whether or not the CNC waits for M3 / M4 confirmation by detecting that the spindle feedback has changed direction when programming a reversal of spindle direction (M3 to M4 or vice-versa).

0 = It does not wait for confirmation. It considers the command executed instantaneously.

1 = It waits for the spindle feedback signals to change direction to confirm that the programmed change has taken place.

When using two spindles with slow reversal, it is recommended to set "**P617(7)=1**".

6.4.1 PARAMETERS RELATED TO SPINDLE ORIENTATION

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

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

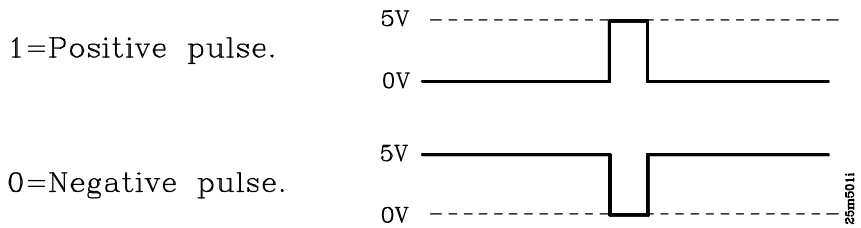
P606(2) Sign of the analog S output associated with spindle orientation

It determines the sign of the analog output associated with spindle orientation. 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 when being oriented.



P709 Minimum spindle analog while being oriented

It determines the minimum spindle analog voltage when being oriented.

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 spindle orientation

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 being oriented

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.

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

7. CONCEPTS

7.1 FEEDBACK SYSTEMS

The CNC has the following feedback inputs:

X axis. Via connector A1. It accepts sine-wave and differential (double ended) square-wave signals. Set machine parameter "P106" and the two dip-switches under this connector accordingly.

Z axis. Via connector A3. It accepts sine-wave and differential (double ended) square-wave signals. Set machine parameter "P306" and the two dip-switches under this connector accordingly.

Spindle encoder. Via connector A5. It accepts differential (double ended) square-wave signals. Assign the number of pulses of the spindle encoder to parameter "P800".

First handwheel. Via connector A6. It accepts single ended (not differential) square-wave signals. Set the following machine parameters accordingly:

P621(7)=1	The machine has no mechanical handwheels
P622(3)=1	Only one electronic handwheel is being used.
P609(1)	The first electronic handwheel is a FAGOR 100P model
P500	Counting direction of the 1st electronic handwheel
P602(1)	Feedback units of the electronic handwheel
P501	Feedback resolution for the electronic handwheel
P602(4)	Multiplying factor for feedback pulses from the handwheel

Second handwheel. Via connector A4. It accepts sine-wave and differential (double ended) square-wave signals. Set the following machine parameters accordingly:

P621(7)=1	The machine has no mechanical handwheels
P622(3)=0	Two electronic handwheels are being used.
P621(6)	Counting direction of the 2nd electronic handwheel
P621(3)	Feedback units of the electronic handwheel
P621(1,2)	Feedback resolution for the electronic handwheel
P621(5)	Multiplying factor for feedback pulses from the handwheel

7.1.1 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 and P303") 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.

Square-wave signals

The maximum counting frequency for differential square-wave signals is 200 KHz (200,000 pulses/sec.), with a 450nsec. separation between flanks (that is 90º ±20º phase shift).

Therefore, the maximum feedrate for each linear axis will depend upon the selected resolution (machine parameters "P103 and P303") 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.

7.1.2 RESOLUTION OF X AND Z AXES

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, P303	They set the counting resolution for each axis.
P602(3), P602(2)	They set the measuring units for each axis feedback signal (mm or inches).
P106, P306	They set the type of feedback signal being used (square-wave or sine-wave) for each axis.
P602(6), P602(5)	They indicate the multiplying factor, x2 or x4 to be applied to the feedback signals of each axis.
P619(1), P619(2)	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=0 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=0 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:

$$\text{Max. Feed} = \frac{200{,}000 \text{ pulses/sec.}}{625 \text{ pulses/rev.}} \text{ x 5 mm/rev.} = 1600 \text{ mm/sec.} = 96 \text{ m/min.}$$

When using a x2 multiplying factor:

$$\text{Max. Feed} = \frac{200{,}000 \text{ pulses/sec.}}{1250 \text{ pulses/rev.}} \text{ x 5 mm/rev.} = 800 \text{ mm/sec.} = 48 \text{ m/min.}$$

Example 2: **Resolution in "mm" with sine-wave encoder**

We want to calculate the multiplying factor required to obtain a resolution of 2µm with a sine-wave encoder mounted on to the X axis which has a 5mm/turn leadscrew pitch.

$$\text{Multiplying factor} = \frac{\text{Leadscrew pitch}}{\text{number of pulses x Resolution}} = \frac{5000 \text{ µm}}{250 \text{ x } 2 \text{ µm}} = 10$$

Since the CNC always applies a x5 multiplying factor to the sine-wave feedback signals, we will need an extra x2 factor. Thus:

P103= 2 P602(3)=0 P106=1 P602(6)=1 P619(1)=0

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:

$$\text{Max. Feed} = \frac{25{,}000 \text{ pulses/sec.}}{250 \text{ pulses/rev.}} \text{ x 5 mm/rev.} = 500 \text{ mm/sec.} = 30 \text{ m/min.}$$

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	0	0
20µm	10	0	0	1

Since the counting frequency of the CNC is limited to 200KHz for square-wave signals, the maximum feedrate obtainable with a 20µm-pitch scale is:

Max. Feed = 20 µm/pulse x 200,000 pulses/sec. = 4000 mm/sec. = **240 m/min.**

However, if FAGOR linear scales are used, the maximum feedrate is limited (by the scales) to 30m/min. (1181 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.

We need to calculate the multiplying factor required for the desired resolution.

$$\text{Multiplying factor} = \frac{\text{Scale pitch}}{\text{Resolution}} = \frac{20\ \mu m}{1\ \mu m} = 20$$

Since the CNC always applies a x5 factor to sine-wave feedback signals, we will need an additional x4 factor. Therefore:

P103= 1 P602(3)=0 P106=1 P602(6)=0 P619(1)=0

Since the counting frequency is limited (by the CNC) to 25KHz for sine-wave signals, the maximum feedrate for this axis will be:

Max. Feed = 20 µm/pulse x 25,000 pulses/sec. = 500 mm/sec. = 30 m/min.

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 P602(3)=1 P106=0 P602(6)=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=0 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 obtain a 0.0001 inch resolution with a 250-line sine-wave encoder on the X axis which has a 4-pitch leadscrew (4 turns/inch or 0.25 inch per turn).

We must calculate the multiplying factor required for that resolution:

$$\text{Multiplying factor} = \frac{\text{Leadscrew pitch}}{\text{Pulses x Resolution}} = \frac{0{,}25}{250 \text{ x } 0.0001} = 10$$

Since the CNC always applies a x5 multiplying factor to sine-wave feedback signals, we need an additional x2 factor. Therefore:

P103= 1 P602(3)=1 P106=1 P602(6)=1 P619(1)=0

Although a FAGOR encoder outputs up to 200KHz, the CNC admits only up to 25KHz. Therefore, the maximum feedrate for this axis will be:

$$\text{Max. Feed.} = \frac{25{,}000 \text{ pulses/sec.}}{250 \text{ pulses/rev.}} \text{ x } 0.25 \text{ inch/rev.} = 25 \text{ inch/sec.} = 1500 \text{ inches/min.}$$

7.2 ADJUSTMENT OF X AND Z AXES

In order to carry out this adjustment, it is necessary to have the feedback devices connected to the CNC.

Before starting this adjustment of the axes it is a good idea to move each one of them to the center of its travel and set the mechanical travel limits (stops), controlled by the electrical cabinet, near that center point in order to avoid possible collisions.

Verify that the axes are not under CONTINUOUS CONTROL (**machine parameters P105 and P305 must be set to "N"**). This means that once the axis is in position, its enable signal to the servo drive is deactivated.

Also, verify that machine parameters **P104 and P304 are set to "Y"** indicating that there must be a delay between the activation of the axis enable signal and the output of its analog voltage.

Once these parameter settings have been verified, proceed with the axis adjustment as follows:

* The axes will be adjusted one at a time.

* Connect the power output of the servo drive corresponding to the axis to be adjusted.

* Jog the axis being adjusted by using the jog keys.

  If the axis runs away, the CNC will issue a Following Error message. Change the setting of the machine parameter for the sign of the analog voltage (P100 for the X axis and P300 for the Z axis) and press RESET to assume the new value.

* If the axis does no longer run away, but it does not count in the desired direction, modify the setting of the parameter corresponding to the COUNTING DIRECTION (P101 for X and P301 for Z) as well as the value of the parameter corresponding to the SIGN OF THE ANALOG VOLTAGE (P100 for X and P300 for Z). Press RESET to assume the new values.

* If the counting direction is correct, but it jogs in the wrong direction, change the setting of the machine parameter corresponding to the JOGGING DIRECTION (P102 for X and P302 for Z).

7.2.1 OFFSET AND MAXIMUM FEEDRATE ADJUSTMENT

These adjustments will be made on the axis servo drives and on the spindle drive.

Offset adjustment

This adjustment will be carried out in two steps:

Pre-adjustment of the drive offset

* Disconnect the analog input and short-circuit it with a wire jumper.

* Turn the offset potentiometer of the drive until the output voltage of the tacho reads 0V. This voltage should be measured with a volt-meter set on the 200mV DC scale.

* Remove the wire jumper of the analog input.

Critical drive-offset Adjustment

* Execute a program in the "point-to-point movement" mode  which moves the axis continuously back and forth. A sample program for the Z axis may be the following:

```
P0 X0 Z100
P1 X0 Z-100
P2 X0 Z100
P3 X0 Z-100
.....
.....
P19 X0 Z-100
P20 X0 Z100
```

While the axis is moving back and forth, monitor the amounts of following error of the axis in both directions and make them the same by turning the **offset potentiometer of the drive. Not at the CNC.**

Maximum feedrate adjustment

It is recommended to adjust the servo drives in such a way that the maximum speed is obtained with 9.5V of analog voltage.

Also, this maximum axis speed (feedrate) must be assigned to the corresponding machine parameter (P111 for X and P311 for Z).

The calculation of the axis maximum feedrate depends on the maximum rpm of the axis motor, the gear reduction and the type of leadscrew being used.

Example for the Z axis:

Having a motor whose maximum rpm is 3000 and the pitch of the leadscrew is 5mm/revolution.

Maximum feedrate = r.p.m. of leadscrew x leadscrew pitch

P311 = 3000 r.p.m. x 5mm/rev. = 15000 mm/minute

Once these values are assigned to the corresponding parameters, it is recommended to proceed with the adjustment of the drive.

To do this, execute a program in the "point-to-point movement" mode  which moves the axis continuously back and forth. A sample program for the Z axis may be the following:

```
P0 X0 Z100
P1 X0 Z-100
P2 X0 Z100
P3 X0 Z-100
.....
.....
P19 X0 Z-100
P20 X0 Z100
```

Before running this program, assign to machine parameter P310 the value of P311 and select "F0" as active feedrate. The 

key lamp must be OFF.

While the axis is moving, set the analog voltage going into the servo drive to 9.5V by turning the gain potentiometer of the drive and **NOT at the CNC.**

7.2.2 GAIN ADJUSTMENT FOR X AND Z AXES

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 or the analog voltage in case the system becomes somewhat unstable.

The CNC has a series of machine parameters which permit adjusting the proportional gain for each axis. These parameters are:

* PROPORTIONAL GAIN K1. Set by machine parameters P114 and P314.

* PROPORTIONAL GAIN K2. Set by machine parameters P116 and P316.

* GAIN BREAK-POINT value. Set by machine parameters P115 and P315.

7.2.2.1 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.

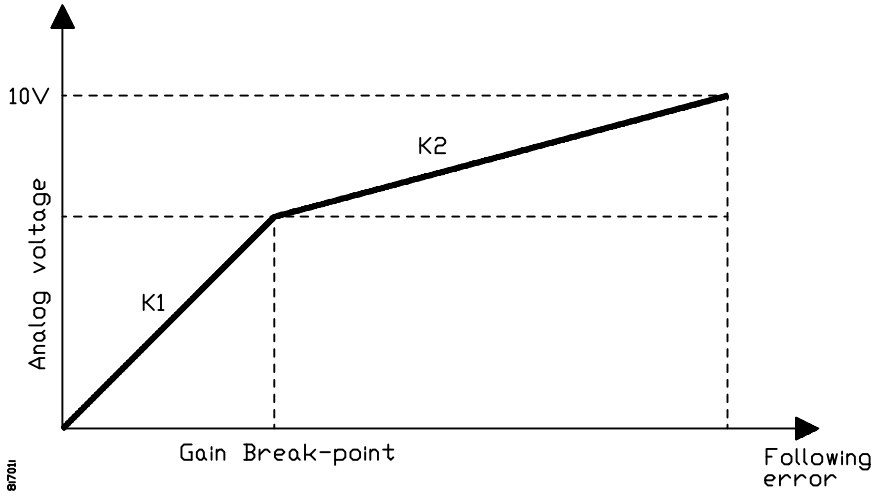
$$\text{Analog output} = \text{Proportional gain} \times \text{following error (microns)} \times \frac{2.5\text{mV}}{64}$$

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 in order to keep the system stable.

The FAGOR 800T 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 of 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.

7.2.2.2 CALCULATION OF K1, K2 AND GAIN BREAK-POINT

The value K1 represents the analog voltage corresponding to 1 micron of following error. It is given by an integer between 0 and 255.

Analog (mV) = K1 x Following error x Kcnc

Where: In **metric**: Following error in microns and Kcnc = 0.039
In **inches**: Following error in thousandths of an inch and Kcnc = 0.993

The amount of following error corresponding to the GAIN BREAK-POINT is given in microns or in ten-thousandths of an inch by parameter P115 for the X axis and P315 for the Z axis. It is recommended to set it to a value slightly greater than the following error corresponding to the maximum machining feedrate F (P110 and P310) (not positioning feedrate, G00: P111 and P311).

The machine parameter for 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.

METRIC example for K1 and gain break-point:

The maximum positioning feedrate (G00) for the Z axis (P311) is 15m/min. the maximum machining feedrate (programmable F) is to be limited to 3m/min and the desired following error for 1m/min is 1 micron (unity gain). The servo drive has been adjusted to obtain 15m/min at 9.5V.

Thus, the analog voltage corresponding to 1m/min would be:

$$\text{Analog} = \frac{9500\text{mV}}{15\text{m/min}} \times 1\text{m/min} = 633\text{mV}$$

And the K1 value corresponding to a gain of 1 would be:

$$K1 = \frac{\text{Analog (for 1m/min)}}{\text{F.E. (for 1m/min)} \times \text{Kcnc}} = \frac{633\text{mV}}{1000\text{ microns} \times 0.039} = 16.2048$$

The integer value to be assigned to **K1 could be 16**.

If K1 has been set for a gain of 1 (K1=16 for this example) and the maximum machining feedrate (F) is 3000 mm/min (P310). At this feedrate, the following error should be about 3000 microns. Thus, the gain break-point value should be slightly larger than 3000 microns; for example: P315 = 3100 microns.
Thus: P310=3000; P311=15000; P314=16; P315=3100; P316=50% - 70% of K1

Example in INCHES for K1 and gain break point:

The maximum positioning feedrate (G00) for the Z axis (P311) is 600 inches/min. the maximum machining feedrate (programmable F) is to be limited to 120 inches/min and the desired system gain (Kv) should be equal to one (unity gain). The servo drive has been adjusted to obtain 600 inches/min at 9.5V.

The following error corresponding to 1 inch/min for this unity gain would be:

$$\text{Following Error} = FE = \frac{\text{Feedrate}}{\text{Kvgain} \times 1000} = \frac{1 \text{ inch/min}}{1 \times 1000} = 0.001"$$

The analog voltage corresponding to 1 inch/min would be:

$$\text{Analog} = \frac{9500mV}{600"/min} \times 1"/min = 15.8333mV$$

And the K1 value corresponding to a gain of 1 would be:

$$K1 = \frac{\text{Analog (for 1"/min)}}{\text{F.E. (for 1"/min)} \times \text{Kcnc}} = \frac{15.8333mV}{1 \text{ (thousandth)} \times 0.993} = 15.94$$

The integer value to be assigned to **K1 could be 16**.

If K1 has been set for a gain of 1 (K1=16 for this example) and the maximum machining feedrate (F) is 120 inch/min (P310). At this feedrate, the following error should be about 0.1200 inches. Thus, the gain break-point value should be slightly larger than 1200 ten-thousandths of an inch; for example: P315 = 1250.

Thus: P310=120; P311=600; P314=16; P315=1250; P316=50% - 70% of K1

To perform a practical axis adjustment at the machine:

it is recommended to first set K1 = K2 and run a program in "point-to-point movement" mode  continuously moving the axis back and forth.

A program of this sort, for the Z axis, may be the following:

```
P0 X0 Z100
P1 X0 Z-100
P2 X0 Z100
P3 X0 Z-100
.....
.....
P19 X0 Z-100
P20 X0 Z100
```

Atention:

Once each axis has been adjusted separately. Both 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 both axes in the K1 area.

Watch the amount of following error reached at maximum **machining** feedrate and assign that value or one slightly larger to the gain break-point.

Once K1 and the gain break-point have been set, change the value of K2 to one between 50% and 70% of K1.

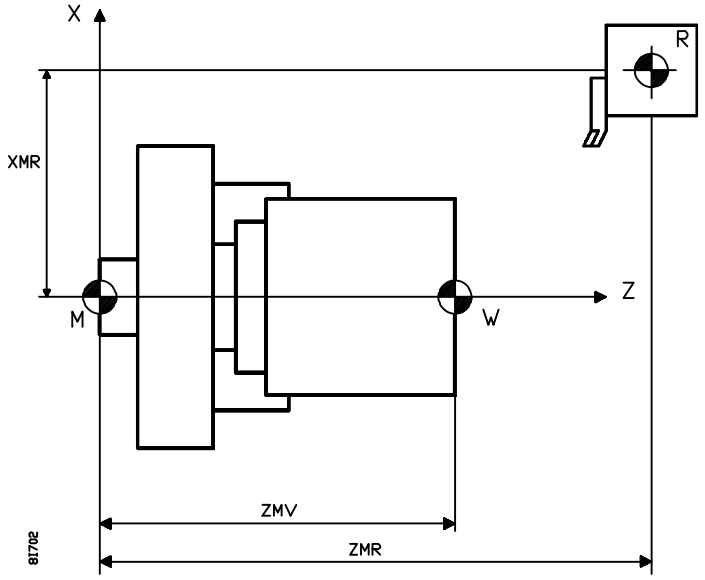
7.3 REFERENCE POINTS FOR X AND Z AXES

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 the physical location of the marker pulse or reference pulse (Io) used as home to synchronize the whole machine coordinate system. The axis moves to this point when being "homed" and the CNC assumes the reference values set at machine parameter "P119" or "P319" accordingly.



M	Machine Reference Zero
W	Part Zero
R	Machine Reference Point
XMW, ZMW	Part Zero coordinates
XMR, ZMR	Machine Reference Point coordinates

7.3.1 MACHINE REFERENCE SEARCH (HOME)

With this CNC it is possible to search home on each axis ([X] [up arrow] and [Z] [up arrow]) as follows:

1.- The CNC starts moving the axis in the direction set by machine parameter P618(8) for X and P618(7) for Z at the feedrate set by machine parameter P112 for X and P312 for Z until the home switch is pressed.

If the selected axis has no home switch (parameters "P600(5), P600(4)"), the CNC will consider it to be pressed and it will go on with paragraph 2.

2.- Once the home switch is pressed, the CNC will continue moving the axis at the feedrate selected by machine parameters P807 for X and P808 for Z until the reference pulse (marker, Io) of the feedback device is found.

Once the home search is completed, the CNC will cancel the selected zero offset and it will display the coordinates of the machine reference point indicated by machine parameter P119 for X and P319 for Z.

7.3.2 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(7)”) until releasing the home switch before starting the actual home search.

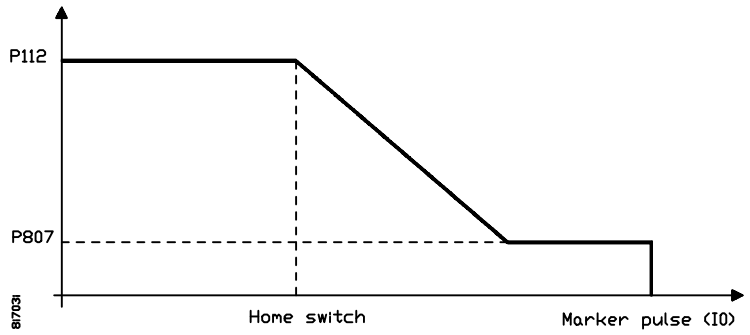
* If the axis is out of the soft travel limits (set by P107,P108 for X and P307,P308 for Z), it has to be jogged into the work area (within limits) and, then, positioned at the correct side from home before starting the actual home search.

* Care must be taken when placing the home switch and when setting the home searching feedrates (P112, P807 for X and P312, P808 for Z) to prevent any overshooting.

* Set machine parameter P600(7) for X and P600(6) for Z to indicate the type of marker pulse (Io) being used.

  FAGOR linear transducers (scales) have a negative marker pulse (Io) every 50 mm (about 2 inches) and FAGOR rotary encoders output a positive marker pulse (Io) 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 and P808).



* If there is no room for that, the first home searching feedrate (set by P112 and P312) must be reduced. This might be the case with those rotary encoders where the marker pulses are very close to each other.

7.3.3 ADJUSTMENT OF THE VALUE CORRESPONDING TO THE MACHINE REFERENCE POINT (HOME)

Before making this adjustment, place the mechanical travel limits (stops) where they are supposed to go.

One of the procedures that can be followed is this:

1.- Define the machine parameters related to home.

Set P600(5) and P600(4) to 1 indicating that both axes have home switches.

Set parameters “P600(7)” and “P600(6)” indicating the type of marker pulse used by the feedback device when searching home. Fagor scales have one negative marker pulse (Io) every 50mm and Fagor rotary encoders have one positive marker pulse per revolution.

Also, set parameters “P618(8)” and “P618(7)” to indicate the direction the axes must move when searching home.

Besides, set parameters "P112" and "P312" to indicate the home switch approaching feedrate (1st home searching feedrate) as well as parameters P807 and P808 to indicate the marker pulse approaching feedrate (2nd home searching feedrate).

Assign a value of "0" to the machine reference point. Parameters P119 and P319.

2.- Position the axis in the proper area for home search and execute the home search command.

Press [X] or [Z] and then, [up arrow] followed by 

The CNC will carry out the home search and when done, it will assign a value of zero to that point.

3.- Move the axis to the physical location where machine zero point will be (or to a position whose distance to machine zero is known), write down the position value displayed by the CNC at that point.

The value to be assigned to parameter P119 or P319 will be:

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

Example for the Z axis:

If the point of known dimensions is 230mm from the machine zero and the CNC shows "-123.5mm", the value to be assigned to P319 will be:
“P319” = 230 - (-123.5) = 353.5 mm.

4.- Assign this new value to the machine parameter and press 

or power the CNC down and back up in order for the CNC to assume this new value.

5.- Perform a new home search in order for the CNC to assume the correct reference values.

7.3.4 AXIS SOFTWARE TRAVEL LIMITS

Once the home search has been done on all the axes, proceed with the setting of the software travel limits for each axis.

This procedure is to be done one axis at a time and can be done as follows:

* Move the axis in the positive direction up to a point close to the mechanical stop keeping a safety distance from it.

* Assign to P107 or P307 (as it corresponds) the position value displayed by the CNC for that point.

* Repeat those two actions in the negative direction and assigned the new displayed value to parameter P108 or P308 (as it corresponds).

* Once this procedure is completed for all the axes, press [RESET] or power the CNC down and back up in order for the new values to be assumed by the CNC.

7.4 ACCELERATION/DECELERATION

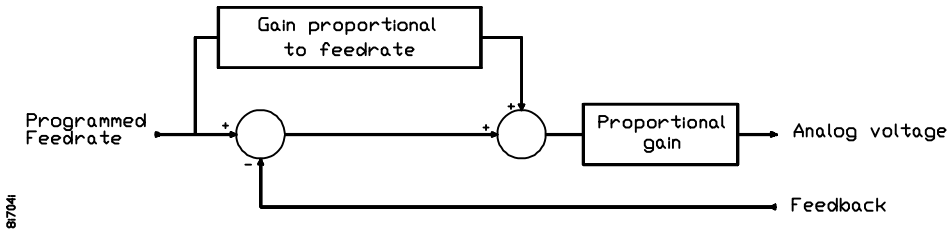
In order to avoid abrupt start-ups and slow-downs of the axes, this CNC offers two ACC./ DEC. control systems.

One of them, acts only on linear interpolations while the other one acts on all types of movements.

It is possible to use the Feed-forward gain in either case in order 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. control.**

7.4.1 CALCULATION OF FEED-FORWARD GAIN

The feed-Forward gain is proportional to the feedrate and is set by machine parameters P720 and P721 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 selected by machine parameter 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 voltage = 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 voltage = (K1 x Ep) + {K2 x [Following error + (Kf x F/6) - Ep]}

Where "Ep" is the value of the gain break-point.

7.4.2 ACCELERATION/DECELERATION ON LINEAR INTERPOLATIONS

When working with this type of acceleration, it is necessary to define by parameters P712 and P713 the acceleration period or time required by **each axis** to reach the positioning feedrate (indicated by parameters P111 and P311).

This time is also valid for the deceleration ramp.

When performing a linear interpolation, the CNC applies to the resulting path the greater of the two acc/dec periods assigned to the axes involved in such interpolation.

It is also possible to apply this type of acc./dec. on all linear interpolations when "P609(4)=1" or only when those movements are carried out at the maximum programmable feedrate (set by parameter P110) when P609(4)=0.

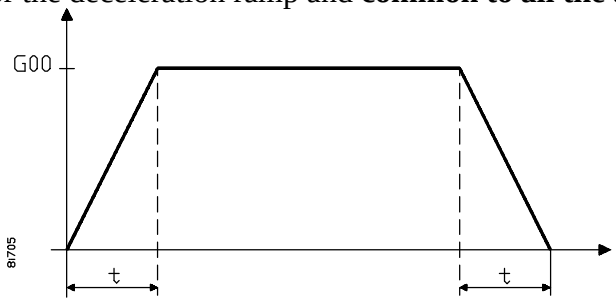
On the other hand, when no acc/dec. is to be applied on G05 block transitions, parameter P616(6) must be set to "1".

7.4.3 ACCELERATION/DECELERATION ON ALL TYPES OF MOVEMENTS

This type of acc./dec. is used on high speed machines.

It is activated by setting parameter P621(8) to "1". **This cancels the other type of acc./dec.** described earlier.

When working with type of acc./dec., it is necessary to define by parameter P731 the ramp period or time required by an axis to reach the selected feedrate (acceleration). This time is the same for the deceleration ramp and **common to all the axes**.



7.5 HANDWHEEL MOVEMENTS

The axes may be moved from the CNC, by means of mechanical handwheels or by means of electronic handwheels.

The available options are:

* The machine only has mechanical handwheels. With this option, no electronic handwheels can be used.

* The machine has one electronic handwheel. With this option, no mechanical handwheels can be used.

* The machine has two electronic handwheels, one for each axis. With this option no mechanical handwheels can be used

7.5.1 THE MACHINE HAS MECHANICAL HANDWHEELS

The following machine parameters must be set:

P621(7)=0	The machine has mechanical handwheels
P105=0	The X axis is not continuously controlled when in position
P305=0	The Z axis is not continuously controlled when in position

This way, when the axes reach position their enables are cancelled so they can be moved by means of the mechanical handwheels.

7.5.2 THE MACHINE HAS A SINGLE ELECTRONIC HANDWHEEL

When the machine has an electronic handwheel, no mechanical handhweel can be used.

When using a FAGOR 100P handwheel, both axes may be governed one at a time. To do this, press the axis selector button on the back of the handwheel as described later on.

Machine parameters P105 and P305 must be set to "1 (YES)", so there is continuous control of the axes. This means that once an axis is in position, its enable signal remains on. (the CNC holds them in position)

Besides, the following parameters must be properly set:

P621(7)=1	The machine does not use mechanical handwheels.
P622(3)=1	The machine
P609(1)=1	The first electronic handwheel is a FAGOR 100P model
P500	Counting direction of the electronic handwheel
P602(1)	Measuring units of the electronic handwheel feedback signals
P501	Counting resolution of the electronic handwheel
P602(4)	Multiplying factor for electronic handwheel feedback signals

To move the axes with the electronic handwheel, select the positions on the Feedrate Override Switch. The available positions are 1, 10 and 100 which indicate the multiplying factor being applied to the pulses supplied by the electronic handwheel.

Then, press any JOG key corresponding to the axis to be moved. The CNC will highlight the selected axis. If only one single electronic handwheel is used and it is a FAGOR 100P model, the desired axis can also be selected by pressing the button on the back of the handwheel.

Turn the electronic handwheel. The distance the axis will move depends on the position selected at the Feedrate Override Switch and the values assigned to the corresponding machine parameters.
Example:

The machine parameters for the electronic handwheel have been set as follows:

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

The Feedrate Override switch is at position "x100".

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

When turning the electronic handwheel very fast, the CNC may be forced to move the axis faster than the maximum allowed feedrate (parameters P111 and P311). In this case, the CNC will limit its speed to the maximum feedrate allowed, ignoring the additional pulses and, therefore, avoiding the possible following error messages that otherwise would have been issued.

To select another axis, just press any of the JOG keys corresponding to the other axis. The CNC will now highlight the new selected axis. If only one single electronic handwheel is used and it is a FAGOR 100P model, the desired axis can also be selected by pressing the button on the back of the handwheel.

To quit this operating mode (with electronic handwheel), select any other position of the Feedrate Override Switch. When using a FAGOR 100P handwheel, this mode can also be quit by holding the axis selector button on its back for a few seconds.

Whenever the Feed-hold input is active (pin 15 of connector I/O 1) or one of the software travel limits (set by parameters P107, P108, P307 and P308) has been overrun, the CNC **will ignore the electronic handwheel** and, therefore, will not move the machine. It will have to be moved by means of the mechanical handwheels.

The handwheel will also be ignored when a CNC error occurs (emergency, etc.).

7.5.3 THE MACHINE HAS TWO ELECTRONIC HANDWHEELS

When having two electronic handwheels, it will not be possible to use the mechanical handwheels of the machine at the same time. They may be used only when the CNC is turned into a DRO by activating its manual input (pin 19 of connector I/O 1) which will deactivate the electronic handwheels. Each electronic handwheel controls only one axis; therefore, the axis selector button of the FAGOR 100P handwheel will have no effect.

The first handwheel will control the X axis and the second one will be associated with the Z axis.

Machine parameters P105 and P305 must be set to "1" (YES), indicating that when the axis is in position its enable signal is maintained.

Besides, the following parameters must be properly set:

P609(1)=0	The first electronic handwheel is not working as a FAGOR 100P model.
P621(7)=1	The machine has no mechanical handwheels
P500, P621(6)	Counting direction of the electronic handwheels (1st and 2nd).
P602(1), P621(3)	Measuring units of the electronic handwheel feedback signals (1st and 2nd).
P501, P621(1,2)	Counting resolution of the electronic handwheels (1st and 2nd).
P602(4), P621(5)	Multiplying factor for electronic handwheel feedback signals (1st and 2nd).

The axes can be moved in all FOS (feedrate override switch) positions and the CNC will apply the corresponding multiplying factor indicated at the handwheel positions and a "x1" factor for any of the other FOS positions.

The distance the axis will move depends on the position selected at the Feedrate Override Switch (FOS) and the values assigned to the corresponding machine parameters.

Example for the 1st handwheel:

The machine parameters for the electronic handwheel have been set as follows:

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

The Feedrate Override switch is at position "x100".

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

When turning the electronic handwheel very fast, the CNC may be forced to move the axis faster than the maximum allowed feedrate (parameters P111 and P311). In this case, the CNC will limit its speed to the maximum feedrate allowed, ignoring the additional pulses and, therefore, avoiding the possible following error messages that otherwise would have been issued.

Atention:

Even when the Feed-hold input is active (pin 15 of connector I/O 1) or one of the software travel limits (set by parameters P107, P108, P307 and P308) has been overrun, the CNC **will be governed by the electronic handwheel** and, therefore, it will move the machine.

The handwheel **will be ignored** when a CNC error occurs (emergency, etc.) and when a part or cycle is being executed.

7.6 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 outputP601(3)=1 and P601(2)=0
For 4-digit BCD outputP601(3)=0 and P601(2)=1

The CNC will issue the code corresponding to the programmed spindle speed at the BCD outputs (pins 20 thru 27 of I/O 1).

It will also activate the "S Strobe" output to indicate to the electrical cabinet that the required auxiliary function must be executed and it will wait for the "M-DONE" signal from the electrical cabinet in order to consider the data transfer has concluded.

When using a 2-digit BCD code, P601(3)=1 and P601(2)=0, The CNC will indicate the selected spindle speed according to the following conversion table:

Programmed S	S BCD	Programmed S	S BCD	Programmed S	S BCD	Programmed S	S BCD
0	S 00	25-27	S 48	200-223	S 66	1600-1799	S 84
1	S 20	28-31	S 49	224-249	S 67	1800-1999	S 85
2	S 26	32-35	S 50	250-279	S 68	2000-2239	S 86
3	S 29	36-39	S 51	280-314	S 69	2240-2499	S 87
4	S 32	40-44	S 52	315-354	S 70	2500-2799	S 88
5	S 34	45-49	S 53	355-399	S 71	2800-3149	S 89
6	S 35	50-55	S 54	400-449	S 72	3150-3549	S 90
7	S 36	56-62	S 55	450-499	S 73	3550-3999	S 91
8	S 38	63-70	S 56	500-559	S 74	4000-4499	S 92
9	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		

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	20	21	22	23		24	25	26	27	
1st stage	0	0	0	1	(Thousands)	0	0	1	0	(Hundreds)
2nd stage	0	0	1	1	(Tens)	0	1	0	0	(Units)

7.6.1 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.

7.6.1.1 MANUAL SPINDLE RANGE CHANGE

To be able to perform the spindle speed range change manually, machine parameter P601(1) must be set to "0".

When the new selected spindle speed involves a range change, the CNC will display a message indicating the range to be selected.

Once this range has been selected, press [ENTER] for the CNC to output the new spindle analog voltage.

Then, the CNC will automatically generate the M function associated with the new spindle range: M41, M42, M43 or M44.

7.6.1.2 AUTOMATIC SPINDLE RANGE CHANGE

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).

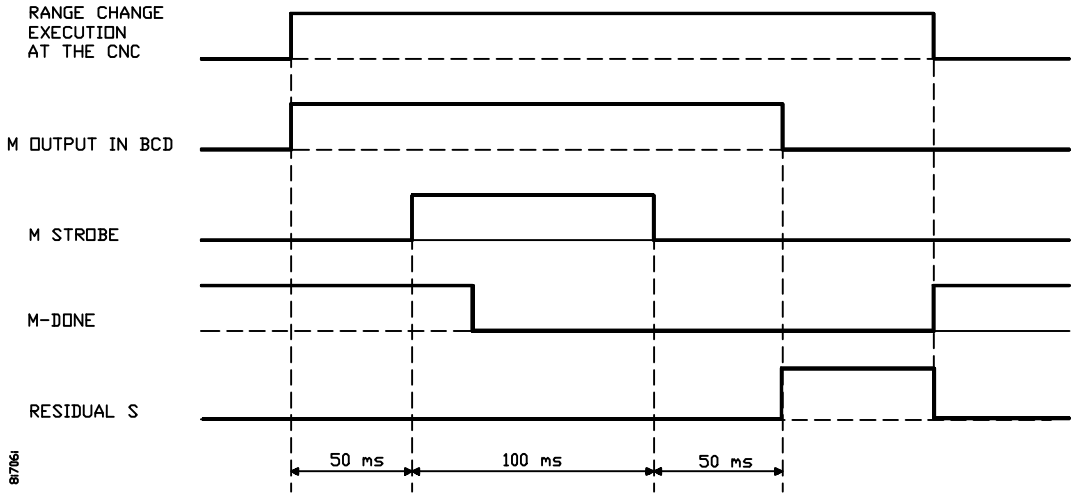
Also, in order to facilitate the range change, the CNC offers the possibility to use a residual analog output during a range change. Machine parameter for the spindle: P601(6).

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.

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 P602(7) must be set "1" (the CNC waits for the down flank of the M-Done signal).

7.6.2 SPINDLE CONTROL

It is necessary to install an encoder on the spindle in order to perform the following operations:

* Automatic threading operation.
* Spindle orientation.

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", the following parameters must be set:

P601(4)	Sign of the spindle analog output.
P606(2)	Sign of the spindle analog output for spindle orientation.
P600(8)	Type of spindle encoder reference (marker) pulse.
P706	Spindle speed "S" when working in spindle orientation.
P707	In position zone for the spindle in spindle orientation.
P708	Spindle proportional gain in spindle orientation.
P709	Minimum spindle analog output in spindle orientation.

To orient the spindle, key in the following keystroke sequence:

* **[S]**  , the bottom of the screen will display: **"S POS =**".

* Key in the desired orientation angle value. for example: S20 or S35.006

* press  .

Every time the spindle is being oriented after working in open loop (in regular rpm mode), the CNC will slow down the spindle speed below the value indicated by parameter P706 (if it was turning); it will home it (search for the marker pulse of the spindle encoder) and it will finally position (orient) it to the specified angle (S POS=).

This spindle position will be displayed in whole degrees and in large characters as:

S320 °

The spindle will only be homed prior to being oriented whenever it is switched from operating in open loop to doing it in closed loop (orientation).

When changing back from closed loop to open loop, the display will show the spindle rpm by replacing the "°" symbol with **"RPM**" characters.

The spindle will switch to open loop when pressing  or  ; after an emergency or on power-up.

7.7 TOOLS AND TOOL MAGAZINE

The machine parameters related with the tools and the tool magazine are the following:

P700	Number of tools
P900	X coordinate for tool change position
P901	Z coordinate for tool change position
P617(3)	The machine has an automatic tool changer

7.7.1 MACHINE WITH AN AUTOMATIC TOOL CHANGER

To select a new tool, press [TOOL] followed by the tool number and press 

The CNC informs the electrical cabinet of the selected tool number via the BCD outputs (pins 20 thru 27 of I/O 1) and it activates the "T Strobe" output to "tell" the electrical cabinet that it must change the tool.

Once the tool change is completed, the CNC assumes the values assigned to the tool offset of the same number and it applies those values (tool length and radius) in all the operations performed with this new tool.

7.7.2 MACHINE WITHOUT AN AUTOMATIC TOOL CHANGER

The CNC must know at all times which tool is being used in a machining operation. To do this, every time a new tool is selected and changed, press [TOOL] followed by the new tool number and press 

The CNC assumes the values assigned to the tool offset of the same number and it applies those values (tool length and radius) in all the operations performed with this new tool.

If while executing a cycle or part previously programmed, a tool must be selected, the CNC will display a message indicating the number of the new tool which must be selected.

It also interrupts the execution of the program until the operator changes the tool and presses [ENTER].

7.7.3 TOOL CHANGE POSITION

It is recommended to use a location away from the workpiece to perform the tool change, especially when making repetitive parts.

With this CNC, it is possible to establish this location by means of machine parameters P900 and P901. These parameter values are absolute X and Z coordinates referred to Machine Reference Zero (home) and the axes will move to this position for a tool change.

This way, every time the operator requests a new tool or the CNC itself needs to change the tool to execute a cycle or a programmed part, the axes will move to this tool change position indicated by parameters P900 and P901.

If both parameters are set to "0", the CNC will act as follows:

* When the operator requests a tool change, the CNC will not move the axes and the tool change will take place at the current position of the axes.

* To execute a programmed cycle with another tool, the CNC will change the tool before starting to execute it, right where the cycle was called upon

* During the execution of a part, the CNC assumes as tool change position the starting point of the part execution. Therefore, whenever a tool change is required, the CNC will move the axes to that point.

The CNC will ignore the values indicated in parameters P900 and P901 when calibrating a tool and it will change tools where the change has been requested.

7.8 FEED-HOLD AND M-DONE SIGNAL PROCESSING

The CNC has one single input (pin 15 of connector I/O 1) to process both signals.

This input must be normally high and these signals are processed as follows:

FEED HOLD

This signal may interrupt the execution of a block.

If while the axes are moving, this input is set low, the CNC maintains the spindle rotating and stops the axes by bringing their analog voltages to "0V" and keeping their enable signals on.

When this input is set back high, the CNC resumes the movement of the axes.

M-DONE or CONFIRMATION FROM ELECTRICAL CABINET

This signal is used as confirmation from the electrical cabinet that the execution of the requested M, S or T function has been completed.

When the CNC sends to the electrical cabinet the BCD output signals corresponding to the M, S or T function, the electrical cabinet must set this M-DONE input low.

The CNC will wait for the electrical cabinet to finish the execution of such function and set this M-DONE input back high. This will "tell" the CNC that the execution of the corresponding auxiliary function has been completed.

7.9 M, S, T FUNCTION TRANSFER

M function transfer:

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 is executed which is not defined on the M function table, the CNC will wait for the "M-DONE" signal before resuming the execution of the program.

S function transfer (BCD format):

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.

7.9.1 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.

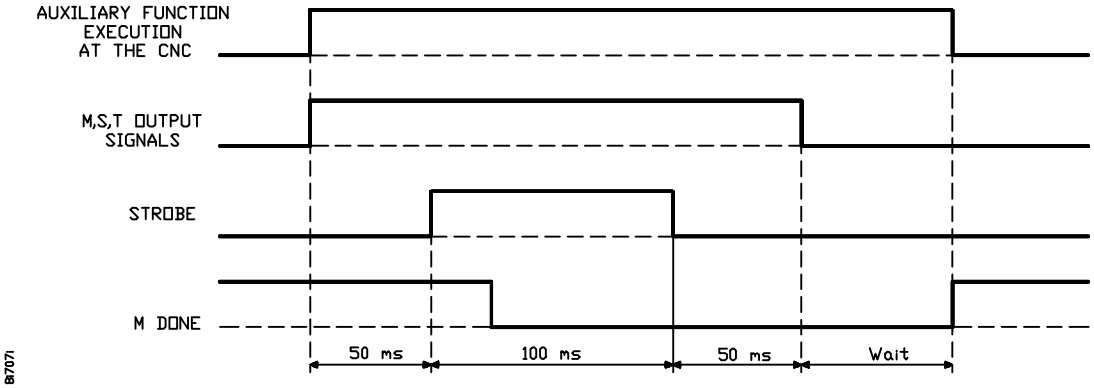
When the electrical cabinet has a device requiring the BCD signals to be active for a longer time, machine parameter P602(7) must be set to "1".

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.

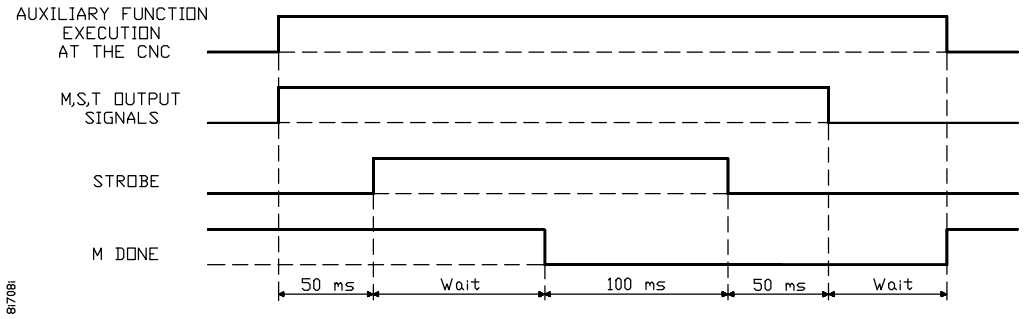
After this time period, it will wait for the M-DONE signal provided by the electrical cabinet indicating to the CNC that the executiong of the "M" function is completed.

“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.

Once the data from the CNC is read and processed, the electrical cabinet deactivates the M-DONE input to "tell" the CNC that the execution of the corresponding function has begun.

3.- The CNC will maintain the Strobe signal for another 100 milliseconds and the BCD outputs for another 150 milliseconds.

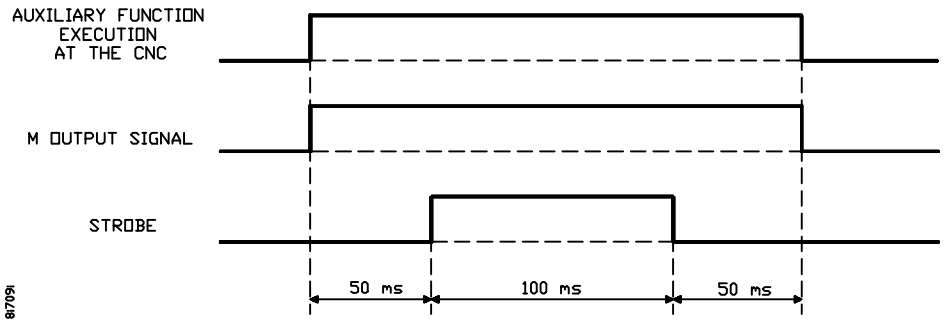
After this time period, it will wait for the electrical cabinet to reactivate the M-DONE input "telling" the CNC that the processing of the required function has concluded.

7.9.2 M FUNCTION TRANSFER WITHOUT M-DONE SIGNAL

This type of transfer is done when the corresponding M function is defined on the decoded M function table and its 17th bit is set in such a way that the CNC does not wait for the M-DONE signal for the execution of the M function to be considered completed.

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 another 100 milliseconds and the BCD signals for another 150 milliseconds.

After this time period, it will resume the execution of the program regardless of whether the function has been completely executed or not by the electrical cabinet.

APPENDIX A

TECHNICAL CHARACTERISTICS OF THE 800T CNC

GENERAL CHARACTERISTICS

Three 8-bit microprocessors
Memory storage of up to 10 programs of up to 20 operations each.
1 RS232C serial communication line.
2 feedback inputs for X and Z respectively.
1 feedback input for the spindle.
2 Handwheel inputs.
Resolution of up to 0.001mm or 0.0001 inch.
Multiplying factor of up to x100 for sine-wave feedback input.
11 optocoupled digital inputs.
32 optocoupled digital outputs.
Approximate weight: Compact model: 12 Kg.
Modular model: Central Unit 9Kg. Monitor 9" 13Kg. Monitor 14" 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.
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.
Minimum separation between flanks: 950 nsec.
Phase shift: 90º ±20º
High threshold (logic state "1"): 2.4V. < V_{IH} < 5V.
Low threshold (logic state "0"): -5V. < V_{IL} < 0.8V.
Vmax. ±7 V.
Hysteresis: 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 insulation by opto-couplers.
Protection against reversed connection up to -30V DC.

ELECTRICAL CHARACTERISTICS OF DIGITAL OUTPUTS

Nominal power supply voltage: +24 V DC.
Maximum nominal voltage: +30V DC.
Minimum nominal voltage: +18V DC.
Output voltage = Power supply voltage - 2V.
Maximum output current: 100 mA.
Protection by means of galvanic insulation by opto-couplers.
Protection by means of 3A. external fuse against reversed connection up to -30V DC and against overvoltage of the external power supply greater than 33V DC.

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 and acceleration of 5G.
Free fall of packaged equipment: 1m.

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 54
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 the 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.

MONITOR

CRT

Monitor	9" monochrome	Deflection:	90 degrees
Screen:	Anti-glare	Phosphor:	Amber
Resolution:	640 points x 480 lines	Screen surface:	160x120 mm.

SWEEP FREQUENCY

Vertical synchronism: 75 Hz negative Horizontal synchronism: 31.25 KHz negative

VIDEO INPUT SIGNALS

Separate video and synchronism signals Differential: RS-422 A (TTL level)
Impedance: 120 Ohm.

POWER SUPPLY

Universal between 100V AC. and 240V AC. (+10%, -15%).
Consumption: 20 W maximum.
Mains frequency: 50 - 60 Hz ±1Hz.
Fuse: 2 of 3.15AF/250V (3,15 Amp. fast-blow)

CONTROLS

Brightness Contrast

CONNECTORS

Monitor supply: bipolar connection base + ground according to IEC-320 and CEE-22 standards.
Video signals: SUB-D type 15-pin male connector.

PACKAGING

Meets the "EN 60068-2-32" standard.

AMBIENT CONDITIONS

Relative humidity: 30-95% non condensing
Operating temperature: 5ºC - 40ºC (41º F - 104ºF) with an average lower than 35ºC (95º F)
Storage temperature : between 25º C (77ºF and 70º C (158º F).
Maximum operating altitude : Meets the “IEC 1131-2” standard.

ELECTROMAGNETIC COMPATIBILITY

See Declaration of Conformity in the introduction of this manual.

SAFETY

See Declaration of Conformity in the introduction of this manual

DEGREE OF PROTECTION

Front panel: IP 54
Rear panel: IPX2
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.

Atention:

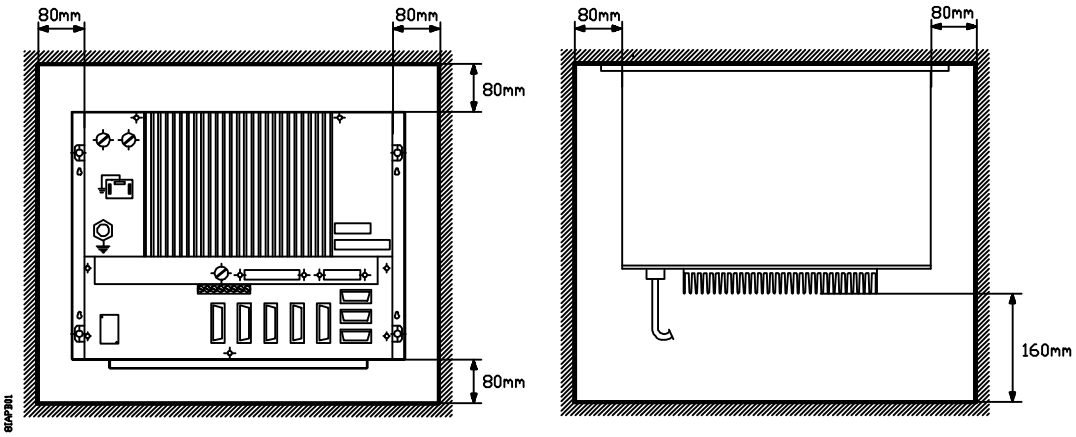
To avoid excessive heating of internal circuits, the several ventilation slits must not be obstructed. It is also necessary to install a ventilation system which extracts hot air from the housing or desk which supports the MONITOR.

APPENDIX B

ENCLOSURES

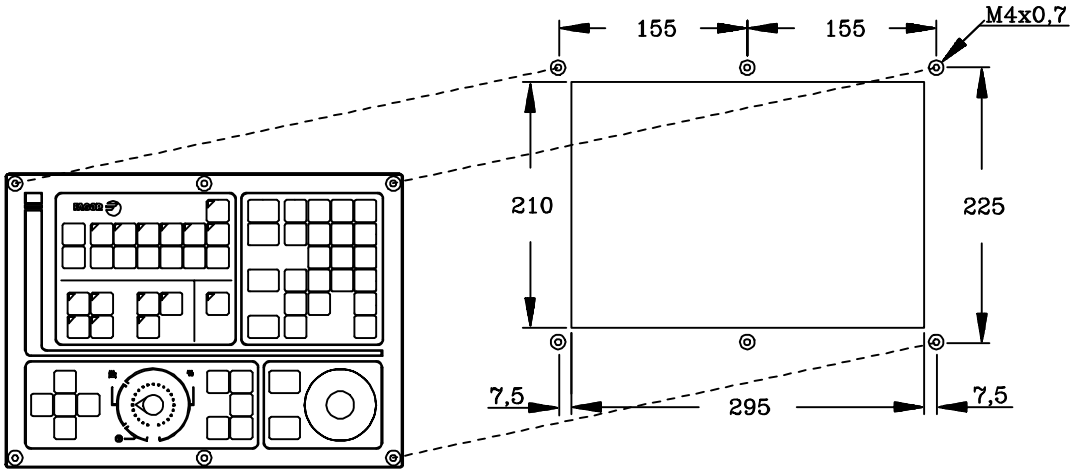
COMPACT MODEL or CENTRAL UNIT OF THE MODULAR MODEL

The minimum distance between the sides of the Central Unit and its enclosure in order to meet the required ambient conditions must be the following:



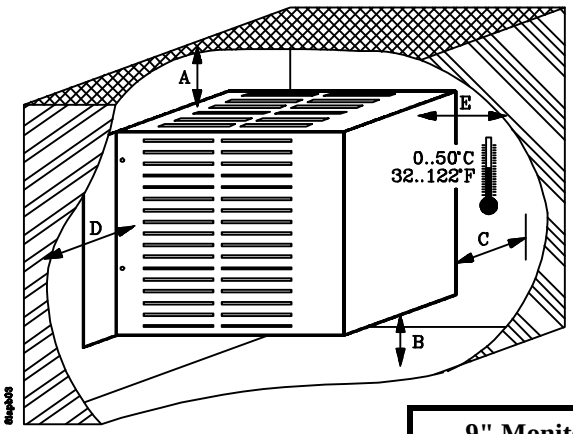
KEYBOARD

The keyboard must be mounted as indicated below: (Dimensions in mm).



MONITOR

The minimum distance, in mm, between the sides of the Monitor and its enclosure in order to meet the required ambient conditions must be the following:

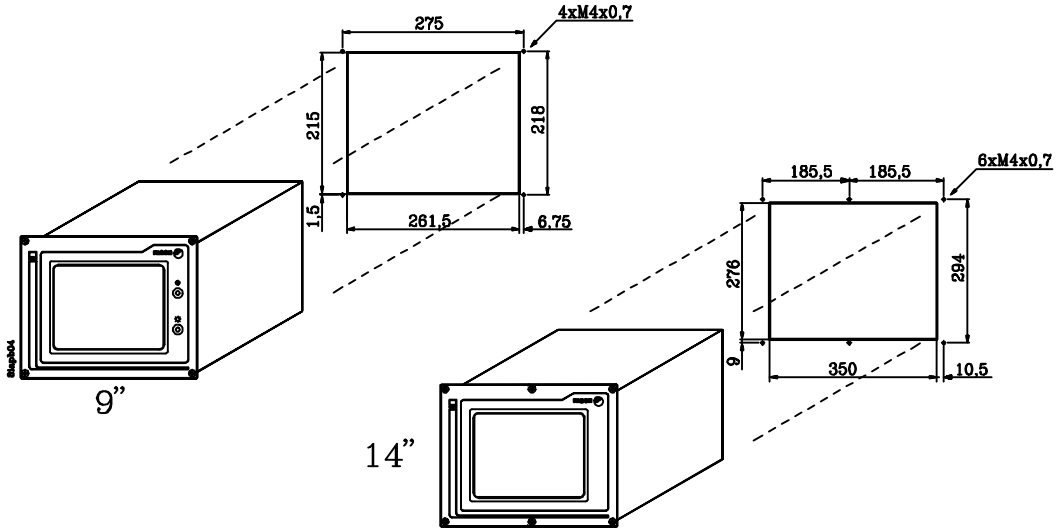


	A	B	C	D	E
9" Monitor	25 mm	25 mm	25 mm	25 mm	150 mm
14" Monitor	100 mm	100 mm	100 mm	100 mm	50 mm

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 Monitor. (Dimensions in mm)

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

The Monitor must be secured as indicated below (dimensions in mm.).



APPENDIX C

LOGIC INPUTS AND OUTPUTS

INPUTS

Pin	Connector	Function
10	I/O 1	X axis Home switch.
12	I/O 1	Z axis Home switch.
14	I/O 1	/Emergency stop
15	I/O 1	/Feed hold - /Transfer inhibit - /M-done
16	I/O 1	/Stop
17	I/O 1	Start
19	I/O 1	Manual (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
34, 35	I/O 1	Z axis analog output
36, 37	I/O 1	Spindle analog output
21	I/O 2	WORK mode indicating 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
22	I/O 2	Decoded M15 output
23	I/O 2	Decoded M14 or G00 indicating output
24	I/O 2	Decoded M13 output
25	I/O 2	Decoded M12 output

APPENDIX D

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	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		

APPENDIX E

MACHINE PARAMETER SUMMARY CHART

GENERAL MACHINE PARAMETERS

P5	Mains (AC power) frequency. 50 / 60	Section 4.3
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)	
P6	Display. 0= Real, 1= Theoretical	
P617(2)	Following error display. 0=No, 1=Yes	
P600(1)	Orientation of the machine axes.	
P606(4.5)	Axis orientation in graphic representation	
P601(1)	Machine with automatic spindle range changer. 0=No, 1=Yes	
P617(3)	Machine with automatic tool changer. 0=No, 1=Yes	

I/O RELATED MACHINE PARAMETERS Section 4.3.1

P604(4)	Normal status of the Emergency output (pin 5 of connector I/O1).0=0V, 1=24V
P604(3)	Pin 23 of connector I/O 2 as G00 indicating output. 0=No, 1=Yes
P605(4)	Pin 6 of connector I/O 1 as THREADING ON (0) or CYCLE ON (1) output
P606(7)	BCD output for decoded M functions. 0=Yes, 1=No
P602(7)	CNC waits for down flank of M-DONE signal. 0=No, 1=Yes
P603(4,3,2,1), P608(1)	Cancel feedback alarm at connectors: A1, A2, A3, A4, A5. 0=No, 1=Yes

MACHINE PARAMETERS FOR THE HANDWHEELS Section 4.3.2

P621(7)	The machine has mechanical handwheels. 0=Yes, 1=No
P622(3)	The machine has a single electronic handwheel. 0=No, 1=Yes
P823	Delay before opening the loop. 1=10ms
P609(1)	First electronic handwheel is a FAGOR 100P model. 0=No, 1=Yes
P500, P621(6)	Counting direction of the electronic handwheels (1st, 2nd)
P602(1), P621(3)	Feedback pulse units for the electronic handwheels (1st, 2nd). 0=mm, 1=Inch.
P501, P621(1,2)	Counting resolution of the electronic handwheels (1st, 2nd)
P602(4), P621(5)	Multiplying factor for signals from electronic handwheels (1st, 2nd). 0=x4, 1=x2
P617(5)	Handwheel inactive if Feedrate Override not in handwheel position. 0=No, 1=Yes
P622(1)	Handwheel settings established by PLC. 0=No, 1=Yes

MACHINE PARAMETERS FOR THE OPERATING MODES Section 4.3.3

P12	Continuous (0) or pulsating (1) axis jog
P601(5)	CYCLE-START key inhibit. 0=No, 1=Yes
P600(2)	Jogging direction assignment to X and Z jog keys
P622(2)	Incremental JOG movements in radius/diameter (0=No, 1=Yes)
P600(3)	Limit of the maximum Feedrate Override Switch value. 0=120%, 1=100%
P4	Feedrate Override Switch also active in rapid positioning movements. 0=No, 1=Yes
P601(7)	Recover initial conditions when switching to standard WORK mode. 0=No, 1=Yes
P617(6)	The "rapid jog" key applies a feedrate override range over 100%. 0=No, 1=Yes
P617(8)	Rounding possibility when defining a profile. 0=No, 1=Yes

TOOL MACHINE PARAMETERS Section 4.3.4

P700	Number of tools. 0...32
P900	X coordinate of the tool changing position
P901	Z coordinate of the tool changing position
P617(3)	Machine with automatic tool changer. 0=No, 1=Yes
P730	Subroutine associated to the T function (only when executing P99996)

MACHINE PARAMETERS FOR RS232C SERIAL LINE Section 4.3.5

P0	Transmission speed (baudrate). 110, 150, 300, 600, 1200, 2400, 4800, 9600
P1	Number of data bits per character. 0=7, 1=8
P2	Parity. 0=No, 1=ODD, 2=EVEN
P3	Stops bits. 1 or 2
P605(5)	DNC active. 0=No, 1=Yes
P605(6)	Communication parameter setting for FAGOR Cassette Unit (0) or Floppy Disk Unit (1).
P605(7)	DNC protocol active on power-up. 0=No, 1=Yes
P605(8)	The CNC does not abort DNC communication (program debugging). 0=Yes, 1=No
P606(8)	Status report by interruption. 0=No, 1=Yes

MACHINE PARAMETERS FOR THE AXES

P100, P300	Sign of the analog output for X and Z	Section 5
P101, P301	Counting direction for X and Z	Section 5
P102, P302	Jogging direction for X and Z	Section 5

MACHINE PARAMETERS FOR AXIS RESOLUTION — Section 5.1

P103, P303	Counting resolution for X and Z
P602(3), P602(2)	Feedback units for X and Z. 0=mm, 1=Inch.
P106, P306	Feedback signal type for X and Z. 0=Square, 1=Sinewave
P602(6), P602(5)	Multiplying factor for X and Z feedback signals. 0=x4, 1=x2
P619(1), P619(2)	Special multiplier for X and Z sine-wave feedback always applied. 0=No, 1=Yes
P604(2), P604(1)	Binary encoder for X and Z axes. 0=No, 1=Yes
P604(7), P604(6)	Pulse/rev. equivalence for X and Z binary encoder

MACHINE PARAMETERS FOR AXIS ANALOG OUTPUT — Section 5.2

P117, P317	Minimum X and Z analog output voltage. 1=2.5mV
P104, P304	Dwell between Enable signal and analog output for X and Z. 0=No, 1=Yes
P118, P318	Width of the in-position zone for X and Z. 0 to 255 microns
P105, P305	Continuous control of X and Z. 0=No, 1=Yes

MACHINE PARAMETERS FOR AXIS TRAVEL LIMITS — Section 5.3.

P107, P307	Positive X and Z travel limit
P108, P308	Negative X and Z travel limit

MACHINE PARAMETERS FOR THE LEADSCREW — Section 5.4.

P109, P309	Leadscrew backlash compensation for X and Z. 0 to 255 microns
P620(1) P620(2)	Sign of the leadscrew backlash compensation for X and Z. 0=Positive, 1=Negative
P113, P313	Additional analog pulse for X and Z. 1=2.5mV
P605(2), P605(1)	Activate leadscrew error compensation for X and Z. 0=No, 1=Yes

FEEDRATE MACHINE PARAMETERS — Section 5.5

P110, P310	Maximum programmable feedrate for X and Z
P111, P311	Feedrate for rapid traverse (in G00) for X and Z
P717	Maximum feedrate for arcs
P703	Feedrate/Override value when the axis analog reaches 10V
P705	Feedrate monitoring between 50% and 200% of programmed value

MACHINE PARAMETERS FOR AXIS CONTROL — Section 5.6

P114, P314	Proportional gain K1 for X and Z
P115, P315	Gain break-point for X and Z
P116, P316	Proportional gain K2 for X and Z
P607(6)	Only K1 applied on threading operations. 0=No, 1=Yes
P607(7)	Only K2 applied on rapid moves. 0=No, 1=Yes
P715	Recovery of programmed position on "non-continuously controlled" axes

MACHINE PARAMETERS FOR HOME SEARCH Section 5.7

P119, P319	Home coordinates for X and Z
P618(8), P618(7)	Home search direction for X and Z. 0=Positive, 1=Negative
P600(7), P600(6)	Home pulse (marker) type for X and Z. 0=Negative, 1=Positive
P600(5), P600(4)	Home switch for X and Z. 0=Yes, 1=No
P112, P312	1st home searching feedrate for X and Z
P807, P808	2nd home searching feedrate for X and Z
P604(8)	Mandatory home search on power-up. 0=No, 1=Yes

MACHINE PARAMETERS FOR ACCELERATION/DECELERATION Section 5.8

P712, P713	Apply ACC/DEC onto X and Z. 1=20ms
P609(4)	ACC/DEC on all linear interpolations. 0=No, 1=Yes
P616(6)	ACC/DEC in G05 (corner rounding). 0=Yes, 1=No
P621(8)	Bell-shaped ACC/DEC ramp. 0=No, 1=Yes
P731	Duration of the bell-shaped ACC/DEC ramp. 1=10ms
P720, P721	FEED-FORWARD gain for X and Z

MACHINE PARAMETERS FOR THE LIVE TOOL Section 5.9

P607(1)	Sign of the analog output for the live tool.
P609(8)	Possibility to vary the live tool speed with spindle speed override keys. 0=No, 1=Yes
P802	Maximum programmable live tool rpm

SPECIAL MACHINE PARAMETERS Section 5.10

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

SPINDLE MACHINE PARAMETERS

P811	Acceleration/Deceleration control for the spindle. 0=No, 1=Yes	Section 6
P617(4)	The actual spindle speed is always shown in RPM. 1=10ms	Section 6

MACHINE PARAMETERS FOR SPINDLE SPEED RANGE CHANGE Section 6.1

P7, P8, P9, P10	Maximum spindle speed for 1st, 2nd, 3rd and 4th RANGES
P601(1)	Machine with automatic spindle range changer. 0=No, 1=Yes
P601(6)	Residual analog S voltage for spindle range change. 0=No, 1=Yes
P701	Value of the residual spindle speed for range change. 1=2.5mV
P702	Oscillation time during a range change

MACHINE PARAMETERS FOR THE ANALOG SPINDLE SPEED OUTPUT Section 6.2

P601(4)	Sign of the analog spindle output S
P607(4)	Unipolar (1) or bipolar (0) analog spindle output S

MACHINE PARAMETERS FOR THE BCD CODED SPINDLE SPEED OUTPUT Section 6.3

P601(3)	2-digit BCD coded S output. 0=No, 1=Yes
P601(2)	4-digit BCD coded S output. 0=No, 1=Yes

MACHINE PARAMETERS FOR SPINDLE CONTROL Section 6.4

P800	Number of pulses of the spindle encoder
P606(3)	Sign of the spindle feedback
P603(8)	Active spindle speed monitoring. 0=Yes, 1=No
P704	Spindle speed stabilizing period. 1=0.1sec.
P617(7)	M3/M4 confirmation by detecting spindle feedback reversal

SPECIFIC MACHINE PARAMETERS FOR SPINDLE ORIENTATION Section 6.4.1

P600(8)	Type of Reference marker pulse (Io) of spindle encoder. 0=Negative, 1=Positive
P606(2)	Direction of spindle orientation
P706	RPM during spindle orientation
P707	Spindle in-position zone (dead-band) during orientation
P708	Spindle gain
P709	Minimum spindle analog output. 1=2.5mV

APPENDIX F

SEQUENTIAL MACHINE PARAMETER LIST

P0	Transmission speed (baudrate). 110, 150, 300, 600, 1200, 2400, 4800, 9600	Section 4.3.5
P1	Number of data bits per character. 0=7, 1=8	Section 4.3.5
P2	Parity. 0=No, 1=ODD, 2=EVEN)	Section 4.3.5
P3	Number of stop bits. 1 or 2	Section 4.3.5
P4	The Feedrate Override Switch active also in rapid moves. 0=No, 1=Yes	Section 4.3.3
P5	Mains (AC power) frequency. 50 or 60	Section 4.3
P6	Theoretical (1) or Real (0) axis position display	Section 4.3
P7	Maximum spindle speed for 1st RANGE	Section 6.1
P8	Maximum spindle speed for 2nd RANGE	Section 6.1
P9	Maximum spindle speed for 3rd RANGE	Section 6.1
P10	Maximum spindle speed for 4th RANGE	Section 6.1
P11	X axis position in radius (0) or diameter (1)	Section 4.3
P12	Continuous (0) or pulsating (1) axis JOG	Section 4.3.3
P13	Measuring units. 0=mm, 1=Inches	Section 4.3
P99	Language. 0= Spanish, 1=German, 2=English, 3=French, 4=Italian	Section 4.3
P100	Sign of the X axis analog output	Section 5
P101	Counting direction of the X axis feedback	Section 5
P102	Jogging direction for the X axis	Section 5
P103	X axis feedback (counting) resolution	Section 5.1
P104	Delay between Enable and analog output for the X axis. 0=No, 1=Yes	Section 5.2
P105	Continuous control of the X axis. 0=No, 1=Yes	Section 5.2
P106	Type of X axis feedback signal. 0=Squarewave, 1=Sinewave	Section 5.1
P107	Positive X axis travel limit	Section 5.3
P108	Negative X axis travel limit	Section 5.3
P109	Leadscrew backlash compensation for the X axis. 0 to 255 microns	Section 5.4
P110	Maximum programmable feedrate for the X axis	Section 5.5
P111	G00 feedrate for the X axis	Section 5.5
P112	1st home searching feedrate for the X axis	Section 5.7
P113	Additional analog pulse for the X axis. 1=2.5mV	Section 5.4
P114	Proportional Gain K1 for the X axis	Section 5.6
P115	Gain break-point of the X axis	Section 5.6
P116	Proportional Gain K2 for the X axis	Section 5.6
P117	Minimum analog voltage for the X axis. 1=2.5mV	Section 5.2
P118	In-position zone for the X axis. 0 to 255 microns	Section 5.2
P119	Home coordinate for the X axis	Section 5.7
P300	Sign of the Z axis analog output	Section 5
P301	Counting direction of the Z axis feedback	Section 5
P302	Jogging direction for the Z axis	Section 5
P303	Z axis feedback (counting) resolution	Section 5.1
P304	Delay between Enable and analog output for the Z axis. 0=No, 1=Yes	Section 5.2
P305	Continuous control of the Z axis. 0=No, 1=Yes	Section 5.2
P306	Type of Z axis feedback signal. 0=Squarewave, 1=Sinewave	Section 5.1
P307	Positive Z axis travel limit	Section 5.3
P308	Negative Z axis travel limit	Section 5.3
P309	Leadscrew backlash compensation for the Z axis. 0 to 255 microns	Section 5.4
P310	Maximum programmable feedrate for the Z axis	Section 5.5
P311	G00 feedrate for the Z axis	Section 5.5
P312	1st home searching feedrate for the Z axis	Section 5.7
P313	Additional analog pulse for the Z axis. 1=2.5mV	Section 5.4
P314	Proportional Gain K1 for the Z axis	Section 5.6
P315	Gain break-point of the Z axis	Section 5.6
P316	Proportional Gain K2 for the Z axis	Section 5.6
P317	Minimum analog voltage for the Z axis. 1=2.5mV	Section 5.2
P318	In-position zone for the Z axis. 0 to 255 microns	Section 5.2
P319	Home coordinate for the Z axis	Section 5.7

P500		Counting direction of the 1st electronic handwheel	Section 4.3.2
P501		Counting resolution of the 1st electronic handwheel. 1=0.001mm or 0.0001".....	Section 4.3.2
P502 thru P519		*Not being used at this time " =0 "*	
P600	(8)	Type of home pulse (marker) for spindle encoder. 0=Negative, 1=Positive	Section 6.4.1
	(7)	Type of home pulse (marker) for the X axis. 0=Negative, 1=Positive	Section 5.7
	(6)	Type of home pulse (marker) for the Z axis. 0=Negative, 1=Positive	Section 5.7
	(5)	Home switch for the X axis. 0=Yes, 1=No	Section 5.7
	(4)	Home switch for the Z axis. 0=Yes, 1=No	Section 5.7
	(3)	Maximum value applied by Feedrate Override Switch. 0=120%, 1=100%	Section 4.3.3
	(2)	Movement direction assignment to X and Z JOG keys. 0=No, 1=Yes	Section 4.3.3
	(1)	Orientation of the axes on the machine.....	Section 4.3
P601	(8)	*Not being used at this time " =0 "*	
	(7)	Recover initial conditions when switching to standard mode. 0=No, 1=Yes	Section 4.3.3
	(6)	Residual spindle analog voltage for range change. 0=No, 1=Yes	Section 6.1
	(5)	Inhibit CYCLE START key. 0=No, 1=Yes	Section 4.3.3
	(4)	Sign of the spindle analog output S	Section 6.2
	(3)	2-digit BCD coded spindle speed output. 0=No, 1=Yes	Section 6.3
	(2)	4-digit BCD coded spindle speed output. 0=No, 1=Yes	Section 6.3
	(1)	Machine with automatic spindle speed range changer. 0=No, 1=Yes.....	Section 6.1
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 4.3.1
	(6)	Multiplying factor for X axis feedback signals. 0=x4, 1=x2	Section 5.1
	(5)	Multiplying factor for Z axis feedback signals. 0=x4, 1=x2	Section 5.1
	(4)	Multiplier for feedback signals from 1st electronic handwheel. 0=x4, 1=x2	Section 4.3.2
	(3)	Measuring units of the X axis feedback signals. 0=mm, 1=Inch	Section 5.1
	(2)	Measuring units of the Z axis feedback signals. 0=mm, 1=Inch	Section 5.1
	(1)	Measuring units of the 1st handwheel feedback signals. 0=mm, 1=Inch ...	Section 4.3.2
P603	(8)	Active spindle speed monitoring. 0=Yes, 1=No	Section 6.4
	(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)	Cancel feedback alarm for input: A1. 0=No, 1=Yes	Section 4.3.1
	(3)	Cancel feedback alarm for input: A2. 0=No, 1=Yes	Section 4.3.1
	(2)	Cancel feedback alarm for input: A3. 0=No, 1=Yes	Section 4.3.1
	(1)	Cancel feedback alarm for input: A4. 0=No, 1=Yes	Section 4.3.1
P604	(8)	Mandatory home search on power-up. 0=No, 1=Yes	Section 5.7
	(7)	Pulse count equivalence for X axis binary encoder	Section 5.1
	(6)	Pulse count equivalence for Z axis binary encoder	Section 5.1
	(5)	*Not being used at this time " =0 "*	
	(4)	Normal status of Emergency output (pin 5 of conn. I/O1). 0=0V 1=24V ...	Section 4.3.1
	(3)	Pin 23 of connector I/O2 as G00 indicator output. 0=No, 1=Yes	Section 4.3.1
	(2)	Binary encoder on X axis. 0=No, 1=Yes	Section 5.1
	(1)	Binary encoder on Z axis. 0=No, 1=Yes	Section 5.1
P605	(8)	The CNC does abort DNC communication (debugging). 0=Yes, 1=No	Section 4.3.5
	(7)	DNC protocol active on power-up. 0=No, 1=Yes	Section 4.3.5
	(6)	Communication settings for FAGOR Cassette (0) or Floppy Disk Unit (1).	Section 4.3.5
	(5)	DNC active. 0=No, 1=Yes	Section 4.3.5
	(4)	Pin 6 of I/O1 as TREADING-ON (0) or CYCLE-ON (1) indicator output ..	Section 4.3.1
	(3)	*Not being used at this time " =0 "*	
	(2)	Activate X axis leadscrew error compensation. 0=No, 1=Yes	Section 5.4
	(1)	Activate Z axis leadscrew error compensation. 0=No, 1=Yes	Section 5.4

P606	(8)	Status report by interruption. 0=No, 1=Yes	Section 4.3.5
	(7)	BCD output for decoded M functions. 0=Yes, 1=No	Section 4.3.1
	(6)	*Not being used at this time " =0 "*	
	(5)	Axis orientation in graphic representation	Section 4.3
	(4)	Axis orientation in graphic representation	Section 4.3
	(3)	Sign of the spindle feedback	Section 6.4
	(2)	Direction of spindle orientation	Section 6.4.1
	(1)	Machine with travels over 8 meters. 0=No, 1=Yes	Section 5.10
P607	(8)	*Not being used at this time " =0 "*	
	(7)	Only K2 gain applied on G00 moves. 0=No, 1=Yes	Section 5.6
	(6)	Only K1 gain applied on threading operations. 0=No, 1=Yes	Section 5.6
	(5)	*Not being used at this time " =0 "*	
	(4)	Unipolar (1) or bipolar (0) spindle analog output S	Section 6.2
	(3)	*Not being used at this time " =0 "*	
	(2)	*Not being used at this time " =0 "*	
	(1)	Sign of the analog output for the live tool	Section 5.9
P608	(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)	*Not being used at this time " =0 "*	
	(3)	*Not being used at this time " =0 "*	
	(2)	*Not being used at this time " =0 "*	
	(1)	Cancel feedback alarm for input A5. 0=No, 1=Yes	Section 4.3.1
P609	(8)	Possibility to vary live tool speed with spindle override keys. 0=No, 1=Yes	Section 5.9
	(7)	Resolution of 0.0001mm and 0.00001 inch. 0=No, 1=Yes.	Section 5.10
	(6)	*Not being used at this time " =0 "*	
	(5)	*Not being used at this time " =0 "*	
	(4)	ACC/DEC applied on all linear interpolations. 0=No, 1=Yes	Section 5.8
	(3)	*Not being used at this time " =0 "*	
	(2)	*Not being used at this time " =0 "*	
	(1)	1st electronic handwheel is a FAGOR 100P. 0=No, 1=Yes	Section 4.3.2
P610		*Not being used at this time " =0 "*	
P611		*Not being used at this time " =0 "*	
P612		*Not being used at this time " =0 "*	
P613		*Not being used at this time " =0 "*	
P614		*Not being used at this time " =0 "*	
P615		*Not being used at this time " =0 "*	
P616	(8)	*Not being used at this time " =0 "*	
	(7)	*Not being used at this time " =0 "*	
	(6)	ACC/DEC applied in G05 (corner rounding). 0=Yes, 1=No	Section 5.8
	(5)	The PLCI uses marks M1801 thru M1899 to send messages to CNC	PLCI Manual
	(4)	*Not being used at this time " =0 "*	
	(3)	*Not being used at this time " =0 "*	
	(2)	*Not being used at this time " =0 "*	
	(1)	*Not being used at this time " =0 "*	
P617	(8)	Rounding possibility when defining a profile. 0=No, 1=Yes	Section 4.3.3
	(7)	M3/M4 confirmation by detecting spindle feedback reversal. 0=No, 1=Yes	Section 6.4
	(6)	The rapid jog key applies a feedrate override over 100%. 0=No, 1=Yes ...	Section 4.3.3
	(5)	Handwheel inactive if Feed. Over. not in handwheel position. 0=No, 1=Yes	Section 4.3.2
	(4)	The actual spindle speed is always shown in RPM. 0=No, 1=Yes	Section 6
	(3)	Machine with automatic tool changer. 0=No, 1=Yes	Section 4.3.4
	(2)	Display the following error. 0=No, 1=Yes	Section 4.3
	(1)	The CNC has a PLCI	PLCI Manual

P618	(8)	Home search direction for the X axis	Section 5.7
	(7)	Home search direction for the Z axis	Section 5.7
	(6)	*Not being used at this time* " =0 "	
	(5)	*Not being used at this time* " =0 "	
	(4)	*Not being used at this time* " =0 "	
	(3)	*Not being used at this time* " =0 "	
	(2)	*Not being used at this time* " =0 "	
	(1)	*Not being used at this time* " =0 "	
P619	(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)	*Not being used at this time* " =0 "	
	(3)	*Not being used at this time* " =0 "	
	(2)	Special multiplying factor for Z axis sine-wave feedback signals	Section 5.1
	(1)	Special multiplying factor for X axis sine-wave feedback signals	Section 5.1
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)	*Not being used at this time* " =0 "	
	(3)	*Not being used at this time* " =0 "	
	(2)	Sign of the leadscrew backlash compensation for Z axis. 0=Pos, 1=Neg. ..	Section 5.4
	(1)	Sign of the leadscrew backlash compensation for X axis. 0=Pos, 1=Neg ...	Section 5.4
P621	(8)	Bell-shaped ACC/DEC ramp. 0=No, 1=Yes	Section 5.8
	(7)	The machine has mechanical handwheels. 0=Yes, 1=No	Section 4.3.2
	(6)	Counting direction of the 2nd electronic handwheel	Section 4.3.2
	(5)	Multiplier for feedback signals from 2nd electronic handwheel. 0=x4, 1=x2	Section 4.3.2
	(4)	*Not being used at this time* " =0 "	
	(3)	Feedback units for 2nd electronic handwheel. 0=mm, 1=Inch	Section 4.3.2
	(2)	Feedback (counting) resolution of the 2nd electronic handwheel	Section 4.3.2
	(1)	Feedback (counting) resolution of the 2nd electronic handwheel	Section 4.3.2
P622	(8,7,6,5,4)	*Not being used at this time* " =0 "	
	(3)	The machine has a single electronic handwheel. 0=No, 1=Yes	Section 4.3.2
	(2)	Incremental JOG movements in radius/diameter. 0=No, 1=Yes	Section 4.3.3
	(1)	Handwheel settings established by PLC. 0=No, 1=Yes	Section 4.3.2
P623		*Not being used at this time* " =0 "	
P700		Number of tools 0...32	Section 4.3.4
P701		Value of residual analog voltage for spindle speed range change. 1=2.5mV	Section 6.1
P702		Oscillation period during spindle range change	Section 6.1
P703		Feedrate/Override value when axis analog voltage reaches 10 V	Section 5.5
P704		Stabilizing period for Spindle analog voltage S	Section 6.4
P705		Error if axis feedrate isn't between 50% and 200% of programmed value	Section 5.5
P706		RPM during spindle orientation	Section 6.4.1
P707		Spindle in-position zone (dead-band) during orientation	Section 6.4.1
P708		Spindle gain	Section 6.4.1
P709		Minimum spindle analog output	Section 6.4.1
P710		*Not being used at this time* " =0 "	
P711		*Not being used at this time* " =0 "	
P712		ACC/DEC for X axis. 1=20ms	Section 5.8
P713		ACC/DEC for Z axis. 1=20ms	Section 5.8
P714		*Not being used at this time* " =0 "	
P715		Recovery of programmed position on "non-continuously controlled" axes	Section 5.6
P716		*Not being used at this time* " =0 "	
P717		Maximum feedrate for arcs	Section 5.5
P718, P719		*Not being used at this time* " =0 "	

P720	FEED-FORWARD gain for X axis	Section 5.8
P721	FEED-FORWARD gain for Z axis	Section 5.8
P722 thru P728	*Not being used at this time “ =0 “*	
P729	Execution frequency for a new PLCI cycle	PLCI Manual
P730	Subroutine associated to T function (only when executing P99996) ..	Section 4.3.4
P731	Duration of bell-shaped ACC/DEC ramp. (1=10ms)	Section 5.8
P732 thru P741	*Not being used at this time “ =0 “*	

P800	Number of pulses from spindle encoder	Section 6.4
P801	*Not being used at this time “ =0 “*	
P802	Maximum programmable live tool rpm	Section 5.9
P803 thru P806	*Not being used at this time “ =0 “*	
P807	2nd home searching feedrate for X axis	Section 5.7
P808	2nd home searching feedrate for Z axis	Section 5.7
P809	*Not being used at this time “ =0 “*	
P810	*Not being used at this time “ =0 “*	
P811	Acceleration/Deceleration control for the spindle. 1=10ms	Section 6
P812 thru P822	*Not being used at this time “ =0 “*	
P823	Delay before opening the loop	Section 4.3.2

P900	X coordinate of tool change position	Section 4.3.4
P901	Z coordinate of tool change position	Section 4.3.4
P902 thru P923	*Not being used at this time “ =0 “*	

APPENDIX G

MACHINE PARAMETER SETTING CHARTS

PARAMETER	VALUE	PARAMETER	VALUE
P0		P8	
P1		P9	
P2		P10	
P3		P11	
P4		P12	
P5		P13	
P6			
P7		P99	

PARAMETER	VALUE	PARAMETER	VALUE
P100		P300	
P101		P301	
P102		P302	
P103		P303	
P104		P304	
P105		P305	
P106		P306	
P107		P307	
P108		P308	
P109		P309	
P110		P310	
P111		P311	
P112		P312	
P113		P313	
P114		P314	
P115		P315	
P116		P316	
P117		P317	
P118		P318	
P119		P319	

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 H

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 :

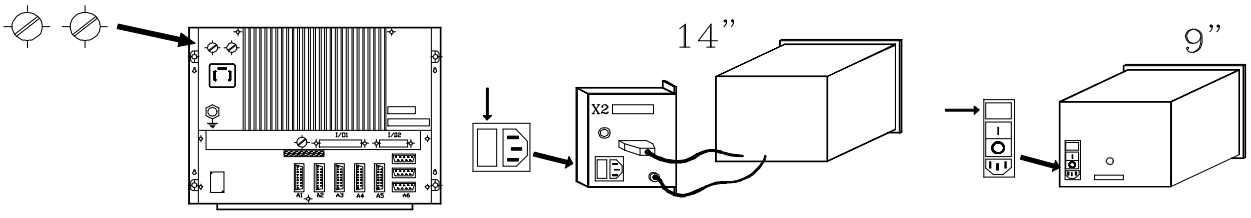
1.- Grease and mineral oils
2.- Bases and bleach
3.- Dissolved detergents
4.- Alcohol

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	800 T 800 TI 800 TG 800 TGI	83370000 83370001 83370004 83370005		
9" Monocrome Monitor		83520000	Fagor Automation	
14" Color Monitor		83520002	Fagor Automation	
Operator Panel		83580000	Fagor Automation	
Cable set for the Monitor	de 5 m de 10 m de 15 m de 20 m	83630001 83630002 83630003 83630007	Fagor Automation	
Cable set for the Operator Panel	de 5 m de 10 m de 15 m de 20 m	83630004 83630005 83630006 83630008	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	83750036 83750015	Fagor Automation	

ERROR CODES

001 This error occurs when the first character of the block to be executed is not an "N".

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 G20, G21, G22, G23, G24, G25, G26, G27, G28, G29, G30, G31, G32, G50, 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 Negative or too large thread pitch value.

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.

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.

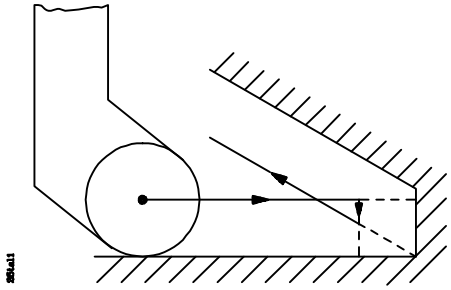
028 An attempt has been made to select a tool offset at the tool table or a non-existent external tool (the number of tools is set by machine parameter).

029 Too large a value assigned to a function.

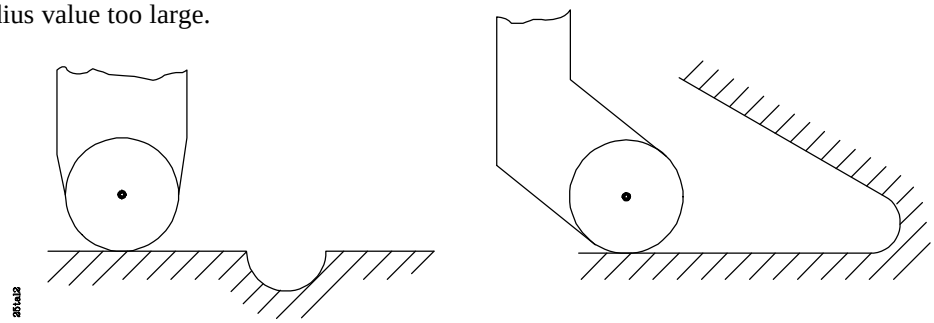
This error is often issued when programming an F value in mm/min (inch/min) and, then, switching to work in mm/rev (inch/rev) without changing the F value.

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:

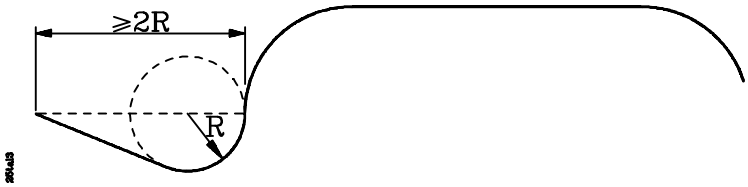
> More than 15 nesting levels when calling subroutines.

> 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:

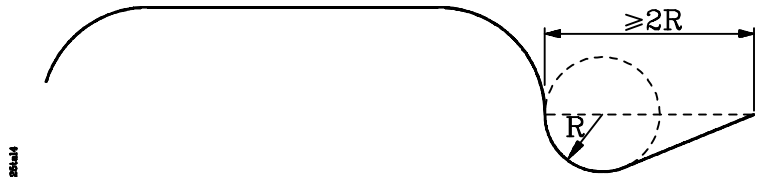
> There is no room to perform the tangential entry. A clearance of twice the rounding radius or greater is required.



> 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 Constant Surface Speed has been selected while the machine uses the BCD coded spindle speed output.

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

055 Parity error when reading or writing a cassette or a disk.

057 Write-protected tape or disk.

058 Sluggish tape or disk movement.

059 CNC communication error with the cassette reader or FAGOR Floppy Disk Unit.

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.

Atention:

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 if, while probing (G75), the programmed position is reached without receiving the probe signal.

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.

072 ** Z axis following error.

074 ** "S" value (spindle speed) too large.

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.

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.

095 ** Parity error in general parameters.

096 ** Parity error in Z axis parameters.

098 ** Parity error in X axis parameters.

099 ** Parity error in M table.

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.

110 ** Error in X axis leadscrew error compensation parameters.

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.