

# FAGOR AUTOMATION

## CNC 101/102(S)

New Features (Version 0110 in)



**FAGOR** 

## ERRORS DETECTED IN THE INSTALLATION MANUAL (REF. 9703)

### Comparison table (page x). General characteristics.

In the "Axes" section" where it says "Axes X + Y + Auxiliary handwheel"

It should say "X Axis + Auxiliary Y axis (not dro) + Auxiliary handwheel"

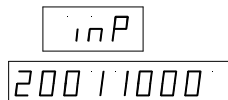
### Comparison table (page xii). Programming.

The programming function G34 is missing:

G34 X axis as an infinite follower of another axis (only for the 101S)

Section 2.3.4 (chapter 2 page 8). Table below.

Digits 7 and 8 are backwards. They should be like this:



Digit	Corresponding Input	Pin
8	External feedrate override 1	10 (I/O1)
7	External feedrate override 2	11 (I/O1)
6	X axis feedback error	
5	Y axis feedback error	
4	X axis sine-wave feedback alarm	
3	Y axis sine-wave feedback alarm	
2	Over-temperature	
1	Not being used at this time	

### Section 3.5 (chapter 3 page 8). Feedback alarm parameter P22(7), P62(7)

The last paragraph is wrong, it should read:

"If the feedback system ..... with a value of "0" (feedback alarm OFF)".

### Section 4.4.2 (chapter 4 page 13). Open positioning loop

The first paragraph is wrong, it should say:

It is necessary to set P23(4)=0.

### Appendix "F" (page 9). I/O related parameters

The first line is wrong, it should read:

P22(7), P62(7) Feedback alarm ON (1) or OFF (0) on the X, Y axis (respectively).

P63(6) The probe is active high, P63(6)=0, or low, P63(6)=1.

### Appendix "G" (page 12). P22(7)

It is wrong, it should read:

P22(7) Feedback alarm ON (1) or OFF (0) on the X axis.

### Appendix "G" (page 13). P62(7)

It is wrong, it should read:

P62(7) Feedback alarm ON (1) or OFF (0) on the Y axis.

P63(6) The probe is active high, P63(6)=0, or low, P63(6)=1.

## MODIFICATIONS TO THE INSTALLATION MANUAL (REF. 9703)

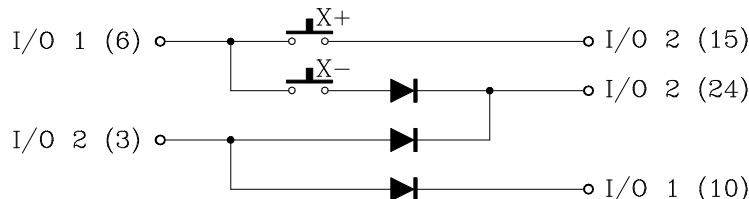
### Section 1.6.1 (chapter 1 page 11). RESET

The last paragraph should read:

When setting machine parameter "P30" with a value other than "0", the CNC behaves as follows: A leading edge (up flank) interrupts the execution and it is redirected to a HOME block, but the HOME function is not executed until a trailing edge (down flank) is detected.

### Section 1.8.1 (chapter 1 page 22). External keys without the "JOG 100" keypad (jog box)

Connection example using only the external control keys "X+", "X-"



### Section 4.1.1.1 (chapter 4 page 11). P63(1) Acceleration/deceleration in all G01 movements

It should read:

P63(1) Acceleration/deceleration also on linear interpolations (G01)

It defines if the CNC, besides applying acc/dec ramps (P16, P17, P91, P92) on all rapid movements at F0, it also applies them on linear interpolations (G01).

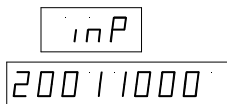
0 = Only on rapid movements (G00) and at F0.

1 = On rapid movements and on all linear interpolations (G01).

## MODIFICATIONS TO THE OPERATION MANUAL (REF. 9703)

Section 2.3.4 (chapter 2 page 8). Table below.

Digits 7 and 8 are backwards. They should be like this:



Digit	Corresponding Input	Pin
8	External feedrate override 1	10 (I/O1)
7	External feedrate override 2	11 (I/O1)
6	X axis feedback error	
5	Y axis feedback error	
4	X axis sine-wave feedback alarm	
3	Y axis sine-wave feedback alarm	
2	Over-temperature	
1	Not being used at this time	

### Section 6.5.4 (chapter 6 page 17). Synchronism (G33)

In the printing example.

- \* If the turning of the roller is controlled with an external device, it is not necessary to program the "N0 S1000 M3" block, but machine parameter P36 must be set to the approximate rpm of the roller.
- \* If the roller does not have an encoder, because no controlled synchronism is needed, pin 5 of connector A2 must be connected to 5V.

## *Software Version 2.02 (May 1998)*

### 1. ASSUME X1 OF THE HANDWHEEL WITH THE "JOG 100" JOG BOX

Machine parameter P102(7) indicates whether the axes can be jogged or not with the handwheel when the Feedrate Override Switch is positioned out of the handwheel markings while using the "JOG 100" jog box and the JOG mode is selected.

- P102(7)=0 It is not possible. The handwheel is active in the handwheel positions only.  
 P102(7)=1 The handwheel is active in any position of the Feedrate Override Switch.

When machine parameter P102(7)=1, the CNC applies the "x1" factor when the switch is positioned out of the handwheel positions.

### 2. THE CNC101 ALSO HAS ARITHMETIC PROGRAMMING.

From this version on, the CNC101 model also has arithmetic programming with conditional jumps (G26, G27, G28, G29).

			101	101S	102	102S
Arithmetic programming			x	x	x	x
Arithmetic parameters			100	100	100	100
<b>PROGRAMMING FUNCTIONS</b>	G26	Jump if zero	x	x	x	x
	G27	Jump if not zero	x	x	x	x
	G28	Jump if less than zero	x	x	x	x
	G29	Jump if equal or greater than zero	x	x	x	x

### 3. AXES NOMENCLATURE.

With bits 1 and 2 of parameter P102, the axes nomenclature may be defined.

P102(2)	P102(1)	Axes Nomenclature
0	0	X Y
0	1	Y C
1	0	X Z
1	1	Y Z

The new denomination selected only affects the display, they will keep being X and Y internally. Therefore, when accessing the CNC via DNC 100, the axes will always be X Y.

### 4. MOVEMENTS IN G75

In previous versions, when a movement programmed in G75 reached position and the CNC had not yet received the probe signal, the CNC would issue error 21.

From this version on, parameter P102(5) indicates whether the CNC issues error 21 or not.

- P102(5)=0 It issues error 21. Like until now.  
 P102(5)=1 It does not issue error 21. It goes on executing the next block.

## 5. DISPLAY OF THE AXIS IN EXECUTION

In previous versions, while in Automatic mode, the CNC could change the axis being displayed depending on the movement programmed:

- If both axes move => it keeps displaying the axis selected with A+ ,A-
- If only the X axis moves => it displays the X axis
- If only the Y axis moves => it displays the Y axis

From this version on, parameter P102(6) determines whether the CNC behaves like before or it does not change the axis being displayed.

- P102(6)=0 Like before
- P102(6)=1 The CNC does not change the axis. It keeps displaying the axis selected with A+ ,A-

# *Software Version 2.03 (January 1999)*

## 1. ADDITIONAL MOVEMENT WITH G75

When probing at high speed, it could stop abruptly making the axis overshoot the programmed position and having to move back into position.

In previous versions of the CNC 101 S, machine parameter P82 could be used to minimize this sometimes undesirable effect.

This parameter indicates to the CNC the distance the axis must move after receiving the probe signal, thus stopping smoothly.

From this version on, this feature will also be available on the "102 S" CNC model and parameter P83 indicates the distance the Y axis must move after receiving the probe signal.

- Therefore:
- P82 indicates the distance the X axis must move after receiving the probe signal.
  - P83 indicates the distance the Y axis must move after receiving the probe signal.
- Possible values:
- From 1 to 65535 microns.
  - From 1 to 25801 tenth-thousandths of an inch.

# *Software Version 2.05 (October 2001)*

## 1. OPERATION WITH 100-LINE HANDWHEELS (U.F.O.)

Until now, the CNC 101/102 was ready to operate with 25-line handwheels. It internally multiplies by 4 in order to obtain 100 pulses per each turn of the handwheel.

From this version on, it is also possible to use 100-line handwheels (Fagor UFO model handwheels)  
Set machine parameter P103(2)=1 so its pulses are not multiplied by 4.

This feature is only available when connecting the handwheel to the CNC's feedback input.  
The auxiliary handwheel, connected to the digital inputs of the CNC must always have 25 lines per turn.

---

<p><b>Headquarters (SPAIN):</b> Fagor Automation S. Coop. B° San Andrés s/n, Apdo. 144 E-20500 Arrasate - Mondragón Tel: (34)-943 71 92 00 Fax: (34)-943 79 17 12 (34)-943 77 11 18 (Service Dept.) <a href="http://www.fagorautomation.com">www.fagorautomation.com</a> E-mail: <a href="mailto:info@fagorautomation.es">info@fagorautomation.es</a></p>
---

**101 / 101S CNC  
102 / 102S CNC**

**OPERATING MANUAL**

9703 (ing)

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

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

# INDEX

---

<u>Section</u>	<u>Page</u>
<hr/>	
Comparison table for FAGOR CNC models: 101/101S/102/102S .....	<i>ix</i>
New Features and Modifications .....	<i>xiii</i>
<hr/>	
<b>INTRODUCTION</b>	
Safety Conditions .....	3
Material Returning Terms .....	6
Fagor Documentation for the 101/101S/102/102S CNC .....	7
Manual Contents .....	8
<hr/>	
<b>Chapter 1            PERIPHERALS</b>	
1.1      Front panel description .....	1
1.2      Peripheral mode .....	2
1.2.1    "1-Output" option. CNC --> peripheral .....	3
1.2.2    "2-Input" option. Peripheral --> CNC .....	4
<hr/>	
<b>Chapter 2            AUXMODE</b>	
2.1      System Input/Output test .....	2
2.2      Machine parameters .....	5
2.2.1    Editing machine parameters .....	5
2.3      Tool table or zero offset table .....	7
<hr/>	
<b>Chapter 3            JOG MODE</b>	
3.1      Jogging the axes .....	1
3.2      Automatic positioning .....	5
3.3      Zero setting or coordinate preset .....	5
3.4      Machine Reference (Home) search .....	6
3.4.1    Considerations about the machine reference zero (home) .....	7
3.5      Operation of the CNC as a DRO .....	7

---

**Chapter 4                    PROGRAMEDITING**

4.1	Editing mode .....	1
4.1.1	Displaying block contents .....	3
4.1.2	Program editing .....	4
4.1.3	Deleting Block contents .....	6
4.1.4	Deleting the whole program memory .....	6
4.1.5	Modifying block contents .....	7
4.1.6	Inserting a new program block .....	9
4.1.7	Eliminating empty blocks (memory compression) .....	9
4.2	Teach-in editing .....	10
4.3	Play-back editing .....	11
4.3.1	Reading points in Play-Back Mode .....	12

---

**Chapter 5                    PROGRAMEXECUTION**

5.1	Program execution .....	1
5.2	Program interruption .....	2
5.3	Display modes .....	3
5.4	Display of the arithmetic parameters .....	6
5.5	Change of operating mode .....	6

---

**Chapter 6                    PROGRAMMING**


6.1	Programming format .....	1
6.1.1	Preparatory G functions .....	2
6.2	Basic concepts .....	3
6.2.1	Block number and conditional block (block skip) .....	3
6.2.2	Metric or inch programming (G70/G71) .....	3
6.2.3	Absolute / incremental programming (G90/G91) .....	3
6.3	Reference systems .....	5
6.3.1	Reference points .....	5
6.3.2	Machine Reference (home) search (G74) .....	6
6.3.3	Coordinate preset and zero offsets .....	7
6.3.3.1	Coordinate preset (G92) .....	7
6.3.3.2	Zero offset loading (G51....G60) .....	8
6.3.3.3	Zero offset selection (T1 ... T10) .....	8
6.4	Complementary functions .....	9
6.4.1	Axis feedrate "F" .....	9
6.4.2	Spindle speed "S" .....	9
6.4.3	Tool number "T" .....	10
6.4.4	Zero offset "T" .....	10
6.4.5	Miscellaneous /auxiliary "M" function .....	10
6.5	Path control .....	12
6.5.1	Rapid positioning (G00) .....	12
6.5.2	Linear interpolation (G01) .....	13
6.5.3	Circular interpolation (G02, G03) .....	14
6.5.4	Synchronization (G33) .....	16
6.5.5	Axis X as an infinite slave of another axis (G34) .....	18
6.5.5.1	Automatic pass compensation .....	20

---

6.6	Additional preparatory functions .....	21
6.6.1	Dwell (G04) .....	21
6.6.2	Increment part counter (G45) .....	21
6.6.3	Round corner (G05) and square corner (G07) .....	22
6.7	Special functions .....	23
6.7.1	Feedback inhibit (G47, G56) .....	23
6.7.2	G47, G48 as opening of the axis loop .....	24
6.7.3	The feedrate "F" is not affected by "P18" (G61, G62) .....	25
6.7.4	Acceleration ramp modification (G93) .....	25
6.8	Other functions .....	26
6.8.1	Batch programming (G81) .....	26
6.8.2	Probing (G75) .....	28
6.8.3	Rigid tapping (G84, G80) .....	29
6.8.4	Loading the punch sizes (G60) .....	32
6.9	Parametric programming .....	33
6.9.1	Assignments .....	33
6.9.2	Operations .....	34
6.9.3	Access to the arithmetic parameter table .....	37
6.9.4	Unconditional jump function (G25) .....	39
6.9.5	Conditional jump functions (G26, G27, G28, G29) .....	41

---

## ERROR CODES



**COMPARISON TABLE  
FOR FAGOR CNC MODELS:  
101/101S/102/102S**

## GENERAL CHARACTERISTICS

		101	101S	102	102S
Feedback inputs	Connector A1 (X axis)	x	x	x	x
	Connector A2 (Y axis)		x	x	x
	x5 multiplier circuit for sine-wave signals		x	x	x
	Feedback correction factor	x	x	x	x
Analog outputs	X axis	x	x	x	x
	Y axis			x	x
	Spindle (S)	x	x	x	x
Axes	X axis	x	x	x	x
	X + Y axis			x	x
	X axis + electronic handwheel		x	x	x
	Axes X + Y + auxiliary handwheel		x	x	x
	Double feedback for X axis		x		
Axis control	Closed Loop	x	x	x	x
	Open Loop	x		x	
	Rigid Tapping				x
Interface with external devices	External operator panel "JOG 100"		x		x
	RS232C Interface		x	x	x
	Fagor Local Area Network (LAN)		x	x	x
	DNC 100		x		x
Operating options	Overtemperature alarm		x	x	x
	Operation in radius or diameter		x	x	x
	Operation with a probe		x	x	x
	Zero offsets		x	x	x
	Tool length compensation		x	x	x
	Acceleration / deceleration	x	x	x	x

## INPUTS AND OUTPUTS

		101	101S	102	102S
<b>INPUTS</b>	X axis home switch	x	x	x	x
	Y axis home switch		x	x	x
	External emergency stop	x	x	x	x
	Feedhold	x	x	x	x
	External Cycle Start	x	x	x	x
	External Cycle Stop	x	x	x	x
	Conditional input (block skip)	x	x	x	x
	Manual input (DRO mode)	x	x	x	x
	External Reset (initial CNC conditions)	x	x	x	x
	2 inputs as Handwheel multiplying factor		JOG100	x	JOG100
	2 inputs for Feedrate override		JOG100	x	JOG100
	5 inputs for parametric programming		x	x	x
	2 inputs for handling the auxiliary handwheel		x	x	x
<b>OUTPUTS</b>	8 outputs for M, S or T in BCD or decoded	x	x	x	x
	M Strobe	x	x	x	x
	S Strobe		x	x	x
	T Strobe		x	x	x
	JOG mode selected at the CNC	x	x	x	x
	Automatic mode selected at the CNC		x	x	x
	Internal CNC emergency	x	x	x	x
	X axis brake	x	x	x	x
	Y axis brake			x	x
	X axis in position	x	x	x	x
	Y axis in position			x	x
	X axis Fast (Non-servocontrolled open loop)	x	JOG 100	x	JOG 100
	X axis Slow (Non-servocontrolled open loop)	x	JOG 100	x	JOG 100
	X direction (Non-servocontrolled open loop)	x	JOG 100	x	JOG 100
	Y axis Fast (Non-servocontrolled open loop)		JOG 100	x	JOG 100
	Y axis Slow (Non-servocontrolled open loop)			x	
	Y direction (Non-servocontrolled open loop)			x	

# PROGRAMMING

	101	101S	102	102S
Number of blocks	900	900	900	900
Conditional blocks (block skip)	x	x	x	x
Parts counter	x	x	x	x
Arithmetic programming		x	x	x
Arithmetic parameters		100	100	100

PROGRAMMI FUNCTIONS						
	G00	G01	G02	G03	G04	G05
	Rapid positioning	x	x	x	x	x
	Linear interpolation	x	x	x	x	x
	Clockwise circular interpolation			x	x	
	Counter-clockwise circular interpolation			x	x	
	Dwell	x	x	x	x	x
	Round corner	x	x	x	x	x
	Square corner	x	x	x	x	x
	Unconditional jump	x	x	x	x	x
	Jump if zero		x	x	x	x
	Jump if not zero		x	x	x	x
	Jump if less than zero		x	x	x	x
	Jump if equal or greater than zero		x	x	x	x
	Synchronization		x			
	Increment part-counter's count	x	x	x	x	x
	Pulse inhibit		x	x	x	x
	Cancel function G47		x	x	x	x
	Load zero offset		x	x	x	x
	F not affected by "P18"	x	x	x	x	x
	Cancel function G61	x	x	x	x	x
	Inch programming	x	x	x	x	x
	Metric programming	x	x	x	x	x
	Machine Reference (home) search	x	x	x	x	x
	Probing		x	x	x	x
	Batch programming		x			x
	Rigid tapping					x
	Absolute coordinate programming	x	x	x	x	x
	Incremental coordinate programming	x	x	x	x	x
	Coordinate presetting	x	x	x	x	x
	Modification of acceleration ramp		x	x	x	x

# NEW FEATURES AND MODIFICATIONS

<i>Date:</i> <b>March 1997</b>	<i>Software Version:</i> <b>2.1 and newer</b>	
<b>FEATURE</b>	<b>AFFECTED MANUAL AND SECTION</b>	
Synchronization of movements (G33)	Operating Manual	Section 6.5.4
Axis X as infinite slave of another axis (G34)	Installation Manual Operating Manual	Section 5.5 Section 6.5.5
G47, G48 as axis loop opener	Installation Manual Operating Manual	Section 5.7 Section 6.7.2
G75 special function	Installation Manual	Section 5.6
Travel limit control taking into account the the punch radius	Installation Manual Operating Manual	Section 5.8 Section 6.8.4
Selection of the Arithmetical Parameters which are required for display.	Installation Manual Operating Manual	Section 3.7 Section 6.9.3
Play-Back, as reading points.	Installation Manual Operating Manual	Section 3.6 Section 4.3.1
Parametrical programming takes the S sign into account	Installation Manual	Sect. 5.1 and 6.7
The axes can be denominated Y, C	Installation Manual	Section 3.4
Auxiliary Handwheel handling by means of 2 digital inputs	Installation Manual Operating Manual	Sect. 1.7, 3.4 and Section 3.1
Braking Control in open loop	Installation Manual	Section 4.4.2
Reading / Writing of machine parameters from the DNC100		
Error elimination by external Reset.		

# **INTRODUCTION**

# 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 45° C (113° 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 (41° F and 113° F)

The storage temperature must be between -25° C and 70° C. (-13° F and 158° 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.

## **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-etherane foam on all sides.
- 5.- Seal the cardboard box with packing tape or industrial staples.

# **FAGOR DOCUMENTATION** **FOR THE 101/101S / 102/102S CNC**

## **101/101S / 102/102S 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.

## **101/101S / 102/102S CNC USER Manual**

Is directed to the end user or CNC operator.

It contains the Operating manual.  
Sometimes, it may contain an additional manual describing New Software Features recently implemented.

# MANUAL CONTENTS

The installation manual consists of the following sections:

Index

Comparative Table for Fagor 101/101S / 102/102S CNC models

Introduction      Safety Conditions  
Shipping conditions  
Fagor documents for the 101/101S / 102/102S CNC  
Manual Contents

Chapter 1    Peripherals

Chapter 2    Aux mode

Chapter 3    Jog mode

Chapter 4    Program editing

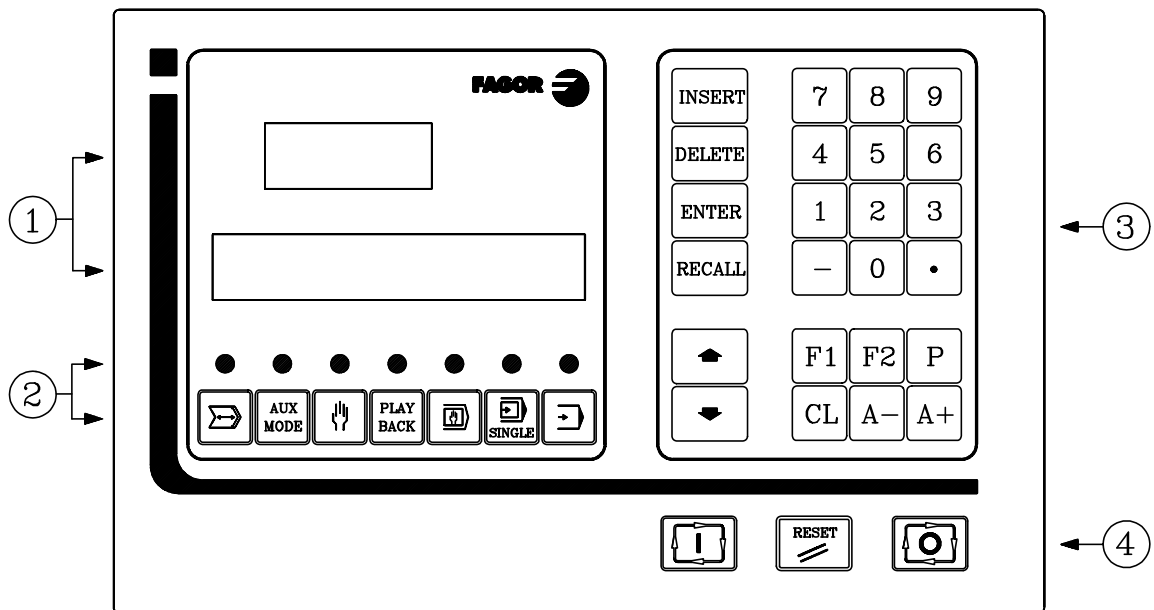
Chapter 5    Program execution

Chapter 6    Programming

Error Codes








# 1. PERIPHERALS

## 1.1 FRONT PANEL DESCRIPTION






1.- These 2 windows show the internal CNC information.

2.- Keys used to select any of the operating modes. When the lamp is lit, it indicates that the corresponding operating mode is currently selected.

-  Peripheral mode.
-  Aux Mode
-  JOG mode
-  Play-Back mode
-  Editing mode
-  Single-block mode
-  Automatic mode

3.- Keys to manipulate internal data, set up tables, parameters, edit programs, preset values, etc.

4.- The  key executes the program. The  key interrupts the execution of the program and the  key resets the CNC to initial conditions assuming the default values assigned to the machine parameters.

## 1.2 PERIPHERAL MODE


In order to access this operating mode, machine parameter "P100(1)" must be set to "0" indicating that the DNC feature is not available.

With this operating mode, it is possible to transfer part-programs, machine-parameters and the tool table out to a peripheral device (cassette reader FAGOR LS80, PC, terminal, etc.) through the RS232C serial line

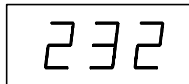
To do this, the machine parameters corresponding to the RS232C serial line must be set accordingly:

P70	Communications speed (baudrate) in baud.
P59(7)	Number of data bits.
P59(5)	Parity
P59(6)	Even or Odd parity
P59(8)	Number of stop bits

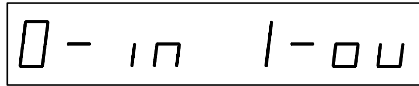
Bear in mind that the CNC must be off when connecting or disconnecting any peripheral device to it.

To access the peripheral mode, press the  key.

The CNC will show the following information:



The upper display indicates that the RS232C line is selected.



The lower display shows the two possible options:  
0 -> Input, 1 -> Output.

Select the "0-Input" option to receive information from a peripheral device and the "1-Output" to send information out to a peripheral device

### **1.2.1 "I-OUTPUT" OPTION. CNC --> PERIPHERAL**

This option must be selected once the peripheral device has been set ready to receive.

The lower CNC display will show the letter "N".

Depending on what has to be transmitted, proceed as follows:

- a) To send the machine parameter table and tool table or zero offset table.

Press the following keystroke sequence: **[8] [9] [9] [A-]**

The transmitted data is illegible and must be used as a back-up copy either to be sent back to the CNC later on or to another similar CNC.

- b) To send the whole content of the part-program memory, that is from N000 to block N899, out to a peripheral:

Press: **[A+]**

- c) To transmit the contents of the part-program memory starting at a particular block number:

Press the number of the first block to be transmitted and, then, press **[A+]**.

For example, The keystroke sequence: **[1] [0] [0] [A+]** will send blocks N100 and all the following ones out to the peripheral.

- d) To transmit a particular set of blocks of the part-program memory:

Indicate the first and last blocks to be transmitted separated by a period and press **[A+]**.

For example, the keystroke sequence: **[2] [0] [0] [.] [2] [2] [0] [A+]** will send all the blocks between N200 and N220 (both included).

The CNC format to send each block consists of 3 digits indicating the block number, the block contents and the Carriage Return (RT) and Line Feed (LF) characters indicating the end of the block.

Once all the blocks have been sent, the CNC sends the ESCAPE character indicating the end of transmission.

Example:   012 G01.91 X130 Y-56.3 F200 RT LF  
          013 X17.9 M6 RT LF  
          -  
          -  
          369 M30 RT LF ESC

The transmission can be aborted at any time by pressing **[CL]**.

Chapter: 1 <b>PERIPHERALS</b>	Section: <b>PERIPHERAL MODE</b>	Page <b>3</b>
----------------------------------	------------------------------------	------------------

## 1.2.2 "0-INPUT" OPTION. PERIPHERAL -->CNC

In order to send part-programs, machine parameter table, tool table or the zero offset table, the CNC memory must be unlocked.

When selecting this option, the lower display of the CNC will show the letter "N".

Depending on what has to be transmitted, proceed as follows:

- a) To receive at the CNC the machine parameter table, tool table or the zero offset table:

Press the keystroke sequence: **[8][9][9][A-]** at the CNC and give the order to transmit at the peripheral device.

- b) To read a program which already has its block numbered:

Press **[A+]**.

The program is loaded into the CNC memory with those block numbers.

Those blocks not involved in the transmission will keep their original data.

- c) To read a program not having its blocks numbered:

Indicate the starting block number from which it must be loaded into the CNC memory and press **[A+]**.

For example: the keystroke sequence **[1][5][0][A+]** indicates that those blocks must be loaded starting at block number N150. The rest of the blocks will be correlative (there will be no empty blocks).

Those blocks not involved in the transmission (the ones before the first block and after the last one) will keep their original data.

The format to be used at the peripheral to send each program block to the CNC must consist of:

3 digits indicating the block number (optional)

The block contents


The Return (RT) and Line feed (LF) characters indicating the end of block.

Once all the blocks have been sent to the CNC, the ESCAPE character must be sent as end of transmission.

The transmission may be aborted at any time by pressing **[CL]**.

Page <b>4</b>	<b>Chapter: 1</b> <b>PERIPHERALS</b>	Section: <b>PERIPHERAL MODE</b>
------------------	---	------------------------------------

## 2. AUX MODE

To access this operating mode, press the  key and the CNC will display the following information.

2-t  
1-P 0-t ESE

To get to the various menus, press:

- [0] To test the CNC's inputs and outputs
- [1] To access the machine parameter mode of the CNC.
- [2] To access the internal CNC table. Depending on the setting of machine parameter "P60(5)" this table may be used either as tool table, "P60(5)=0", or as zero offset table, "P60(5)=1".

With this operating mode it is also possible to:

**Lock machine parameters.** Press the following keystroke sequence:

[1] [CL] [CL] [9] [9] [9] [A+] [1] [ENTER]

If the machine parameters are locked, the CNC will display their values but it will not allow to change them.

**Unlock machine parameters.** Press the following keystroke sequence:

[1] [CL] [CL] [9] [9] [9] [A+] [0] [ENTER]

**Lock part-program memory** Press the following keystroke sequence:

[1] [CL] [CL] [8] [8] [8] [A+] [1] [ENTER]

If the part-program memory is locked, the CNC will display the contents of each block but it will not possible to modify it.

**Unlock part-program memory.** Press the following keystroke sequence:

[1] [CL] [CL] [8] [8] [8] [A+] [0] [ENTER]

**Clear the whole part-program memory.** Press the following keystroke sequence:

[1] [CL] [CL] [DELETE] [1] [DELETE]

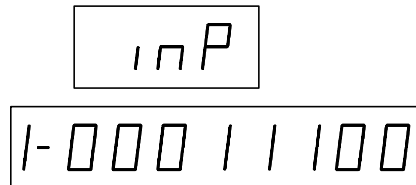
In order to erase the whole part-program memory, it must be unlocked.

## 2.1 SYSTEM INPUT/OUTPUT TEST

With this option, it is possible to analyze the status of the logic inputs and outputs of the CNC as well as activating and deactivating each logic output.

To do this, press the following keystroke sequence: **[AUX MODE] [0]**

The CNC displays the status of the first set of inputs (INP)



The lower display shows the status of 8 logic inputs, one digit per input.

Bottom Display	Digit 8	Digit 7	Digit 6	Digit 5	Digit 4	Digit 3	Digit 2	Digit 1
	1	0	0	0	1	1	1	0

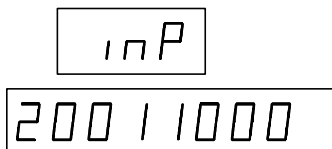
The CNC, shows at all times and dynamically, the status of the inputs. To check a particular one, actuate the corresponding external push button or switch and observe the status change on the corresponding digit on the CNC display.

A "1" on the display digit indicates that its corresponding input is receiving 24 Vdc. and a "0" means that it is not receiving 24 Vdc.

The inputs appearing at the lower window are:

Digit	Corresponding Input	Pin
8	Manual Input (DRO mode)	19 (I/O1)
7	Conditional Stop (M01) / Block Skip	18 (I/O1)
6	Cycle Start	17 (I/O1)
5	/ Cycle Stop	16 (I/O1)
4	/ Feed Hold	15 (I/O1)
3	/ Emergency Stop	14 (I/O1)
2	X axis Home switch	13 (I/O1)
1	Reset	12 (I/O1)

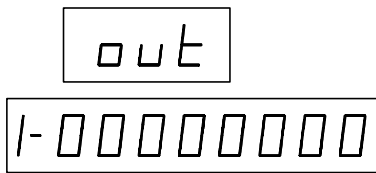
To display the next set of inputs, press **[A+]**



Digit	Corresponding Input	Pin
8	External feedrate override 2	11 (I/O1)
7	External feedrate override 1	10 (I/O1)
6	X axis feedback error	
5	Y axis feedback error	
4	X axis sine-wave feedback alarm	
3	Y axis sine-wave feedback alarm	
2	Over-temperature	
1	Not being used at this time	

The data shown at digits 6, 5, 4, 3, 2 and 1 is internal CNC information.

To display the first set of outputs, press [A+]



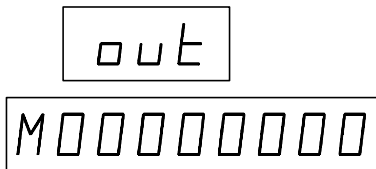
<i>Digit</i>	<i>Corresponding Output</i>	<i>Pin</i>
8	X axis In Position	9 (I/O1)
7	X axis Direction (Open Loop)	8 (I/O1)
6	X axis Slow (Open Loop)	7 (I/O1)
5	X axis Fast (Open Loop)	6 (I/O1)
4	/ Emergency	5 (I/O1)
3	M Strobe	4 (I/O1)
2	X axis Brake	3 (I/O1)
1	Jog mode selected	2 (I/O1)

To check a particular output, select it by using the up and down arrow keys.

Once the desired output has been selected, it could be turned on (1) or off (0) by pressing [1] or [0].

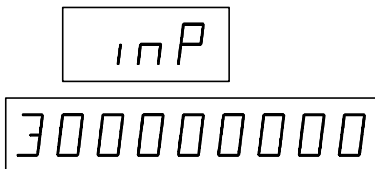
Several outputs may be active at the same time and they will all output 24 Vdc. at their corresponding connector pin.

To display the next set of outputs, press [A+]



<i>Digit</i>	<i>Corresponding Output</i>	<i>Pin</i>
8	MST80	20 (I/O1)
7	MST40	21 (I/O1)
6	MST20	22 (I/O1)
5	MST10	23 (I/O1)
4	MST8	24 (I/O1)
3	MST4	25 (I/O1)
2	MST2	26 (I/O1)
1	MST1	27 (I/O1)

If [A+] is pressed again, the CNC will display the third set of inputs.



<i>Digit</i>	<i>Corresponding Input</i>	<i>Pin</i>
8	E5	17 (I/O2)
7	E4	25 (I/O2)
6	E3	22 (I/O 2)
5	E2	23 (I/O 2)
4	E1	21 (I/O 2)
3	Mult. factor, handwheel or JOG type posit.	24 (I/O 2)
2	Mult. factor, handwheel or JOG type posit.	15 (I/O 2)
1	Y axis home switch	18 (I/O 2)

Press [A+] to show the next set outputs.

out  
2000000000

Digit	Corresponding Output	Pin
8	Y axis In Position	9 (I/O 2)
7	Y axis direction (Open Loop)	8 (I/O 2)
6	Y axis Slow (Open Loop)	5 (I/O 2)
5	Y axis Fast (Open loop)	3 (I/O 2)
4	T Strobe	4 (I/O 2)
3	S Strobe	6 (I/O 2)
2	Y axis Brake	7 (I/O 2)
1	Automatic	10 (I/O 2)

By pressing [A+] again, the CNC shows the checksum corresponding to the software version installed on this unit (in this example: 08AF) and the CNC model (in this example: 102S).

CSU  
08AF.102S

If [A+] is pressed again and the CNC belongs to the Fagor LAN, the display will show the element connected to each node. Use [A+] to scroll them.

The appearing information looks like this:

LAn  
Nod.0 PLC

The top window displays the message "Lan" referring to the Local Area Network.

The bottom window shows the node number (in this case "Nod.0" and the element occupying it (in this case: "PLC").

The CNC models: 82, 101S, 102 and 102S are considered as a single family by the LAN. That is why they appear as "Nod.? CN82".

By pressing [A+] again and if the CNC belongs to the Fagor LAN, the display will show the number of lost tokens. The appearing information looks like this:

LAn  
T0. 00000

By pressing [A+] again, the CNC lights all the LEDs indicating the end of the system I/O test.

1000  
☒ 0.0.0.0.0.0.0.0.

By pressing [A+] again, the CNC will start the system I/O test all over.

## 2.2 MACHINE PARAMETERS

With this option, it is possible to analyze the machine parameters of the CNC and modify their settings if the machine parameter memory is not locked.

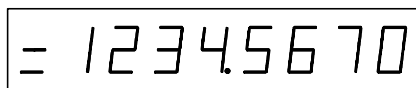
These machine parameters are set by the manufacturer and are used to adapt the CNC to the machine.

To access the machine parameters, press the keystroke sequence: **[AUX MODE] [1]**

The display information looks like this:





The top display shows the number of the machine parameter.



The bottom display shows the current value of that parameter.

To have another parameter display, use one of the following methods:

- \* Use the   keys
- \* Press **[CL]** twice to clear the current display. Then, key in the desired parameter number and press **[RECALL]**. The CNC will show that parameter.

Bear in mind that once the machine parameters have been set, either the **[RESET]** key must be pressed or the CNC powered-down and back up in order for the new values to be assumed by the CNC.

The installation manual describes all these machine parameters.

### 2.2.1 EDITING MACHINE PARAMETERS

The machine parameters must be edited one by one. Depending on the type of machine parameter selected, it will take one of the following types of values:

A number	P12 = 30000
A group of 8 bits	P19 = 00001111



To edit a parameter which is assigned a number, proceed as follows once the desired parameters has been selected:

- \* Press **[CL]**. The CNC clears the bottom display.
- \* Key in the desired value.
- \* Press **[ENTER]** to "enter" this value in the machine parameter table.

If the CNC does not assume this new value and it displays the previous one instead, it means that the machine parameter table is locked.

To edit a parameter defined with a group of 8 bits, once it has been selected, press **[CL]** and enter the 8 bits or press **[RECALL]** and change the bits one by one.

To change the bits one by one, proceed as follows:

- \* Press **[RECALL]**. The display will blink the first parameter bit.
- \* To change this bit , press **[0]** or **[1]** accordingly.
- \* Use the   keys to move from bit to bit (making them blink).
- \* To change a particular bit, select it by making it blink and set it to the desired value.
- \* Once all the bits have been set, press **[ENTER]** for this value to be entered in the machine parameter memory.

If the CNC does not assume this new value and it displays the previous one instead, it means that the machine parameter table is locked.

To enter all 8 bits at once, proceed as follows:

- \* If the parameter was selected by using the **[RECALL]** key, the CNC will blink the first parameter bit (editing mode).

To quit this mode, press **[CL]** and the bit will stop blinking.

- \* Press **[CL]** again and the bottom display will clear out.
- \* Key in the desired value (series of 1s and 0s).
- \* Press **[ENTER]** so this new value is "entered" in the machine parameter table.

If the CNC does not assume this new value and it displays the previous one instead, it means that the machine parameter table is locked.

### 2.3 TOOL TABLE OR ZERO OFFSET TABLE

The CNC has an internal table which, depending on the setting of machine parameter "P60(5)", may be used as tool table, "P60(5)=0", or zero offset table, "P60(5)=1".

Each one of 10 table addresses (1 through 10) has 2 fields: one for the X axis and the other one for the Y axis.

When setting "P60(5)=0", tool table, the address number matches the tool number. Both fields indicate tool length along X and Y (respectively). The CNC will take the table values and will apply tool length along both axes.

When setting "P60(5)=1", zero offset table, each table address can be allocated a new Part Zero. This part zero will be referred to Machine Reference Zero (home) at the distance indicated in the X and Y fields.

To access this table, press: **[AUX MODE] [2]**

The displayed information looks like this:

A rectangular digital display showing the number "001" in a seven-segment font.



The upper display shows the first table address.

A rectangular digital display showing "X0.2350" in a seven-segment font.

The lower display shows the X field value for this address.

When pressing  , the CNC shows the value for the Y field.


Use one of the following methods to display another table address:

- \* Use the   keys.
- \* Press the **[CL]** key twice to clear the current display. Then, key in the desired table address and press **[RECALL]**. The CNC will display this new address.

To edit the values of the X or Y fields, proceed as follows:

- \* Press **[CL]** and the CNC will clear the lower display.
- \* Key in the desired value.
- \* Press **[ENTER]** for the new value to be "entered" in the table.

# 3. JOG MODE

To access this mode, press the  key. The top display will appear blank and the bottom one will show the X axis position.

To change the axis being displayed, press [A+]. The lower display will now show the Y axis position.

With this operating mode it is possible to:

- Move the axes of the machine.
- Set Zero or preset a coordinate value (position).
- Automatically search home (machine referencing)
- Select the CNC to operate as a DRO.

## 3.1 JOGGING THE AXES



The CNC may be configured in the following ways:

- CNC with 1 or 2 axes.
- CNC with 1 axis and 1 electronic handwheel.
- CNC with 1 or 2 axes and the external operator panel "JOG 100".
- CNC with 1 axis, 1 electronic handwheel and the external operator panel "JOG 100".
- CNC with 2 axes and auxiliary handwheel
- CNC with 2 axes, auxiliary handwheel and the external operator panel "JOG 100"

When moving the axes, the CNC limits their travels according to the setting of machine parameters P0, P1, P40 and P41.

### CNC with 1 or 2 axes

The axes are jogged one at a time. To do that, proceed as follows:

- \* Select at the lower display, by using the [A+] key, the axis to be jogged (X or Y).
- \* Use the   keys to jog the axis.

The movement will be continuous. In other words, the axis will keep moving while its jog key is kept pressed.

If pins 14 and 25 of connector I/O2 are used the movements could be continuous or incremental. In incremental movements the axis will move the predetermined distance.

Chapter: 3 <b>JOG MODE</b>	Section: <b>JOGGING THE AXES</b>	Page <b>1</b>
-------------------------------	-------------------------------------	------------------

When using the "JOG/handwheel 1 and 2" inputs of connector I/O2, the movements will be either continuous or incremental and the axis will move the set distance every time its jog key is pressed.

The table below shows the available options:

<i>Pin</i>		JOG type	Moving distance
24	15		
0	0	Continuous	
0	1	Incremental	0.001 mm or 0.0001"
1	0	Incremental	0.010 mm or 0.0010"
1	1	Incremental	0.100 mm or 0.0100"

Activated pin = 1  
Deactivated pin = 0

Furthermore, pins 10 and 11 of connector I/O1 allow the axis feedrate to be altered as shown below.

<i>Pin</i>		<i>Servocontrolled axes</i>		<i>Non-servocontrolled axes</i>
10	11	(G01/G02/G03)	(G00)	Range
1	1	25%	25%	Slow range
1	0	50%	50%	Ignored
0	0	100%	100%	Ignored
0	1	If "P101(8)=0" 200% If "P101(8)=1" 0%	100%	Slow range

Activated pin = 1, deactivated pin = 0

### ***CNC with electronic handwheel***

The positioning of axis X can either be carried out with the electronic handwheel or from the keyboard, by using keys  

Pins 15 and 24 of connector I/O2 display the multiplying factor applied by the CNC to the pulses of the electronic handwheel.

This way, the actual axis jogging units result from multiplying the number of handwheel pulses by this factor.

<i>Pin</i>		<i>Multiplying Factor</i>	<i>Example: 250 lines/turn</i>
24	15		Distance per turn
0	0	x 1	0.250 mm or 0.0250"
0	1	x 10	2.500 mm or 0.2500"
1	0	x 50	12.50 mm or 1.2500"
1	1	x 100	25.00 mm or 2.500"

Activated pin = 1  
Deactivated pin = 0

Furthermore, pins 10 and 11 of connector I/O1 allow the axis feedrate to be altered as shown below.

<i>Pin</i>		<i>Servocontrolled axes</i>		<i>Non-servocontrolled axes</i>
10	11	(G01/G02/G03)	(G00)	Range
1	1	25%	25%	Slow range
1	0	50%	50%	Ignored
0	0	100%	100%	Ignored
0	1	If "P101(8)=0" 200% If "P101(8)=1" 0%	100%	Slow range

Activated pin = 1, deactivated = 0

### **CNC with 1 or 2 axes and the external operating panel "JOG100"**

The axes are jogged one at a time and may be moved by using either the CNC keyboard or the "X+, X-, Y+, Y-" push-buttons associated to the external operator panel "JOG100".

When jogging from the CNC keyboard, proceed as follows:

\* Select, at the lower display, the axis to be jogged (X or Y) by using the [A+] key.

\* Use the   keys to jog the axis.

The jogging direction corresponding to each key is determined by machine parameter.

With the external operator panel "JOG 100", it is possible to select the type of movement as well as the axis feedrate:



\* While in the "% FEED" positions, the axis movement will be continuous. In other words, they will move while their jog key is kept pressed.

Each of the positions of this zone indicates the percentage of feedrate applied by the CNC in the movement shown.

\* While in the "JOG" positions, the movements will be incremental. In other words, the axis will move a set distance every time its jog key is pressed.

**CNC with 1 axis, 1 electronic handwheel and the external operator panel "JOG 100"**


The X axis positioning can be done:

- \* With the electronic handwheel
- \* By means of push-buttons "X+, X-" associated with the "JOG 100" external operator panel.
- \* With the keys   on the keyboard.

With the external operator panel "JOG 100", it is possible to select the type of movement as well as the axis feedrate and the multiplying factor applied by the CNC to the feedback pulses output by the electronic handwheel.

- \* While in the "% FEED" positions, the axis movement will be continuous. In other words, they will move while their jog key is kept pressed.

Each of the positions in this zone displays the percentage feedrate applied by the CNC in the movement indicated.


- \* While in the "JOG" positions, the movements will be incremental. In other words, the axis will move a set distance every time its jog key is pressed.
- \* While in the  position, the axis can only be moved with the electronic handwheel and each position indicates the multiplying factor applied by the CNC to the feedback pulses output by the electronic handwheel.

This way, the actual axis jogging units result from multiplying the number of handwheel pulses by this factor.

<i>Position Multiplying factor</i>	<i>Example: 250 pulses/turn</i>
	Distance per turn
x 1	0.250 mm or 0.0250"
x 10	2.500 mm or 0.2500"
x 100	25.00 mm or 2.500"

### **3.2 AUTOMATIC POSITIONING**

It is carried out one axis at a time by following these steps:

- \* Select, at the lower display, the axis to be positioned (X or Y) by means of the [A+] key.
- \* Press [CL] to clear the lower display.
- \* Key in the desired destination coordinate (position).
- \* Press 

The CNC will position the axis at the indicated position.

### **3.3 ZERO SETTING OR COORDINATE PRESET**

The coordinate preset is performed one axis at a time by following these steps:

- \* Select at the lower display, with the [A+] key, the axis to be preset (X or Y).
- \* Press [CL] to clear the display.
- \* Key in the desired position value (coordinate).
- \* Press [ENTER] for the CNC to assume this value.

Press [CL] to cancel this preset before pressing [ENTER] if so desired. In this case, the CNC will display the previous value again.

### 3.4 MACHINE REFERENCE (HOME) SEARCH

This home search is done one axis at a time by following one of these methods:

- a) Press **[F1] [0]**. The CNC references (homes) the X axis.
- b) Press **[F1] [1]**. The CNC homes the Y axis.
- c) \* Select, at the lower display and by means of the **[A+]** key, the axis to be homed (X or Y).
  - \* Press **[CL]** to clear the display
  - \* Press **[RECALL]**

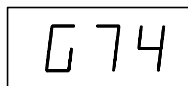
The axis will move the axis in the direction set by axis machine parameters "P62(4), P62(5)".

This homing movement will be carried out at the feedrate established by machine parameters "P25, P65", until the home switch is pressed, if any, machine parameters "P22(8), P62(8)".

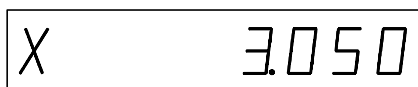
Once the home switch is pressed, the search will continue at 100mm/min. (3.9 inches/min.) until the reference mark (marker pulse I<sub>o</sub>) of the feedback device is detected, thus concluding the home search for that axis..

The CNC will cancel the currently active zero offset (part zero) and it will display the home coordinates allocated to machine parameters "P2, P42".


- \* If during the home search, the  key is pressed, the CNC interrupts the movement of the axis and it displays the following information:



The upper display shows function G74 indicating that a home search is in progress.



The lower display shows the axis position at the time.

To resume the home search, press 

To quit the home search, press **[CL]** twice.

### **3.4.1 CONSIDERATIONS ABOUT THE MACHINE REFERENCE ZERO (HOME)**

- \* If when initiating the home search, the home switch is pressed, the axis will move in the opposite direction to the one determined by machine parameters "P62(4), P62(5)", until the home switch is released and, then, it will start the home search.
- \* If the axis is positioned outside the travel limits specified by machine parameters "P0, P1, P40 and P41", the axis must be jogged by hand into the work area and, then, position it in the appropriate area for homing it.
- \* If the selected axis does not have a home switch installed, machine parameters "P22(8)" and "P62(8)", only the marker pulse searching move at 100 mm/min will be carried out until the marker pulse (Io) of the feedback device is detected, thus concluding the home search.

### **3.5 OPERATION OF THE CNC AS A DRO**

When the Manual (DRO) input at pin 19 of connector I/O1 is set high (24 Vdc), the CNC behaves like a DRO.

This means that:

- \* That the axes must be moved by means external to the CNC.
- \* The axis enable signals must be deactivated so they can be moved freely.
- \* If when operating in this mode, the axis travel limits set by machine parameters P0, P1, P40 and P41 are overrun, the CNC will issue the corresponding error message.

Chapter: 3 <b>JOG MODE</b>	Section: <b>HOME SEARCH AND DRO MODE</b>	Page <b>7</b>
-------------------------------	---	------------------

# 4. PROGRAM EDITING


This CNC offers three program editing modes as described in this chapter and they are:

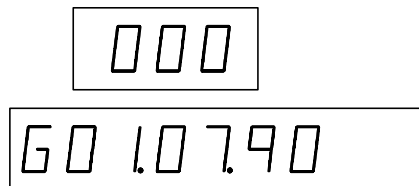
Regular editing. Press  to access this mode.

Teach-in editing. Press  to access this mode.

Play-back editing. Press  to access this mode.

## 4.1 EDITING MODE

Press  to access this operating mode. The CNC will display the following type of information:



The upper display shows the number of the block currently selected.

The lower display shows the functions programmed in this block.

The CNC memory may store up to 900 part-program blocks (000/899).

A program starts at a specific block and it ends at the one containing the M30 function indicating the end of the program.

Each block may have up to 5 ISO-coded functions. These functions may be:

- G Preparatory functions. Up to a maximum of 4 per block.
- X X axis coordinates
- Y Y axis coordinates
- I Arc center's X coordinate of a circular interpolation.
- J Arc center's Y coordinate of a circular interpolation.
- F Feedrate of the axes.
- S Spindle speed (r.p.m).
- T Tool number of zero offset.
- M Auxiliary functions. Up to a maximum of 4 per block.
- N Jump or Call functions (G25 through G29).
- P Parametric programming.

When using parametric programming, the block cannot have any more ISO-coded data. The parametric programming may use the following functions:

- X Assign the X axis coordinate value to a parameter.
- Y Assign the Y axis coordinate value to a parameter.
- F Parametric function.
- P Reference to another parameter.
- K Assign a constant value to a parameter.
- E Assign the status of inputs E1 to E5 (of connector I/O1) to a parameter.

The example below shows:

- \* A possible part-program lay-out in memory indicating the beginning and the end of each program.
- \* Various program blocks containing 2, 3, 4 and up to 5 functions. The programming method does not correspond to this chapter; but it is fully described in the "Programming" chapter later on in this manual.

N000	G1.7.90	F5000	S250	T01	M 3	Beginning of program
N001	X60	Y40				
N002	G3	X110	Y90	I0	J50	
N003	G3	X160	Y40	I50	J0	
N004	M30					End of program

N050	P17	X				Beginning of program
N051	P18	Y				
N052	P4	F7	P3			
N053	P5	F8	P3			
N054	P6	P0	F3	P4		
N055	P7	P1	F3	P5		
N056	G0.5	XP6	YP7			
N057	P3	P3	F1	P19		
N058	G1	XP6	YP7	F200		
N059	P3	F11	K360			
N060	G28	N57				
N061	G0	XP17	YP18			
N062	M30					

N100	G0	X10				Beginning of program
N101	Y20					
N102	G1	X50	M3			
N103	G0	Y0				
N104	X0					
N105	G25	N100.104.81				
N106	M30					End of program


In this mode, it is possible to:

- \* Display the contents of all the memory blocks.
- \* Create a program (edit all program blocks).
- \* Delete a particular program block.
- \* Modify the contents of specific block.
- \* Replace a particular program block.
- \* Insert a new program block, shifting the rest of the blocks back.

In the "Aux Mode" it is also possible to:


- \* Lock the part-program memory.
- \* Unlock the part-program memory.
- \* Erase the whole part-program memory.

### 4.1.1 DISPLAYING BLOCK CONTENTS



While in this operating mode, when pressing , the CNC shows the following type of information:



The top display shows the number of the block currently selected.






The bottom display shows one of the functions programmed in this block.

To see the contents of the blocks, use the   keys. The CNC will show all the functions of each block one by one and, when it is done with one block it will go on to display the next one.

The CNC will only show the blocks containing some type of information and it will skip the empty ones.

To see the contents of a particular block, proceed as follows:

- \* Press [CL] twice. The CNC will clear both displays.
- \* Enter the number of the block to be displayed.
- \* Press 

The CNC will display the first function of this block. Use the   keys to see the rest of the functions and the previous blocks as well as the next ones.

If the requested block is empty, the lower display will be blank (empty).

## 4.1.2 PROGRAM EDITING

A program is edited one block at a time starting from the first block. To do this proceed as follows:

- 1.- If the selected block (appearing at the top display) is not the desired one, do the following:
  - \* Press **[CL]** twice to clear both displays.
  - \* Enter the desired block number.
  - \* Press **[RECALL]**
- 2.- If the block is not empty (bottom display), do the following:
  - \* Press **[CL]** once to clear the bottom display (block contents).
- 3.- Edit the block contents by defining all its functions. Use the **[A+]** and **[A-]** keys to select these functions.

The way to program a block is fully described in the "Programming" chapter later on in this manual.

Once the whole block has been edited, press **[ENTER]**. The CNC will show the next block.

Example: N000 G1.7.90 F5000 T1

- \* Select the block number N000 at the upper display. Make sure the block is empty (blank), if not empty, press **[CL]**.
- \* Press **[A+]**. The block number at the upper display starts to blink (block editing mode).

The lower display shows the first function that may be selected. The "G" character corresponds to a G function.
- \* Key in "1.7.90" which correspond to G01, G07 and G90.
- \* Press **[A+]** and the CNC will display the next available function which, in this case, is "X".
- \* Press **[A+]** and **[A-]** until the lower display shows the "F" character which corresponds to the "F" function (feedrate).
- \* Key in "5000", corresponding to a feedrate value of F5000.
- \* Press **[A+]** and the CNC will show the next available function which, in this case, is the "S" function (spindle).
- \* Press **[A+]** and **[A-]** until the lower display shows the "T" character corresponding to the "T" function (tool).

Page <b>4</b>	Chapter: 4 <b>PROGRAM EDITING</b>	Section: <b>PROGRAM EDITING</b>
------------------	--------------------------------------	------------------------------------

- \* Key in [1], number of the tool to be selected T1.
- \* Press [ENTER], The CNC concludes the editing of the block and the upper display will show the next block number without blinking. In this case is block "001".

4.- Repeat the operations described in the previous procedure to edit the rest of the blocks.

### **Attention:**

While editing a block, if when pressing [A+] or [A-] the CNC does not show any more functions, it means that the block does not admit any more functions either because of the type of block it is or because all the permitted ones have already been defined (a maximum 5 functions).

The block numbers need not be sequential nor contain all the possible information.

The following example shows 2 different ways to define the same program.



N000	G1.7.90	F5000	S250	T01	M 3	N000	G1.7.90
N010	X60	Y40				N001	F5000
N020	G3	X110	Y90	I0	J50	N002	S250
N030	G3	X160	Y40	I50	J0	N003	T01
N040	M30					N004	M 3
						N005	X60 Y40
						N006	G3 X110 Y90 I0 J50
						N007	G3 X160 Y40 I50 J0
						N008	M30

The program must always end with a block containing the M30 function.

If this function is not programmed, the CNC will keep executing the rest of the blocks (when in AUTOMATIC) and if there are no other blocks edited, it issues error 15.

### **4.1.3 DELETING BLOCK CONTENTS**

Two cases may occur when attempting to delete the contents of a program block:

- 1.- The currently selected block number (upper display) is not desired one. Proceed as follows:
  - \* Press **[CL]** twice to clear both displays.
  - \* Enter the desired block number and...
  - \* Press **[DELETE]**.
- 2.- The currently selected block is the desired one. The upper display may be blinking (insert mode) or not. In either case...
  - \* Press **[DELETE]**

In both instances, the block will stay selected but empty (the lower display being blank). When scrolling the program blocks, this block will no longer be displayed.

### **4.1.4 DELETING THE WHOLE PROGRAM MEMORY**

When you wish to delete the whole program memory take the following steps:

- \* Press key **AUX MODE** to access the Aux-Mode operation mode.
- \* Press the following key sequence:

[1] [CL] [CL] [DELETE] [1] [DELETE]

To be able to delete the program memory this has to be unlocked.

## 4.1.5 MODIFYING BLOCK CONTENTS

If the currently selected block number is blinking (editing mode), press [CL] twice to quit this mode.



When modifying the contents of a program block, the following cases may come up:

### a) We would like to change the value of a function.

Example, modify a coordinate value:

Current block: N000 G1.7.90 **X100** F1000

New block: N000 G1.7.90 **X250** F1000



- \* Select block N000 and press [RECALL]. The upper display blinks this block number.
- \* Use the   keys to position at function "X100".
- \* Press [CL]. The CNC erases the value of the function; but it keeps its indicator character (X).
- \* Key in the desired value which, in this case, is "250".
- \* Press [ENTER] so the CNC assumes the new block.

### b) We would like to eliminate one of the functions defining the block.

Example, eliminate the X coordinate:

Current block: N000 G1.7 **X100** Y20 F1000

New block: N000 G1.7 Y20 F1000



- \* Select block N000 and press [RECALL]. The upper display blinks this block number.
- \* Use the   keys to position at "X100".
- \* Press [CL] twice to clear the lower display.
- \* Press [ENTER] for the CNC to assume the new block.

### c) We would like to change a function.

Example, replace the X coordinate with Y:

Current block: N000 G1 **X100** F1000 M33

New block: N000 G1 **Y120** F1000 M33

- \* Select block N000 and press [RECALL]. The upper display will blink this block number.
- \* Use the   keys to position at "X100".
- \* Press [CL] twice so the CNC clears the lower display.


Chapter: 4 <b>PROGRAM EDITING</b>	Section: <b>BLOCK MODIFICATION</b>	Page <b>7</b>
--------------------------------------	---------------------------------------	------------------

- \* Press [A+] and [A-] to select the new function which, in this case, is "Y".
- \* Key in the new value which, in this case, is "120".
- \* Press [ENTER] for the CNC to assume this new block.

**Attention:**

To change more than 2 functions on the same block, proceed as follows:



- Modify the first function.
- Press  to find the next function.
- Modify the other function.
- Press [ENTER] for the CNC to assume these changes.

The CNC does not assume the changes until [ENTER] is pressed.

If when pressing [ENTER], the block number keeps blinking (top display), it means that there is an error in the edited block and it is not accepted into memory.

#### 4.1.6 INSERTING A NEW PROGRAM BLOCK

To insert a new block into a previously edited program, proceed as when editing blocks; but instead of pressing [ENTER], press [INSERT].

The new edited block is inserted into memory shifting the ones behind it one position back.

Example: To insert block N051 containing function "G4.2".

Current program	New program
N050 G90.71	N050 G90.71
N051 G74	<b>N051 G4.2</b>
N052 X-15.363	N052 G74
N053 M9.6	N053 X-15.363
	N054 M9.6

- \* Select block N051 and press [CL] to clear the lower display.
- \* Edit the block contents. Use [A+] to select the "G" function and key in the value: "4.2".
- \* Press [INSERT]. The CNC assumes the new block and "pushes" the following ones back one position.

If the block insertion affects blocks containing jump or call functions (G25 to G29), the CNC will take them into consideration and will update them accordingly.

#### 4.1.7 ELIMINATING EMPTY BLOCKS (MEMORY COMPRESSION)

To eliminate an empty block, first select the block number and, then, press [INSERT]. The CNC will shift ("pull") all the following blocks one position forward.


Example: To eliminate N152 and N153 which are empty.

Current program	New program
N151 X12 Y13 F500 S1000 M3	N151 X12 Y13 F500 S1000 M3
N154 M40	<b>N152 M40</b>
N155 X20 Y0	<b>N153 X20 Y0</b>


- \* Select block N152 and press [INSERT]. The CNC "pulls" the following blocks up one position.
- \* Press [INSERT] again to shift them up one more position.

If the block elimination affects blocks containing jump or call functions (G25 to G29), the CNC will take them into consideration and will update them accordingly.

## 4.2 TEACH-IN EDITING

Press  to access this operating mode.

In this programming mode, it is possible to execute the program blocks as they are being edited before entering them into memory. It is also possible to execute functions or movements outside the program.

To do this, once the block has been edited, press .

The CNC will execute the block. When it is executed (blinking block number), the following keys may be used:

**[ENTER]** To enter this block into memory. The CNC will now be ready for the next block to be edited.

**[CL]**. The block will **not** be entered into memory being possible to modify it or edit it.




The CNC will execute the block again.

With Teach-in programming, it is possible to run a part one block at a time as it is being programmed (edited) since the CNC keeps the history of all the executed blocks.

For example, when executing block N100 G1.5.90 X100 F1000, the CNC assumes as history functions G1, G5, G90 and F1000. In other words, it is the same to execute afterwards N101 X120 or block N101 G1.5.90 X120 F1000.


### **Attention:**



Teach-in programming allows executing blocks as they are being edited. For this reason, care must be taken before pressing  and make sure that the block number appearing on the upper display is blinking. If it is not blinking, press **[RECALL]**.

It is not possible to execute, in this mode, blocks containing jump functions (G25 to G29). The CNC will issue error 1 when attempting to execute them.

### 4.3 *PLAY-BACK EDITING*

Press  to access this operating mode.

In this editing mode and while editing a block, it is possible to jog the axes to the desired point and then enter the coordinate values reached as program data.



The rest of the functions are edited as described for the Editing mode earlier in this chapter.

The following example shows how to program a block of the G1 X\_\_\_ Y\_\_\_ F100 type.



The screen displays field "G"

- Press key [1] and then key [A+] to go on to the next field.

The screen shows field "X"

- Press key [Recall] and the CNC will display the present position of axis X.
- Move the axis to the required position by means of the electronic handwheel, keys   or push-buttons "X+, X-" associated with the "JOG100" external operator panel.
- Press key [Enter] for the CNC to assume this value.
- Press key [A+] to go on to the next field.

The screen displays field "Y"

- Press key [Recall] and the CNC will display the present position of axis X.
- Move the axis to the required position by means of the electronic handwheel, keys   or push-buttons "Y+, Y-" associated with the external "JOG100" operator panel.
- Press key [Enter] for the CNC to assume this value.
- Press key [A+] to go on to the next field.

The screen displays field "F"

- Move the axis to the required point
- Press the keys [1] [0] [0]
- Press key [Enter] for the CNC to assume this value and take the block as having finished


Chapter: 4 <b>PROGRAM EDITING</b>	Section: <b>PLAY-BACK EDITING</b>	Page <b>11</b>
--------------------------------------	--------------------------------------	-------------------

### 4.3.1 READING POINTS IN PLAY-BACK MODE

The CNC allows points to be read in Play-Back mode more easily than the way shown above

```
N100 X__ Y__  
N101 X__ Y__  
N102 X__ Y__  
N103 X__ Y__
```

To do this define parameter "P100(5)=1" and take the following steps

After pressing  and selecting the block number:

The screen displays the field "X"

- Move the axis to the point required.
- Press key [A+] or the external "Y+", "Y- " push-buttons for the CNC to assume this value and go on to the following field .

The screen displays field "Y"

- Move the axis to the required point
- Press key [Enter] or external push-button "Start" for the CNC to assume this value and transfer the block to its memory.

The CNC increments the block number and the screen displays field "X".


- Repeat the aforementioned operations.


#### **Attention:**




In models with 2 axes available, the blocks are always stored in memory with the coordinates of both axes (X\_\_ Y\_\_ ).


# 5. PROGRAM EXECUTION

This CNC offers 2 program execution modes. The Automatic mode to run the whole program all the way through the last block and the Single Block mode to run it block by block requiring the pressing of the  key to execute each block.

Press  to select the "Single-Block" mode.

Press  to select the "Automatic" mode.


Depending on the setting of machine parameter "P23(3)", the execution will be either Automatic, "P23(3)=0", or Semi-automatic, "P23(3)=1".

When operating in Semi-automatic mode, whenever the CNC runs into a block which contains a movement, it interrupts the program execution and waits for the  key to be pressed or for the external Cycle Start input to be activated (pin 17 of connector I/O1) before resuming the execution of the program..

All three operating modes, Single Block, Fully Automatic and Semi-Automatic, are described next.

## 5.1 PROGRAM EXECUTION

To execute a program, follow these steps:


- 1.- Select the desired operating mode, Single Block or Automatic.
- 2.- To select the first program block do the following:
  - \* Press [CL] twice to clear both displays.
  - \* Enter the desired block number.
  - \* Press [RECALL].
- 3.- Press  or activate the external Cycle Start input, pin 17 of connector I/O1, to begin running the program.
- 4.- During the execution of the program, it is possible to vary the feedrate of the axes by using the external operator panel "JOG100" or, when not available, the "Feedrate override" inputs (pins 10 and 11 of connector I/O1).

While the program is running, it is also possible to change the execution mode from Automatic to Single-Block and vice versa.

Chapter: 5 <b>PROGRAM EXECUTION</b>	Section: <b>PROGRAM EXECUTION</b>	Page <b>1</b>
--	--------------------------------------	------------------

## 5.2 PROGRAM INTERRUPTION

The program is interrupted whenever:

 is pressed



The external Cycle Stop input is activated, pin 16 of connector I/O1.

The CNC executes the M00 function (program stop).

The conditional input (pin 18 of connector I/O1) is active when executing function M01 (conditional stop).

The external Feed-hold is active, pin 15 of connector I/O1. The CNC will continue executing the program when feed-hold is released (deactivated).

When the program execution is interrupted, the CNC stops the execution of the block and it allows to:

- \* Change the mode of execution. From Automatic to Single Block and vice-versa.
- \* Quit the execution mode and select another operating mode.
- \* Analyze the contents of previous and following blocks by using  

Regardless of the block being displayed, the CNC "remembers" the block where the program was interrupted and it resumes execution from that block on.

- \* Select another block to resume the execution of the program from that block on.

Press [CL] twice, enter the desired block number and press [RECALL].

- \* Etc.

To resume program execution, press  or activate the external Cycle Start input, pin 17 of connector I/O1.

The execution of the program ends when:

The CNC executes function M30. code indicating the end of the program with return to the first block of the program.

The external Emergency Stop signal is activated (pin 14 of connector I/O1). The CNC issues the corresponding error and program must be executed from the beginning.

### **Attention:**



It must be borne in mind that if machine parameter "P30" is set with a value smaller than 900; when activating the Reset input (pin 12 of connector I/O1) the CNC interrupts the execution of the program and starts executing from the block number indicated by machine parameter "P30".

Page 2	Chapter: 5 <b>PROGRAM EXECUTION</b>	Section: <b>PROGRAM INTERRUPTION</b>
-----------	--	---

### 5.3 DISPLAY MODES

Once the program is running, the lower display shows the X axis position. It is possible to select the type of information to appear at the lower display by using the [A+] and [A-] keys.

The possible options are:

X 12345.678	X axis position value (coordinate)
Y 12345.678	Y axis position value
M 3.41.	M functions currently active
Con = 60	Number of parts (counter count)
G 1.5.91.	G functions currently active
E 0.012	Amount of X axis lag (following error)
e 0.025	Amount of Y axis lag (following error)
F 01000	Axis feedrate F
N 010.025.12	Execution status of a call
S 0250 T01	S value and tool number or zero offset currently active

#### Display of the position of the axes (X 12345.678 Y 12345.678)

It indicates the position of the axes at all times. Depending on the setting of machine parameter "P23(1)" this value may correspond to the theoretical axis coordinate, "P23(1)=1", or to the real one, "P23(1)=0".

#### Display of the M functions currently active (M3.41)

It displays the auxiliary M functions active at the time, even the M00, M01 or M30 functions.

The M functions that may be displayed are:

M00, M01, M30	Related to stopping the program
M03, M04, M05	Related to starting and stopping the spindle
M41, M42, M43, M44	Related to changing spindle ranges (gears)

When pressing [RESET] or activating the external Reset input (pin 12 of connector I/O1), the CNC assumes the initial conditions and generates function M30.

#### Display of the parts-counter count (Con= 60)

The CNC increments this counter in one unit in the following instances:

- \* Every time function G45 is executed.
- \* Every time function M30 is executed while machine parameter "P21(4) = 1".

To reset the counter count to "0", select this display mode and, while the program is **not** running, press [DELETE].

The maximum count value is 9999. Beyond this value, the counts rolls over starting at 0, 1, 2 and so forth.

Chapter: 5 <b>PROGRAM EXECUTION</b>	Section: <b>DISPLAY MODES</b>	Page <b>3</b>
--	----------------------------------	------------------

While executing function G81 (batch programming), the lower display shows the letters "Con" followed by the number of parts programmed for that batch and the right hand side of the display shows the number of parts already executed.

**Display of the currently active G functions (G 1.5.91.)**

The G functions that may be displayed are:

G01, G02, G03	Function G00 is not displayed
G05	G07 is not displayed
G70	G71 is not displayed
G91	G90 is not displayed

Functions: G04, G25, G26, G27, G28, G29, G45, G74, G75, G92 are not displayed

**Display of the axis lag (following error) (E 0.012, e 0.025)**

This display mode is mainly used when setting the machine up to help adjust the axes of the machine.

They display the amount of following error (lag); that is, the difference between the theoretical position of the axes (where they should be) and their real position (where they actually are).

**Display of the feedrate of the axes (F 01000)**

It displays the axis **feedrate being applied** at the time. That is, the programmed feedrate F multiplied by the % override.

**Display of the execution status of a call (N 010.025.12)**

The CNC uses this display mode when executing a jump function: G25, G26, G27, G28 or G29.

When not executing this type of function, it displays the following information:

N---.----.--

When executing a jump function, the CNC executes the indicated program section a number of times. The information appearing at the lower display looks like this:

**N 050 . 120 . 16**

The first 3-digit figure (050, in this example) indicates the number of the first block being executed repeatedly.

The second 3-digit number (120, in this example) indicates the number of the last block being executed repeatedly.

The third figure (2 digits. 16, in this example) indicates the number of repetitions **left** for the program section being executed repeatedly.

**Display of the selected spindle speed "S" and table (S 0250 T01)**

It displays the spindle speed "S" currently selected as well as the "T" table address currently selected.

Depending on the setting of machine parameter "P60(5)", the internal CNC table may be used as tool table, "P60(5)=0", or as zero offset table, "P60(5)=1".

Chapter: 5 <b>PROGRAM EXECUTION</b>	Section: <b>DISPLAY MODES</b>	Page <b>5</b>
--	----------------------------------	------------------

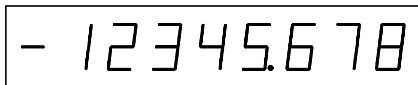
## 5.4 DISPLAY OF THE ARITHMETIC PARAMETERS

Once one of the execution modes is selected, Automatic or Single-Block, this mode may be access by pressing [P]. The information appearing on the displays looks like this:



P00

The top display shows the number of the parameter currently selected.



- 12345.678



The bottom display shows the arithmetic value of that parameter.

The CNC has 100 arithmetic parameters (00/99). Their values may have a sign, decimals and may be represented in exponential form.

0.98700 -1  
1298700 7

Is the same as  $0.98700 \cdot 10^{-1}$   
Is the same as  $1298700 \cdot 10^7$

If the exponent of the arithmetic parameter to be displayed is greater than 9, the first digit on the right will show the "-" sign.



Use the   keys to view the previous and following parameters.

To quit this display mode:

Press [CL]. The CNC returns to the previous display mode (either the standard one or a specific one).

Press [A+] or [A-]. The CNC will show one of the specific display modes.

## 5.5 CHANGE OF OPERATING MODE

With this CNC it is possible to change from Automatic mode to Single-Block mode or vice-versa while running a program. To do this, press the key corresponding to the desired operating mode:  or 

To quit the execution mode and switch to another operating mode, the program **must not** be running. Therefore, the program must be interrupted or ended.

# 6. PROGRAMMING

## 6.1 PROGRAMMING FORMAT

The part program must contain all the necessary geometric and technological data in order for the machine to carry out all the desired functions and movements.

To do this, the CNC offers up to 900 blocks of part-program memory (000/899).

A program begins at a particular block and ends at a block containing the M30 function (indicating the end of the program).

Each block may have up to 5 ISO-coded functions. These functions may be:

- G Preparatory functions. Up to a maximum of 4 per block.
- X X axis coordinates
- Y Y axis coordinates
- I Arc center's X coordinate of a circular interpolation.
- J Arc center's Y coordinate of a circular interpolation.
- F Feedrate of the axes.
- S Spindle speed (r.p.m).
- T Tool number of zero offset.
- M Auxiliary functions. Up to a maximum of 4 per block.
- N Jump or Call functions (G25 through G29).
- P Parametric programming.

When using parametric programming, the block cannot have any more ISO-coded data. The parametric programming may use the following functions:

- X Assign the X axis coordinate value to a parameter.
- Y Assign the Y axis coordinate value to a parameter.
- F Parametric function.
- P Reference to another parameter.
- K Assign a constant value to a parameter.
- E Assign the status of inputs E1 to E5 (of connector I/O1) to a parameter.

Chapter: 6 <b>PROGRAMMING</b>	Section: <b>PROGRAMMING FORMAT</b>	Page <b>1</b>
----------------------------------	---------------------------------------	------------------

## 6.1.1 PREPARATORY G FUNCTIONS

The preparatory functions are always programmed at the beginning of the block and they serve to determine the geometry and work conditions of the CNC.

They are defined by means of the G function and each block may contain up to 4 of them. The functions must be separated by a period G 1.7.90

If incompatible G functions are programmed in the same block, the CNC assumes the last one.

The possible G functions are:

Function	Description	M	D	101	101S	102	102S
G00	Rapid positioning	*	*	*	*	*	*
G01	Linear interpolation	*		*	*	*	*
G02	Clockwise circular interpolation	*				*	*
G03	Counter-clockwise circular interpolation	*				*	*
G04	Dwell			*	*	*	*
G05	Round corner	*		*	*	*	*
G07	Square corner	*	*	*	*	*	*
G25	Unconditional jump			*	*	*	*
G26	Jump if zero				*	*	*
G27	Jump if not zero				*	*	*
G28	Jump if less than zero				*	*	*
G29	Jump if equal or greater than zero				*	*	*
G33	Synchronization	*			*		
G34	Axis X as infinite slave of another axis.	*			*		
G45	Increment parts counter			*	*	*	*
G47	Feedback pulse inhibit Open the position loop of the axes.	*			*	*	*
G48	Cancel function G47 Close the position loop of the axes.	*	*		*	*	*
G51 to G60	Load zero offsets				*	*	*
G60	Load the dimensions of the punch.	*		*	*	*	*
G61	F not affected by "P18"	*		*	*	*	*
G62	Cancel function G61	*	*	*	*	*	*
G70	Inch programming	*	*	*	*	*	*
G71	Metric programming	*	*	*	*	*	*
G74	Home search			*	*	*	*
G75	Probing				*	*	*
G81	Batch programming				*		*
G84, G80	Rigid tapping						*
G90	Absolute programming	*	*	*	*	*	*
G91	Incremental programming	*	*	*	*	*	*
G92	Coordinate preset			*	*	*	*
G93	Modify acceleration ramp	*			*	*	*

### Attention:



The letter "M" means modal. This means that once the G function is executed, it remains active until another incompatible G function or M30 is executed or on an Emergency or a Reset.

The letter "D" means "By Default". This means that the it is assumed by the CNC on power-up, after executing an M30 or after an Emergency or a Reset.

## 6.2 BASIC CONCEPTS

### 6.2.1 BLOCK NUMBER AND CONDITIONAL BLOCK (BLOCK SKIP)

The block number serves to identify each block of the program.

It consists of 3 digits and it must be between 000 and 899.

A conditional block is one that is executed only when the conditional input (pin 18 of connector I/O1) is high (24 Vdc).

To define a conditional block, once the block number has been defined, press [.] and the CNC will show at the upper display the block number with a decimal point (period).

125.
------

### 6.2.2 METRIC OR INCH PROGRAMMING (G70, G71)

Machine parameter "P23(8)" determines the programming units assumed by the CNC on power-up, after executing an M30 or after an Emergency or a Reset.

Nevertheless, it is possible to modify the programming units by using these functions:

G70	Inch programming
G71	Metric programming (in millimeters)

The CNC assumes the selected units as new programming units.

The G70 and G71 functions are modal and incompatible with each other.

### 6.2.3 ABSOLUTE / INCREMENTAL PROGRAMMING (G90 / G91)

The coordinates of a point (position) may be absolute (G90) or incremental (G91).

When using absolute coordinates (G90), they are referred to the coordinate system established as part zero.

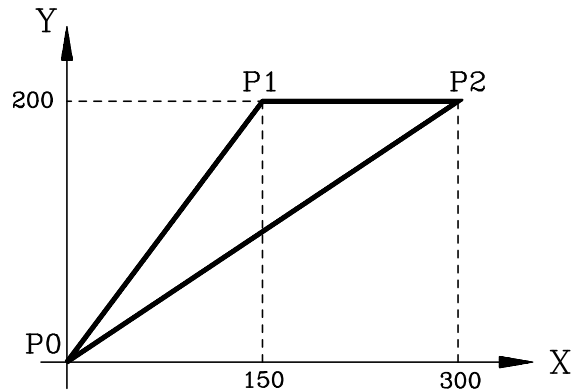
When using incremental coordinates (G91), they indicate the distance to the target point. The sign of the coordinate indicates the direction the tool must be moved to reach the target point.

The G90 and G91 functions are modal and incompatible with each other.

On power-up, after executing an M30 or after an Emergency or a Reset, the CNC assumes the code set by machine parameter "P21(3)". G90 if "P21(3)=0" and G91 if "P21(3)=1"

Chapter: 6 <b>PROGRAMMING</b>	Section: <b>BASIC CONCEPTS</b>	Page <b>3</b>
----------------------------------	-----------------------------------	------------------

Programming example taking P0 (0,0) as starting point.



Program in absolute coordinates (G90)

N250G90	X0	Y0	Point P0
N251	X150	Y200	Point P1
N252	X300		Point P2
N253	X0	Y0	Point P0

Program in incremental coordinates (G91)

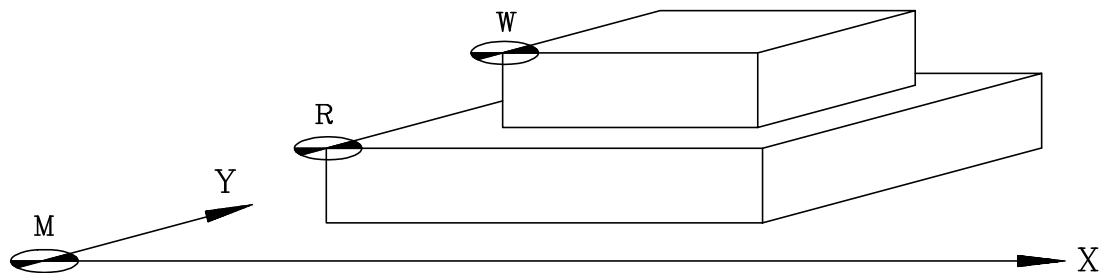
N250G90	X0	Y0	Point P0
N251G91	X150	Y200	Point P1
N252	X150		Point P2
N253	X-300	Y-200	Point P0

## 6.3 REFERENCE SYSTEMS

### 6.3.1 REFERENCE POINTS

A CNC machine must have the following reference points established:

- \* **Machine Reference Zero** or origin point of the machine. It is set by the machine manufacturer as the origin of the coordinate system of the machine.
- \* **Part Zero** or origin point for the part. It is the origin point set to program the measurements of the part. It can be chosen freely by the programmer and its reference to the machine reference zero is set by means of a zero offset.
- \* **Machine Reference Point.** It is the physical location of the marker pulse or reference pulse (I<sub>o</sub>) 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 parameters “P2, P42”.



M	Machine Reference Zero.
W	Part zero.
R	Machine reference point.

### 6.3.2 MACHINE REFERENCE (HOME) SEARCH (G74)

The axes may be programmed to be homed separately or together.

#### Home search on one axis.

Program function G74 followed by the axis to be referenced: G74 X or G74 Y.

#### Home search on both axes

Program G74 alone in a block.

The CNC starts homing the axes in the order established by machine parameter P61(3).

If "P61(3)=0"    First X and then Y  
If "P61(3)=1"    First Y and then X

#### **Attention:**

A block containing function G74 may **not** contain any other G function.



When homing in JOG mode, the currently active zero offset (part zero) will be cancelled and the CNC will display the home coordinates set by machine parameters "P2, P42".

In all other cases, the selected part zero will be maintained. Therefore, the coordinates being displayed will be referred to that part zero.

Function G74 implies function G01 which means that after executing G74 the CNC assumes function G01.

### 6.3.3. COORDINATE PRESET AND ZERO OFFSETS

With this CNC it is possible to apply zero offsets in order to use the coordinates relative to the part's blueprint without having to modify the coordinates of the point on the part when editing a program.

A "Zero Offset" is the distance between the part zero and the Machine Reference Zero (home).

The zero offset may be applied as follows:

- \* In the jog mode, by presetting a coordinate value or "0".
- \* By means of function G92 (coordinate preset) where the CNC takes the programmed values as new axis position values (coordinates).
- \* By loading the zero offset table (G51.....G60) and then selecting one of them (T function). The CNC takes as new part zero the point located at the distance indicated by the table from Machine Reference Zero (home).

#### 6.3.3.1 COORDINATE PRESET (G92)

When applying a zero offset by means of function G92, the CNC does not move the axes. It assumes the coordinates programmed after G92 as the new axis position values.

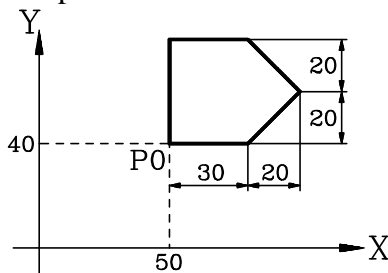
If machine parameters "P100(8)=0", "P100(7)=0", the CNC stores internally the distance from the new Part Zero to Home.

On certain applications as cyclic rollers, the accumulated distance from home after many continuous presets may overflow the CNC memory.

In this type of applications, it is possible to use G92 as a "coordinate setting" (versus presetting), (P100(8)=1, P100(7)=1, where the CNC selects a new Part Zero without storing its distance from home.

The programming format in either case is: G92 X — Y —

Example:



```

N125 G90 X50 Y40 Positioning at P0
N126 G92 X0 Y0 Presetting P0 as Part Zero
N127 G91 X30 Programming according to blue-
print
N128 X20 Y20
N129 X-20 Y20
N130 X-30
N131 Y-40
N132 M30
    
```

#### **Attention:**



A block containing G92 may not contain any other G code.

Function G92 is incompatible with G51, G52, G53, G54, G55, G56, G57, G58, G59 and G60.

### 6.3.3.2 ZERO OFFSET LOADING (G51.....G60)

In order to use these functions, the internal CNC table must be set as zero offset table, "P60(5)=1".

To see how to access this table, refer to the chapter on "AUX MODE" described earlier in this manual.

Table address 1 corresponds to function G51, 2 to G52 and so on up to address 10 which corresponds to G60.

Functions G51, G52, G53, G54, G55, G56, G57, G58, G59 and G60 must be programmed alone in the block and are incompatible with G92.

Their programming format is: G51 ... G60 X Y

The X and Y values indicate the distance from Machine Reference Zero (home) where the part zero is to be set.

When executing one of these functions: G51, G52, G53, G54, G55, G56, G57, G58, G59, G60, the CNC loads the new XY values at its corresponding table address.

### 6.3.3.3 ZERO OFFSET SELECTION (T1 ... T10)

To select a particular zero offset, use the "T" function.

The CNC cancels the previous zero offset and applies the new one.

The new part zero will be located at a distance from home set by the XY values of the corresponding table address.

Example. G52 X100 Y50 Loads the table with X100 Y50  
T2 Selects zero offset T2. The new part zero is at X100 Y50 from home.

This type of programmable zero offsets are very useful for repetitive operations to be performed at different positions along the axes.

#### **Attention:**



"T0" cancels the previously active zero offset and does not apply a new one. Therefore, the new part zero will coincide with the Machine Reference Zero (home).

## 6.4 COMPLEMENTARY FUNCTIONS

The CNC offers the following complementary functions.

<b>F</b>	Feedrate for the axes
<b>S</b>	Spindle speed.
<b>T</b>	Tool number / Zero offset
<b>M</b>	Miscellaneous (auxiliary) function

This programming order must be maintained within a block; but all of them need not be programmed in the block.

### 6.4.1 AXIS FEEDRATE "F"

This function indicates the feedrate of the axes when operating in closed positioning loop.

When operating in servo-controlled open loop, the path is not controlled.

If "F" is programmed while in non-servo-controlled open loop, the CNC will issue error 45.

With the "F" function ,it is possible to program the feedrate of the axes for G01, G02 and G03 movements.

In rapid positioning, G00, the feedrates for the axes are set by machine parameters "P27" and "P67".

If "F" is not programmed or "F0" has been programmed, the CNC carries out the G01, G02 and G03 interpolations at the feedrates set by machine parameters "P27" and "P67".

It is possible to vary the feedrate of the axes by means of the external operator panel "JOG 100" or by using the Feedrate Override inputs, pins 10 and 11 of connector I/O1.

### 6.4.2 SPINDLE SPEED "S"

With the "S" function, the spindle speed is programmed directly in revolutions per minute.

It accepts values between S0 and S9999, corresponding to 0 rpm. and 9999 rpm.

The actual maximum values are limited by machine parameters "P36", "P37", "P38" and "P39", depending, in each case, upon the selected spindle speed range.

Chapter: 6 <b>PROGRAMMING</b>	Section: <b>COMPLEMENTARY FUNCTIONS</b>	Page <b>9</b>
----------------------------------	--	------------------

### 6.4.3 **TOOL NUMBER "T"**

In order to use this function, it is necessary to set the internal CNC table as a tool table, machine parameter "P60(5)=0".

To see how to access this table, refer to the chapter on "AUX MODE" described earlier in this manual.

Every time a "T" function is executed, the CNC assumes the XY values indicated on the table and applies the corresponding tool length compensation to each axis.

If "T0" is programmed, the CNC applies no compensation and it cancels any one that may be active at the time.

Also, if machine parameter "P61(5)=0", every time a "T" function is executed, pins 20 through 27 of connector I/O1 show the BCD code of the selected tool and pin 4 of connector I/O2 outputs a "T Strobe".

### 6.4.4 **ZERO OFFSET "T"**

In order to use this function, the internal CNC table must be set as a zero offset table, machine parameter "P60(5)=1".

Every time a "T" function is executed, the CNC cancels the zero offset currently active and it applies the new one.

The new part zero will be located at a distance from home set by the XY values of the corresponding table address.

This type of programmable zero offsets are very useful for repetitive operations to be performed at different positions along the axes.

### 6.4.5 **MISCELLANEOUS (AUXILIARY) "M" FUNCTION**

After the "M" code, indicate the number of the miscellaneous function to be executed. There are 100 different miscellaneous functions (M00 through M99).

Up to 4 M functions may be programmed in a block and they must be separated by a period (M03.18.27).

When the new spindle speed requires a range change, the CNC automatically executes the corresponding M function: M41, M42, M43 or M44. Therefore, when a block contains an "S" code which implies a range change, only up to 3 more "M" functions may be programmed.

Functions M41, M42, M43 and M44 cannot be programmed by the machine operator.

The M functions are executed at the beginning of the block except M00, M01 and M30 which are the last ones to be executed in the block.


When more than one "M" function has been programmed in a block, the CNC executes them in the order they are programmed.

Page <b>10</b>	Chapter: 6 <b>PROGRAMMING</b>	Section: <b>COMPLEMENTARY FUNCTIONS</b>
-------------------	----------------------------------	--

The CNC always outputs the "M" function via pins 20 through 27 of connector I/O1. These outputs may be either binary coded or BCD coded as described in the section on "Auxiliary M function" of the chapter on "Concepts" in the Installation Manual.

The following "M" functions have a specific internal meaning at the CNC.

***M00. Program stop***


When the CNC executes the M00 code, it interrupts the program. To resume it, press 

***M01. Conditional program stop***

Same as M00 except that it is ignored unless the "Conditional Input" is active (high -> 24Vdc), pin 18 of connector I/O1.

***M30. End of program with return to first program block***

This code indicates the end of the program and performs a "General Reset" of the CNC setting it to initial conditions. It also stops the spindle (M05).

Plus, the CNC returns to the first block of the program in such way that if  is pressed again, the CNC will execute the program again from the beginning.

The M30 must always be programmed at the end of each program. Otherwise, the CNC will keep on executing all the following blocks.

If machine parameter "P21(4)=1", the CNC increments the count of the parts counter in one unit every time function M30 is executed.

***M03. Clockwise spindle rotation***

This code starts the spindle clockwise.

***M04. Counter-clockwise spindle rotation***

This code starts the spindle counter-clockwise.

***M05. Spindle stop***

This code stops the spindle.

***M41,M42,M43,M44. Spindle speed range selection***

The CNC offers 4 spindle speed ranges (gears) M41, M42, M43 and M44, with their maximum speed limits set by machine parameters "P36", "P37", "P38" and "P39".

When the new "S" value involves a range change, the CNC automatically outputs its corresponding M function (M41 through M44).

These codes cannot be programmed by the machine operator.

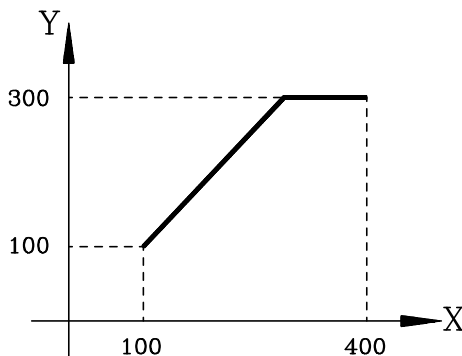
## 6.5 PATH CONTROL

### 6.5.1 RAPID POSITIONING (G00)

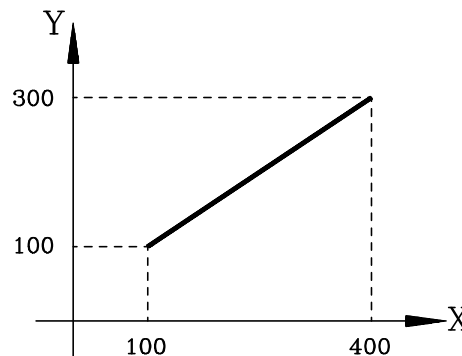
The G00 movements are executed at the feedrates set by machine parameters "P27" and "P67".

Depending on the setting of machine parameter "P60(1)", the rapid positioning moves will be interpolated (vectored): "P60(1)=1", or not: "P60(1)=0".

For example: G00 X400 Y300



"P60(1)=0"



"P60(1)=1"

When the G00 moves are not vectored, "P60(1)=0", each axis will position at the feedrate established by its corresponding machine parameter "P27" or "P67".

When the G00 moves are vectored, "P60(1)=1", both axes are interpolated following a straight line from the current position to the end point. In this case, the feedrate for the resulting path is the slower one of the values of machine parameters "P27" and "P67".

#### **Attention:**

When programming G00, the previous F value is not cancelled. Therefore, when G01, G02 or G03 is programmed again, that F value is recovered.



Function G00 is modal and incompatible with G01, G02, G03, G33, G74 and G75. G00 may also be programmed as G or G0.

It is possible to vary the actual feedrate by using the external operator panel "JOG 100" or by means of the feedrate override inputs at pins 10 and 11 of connector I/O1.

On power-up, after executing an M30 or after an Emergency or Reset, the CNC assumes G00.

## 6.5.2 LINEAR INTERPOLATION (G01)

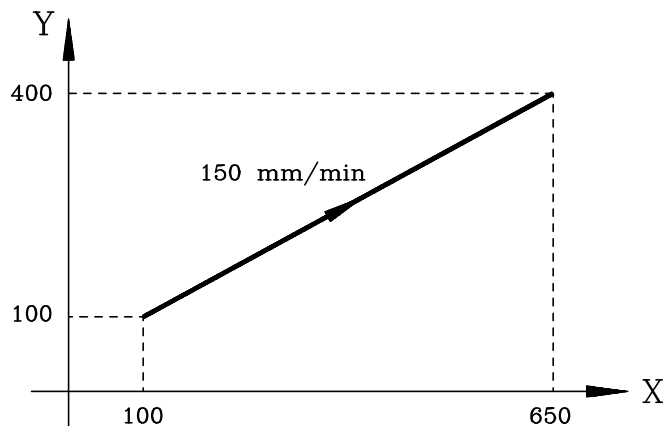
This function cannot be executed when operating in non-servo-controlled open positioning loop (error 14). When operating in servo-controlled open loop, the path is not controlled.

When operating in Closed Loop, the G01 movements are carried out following a straight line at the programmed feedrate "F".

When moving both axes, the resulting path is a straight line between the current position and the target point.

The axes move along this path at the programmed Feedrate F. The CNC calculates the feedrates for each axis so the resulting path is the programmed feedrate F.

Example:



**G1.90 X650 Y400 F150**

### **Attention:**



If no "F" value has been programmed, the CNC assumes the "F" programmed previously.

If none is selected, the CNC assumes the smaller of the values set by machine parameters "P27" and "P67".

It is possible to vary the actual feedrate by using the external operator panel "JOG 100" or by means of the feedrate override inputs at pins 10 and 11 of connector I/O1.

Function G01 is modal and incompatible with G00, G02, G03 and G33. G01 may also be programmed as G1.

### 6.5.3 CIRCULAR INTERPOLATION (G02, G03)

In order to perform a circular interpolation, the system must operate in Closed Positioning Loop.

There are two types of circular interpolations.

- G02:** Clockwise interpolation.
- G03:** Counter-clockwise interpolation.

The movements programmed after G02 and G03, are carried out following a circular path at the programmed feedrate F.

The programming format is: "G02 X Y I J" or "G03 X Y I J", where:

- G02 Clockwise interpolation.
- G03 Counter-clockwise interpolation.
- X X coordinate of the end point.
- Y Y coordinate of the end point.
- I Distance from the arc's starting point to the center of the arc along X.
- J Distance from the arc's starting point to the center of the arc along Y.

The X and Y coordinates may be either absolute or incremental and the I and J values always define the distance from the starting point to the center of the arc.

When any of the X or Y coordinates of the end point is the same as that of the starting point, it is not necessary to program it. However, the I and J values must always be programmed even when their values are "0".

#### **Attention:**



G02 and G03 are modal and incompatible with each other and with G00 and G01.

G02/G03 may also be programmed as G2/G3.

It is possible to vary the actual feedrate by using the external operator panel "JOG 100" or by means of the feedrate override inputs at pins 10 and 11 of connector I/O1.

Programming example taking X60 Y40 as starting point:



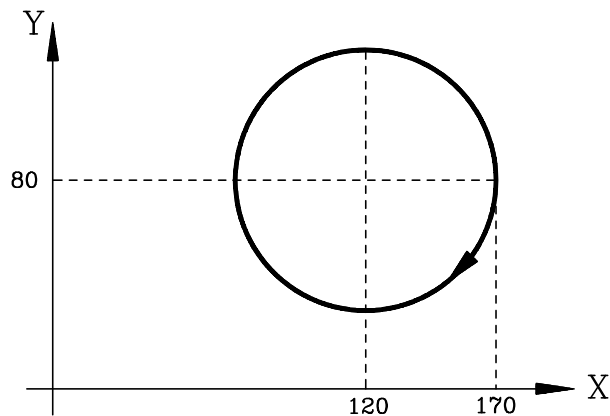
Absolute programming (G90)

```
N220 G90.3 X110 Y90 I0 J50
N230 G3 X160 Y40 I50 J0
```

Incremental programming (G91)

```
N220 G91.3 X50 Y50 I0 J50
N230 G3 X50 Y-50 I50 J0
```

Programming example for a complete circle in a single block starting at X170 Y80.



Different ways to program in absolute coordinates (G90)

```
N330 G90.2 X170 Y80 I-50 J0
N330 G90.2 I-50 J0
```

Different ways to program in incremental coordinates (G91)

```
N330 G91.2 X0 Y0 I-50 J0
N330 G91.2 I-50 J0
```

### 6.5.4 SYNCHRONIZATION (G33)

With function G33 it is possible to synchronize the X axis with an external device.

When this feature is used the machine cannot use a Y axis, with the A2 feedback connector being used as feedback input for the external device.

Typical applications for this feature are: Lathe style threading, paper feeder synchronized with the rotation of the roller, etc.

Its programming format is: G33 X K

Where X End coordinate  
K Synchronism pitch or displacement of the X axis per turn of the external device.

#### **Attention:**



The programmed feedrate cannot be overridden while function G33 is active.

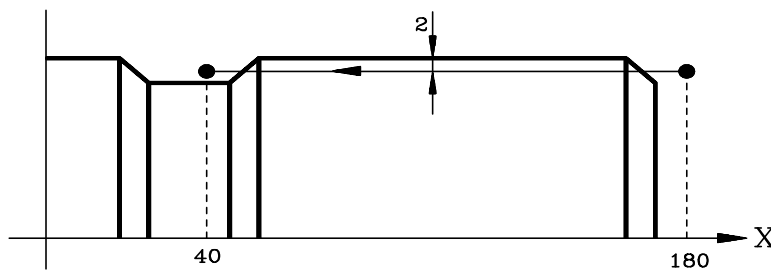
G33 is modal and incompatible with G00, G01, G02, G03, G34 and G75.

On power-up, after executing an M30 or after an Emergency or Reset, the CNC assumes G00.

#### **Example of threading**

The machine must have a spindle encoder.

Take the X axis feedback to connector A1 and the spindle feedback to connector A2



To make in only one pass the cylindrical thread 2 mm in depth and with 5 mm pitch, as shown in the figure, program as follows:

G33 X40 K5 in absolute coordinates or G33 X-140 K5 in incremental coordinates.

The value in X indicates the final positioning coordinate

The value of K indicates the positioning of the X axis per spindle turn. As in this case a 5 mm thread pitch is required K5 should be programmed.

The proper programming is as follows:

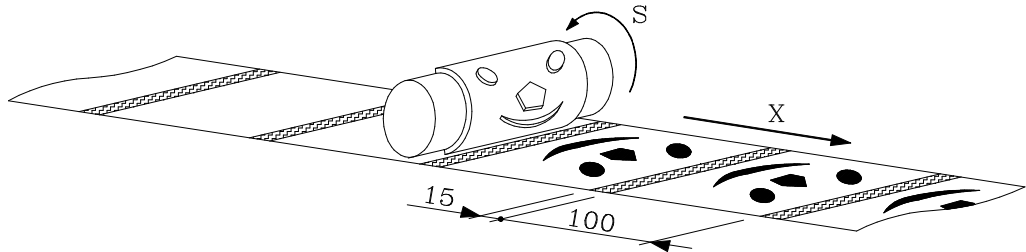
G1.90 X180 ..... ; Movement in operating feedrate to the starting point

G33 X40 K5 .... ; Threading

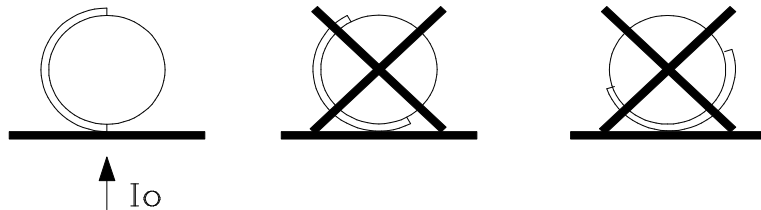
M30 ..... ; End of program

**Example of synchronized stamp printing**

A roller with half of its surface covered with a stamp is being used.  
 This figure has to be stamped on continuous paper.  
 The resulting image takes up 100 mm and a space of 15 mm between each two images is to be left.



The feedback for the paper feeder will be taken to connector A1 and will correspond to axis X.  
 The roller covered with the stamp should have an encoder. Said feedback should be taken to connector A2.  
 Make the Io signal for the feedback from the roller covered with the stamp coincide with the start of the printing process.



To carry out printing as shown in the figure the following should be programmed:

- N0 S1000 M3 ..... ; Turning direction and speed of the roller covered with the stamp
- N1 G0.91 X15 ..... ; Fast feed across the space to be left between 2 images.
- N2 G33 X100 K200 . ; Printing of an image  
 The value in X indicates the distance to be moved, 100 mm  
 The value of K indicates the positioning of axis X per spindle turn. As in this case 100mm has to be moved for each half turn of the roller, K200 should be programmed.
- N3 G25 N1 ..... ; Repetition of the process.

**Operation:**

The paper fast feeds across the space that has to be left between 2 images, 15 mm.

Waiting stage until the Io signal for the feedback from the roller covered with the stamp is received.

Printing of the image. The positioning of axis X is synchronized with the roller turn.

After covering the 100 mm the paper again fast feeds and the cycle is repeated.

### 6.5.5 AXIS X AS AN INFINITE SLAVE OF ANOTHER AXIS (G34)

Function G34 enables indefinite synchronization of axis X with an external device.

When this feature is to be used one should:

Set the machine parameter "P100(4)=1"

The machine cannot have a Y axis, using the A2 feedback connector A2 as input for feedback from the external device.

The external device must have an encoder.

Set machine parameter "P77" with the number of pulses of said encoder.

The programming format is: G34 X

Where X is the synchronization or positioning pass of axis X per turn of the external device.

**Attention:** While function G34 is activated, programmed feedrate F cannot be altered. If the Feed-Hold signal is activated the CNC stops the axis feed and starts this again when the Feed-Hold is deactivated.

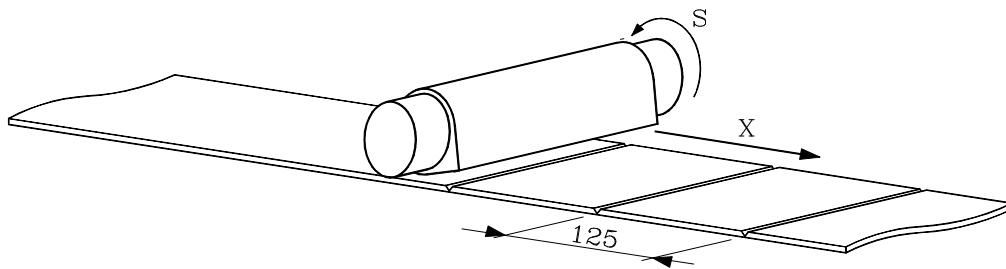


Function G34 is modal and incompatible with G00, G01, G02, G03, G33 and G75.

On power-up, after executing M30, G74 X or after an Emergency or reset, the CNC assumes code G00.

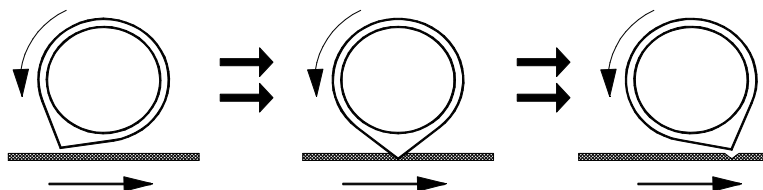
#### Example of a plastic cutter

There is a roller with a wedge which acts like a shear.  
Cuts are to be made every 125 mm along a plastic strip.



The feedback for the plastic feeder will be taken to connector A1 and correspond to axis X.

The roller with wedge must have an encoder. This feedback will be taken to connector A2.



Example of programming:

```

N0 G74 X ..... ; Search for reference point of axis X
N1 G34 X125 ..... ; Activation of slave mode
N2 S1000 M3 ..... ; Turning direction and speed of the roller with wedge
                        Axis X will move as infinite slave, making cuts every 125
                        mm.
N3 M0 ..... ; Program stop
                        Function G34 is still active and cuts are still made every 125
                        mm
                        If the START key is pressed the CNC will continue with
                        program execution .
N4 M30 ..... ; End of the program

```

The following example displays how one can program cutting of one batch of parts 150 mm in length and then another batch of 80 mm pieces.

```

N0 G74 X ..... ; Search for reference point of axis X
N1 G34 X150 ..... ; Activation of slave mode
N2 S1000 M3 ..... ; Turning direction and speed of the roller with wedge.
                        Axis X will move as infinite slave, making cuts every 125
                        mm.
N3 M0 ..... ; Program stop
                        Function G34 is still active and cuts are still made every 125
                        mm
                        The CNC does not continue with the execution of the
                        program until the START key is pressed
N4 M5 ..... ; Program stop. End of the first batch.
                        If required the material to be cut can be changed.
N5 M0 ..... ; Program stop.
                        The CNC does not continue with the execution of the
                        program until the START key is pressed.
N6 G74 X ..... ; Search for reference point of axis X
N7 G34 X80 ..... ; Activation of the slave mode. Cuts every 80 mm
N8 S1000 M3 ..... ; Turning direction and speed of the roller with wedge.
                        Axis X will move as infinite slave, making cuts every 80
                        mm.
N9 M0 ..... ; Program stop.
                        Function G34 is still active and cuts are still made every 80
                        mm
                        The CNC does not continue with the execution of the
                        program until the START key is pressed.
N10 M30 ..... ; End of program

```

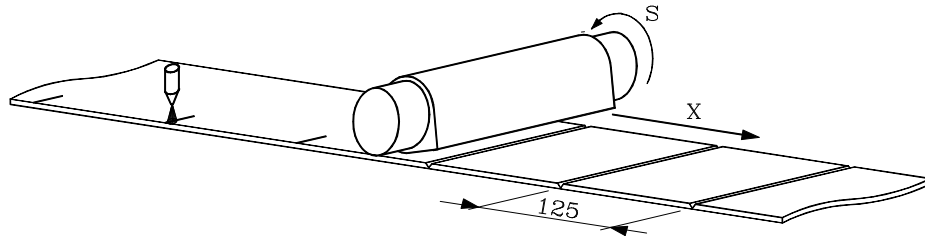
### 6.5.5.1 AUTOMATIC PASS COMPENSATION

The CNC allows any variations that may arise in the material through temperature dilation, roller tension, etc. to be automatically compensated.

To carry out these compensations the following is required:

- The material has to have reading marks in the cutting zones.
- An optical reader has to be used for reading these marks.
- The signal given by this reader must be compatible with the I<sub>o</sub> signal given by the Fagor feedback devices.
- Connect the signal given by the optical reader to pin 6 of connector A2 and properly set "P63(6)" machine parameter.
- Set machine parameter "P100(3)=1" to be able to use this feature.
- Set machine parameter "P81" with the maximum deviation allowed.

Operation:



- 1.- Activate the slave mode function. The machine will start to cut.
- 2.- When variations are detected in the material, and only then, the compensator function must be activated. To do this set input E1 high (pin 21 of connector I/O1 at 24V).
- 3.- The optical reader tells the CNC the location of the marks and this displays the distance between 2 consecutive marks in the window for the X axis.
- 4.- The CNC compares this value and the one programmed in function G34 X???? and does the following:
  - \* If parameter "P81=0" is set, the CNC does nothing.
  - \* If the difference is under 100 microns, the CNC does nothing.
  - \* If the difference is between 100 microns and the value assigned to parameter P81, the CNC automatically compensates the value set in function G34 X????.
  - \* If the difference is greater than the value assigned to parameter P81, the CNC will display error 53.
- 5.- Compensation has already been carried out. Deactivate the compensator function, input E1 (pin 21 of connector I/O1) at 0V.

The CNC will stop handling the signals coming from the optical reader and continue working with the last step calculated.



**Attention:** Do not modify the spindle revolutions during automatic pass compensation.  
Automatic pass compensation should not be done continuously. It should be used when variations in the material are found.

## 6.6 ADDITIONAL PREPARATORY FUNCTIONS

### 6.6.1 DWELL (G04)

A dwell (delay) may be programmed by using function G04.

To do this, after the G04 code, press the decimal point and indicate the desired dwell in seconds. The maximum dwell programmable in a block is 99.99 seconds.

Examples:

G04.99.99	99.99 second dwell
G04.2	2 second dwell.
G04.0.01	0.01 second dwell.

#### **Attention:**



G04 may also be programmed as G4.

The dwell is always executed at the beginning of the block containing it.

The block containing function G04 cannot have any more G codes; but it may have other functions (X, Y, F, S, T, etc)

### 6.6.2 INCREMENT PART COUNTER (G45)

The CNC has an internal part counter which increments its count in one unit in the following cases:

- \* Every time G45 is executed.
- \* Every time M30 is executed if machine parameter "P21(4)=1".

Its maximum count value is 9999 and it rolls over starting at 0, 1, 2, and so forth.

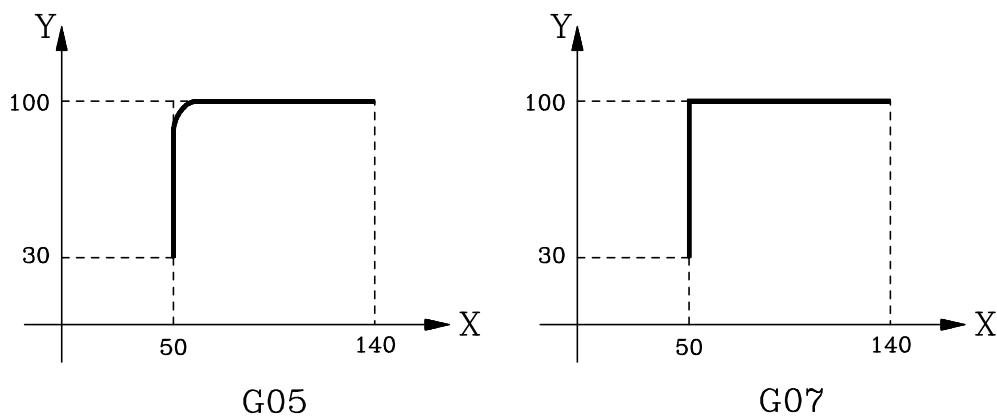
To see the value of this counter or reset it to "0", proceed as described in the section on "Display Modes" of the chapter on "Program Execution" in this manual.

### 6.6.3 ROUND CORNER (G05) AND SQUARE CORNER (G07)

Functions G05 and G07 are modal and incompatible with each other. With them, it is possible to execute the transition between motion blocks in a different way.

G05. When in this mode, the CNC starts executing the next block as soon as the axis programmed in the current block starts decelerating to get into position.

G07. When in this mode, the CNC does not start executing the next block until the axis programmed in the current block has reached position (in the in-position zone: "P28", "P68").



#### **Attention:**

G05 and G07 may also be programmed as G5 and G7.



On power up, after executing an M30, Emergency or a Reset, the CNC assumes G07.

When operating in non-servo-controlled open loop, executing a program in Single-Block mode or in semi-automatic mode, all the movements are carried out in G07.

## 6.7 SPECIAL FUNCTIONS

### 6.7.1 FEEDBACK INHIBIT (G47, G48)

In some applications, after positioning the part, sheetmetal, etc. it is necessary to correct its position.

This correction is made by an external device not controlled by the CNC.

When executing function G47, the CNC ignores the possible movements caused by that device, thus inhibiting the feedback pulses.

The programming formats for this function are:

G47 X It ignores the X axis feedback pulses.

G47 Y It ignores the Y axis feedback pulses.

G47 It ignores the feedback pulses from both axes.

To cancel this feature and return to normal feedback monitoring, program function G48.

#### **Attention:**

G47 is modal.

When G47 is active, the CNC ignores the feedback pulses once the axis is in position.



If a movement block is executed afterwards (G00, G01, G02, G03,...) the CNC switches to control the feedback thus preventing the machine from moving out of control.

Once the programmed movement has ended (reaching position), the CNC switches back to ignoring the feedback pulses.

If while G47 is active, one or both axes are moved and, then, G48 is programmed, the CNC **does not** recover the lost position.

On power-up, after executing an M30, after an Emergency or a Reset, the CNC assumes G48.

Chapter: 6 <b>PROGRAMMING</b>	Section: <b>SPECIAL FUNCTIONS</b>	Page <b>23</b>
----------------------------------	--------------------------------------	-------------------

## 6.7.2 G47, G48 AS OPENING OF THE AXIS LOOP

In particular applications such as presses, metal forming machines, sheetmetal forming machines, etc., the part undergoes a movement or pull while it is machined.

By means of functions G47, G48 the CNC allows the loop of the axis undergoing the pull to be opened or closed.

When the position loop is closed, in normal operating mode, the CNC controls the axis feedrate at all times.

When the position loop is opened the axis operates in DRO mode, that is, the CNC does not control the feedrate of the axes but does take into account their movements.

Example of sheetmetal forming machines:

Normal operation is with the position loop closed.

When the tube forming is to be done the following steps must be taken:

- Open the position loop of the axis undergoing the pull and the axis starts to operate in DRO mode.
- Form the tube. The CNC displays at all times the position coordinate of both axes, that is, it accurately reflects the tugs undergone by the tube.

After forming the tube close the axis position loop again.

In order to be able to use functions G47, G48 as opening for the axis loop machine parameter "P101(3)=1" has to be set.

The programming formats of this function are:

- G47 X      Opens the position loop of axis X.
- G47 Y      Opens the position loop of axis Y.
- G47        Opens the position loop of both axes
- G48 X      Closes the position loop of axis X.
- G48 Y      Closes the position loop of axis Y.
- G48        Closes the position loop of both axes.

### **Attention:**

Functions G47 and G48 are modal.

The block containing function G47 or G48 cannot contain other functions (X, Y, F, S, T, etc.)



If a movement block of an axis operating in open position loop is executed, the CNC will display the corresponding error code.

On power-up, after executing an M30 or after an Emergency or Reset, the CNC assumes function G48.

### 6.7.3 THE FEEDRATE F IS NOT AFFECTED BY "P18" (G61, G62)

When operating with feedrates slower than 1 mm/min. or 1 inch/min. machine parameter "P18" must be set to a value other than "0".

The CNC divides the programmed feedrate by the P18 value and applies the result as axis feedrate.

It is possible to program whether this "P18" is to be considered or ignored by means of functions G61 and G62.

- G61 The value of P18 is ignored. The CNC applies the programmed F value.
- G62 The value of P18 is not ignored. The programmed feedrate F value is affected by this value as described above.

Example: Machine parameter "P18 = 100"

N0	F1		Programmed F 1 mm/min.
N1	G62.1	X20	Movement at 0.01 mm/min.
N2		X-100	Movement at 0.01 mm/min.
N3	G61		F not affected by P18
N4		X0	Movement at 1 mm/min.
N5	G62		F affected by P18
N6		X30	Movement at 0.01 mm/min.
N7	M30		

#### **Attention:**



G61 and G62 are modal and incompatible with each other.

On power-up, after executing an M30, after an Emergency or a Reset, the CNC assumes G62.

### 6.7.4 ACCELERATION RAMP MODIFICATION (G93)

When the axes operate in Closed Loop, machine parameters "P16, P17", "P91" and "P92" set the acceleration and deceleration times for each axis.

In certain situations it may be interesting to apply other acc./dec. times.

With this function, it is possible to program the desired acc./dec. time.

Its programming format is: **G93 X Y**

Parameters X Y indicate in seconds, with millisecond accuracy, the acceleration time required for setting for each one of the axes. Maximum value 6 seconds.

Example: G93 X1.500 Y0.700 fixes an acceleration and deceleration ramp of 1.5 seconds for axis X and another one of 700 milliseconds for axis Y.

On power-up, after executing an M30, after an Emergency or a Reset, the CNC assumes the settings of machine parameters "P16, P17", "P91" and "P92".

## 6.8 OTHERS FUNCTIONS

### 6.8.1 BATCH PROGRAMMING (G81)

This function is especially designed for feeders of sheetmetal, cardboard, paper, plastic, etc.

It allows programming batches of parts having the same length.


Its programming format is: **G81 L C A**

- L** Length of each part of the batch.
- C** Number of parts to make.
- A** Forewarning part number

Example: To make a batch of 100 parts, each being 200mm long and warning when there are 3 parts left to finish the batch: G81 L200 C100 A2

Function G81 executes a G92 after each part (after each feed), thus being necessary to set "P100(8)=1" so the G92 does not accumulate count (coordinate setting)

#### **Basic operation:**

- 1.- When pressing  or receiving the external Cycle Start command.
- 2.- The CNC executes function G92 X0 setting the X axis coordinate to "0".
- 3.- The X axis moves the distance indicated by parameter "L" in G01 and the programmed feedrate F. It assumes the G05 or G07 previously active.
- 4.- Once the axis is in position, the CNC outputs an "M07" for the machine to cut the material. The part counter is incremented and the cycle is interrupted.
- 5.- The CNC verifies the following:
  - a) If the whole lot has been cut, number of parts equal to "C", the CNC outputs an "M09" and concludes the execution of function "G81".
  - b) When there are "A" parts left to go, the CNC outputs function "M08" so the forewarning signal is activated.
- 6.- If all the parts have not been made yet, the CNC waits for another Cycle Start command in order to resume the execution of the cycle from step "2".

#### **Attention:**



G81 is not modal. Therefore, it must be programmed every time.

It uses the internal CNC part counter. It resets it to zero at the beginning of the execution and it increments it every time it executes a part.

In the counter display mode, the CNC shows, on the left-hand side, the number of parts programmed for the batch and, on the right-hand side, the number of parts already made.

Once the cycle has been defined, the CNC stores the definition parameter values, thus not being necessary to redefine them.

```
N00 G81 L255 C200 A10 F1000
N01 G81 L387.5 C350           Takes A10
N02 G81                       Takes L387.5 C350 A10
N03 M30
```

The definition parameters "L", "C" and "A" are associated to arithmetic parameters "P0", "P1" and "P2" respectively.

Therefore, function "G81" may be defined by using the definition parameters "L", "C" and "A" and/or the arithmetic parameters "P0", "P1" and "P2", as illustrated by the following example:

```
N000 G81 L255 C200 A10      N000 P0=255
                             N001 P1=200
                             N002 P2=10
                             N003 G81
N002 G81 L387 C350         N004 P0=387
                             N005 P1=350
                             N006 G81
N004 G81 L104.5 C80 A8     N007 P0=104.5
                             N008 P1=80
                             N009 G81 A8
N006 M30                   N010 M30
```

If the external START signal is activated automatically after each cut, the CNC will start executing the following block after the last cut.

**Example**

```
N00 G81 L25500 C200 A10 F1000
N01 G81 L38700 C350 A20
N02 M0                       Interrupts the program.
N03 G81 L10450 C80 A8
N04 M30
```

## 6.8.2 *PROBING (G75)*

With function G75, it is possible to make movements ending when the CNC receives the signal from a touch probe.

The definition format is:     G75 X Y

After G75, the desired axes must be programmed followed by the coordinates of the target point.

The axes will move in G01 and at the programmed feedrate F until the external signal from the touch probe is received. At that point, the CNC will conclude the block assuming as theoretical position of the axes their real position at the time.

If the axes reach their target position before the CNC receives the probe signal, the CNC stops the axes and issues error 21.

### **Attention:**



G75 is not modal. Therefore, it must be programmed every time. It also implies function G01 which means that after a G75 block, the CNC assumes function G01.

The programmed feedrate cannot be overridden during the probing move.

Once G75 is executed, it is possible to assign the position of each axis to any arithmetic parameter and, by using arithmetic programming, generate special programs for measurement and calibration of tools and parts.

### 6.8.3 RIGID TAPPING (G84, G80)

With function G84, it is possible to tap a part by interpolating the spindle rotation with the movement of the axis.

The machine must have an X axis and a servo-controlled spindle (with servo-drive and spindle encoder).

The spindle feedback is connected to input A2 and the analog output for the spindle is provided at pins 34 and 35 of connector I/O1.

When the machine has a rigid tapping axis Y cannot be used. If you wish to use an electronic handwheel this must be connected to pins 17 and 25 of connector I/O2

Nevertheless, the CNC uses the Y function to define the thread pitch. Consequently, machine parameter "P60(8)" must be set to "0" to enable this function

P63(3)=1	The machine uses rigid tapping.
P60(8)=0	Enable the Y function.
P77	Number of spindle encoder pulses.

Its programming format is: **G84 X Y F S**

**X** Defines the depth of the tap.

**Y** Defines the tapping pitch.

When assigning a positive value, it must be defined in turns/mm or turns/inch. and when assigning a negative value, in mm/turn or inches/turn.

**F** Defines the tap-in speed in rpm.

**S** Defines the tap-out speed in rpm.

If parameter "F" is programmed and not the "S" parameter, the tap-in and tap-out movements are performed at the same speed, the one indicated by parameter "F".

If parameters "F" and "S" are not programmed, the CNC assumes the values used on the previous tap. If no tapping operation was performed previously, the CNC will issue the corresponding error.

Parameters "F" and "S" are reset to "0" on power-up, after an Emergency or a Reset.

## General Considerations

- \* When the machine uses rigid tapping "P63(3)=1" the spindle control is done in closed loop.
- \* To switch it to open loop, G80 must be executed.

From this point on, the spindle control is done in open loop and it is possible to program the "S" function.

Function G80 must be programmed alone in the block.

- \* The spindle will operate in closed loop again, if G84 is programmed, on power-up, after executing an M30 or after an Emergency or Reset.
- \* The spindle turning direction during rigid tapping must be defined before function G84 and by means of M3, M4.

If when executing G84, no turning direction has been selected, the CNC assumes the M03 (clockwise).

- \* All the various movements involved in rigid tapping are treated as a single block. In other words, when executing a G84 cycle in the Single-Block, the CNC makes the whole tap.
- \* When executing function G84, the CNC assumes functions G01 and G07.
- \* If due to possible spindle backlash, an additional spindle rotation is required at the bottom of the tap, it must be indicated by arithmetic parameter P19 in the number of spindle encoder pulses to move.

The CNC recovers this additional movement by rotating in the opposite direction at the end of the tap.

- \* A dwell at the bottom of the tap is also possible by indicating it, in milliseconds, with machine parameter "P32".

Page <b>30</b>	Chapter: 6 <b>PROGRAMMING</b>	Section: <b>OTHER FUNCTIONS</b>
-------------------	----------------------------------	------------------------------------

### Example and way to operate

A hole is to be tapped 20 mm deep with a thread pitch of 0.5 mm at a tap-in speed of 300 rpm and tap-out speed of 600 rpm.

The spindle encoder has 1000 lines and its pulses require a x4 multiplying factor.

Also, due to spindle backlash, it must turn 20 pulse units at the bottom of the tap.

It also requires a 100 millisecond dwell at the bottom of the tap.

The following CNC machine parameters must be set accordingly:

P63(3)=1	The machine uses rigid tapping.
P60(8)=0	Enable the Y function
P77=1000	Spindle encoder line count (number of pulses)
P61(6)=0	Feedback multiplying factor for Y (S)
P32=100	100 millisecond dwell at the bottom of the tap.

The tapping program looks like this:

N000 M3	Start the spindle clockwise.
N010 P19 K20	Additional spindle movement at the bottom of the tap.
N020 G84 X20 Y0.5 F300 S600	Definition of the canned cycle.
N030 M30	End of program.

The CNC carries out the following machining steps:

1.- Tap-in movement with the spindle turning at 300 rpm.

The X axis moves to X 20.000  
The Y (S) axis moves to Y 40.000 (20000 x 0.5 x 1000 x4)

2.- Once at the bottom of the tap, the spindle moves the amount indicated by arithmetic parameter P19, 20 pulses.

The Y (S) moves to Y 40.020

3.- 100 millisecond dwell as indicated by machine parameter P32.

4.- Tap-out move to clear the tap with the spindle turning at 600 rpm.

The X axis moves 20mm up to X 00.000  
The Y (S) axis moves (20000 x 0.5 x 1000 x4) up to Y 00.020

5.- Recovery of the additional move made at the bottom of the tap

The Y (S) axis moves up to Y 00.000

6.- End of tap and end of program.

#### 6.8.4 LOADING THE PUNCH SIZES (G60)

Function G60 can be used in two different ways.

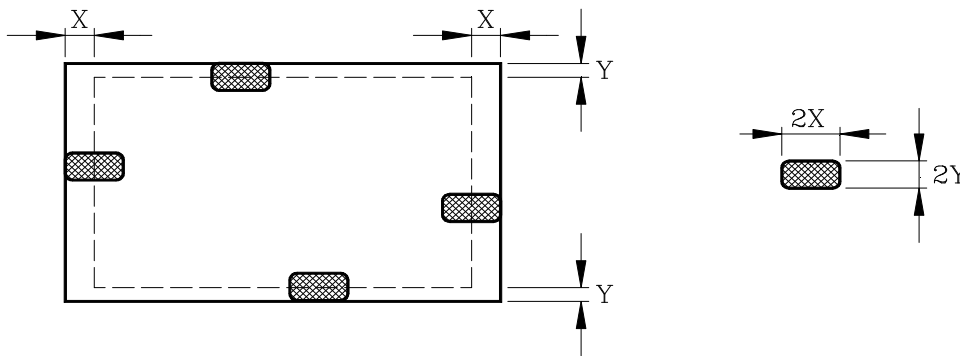
For loading zero offsets

This is explained in section 6.3.3.2 of this same manual.

For loading the punch dimensions

This is particularly meant for punching machines. Machine parameter "P101(7)=1" must be set.

The CNC takes into account the dimensions of the punch during program execution, preventing its extremes from exceeding the travel limits fixed by machine parameter.



The programming format is G60 X Y. Values X and Y indicate half of the punch dimensions according to axes X and Y.

When the punch is round values X and Y coincide with the punch radius.

#### **Attention:**



On power-up, after executing M30 or after an Emergency or Reset, the CNC assigns value 0 to both dimensions of the punch.

## 6.9 PARAMETRIC PROGRAMMING

A parameter takes the place of a numeric value in the various functions of the program.

The programmer may use these parameters when editing his/her own programs. Later on, and during execution, the CNC will replace these parameters with the values allocated to them at the time.

Up to 100 (P0 through P99) arithmetic parameters may be used.

While editing a program, proceed as follows in order to assign an arithmetic parameter to a particular function (G, X, Y, F, S, etc):

- \* Select the desired function (G, X, Y, F, S, etc).
- \* Press [P]. The CNC shows the letter "P" next to the selected function. For example: "XP".
- \* Key in the number of the arithmetic parameter.

### **Attention:**



When defining, in the same block, several functions of the same type, all of them must be programmed with or without parametric format. **The two formats cannot be combined.**

For example, block MP17.5.0 is the same as MP17 MP5 MP0 and not the same as MP17 M5 M0.

When defining function "G" in parametric, it only assumes the first indicated value. Therefore, programming GP10.11.12 will be the same GP10.

### 6.9.1 ASSIGNMENTS

When assigning values to an **arithmetic parameter**, use one of the following methods:

**N001 P1 P2** P1 takes the value of P2, while P2 maintains its own value.

**N002 P3 K1.5** P3 takes the numeric constant value of K which, in this case, is 1.5.

The constant values may be within  $\pm 99999.99$ .

**N003 P4 X** P4 takes the value of the theoretical position of the X axis at the time.

**N004 P5 Y** P5 takes the value of the theoretical position of the Y axis at the time.

**N005 P6 E1** P6 takes the value corresponding to input E1. Pin 21 of connector I/O2. A value of "0" when E1 = 0V and "1" when E1 =24V.

**N006 P7 E2** P7 takes the value corresponding to input E2. Pin 23 of connector I/O2. A value of "0" when E2 = 0V and "1" when E2 =24V.

**N007 P8 E3** P8 takes the value corresponding to input E3. Pin 22 of connector I/O2. A value of "0" when E3 = 0V and "1" when E3 =24V.

Chapter: 6 <b>PROGRAMMING</b>	Section: <b>PARAMETRIC PROGRAMMING</b>	Page <b>33</b>
----------------------------------	---	-------------------

**N008 P9 E4** P9 takes the value corresponding to input E4. Pin 25 of connector I/O2. A value of "0" when E4 = 0V and "1" when E4 =24V.

**N009 P10 E5** P10 takes the value corresponding to input E5. Pin 17 of connector I/O2. A value of "0" when E5 = 0V and "1" when E5 =24V.

**N010 P11 E0** P11 takes the value corresponding to the status of inputs E1, E2, E3, E4 and E5.

For example  $P11 = 2^4 + 2^2 + 2^1 = 16 + 4 + 2 = 22$

Input	E5	E4	E3	E2	E1
Voltage	24V	0V	24V	24V	0V
Logic State	1	0	1	1	0
Value	$2^4$	$2^3$	$2^2$	$2^1$	$2^0$

## 6.9.2 OPERATIONS

The operations that may be performed between arithmetic parameters or between arithmetic parameters and numeric constants (K) are:

- F1 Addition
- F2 Subtraction
- F3 Multiplication
- F4 Division
- F5 Square root
- F6 Square root of the sum of squares
- F7 Sine
- F8 Cosine
- F9 Tangent
- F10 Arc tangent
- F11 Comparison
- F12 Integer
- F13 Integer plus one
- F14 Integer minus one
- F15 Absolute
- F16 Complement

### F1 Addition

- N101 P10 P11 F1 P12 Means  $P10 = P11 + P12$
- N102 P10 P11 F1 K2 Means  $P10 = P11 + 2$
- N103 P10 K8 F1 K2 Means  $P10 = 8 + 2 = 10$
- N104 P10 P10 F1 K2 Means  $P10 = P10 + 2$

### F2 Subtraction

- N101 P10 P11 F2 P12 Means  $P10 = P11 - P12$
- N102 P10 P11 F2 K2 Means  $P10 = P11 - 2$
- N103 P10 K8 F2 K2 Means  $P10 = 8 - 2 = 6$
- N104 P10 P10 F2 K2 Means  $P10 = P10 - 2$

### F3 Multiplication

N101	P10	P11	F3	P12	Means P10 = P11 x P12
N102	P10	P11	F3	K2	Means P10 = P11 x 2
N103	P10	K8	F3	K2	Means P10 = 8 x 2 = 16
N104	P10	P10	F3	K2	Means P10 = P10 x 2

### F4 Division

N101	P10	P11	F4	P12	Means P10 = P11 ÷ P12
N102	P10	P11	F4	K2	Means P10 = P11 ÷ 2
N103	P10	K8	F4	K2	Means P10 = 8 ÷ 2 = 4
N104	P10	P10	F4	K2	Means P10 = P10 ÷ 2

### F5 Square root

N101	P10	F5	P12	Means P10 = $\sqrt{P12}$
N102	P10	F5	K9	Means P10 = $\sqrt{9} = 3$
N104	P10	F5	P10	Means P10 = $\sqrt{P10}$

### F6 Square root of sum of squares

N101	P10	P11	F6	P12	Means P10 = $\sqrt{P11^2 + P12^2}$
N102	P10	P11	F6	K2	Means P10 = $\sqrt{P11^2 + 4}$
N103	P10	K3	F6	K4	Means P10 = $\sqrt{9 + 16} = 5$
N104	P10	P10	F6	K2	Means P10 = $\sqrt{P10^2 + 4}$

### F7 Sine

N101	P10	F7	P12	Means P10 = sine P12
N102	P10	F7	K30	Means P10 = sine 30° = 0,5

### F8 Cosine

N101	P10	F8	P12	Means P10 = cos P12
N102	P10	F8	K60	Means P10 = cos 60° = 0,5

### F9 Tangent

N101	P10	F9	P12	Means P10 = tan P12
N102	P10	F9	K45	Means P10 = tan 45° = 1

### F10 Arc tangent

N101	P10	F10	P12	Means P10 = arc tan P12
N102	P10	F10	K1	Means P10 = arc tan 1 = 45°

## F11 Comparison

It compares an arithmetic parameter with another one or with a numeric constant indicating whether the first operand is equal, different, greater, or smaller than the second operand.

This operation is greatly used when working with conditional jumps G26, G27, G28 and G29, which are described later on.

N101 P10 F11 P12      Compares the values of P10 and P12  
N102 P10 F11 K100      Checks whether the value of P10 is equal, different, greater than or smaller than 100

## F12 Integer

N101 P10 F12 P12      P10 takes the integer part of P12  
N102 P10 F12 K12.34    P10 = 12, integer of 12.34

## F13 Integer plus one

N101 P10 F13 P12      P10 takes the integer value of P12 plus one  
N102 P10 F13 K12.34    P10 = 13, (integer of 12.34) + 1

## F14 Integer minus one

N101 P10 F14 P12      P10 takes the integer value of P12 minus 1  
N102 P10 F14 K12.34    P10 = 11, (integer of 12.34) - 1

## F15 Absolute value


N101 P10 F15 P12      P10 takes the absolute (unsigned) value of P12  
N102 P10 F15 K-12.3    P10 = 12.3  
N103 P10 F15 K4.5      P10 = 4.5

## F16 Complement

N101 P10 F16 P12      P10 takes the value of P12 with the opposite sign  
N102 P10 F16 K-12.3    P10 = -12.3  
N103 P10 F16 K4.5      P10 = -4.5

### 6.9.3 ACCESS TO THE ARITHMETIC PARAMETER TABLE

The CNC stores the values of the arithmetic parameters in a table. These values may be modified when executing parametric blocks or by accessing this table in the editing mode.

Press  to access the Editing mode and then press [F1] [P]. The information displayed by the CNC will look like this:

P00

The upper display shows the parameter number currently selected.

- 12345.678



The lower display shows the value of this parameter.

The CNC offers 100 (00/99) arithmetic parameters. They may have a sign, decimals and may be expressed in exponential format.

When the CNC uses the exponential format, the last two digits indicate that exponent.

0.98700 -1      Is the same as 0.98700 10<sup>-1</sup>  
1298700 7      Is the same as 1298700 10<sup>7</sup>

#### Displaying parameters

To see the previous and following parameters use keys  

When few arithmetic parameters are used, the CNC allows, in order to simplify operator handling, using only the first parameters of each ten and properly setting machine parameter "P93".

Examples:

With machine parameter "P93=0" the CNC will display all the arithmetic parameters.

With machine parameter "P93=1" the CNC will display the following parameters:

P0-P1 P10-P11 P20-P21 P30-P31 ..... P90-P91

With machine parameter "P93=2" the CNC will display the following parameters:

P0-P1-P2 P10-P11-P12 P20-P21-P22 P30-P31-P32 .... P90-P91-P92

#### Access to a particular parameter

To access a particular parameter, do the following:

- Press key [CL] twice. The CNC will clear the 2 windows.
- Introduce the number of the parameter that you wish to display.
- Press key [RECALL]

The CNC will display the value of this parameter.

*Assign a value to a parameter*

To assign a value to a specific parameter, do the following:

- Select the parameter required.
- Press key [CL]. The CNC will clear the lower window.
- Introduce the value required.
- Press key [ENTER]

#### 6.9.4 UNCONDITIONAL JUMP FUNCTION (G25)

Function G25 may be used in two ways:

- \* To jump to the desired block within the same program and continue executing from that block on.
- \* To execute a particular section (between any two blocks) of the program several times.

After executing this program section the desired number of times, the program will continue executing from the block after the one containing the jump.

The block containing function G25 may not have any other functions.

##### G25 as jump to another jump

With this function, it is possible to have the program jump to the desired block within the same program and continue executing from that block on.

Write "G25" followed by the destination block number.

The destination block number may be indicated by either a number (N010 G25 N50) or an arithmetic parameter (N010 G25 NP10).

Programming example:

```
N000 G00 X100
N005     Y50
N010 G25 N50
N015     X50
N020     Y70
N050 G01 X20
```

When reaching block N010, the program jumps to block N050 and it continues running from this block on, thus skipping blocks N015 and N020.

##### G25 to execute a section of the program

With this function, it is possible to execute a particular section (between any two blocks) of the program several times.

After executing this program section the desired number of times, the program will continue executing from the block after the one containing the jump.

Write "G25" followed by the first block number of the section to be executed. For example: N030 G25 N5

After that, press [.] and key in the last block number of the section to be executed. For example: N030 G25 N5.20

Then, press [.] and key in the number of times this section is to be executed. For example: N030 G25 N5.20.81

If the number of repetitions is not indicated, the CNC assumes a value of "1" meaning that it will only execute it once. For example: N030 G25 N5.20 is the same as N030 G25 N5.20.1

When using a parameter to indicate the program section to be executed, all three fields of the G25 statement will be parametric. For example: when programming N123 G25 NP10.20.30 the CNC interprets the first block of the section to be the one indicated by parameter P10, the last block of the section will be the one indicated by P20 (not necessarily block N020) and it will be executed the number of times indicated by parameter P30 (not necessarily 30 times).

Up to 15 nesting levels may be programmed, which means that it is possible to call from one section to another and from this one to another and so on up to 15 levels. Each level may be repeated 99 times.

**Attention:**



If the last block of the program section to be executed is empty, the CNC will issue error 7.

Programming example:

```
N000 G00 X10 M3
N005 Y20
N010 G01 X50
N015 G00 Y0
N020 X0
N025 Y10
N030 G25 N5.20.81
N035 Y22
```

The program starts running at block N000. When it reaches block N030, the CNC executes the section between blocks N005 and N020 (both included) 81 times.

Then, it continues running from block N035.

### 6.9.5 *CONDITIONAL JUMP FUNCTIONS (G26, G27, G28, G29)*

They are similar to function G25 (unconditional jump).

Functions G26, G27, G28 and G29, before jumping to the indicated block or executing the indicated program section, check whether the required condition is met or not.

G26 Jump if zero.	The "if zero" condition must be met.
G27 Jump if not zero.	The "if zero" condition must not be met.
G28 Jump if less than zero.	The "less than" condition must be met.
G29 Jump if equal or greater than zero.	The "less than" condition must not be met.

The "If zero" condition ("equal to" flag) is met in the following cases:

- \* When the result of an operation is zero.  
Example: N001 P1 P3 F2 K5 If P3 = 5
- \* When both sides of a comparison are identical.  
Example: N002 P1 F11 K8 If P1 = 8

The "Less than" (negative flag) condition is met in the following cases:

- \* When the result of an operation is less than zero (negative).  
Example: N001 P1 P3 F2 K5 If P3 < 5
- \* When the first operand of a comparison is smaller than the second operand.  
Example: N002 P1 F11 K8 If P1 < 8

#### **Attention:**



The assignments and non-parametric functions do not alter the status of the condition flags.

Programming example:

```
N060 P2 F11 K22
N065 G01 X10
N070 Y20
N071 G26 N100
N072 G28 N200
N073 G29 N300
```

Block N060 makes a comparison.

Blocks N65 and N70 do not alter the condition flags.

Therefore: If P2 = 22, The program continues at block N100  
If P2 < 22, The program continues at block N200  
If P2 > 22, The program continues at block N300

Care must be taken when programming functions G26 and G29. If in the previous example, the following were programmed, instead:

```
N071 G28 N200
N072 G29 N300
N073 G26 N100
```

The program would not execute block N073 since with P2 < 22 it jumps to N200 and with P ≥ 22 it jumps to N300.

Parametric programming example to calculate the coordinates of the various points forming an ellipse.

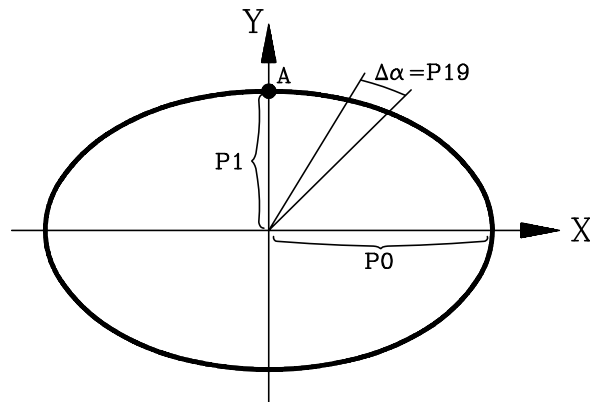
The formula defining an ellipse is:

$$X = P0 \sin \beta$$

$$Y = P1 \cos \beta$$

Where:

P0 = Long axis  
P1 = Short axis



The definition parameters are:

P0 = Long axis.  
P1 = Short axis.  
P2 = Angular position of the first point (A).  
P3 = Angular step between consecutive points.

The parameters used by the program are:

P2 = Angular position of the new point ( $\beta$ )  
P4 = X coordinate of the new point.  
P5 = Y coordinate of the new point.

Program:

N000	P0 K20	Setting of definition parameters (Example).
N001	P1 K10	
N002	P2 K0	
N003	P3 K2	
N004	G25 N20	Jump to subroutine to calculate the ellipse.
N020	P4 F7 P2	Calculation of the new point of the ellipse.
N021	P5 F8 P2	
N022	P4 P0 F3 P4	$P4 = P0 \sin \beta$
N023	P5 P1 F3 P5	$P5 = P1 \cos \beta$
N024	G0.5.90 XP4 YP5	G00 move to first point of ellipse
N025	P2 P2 F1 P3	Angular increment $\beta = \beta + P3$
N026	G25 N20.23	Calculation of new point of the ellipse.
N026	G1 XP4 YP5 F200	Positioning at new point.
N027	P2 F11 K360	Compare $\beta$ with $360^\circ$
N028	G28 N25	If $\beta < 360^\circ$ , still not done, new point, jump to N25
N029	M30	If $\beta = 360^\circ$ , End of program.

# **ERROR CODES**

- 01 The execution of JUMP/CALL function has been attempted in the TEACH IN mode.
- 02 Axis feedrate F too high during execution.
- 03 EPROM memory error.
- 04 RAM memory error.
- 05 Parameter CHECKSUM error.
- 06 Strange data in memory.
- 07 Jump/Call programmed incorrectly.
- 08 Battery Error.
- 09 X axis machine parameter error:
  - > The braking distance (P3) is smaller than the In-position zone (P28).
  - > The P3 value, is greater than 8388.607 mm or 330.259 inches.
- X axis execution error:
  - > The movement cannot be executed since, due to the programmed feedrate, the braking distance is smaller than the In-Position zone (Open Positioning Loop).
- 10 A block has been programmed which forces the X axis to overrun its travel limits.
- 11 A value outside the permissible limits has been programmed.
- 12 An invalid value has been programmed:
  - > The rotary ROLLOVER axis via the shortest way, does not admit negative absolute coordinates.
  - > An axis with HIRTH tothing does not admit decimal values.
- 13 A jump/call has been programmed with more than 15 nesting levels.
- 14 An inappropriate G function has been programmed.
- 15 Function M30 has not been programmed at the end of the program.
- 16 X axis not controlled:
  - > The X axis has shifted a distance 16 times the in-position zone once the brake has been activated.
  - > The moving direction does not correspond to the sign of the analog voltage.
  - > The actual X axis feedback pulses are not between 50% and 200% of the theoretical ones within the time window set by machine parameter P12.
- 17 Emergency.
- 18 X axis feedback alarm.
- 19 X axis travel limit overrun.
- 20 X axis following error.
- 21 Probing Error.
- 22 Features not available at this CNC model have been defined, such as:
  - > Non-servo-controlled Open Positioning Loop on 101S and 102S models.
  - > External operator panel "JOG 100" on 101 nd 102 models.

There is a certain incompatibility in the machine parameters such as:

- > Fast and Slow signals when operating in Closed Positioning Loop.
- > Rotary ROLLOVER axis when operating in Open Positioning Loop.
- > Rotary HIRTH axis or following the shortest way while not being a ROLLOVER axis.

- 23 Too large a value assigned to an arithmetic parameter in parametric programming.
- 24 Arithmetic division by 0.
- 25 Square root of a negative figure.
- 26 Parametric block edited wrong.
- 27 A tool too large has been selected. For example, when assigning an arithmetic parameter value greater than 10 to a "T" function.
- 28 More than 4 "M" functions have been programmed in a block or 4 "M" functions in a block containing an "S" function which involves a gear (range) change (M41,M42,M43 or M44).
- 29 There are no selected spindle ranges (P36, P37, P38 and P39=0), or the "S", spindle speed, value is too high.
- 30 Value of machine parameter P24 or P25 greater than that of P27.
- 31 Value of machine parameter P64 or P65 greater than that of P67.
- 32 Y axis machine parameter error:
- > The braking distance (P43) is smaller than the In-position zone (P68).
  - > The P43 value, is greater than 8388.607 mm or 330.259 inches.
- Y axis execution error:
- > The movement cannot be executed since, due to the programmed feedrate, the braking distance is smaller than the In-Position zone (Open Positioning Loop).
- 33 RS 232 C transmission error.
- 34 Too long a block transmitted via RS 232 C.
- 35 Wrong block in the RS 232 C transmission.
- 36 A block has been program which forces the Y axis to overrun its travel limits.
- 37 Internal temperature limit exceeded.
- 38 Y axis not controlled:
- > The Y axis has shifted a distance 16 times the in-position zone once the brake has been activated.
  - > The moving direction does not correspond to the sign of the analog voltage.
  - > The actual Y axis feedback pulses are not between 50% and 200% of the theoretical ones within the time window set by machine parameter P12.
- 39 Y axis feedback alarm.
- 40 Y axis travel limit overrun.
- 41 Y axis following error.
- 42 The programmed circle does not go through the defined end point (tolerance 0.01 mm).
- 43 The circular interpolation is missing the I, J values or has been defined wrong.
- 44 A wrong block has been programmed, either in itself or with respect to the program history up to that point.
- 45 A function has been programmed which is not permitted by machine parameters.
- 46 Tool checksum error.
- 47 LAN error, It is not working.
- 48 Wrong command, the destination node cannot interpret it.
- 49 Misconfigured LAN, a certain node has not been accepted in the LAN.
- 50 The destination node cannot attend to the command (in execution).
- 51 The axes must be referenced (homed) after power-up.

- 52 While monitoring the "double feedback", the actual deviation (slippage) has exceeded its maximum value set by machine parameter P110.
- 101 DNC error. Wrong command.
- 102 DNC error. Error when reading a key code.
- 103 DNC error. An incorrect block has been sent.
- 104 DNC error. Error when reading the number of the first block of the active program.
- 105 DNC error. Incorrect block number.
- 106 DNC error. The CNC is in execution
- 107 DNC error. Locked key.
- 108 DNC error. Wrong key code.
- 109 DNC error. Negative coordinate for a rollover axis.
- 110 DNC error. Wrong arithmetic parameter number.
- 111 DNC error. Arithmetic parameter exponent too large.
- 112 DNC error. Wrong arithmetic parameter value.
- 113 DNC error. When requesting the contents of several blocks (RN code), the first block number is greater than the last block number. For example: RN100.50
- 114 DNC error. The CNC machine parameters are locked.
- 115 DNC error. The CNC part-program memory is protected.
- 116 DNC error. A block has been sent which contains too many characters. The maximum is 80 characters when using blank spaces as separators and 45 when otherwise.
- 117 DNC error. A program block has been sent in disagreement with the machine parameters.