8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

Installation manual (M & T models)

Ref. 0204-ing

INDEX

4. Concepts

5. Introduction to the PLC

6. PLC Resources

7. PLC Programming

8. CNC-PLC Communication

9. Logic CNC inputs and outputs

10. Access to the internal CNC variables

11. Axes controlled from the PLC

12. Customizable screens

13. Configurable work mode

14. PLC programming example

Appendix

8040 CNC

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

INTRODUCTION

0

0. INTRODUCTION

0.1 About this manual

Title Installation and Setup manual.

Type of documentation

Describes how to install and set the CNC up.

Version Corresponds to software versions:
5.2x for milling and 6.2x for turning (lathe).

Abbreviations used in this manual

	meaning	example
g.m.p.	general machine parameter	g.m.p. CUSTOMY (P92)
a.m.p.	axis machine parameter	a.m.p. LOOPCHG (P26)
s.m.p.	spindle machine parameter	s.m.p. MAXVOLT (P37)
plc.m.p.	PLC machine parameter	plc.m.p IOCANSPE (P88)

Start-up

Check that machine carrying this CNC meets the 89/392/CEE ruling.

Before starting this CNC up, read the instructions described in this chapter ***2. Machine and Power connection***

Warning

The information described in this manual may be modified for technical reasons.

FAGOR AUTOMATION S. COOP. Reserves the right to modify the contents of this manual without having to communicate such modifications.

Headquarters

Fagor Automation, S. Coop.
Bº San Andrés s/n, Apdo. 144
E-20500 Arrasate-Mondragón
www.fagorautomation.com
info@fagorautomation.es
Telephone 34-943-719200
+34 943 771118 (Technical Service Department)

0.2 About the product

Basic characteristics:

Block processing time without CPU turbo	12 ms
RAM memory	256 Kb expandable to 1Mb
Memkey Card memory	512 Kb expandable to 2 Mb
Minimum position loop	4 ms

Hardware options:

RS-232 serial line.	Standard
16 digital inputs and 8 outputs (I1 to I16 and O1 to O8)	Standard
Another 40 digital inputs and 24 outputs (I65 to I104 and O33 to O56)	Option
Probe inputs	Standard
Spindle (feedback input and analog output)	Standard
Electronic handwheels	Standard
4 axes (feedback and analog voltage)	Option
CAN for digital I/O expansion (RIO).	Option

Software options:

	Model						
	GP	M	MC	MCO	T	TC	TCO
Number of axes with standard software	4	4	4	4	2	2	2
Number of axes with optional software	-----	-----	-----	-----	4	4	4
Electronic threading	-----	Stand	Stand	Stand	Stand	Stand	Stand
Tool magazine management:	-----	Stand	Stand	Stand	Stand	Stand	Stand
Machining canned cycles	-----	Stand	Stand	-----	Stand	Stand	-----
Multiple machining	-----	Stand	Stand	-----	-----	-----	-----
Solid graphics	-----	Opt	Opt	Opt	Stand	Stand	Opt
Rigid tapping	Opt	Opt	Opt	Opt	Opt	Opt	Opt
Tool live monitoring	-----	Opt	Opt	Opt	Opt	Opt	Opt
Probing canned cycles	-----	Opt	Opt	Opt	Opt	Opt	Opt
DNC	Opt	Opt	Opt	Opt	Opt	Opt	Opt
COCOM version	Opt	Opt	Opt	Opt	Opt	Opt	Opt
Profile editor	Opt	Opt	Stand	Opt	Opt	Stand	Opt
Tool radius compensation	Opt	Stand	Stand	Opt	Stand	Stand	Stand
Tangential control	Opt	Opt	Opt	Opt	Opt	Opt	Opt
Retracing	-----	Opt	Opt	Opt	Opt	Opt	Opt
Setup assistance	-----	Opt	Opt	Opt	Opt	Opt	Opt
Irregular pockets with islands	-----	Opt	Stand	Opt	-----	-----	-----
1M RAM - 2M Flash	Opt	Opt	Opt	Opt	Opt	Opt	Opt

0.3 Version history. M model

April 2002
Software: 5.3x.

Feature	Manual
New expansion board models	Installation Programming
Bus CAN OPEN to control remote digital I/Os	Installation Operation Error solutions
New PLC instructions: IREMRD and OREMWR:	Installation
Leadscrew error compensation on rotary axes between 0-360 degrees	Installation
PLC statistic deletion with a single softkey	Operation
Show only the XY plane in top-view graphics	Operation
Absolute reference mark management via Sercos (see a.m.p. REFVALUE)	Installation

0.4 Version history. T model

April 2002
Software: 6.3x.

Feature	Manual
New expansion board models	Installation Programming
Bus CAN OPEN to control remote digital I/Os	Installation Operation Error solutions
New PLC instructions: IREMRD and OREMWR:	Installation
Leadscrew error compensation on rotary axes between 0-360 degrees	Installation
PLC statistic deletion with a single softkey	Operation
Absolute reference mark management via Sercos (see a.m.p. REFVALUE)	Installation

0.5 Declaration of conformity

Manufacturer:

Fagor Automation, S. Coop.

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

We declare

under our exclusive responsibility the conformity of:

Numerical Control Fagor
8040 CNC

referred to by this declaration with following directives:

Safety:

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

Electromagnetic compatibility

EN 50081-2	Emission.
EN 55011	Radiated. Class A, Group 1.
EN 50082-2	Immunity.
EN 61000-4-2	Electrostatic Discharges.
EN 61000-4-3	Radiofrequency Radiated Electromagnetic Fields.
EN 61000-4-4	Bursts and fast transients.
EN 61000-4-6	Conducted disturbance induced by radio frequency fields.
EN 61000-4-11	Voltage fluctuations and Outages.
ENV 50204	Fields generated by digital radio-telephones

As instructed by the European Community Directives: 73/23/CEE on low voltage, 89/392/CEE on machine safety, 89/336/CEE on electromagnetic compatibility and their updates.

In Mondragón, on February 1st, 2001

Fagor Automation, S. Coop. Ltda.
Director Gerente

Fdo.: Julen Busturia

0.6 Safety conditions

Read the following safety measures in order to prevent damage to personnel, to this product and to those products connected to it.

This unit must only be repaired by personnel authorized by Fagor Automation.

Fagor Automation shall not be held responsible for any physical or material damage derived from the violation of these basic safety regulations.

Precautions against personal damage

- Interconnection of modules

  Use the connection cables provided with the unit.

- Use proper Mains AC power cables

  To avoid risks, use only the Mains AC cables recommended for this unit.

- Avoid electrical overloads

  In order to avoid electrical discharges and fire hazards, do not apply electrical voltage outside the range selected on the rear panel of the Central Unit.

- Ground connection.

  In order to avoid electrical discharges, connect the ground terminals of all the modules to the main ground terminal. Before connecting the inputs and outputs of this unit, make sure that all the grounding connections are properly made.

- Before powering the unit up, make sure that it is connected to ground

  In order to avoid electrical discharges, make sure that all the grounding connections are properly made.

- Do not work in humid environments

  In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45º C (113º F).

- Do not work in explosive environments

  In order to avoid risks, damage, do no work in explosive environments.

Precautions against product damage

- Working environment

  This unit is ready to be used in Industrial Environments complying with the directives and regulations effective in the European Community

  Fagor Automation shall not be held responsible for any damage suffered or caused when installed in other environments (residential or homes).

- Install this unit in the proper place

  It is recommended to install the CNC away from coolants, chemical products, possible blows etc. which could damage it.

  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 decoupled all those elements capable of generating interference (relay coils, contactors, motors, etc.)

  - DC relay coils. Diode type 1N4000.
  - AC relay coils. RC connected as close as possible to the coils. Their approximate values should be: R 220Ω/1W and C 0.2µF/600V.
  - AC motors. RC connected between phases with values: R 300Ω/6W and C 0.47µF/600V

- 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. Machine and Power connection***

- Ambient conditions

  The working temperature must be between +5° C and +40° C (41ºF and 104º F)

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

- Central Unit (CPU) enclosure

  Guarantee the required gaps between the central unit and each wall of the enclosure. See chapter ***1. 8040 CNC configuration***

  Use a DC fan to improve enclosure ventilation.

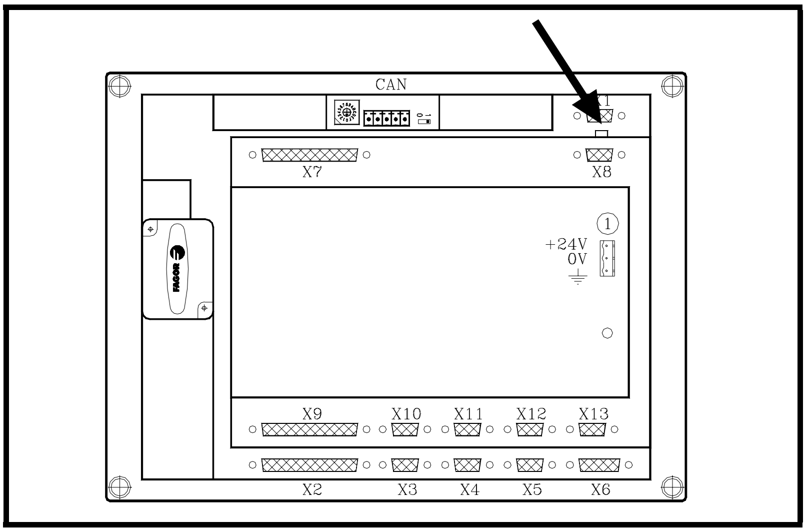
- Power switch

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

Protections of the unit itself

- Central Unit

It has a 4 Amp /250V external fast fuse (F).



- Inputs-Outputs

  All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.

Precautions during repair

Do not open this unit

- *Only personnel authorized by Fagor Automation may open this module.*

Do not handle the connectors with the unit connected to main 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

Danger or forbidding symbols.

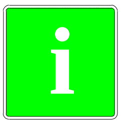
It indicates actions or operations that may cause damage to people or to units.

Warning or caution symbol.

It indicates situations that may be caused by certain operations and the actions to be taken to prevent them.

Obligation symbol.

It indicates actions and operations that must be carried out.



Warning symbol.

It indicates notes, warnings and suggestions.

0.7 Warranty terms

All products manufactured or marketed by Fagor Automation has a warranty period of 12 months from the day they are shipped out of our warehouses.

The mentioned warranty covers repair material and labor costs, at Fagor facilities, incurred in the repair of the products.

Within the warranty period, Fagor will repair or replace the products verified as being defective.

Fagor is committed to repairing or replacing its products from the time when the first such product was launched up to 8 years after such product has disappeared from the product catalog.

It is entirely up to Fagor to determine whether a repair is to be considered under warranty.

Excluding clauses

The repair will take place at our facilities. Therefore, all shipping expenses as well as travelling expenses incurred by technical personnel are NOT under warranty even when the unit is under warranty.

This warranty will be applied so long as the equipment has been installed according to the instructions, it has not been mistreated or damaged by accident or negligence and has been manipulated by personnel authorized by Fagor.

If once the service call or repair has been completed, the cause of the failure is not to be blamed the FAGOR product, the customer must cover all generated expenses according to current fees.

No other implicit or explicit warranty is covered and Fagor Automation shall not be held responsible, under any circumstances, of the damage which could be originated.

Service agreements

Service and Maintenance Contracts are available for the customer within the warranty period as well as outside of it.

0.8 Material returning terms

When returning the Remote modules or the Central Unit, pack it in its original package and with its original packaging material. If not available, pack it as follows:

1. Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg. (375 pounds).
2. Attach the unit label indicating the owner of the unit, his/her address, the name of the contact person, the type of unit and the serial number.

   In case of failure, also indicate the symptom and a short description.
3. Wrap the unit in a polyethylene roll or similar material to protect it.

   When sending the central unit, above all protect the screen
4. Pad the unit inside the cardboard box with poly-utherane foam on all sides.
5. Seal the cardboard box with packing tape or industrial staples.

0.9 Additional remarks

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

Before turning the unit on, verify that the ground connections have been properly made. See chapter *2. Machine and Power connection*

In case of a malfunction or failure, disconnect it and call the technical service. Do not open this unit.

0.10 Fagor documentation for the CNC

OEM Manual	It is directed to the machine builder or person in charge of installing and starting-up the CNC.
USER-M manual	Directed to the end user. It describes how to operate and program in M mode.
USER-T manual	Directed to the end user. It describes how to operate and program in T mode.
MC Manual	Directed to the end user. It describes how to operate and program in MC mode. It contains a self-teaching manual.
TC Manual	Directed to the end user. It describes how to operate and program in TC mode. It contains a self-teaching manual.
MCO/TCO model	Directed to the end user. It describes how to operate and program in MCO and TCO mode.
Examples-M manual	Directed to the end user. It contains programming examples for the M mode.
Examples-T manual	Directed to the end user. It contains programming examples for the T mode.
Floppy Disk manual	Directed to people who use the Fagor floppy disk unit. It indicates how to connect and use that floppy disk unit.
WINDNC Manual	It is directed to people using the optional DNC communications software. It is supplied in a floppy disk with the application.
WGDRAW Manual	Directed to people who use the WGDRAW to create screens. It is supplied in a floppy disk with the application.

User notes:

8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

8040 CNC CONFIGURATION

1

1. 8040 CNC CONFIGURATION

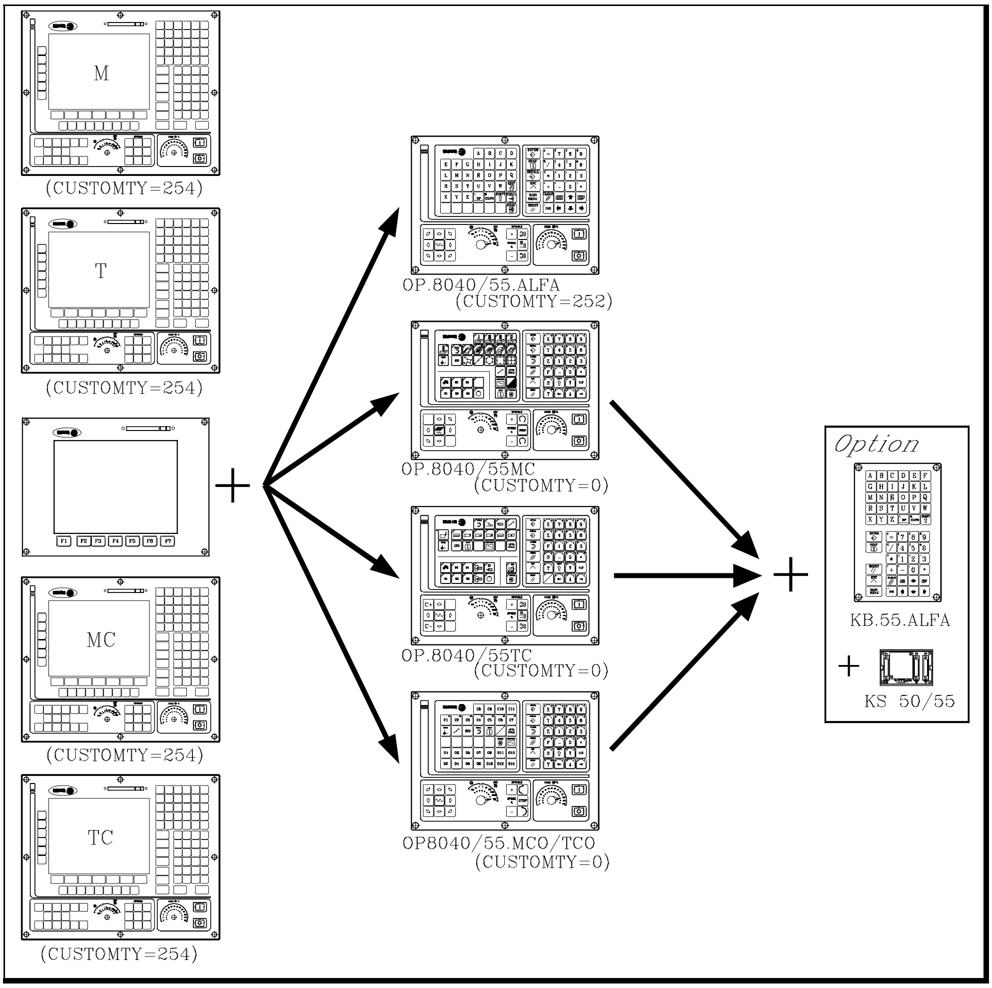
The CNC is prepared to be used in Industrial Environments, especially on milling machines, lathes, etc.

It can control machine movements and devices.

1.1 CNC structure.

The central unit (cpu) is on the back of the monitor.

The monitor may be either integrated into the operator panel or separate from it.



Operator panels with integrated monitor

CNC 8040-M-COL-K	CNC 8040-T-COL-K	CNC 8040-MC-COL-K	CNC 8040-TC-COL-K
CNC 8040-M-MON-K	CNC 8040-T-MON-K	CNC 8040-MC-MON-K	CNC 8040-TC-MON-K

Independent monitors

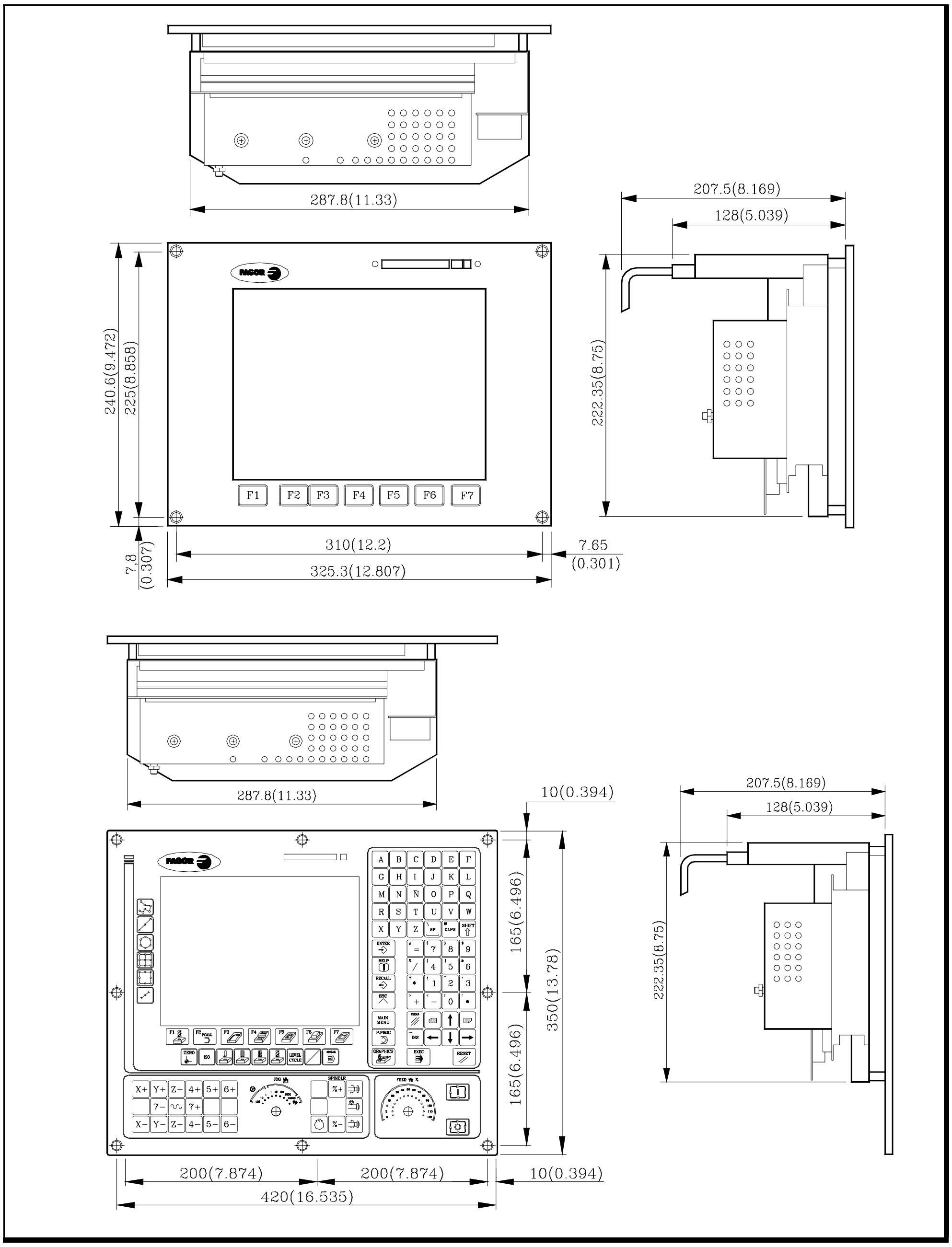
CNC 8040-M-COL	CNC 8040-T-COL
CNC 8040-M-MON	CNC 8040-T-MON

Independent operator panels

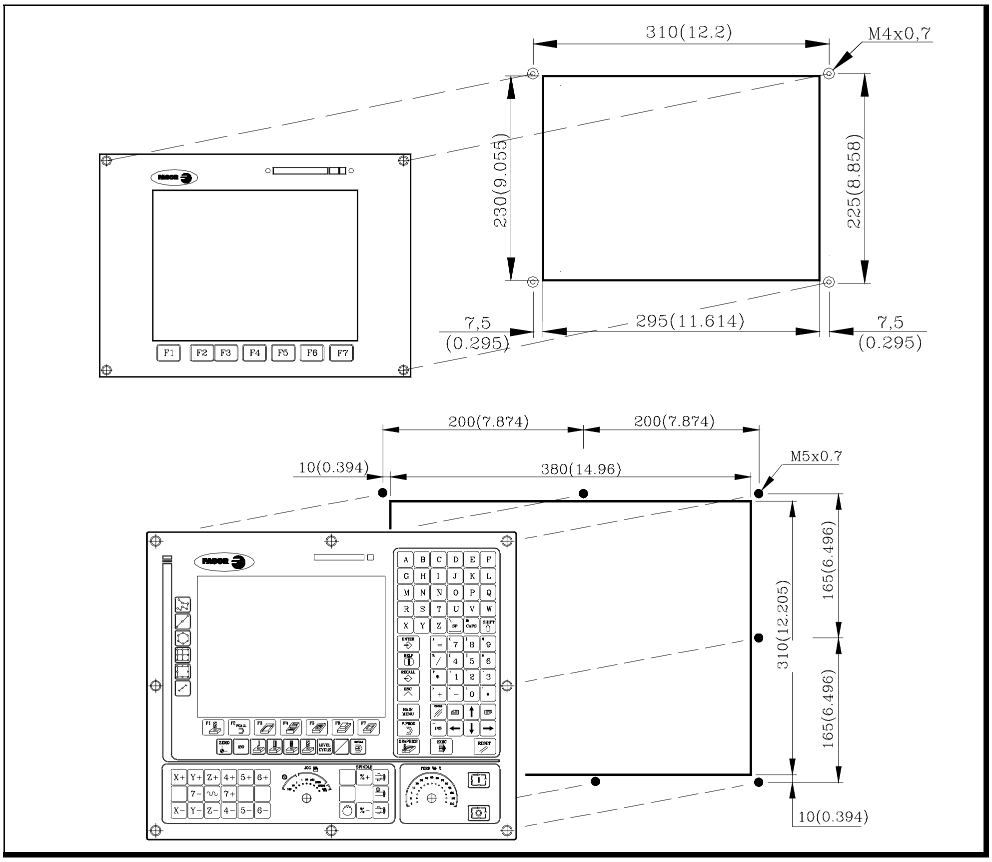
M model	Operator panel: OP.8040/55.ALFA
T model	Operator panel: OP.8040/55.ALFA
MC model	Operator panel: OP.8040/55.MC
TC model	Operator panel: OP.8040/55.TC
MCO/TCO model	Operator panel: OP.8040/55.MCO/TCO

MC, TC and MCO/TCO operator panels can have an optional alphanumeric keyboard where each key has a single letter or number assigned to it (KB55.ALFA). This keyboard is connected to the operator panel through the KS50/55 adapter.

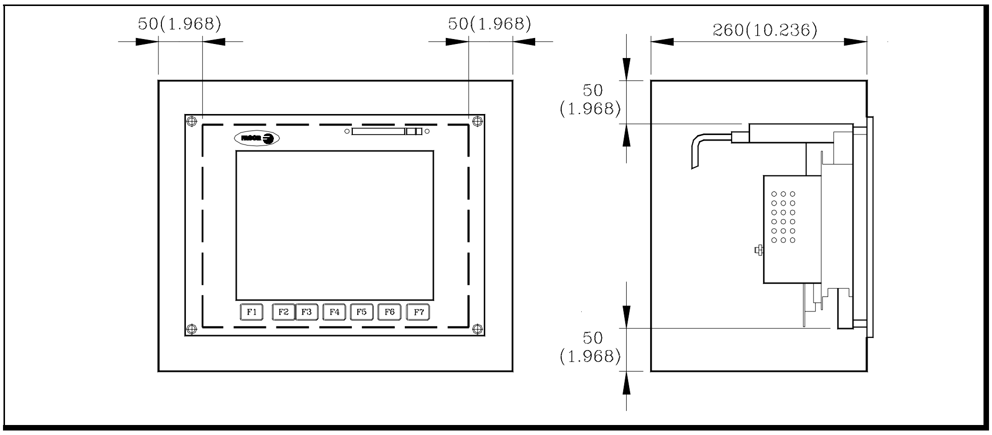
Dimensions.



Enclosure:



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



It is up to the installer to make sure that the enclosure has forced ventilation or ventilation grooves in order to prevent the inside temperature to exceed the specified ambient temperature.

Between 5º C and +50º C (41º F and 122º F)

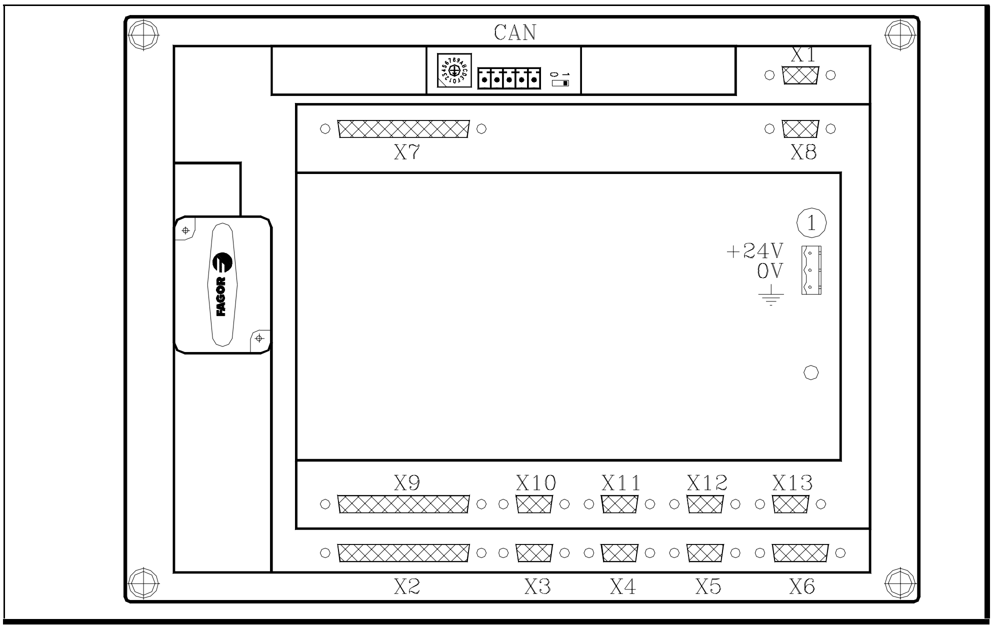
Relative humidity between 5% and 95% non condensing

When using a fan to better ventilate the enclosure, a DC fan must be used since an AC fan may generate electromagnetic interference resulting in distorted images being displayed by the CRT.

Brightness and contrast may be adjusted on monochrome monitors. See the Operation manual, chapter on Diagnosis, section on Hardware configuration.

1.1.1 Connectors

They are located in the rear.



(1) Power supply

X1. For serial communication line RS232.

X2. For digital I/O connection (I1 to I16 and O1 to O8)

X3. For probe connection

X4. For analog spindle connection

X5. For electronic handwheel connection

X6. For Operator Panel connection

X7. For digital I/O connection (I97 to I104 and O33 to O56)

X8. For axis analog voltage connection

X9. For digital I/O connection (I65 to I96)

X10. For feedback connection of the first axis

X11. For feedback connection of the second axis

X12. For feedback connection of the third axis

X13. For feedback connection of the fourth axis

CAN Remote modules for digital I/O expansion.

Warning:

Do not open this unit

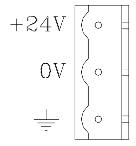
- *Only personnel authorized by Fagor Automation may open this module.*

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

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

The machine manufacturer must comply with the EN 60204-1 (IEC-204-1) standard in terms of protection against electrical shock due to faulty I/O contacts with external power supply.

Power supply 3-prong male Phoenix connector, 7.65 mm pitch

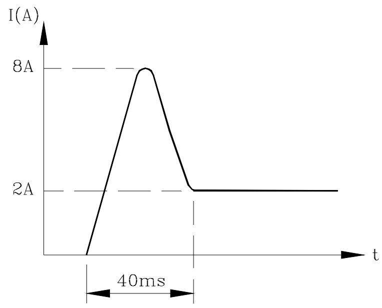
	Pin	Signal and Function	
	1 2 3	+24V 0V Chassis	Power supply Power supply Shield

Use an independent external power supply with the following specifications:

Nominal voltage:	20 V minimum 30V maximum
Ripple:	4V
Nominal current:	2A
Current peak on power-up:	8A

The central unit has a protection against overvoltage that activates at 36 V.

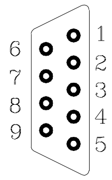
The supply current has the following shape on power-up



Connector X1 (RS232)

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

The cable shield must be connected to the metallic hood at each end.



Pin	Signal
1	DCD
2	RxD
3	TxD
4	DTR
5	GND ISO
6	-----
7	RTS
8	CTS
9	+5V ISO

All the pins of this connector are opto-isolated.

Cable length

EIA RS232C standards specify that the capacitance of the cable must not exceed 2500pF; therefore, since average cables have a capacitance between 130pF and 170pF per meter, the maximum length of the cable should not be greater than 15m (49ft).

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

Use shielded 7 conductor cable of 0.14 mm² section.

Transmission speed (baudrate)

The CNC can operate at up to 115,200 Baud.

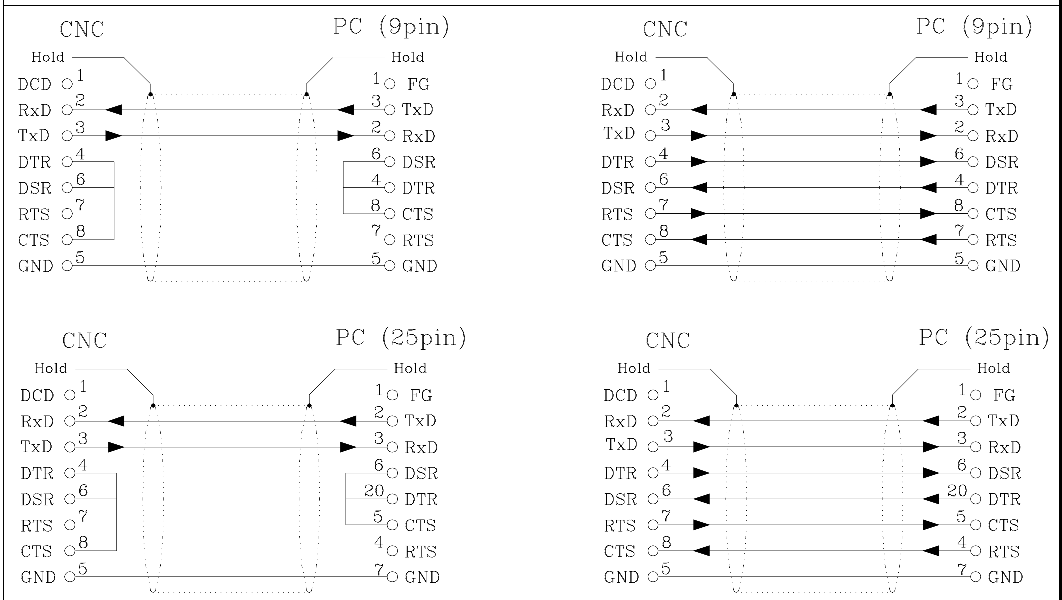
It is recommended to ground the unused pins in order to avoid erroneous control and data signal interpretations.

Ground connection

It is suggested to reference all control and data signals to the same ground cable (pin 7 GND) thus, avoiding reference points at different voltages especially in long cables.

Recommended RS232C interface connection

Simplified connection	Full connection.



Connectors X2, X7 and X9

Digital inputs and outputs

X2	Inputs I1 to I16 and outputs O1 to O8
X9.	Inputs I65 to I96
X7.	Inputs I97 to I104 and outputs O33 to O56

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

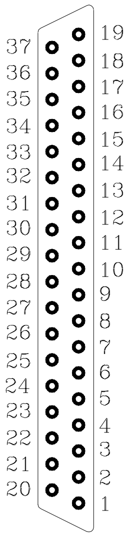
All shields must only be connected to ground at the CNC end through pin 37 of the connector leaving the other end free. The wires of the shielded cables cannot be unshielded for more than 75mm (about 3 inches).

Warning:

Since the response time of the EMERGENCY signal must be very short, the CNC has assigned input I01 for this purpose. Thus, the CNC will treat this input immediately regardless of how the PLC program uses it.

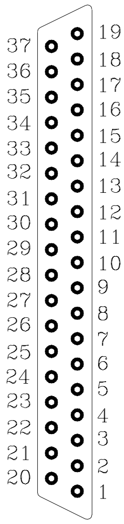
The Emergency output, which coincides with O1 of the PLC, will be activated (change from logic level 1 to 0) when an ALARM or ERROR occurs at the CNC or when the PLC output O1 is set to 0 (logic level 0).

Connector X2, 37-pin normal density SUB-D type female connector.



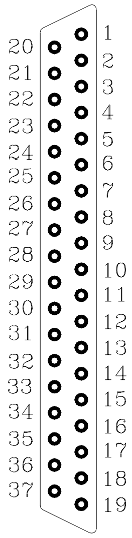
Pin	Signal and Function	
1	24V	External power supply
2	O 1	/ Emergencia output
3	O 3	
4	O 5	
5	O 7	
6	----	
7	----	
8	----	
9	----	
10	I 1	
11	I 3	
12	I 5	
13	I 7	
14	I 9	
15	I11	
16	I13	
17	I15	
18	0 V	External power supply
19	0 V	External power supply
20	24V	External power supply
21	O 2	
22	O 4	
23	O 6	
24	O 8	
25	----	
26	----	
27	----	
28	----	
29	I 2	
30	I 4	
31	I 6	
32	I 8	
33	I10	
34	I12	
35	I14	
36	I16	
37	Chassis	Shield

Connector X7, 37-pin normal density SUB-D type female connector.



Pin	Signal	Function
1	24V	External power supply
2	O33	
3	O35	
4	O37	
5	O39	
6	O41	
7	O43	
8	O45	
9	O47	
10	O49	
11	O51	
12	O53	
13	O55	
14	I97	
15	I99	
16	I101	
17	I103	
18	0 V	External power supply
19	0 V	External power supply
20	24V	External power supply
21	O34	
22	O36	
23	O38	
24	O40	
25	O42	
26	O44	
27	O46	
28	O48	
29	O50	
30	O52	
31	O54	
32	O56	
33	I98	
34	I100	
35	I102	
36	I104	
37	Chassis	Shield

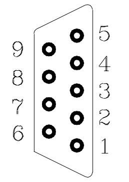
Connector X9, 37-pin normal density SUB-D type male connector.



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

Connector X3 For probe connection

9-pin normal density SUB-D type female connector.



Pin	Signal and Function	
1	Chassis	Shield
2	+5 V	+5 V output for the probe
3	Palp1_5	Probe 1. 5V TTL input
4	Palp1_24	Probe 1. 24V input
5	GND	0V probe input.
6	+5 V	+5 V output for the probe
7	Palp2_5	Probe 2. 5V TTL input
8	Palp2_24	Probe 2. 24V input
9	GND	0V probe input.

2 probes may be connected, each has 2 capture inputs (5V and 24V).

For further information on the characteristics of the probe input, see appendix ***Technical specifications of the CNC***

To see the recommended connection circuits, see appendix ***Probe connection***

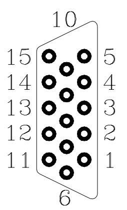
All shields must only be connected to ground at the CNC end through pin 1 of the connector leaving the other end free. The wires of the shielded cables cannot be unshielded for more than 75mm (about 3 inches).

Warning:

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

Connector X4 For analog spindle connection

15-pin high density SUB-D type female connector.

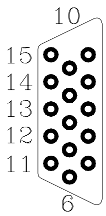


Pin	Signal and Function	
1	Ac	Feedback signals
2	/ Ac	
3	Bc	
4	/ Bc	
5	I0c	
6	/ I0c	
7	ALc	
8	/ ALc	
9	+5 V	+5V output for feedback
10	ConsCab	Analog voltage output
11	GND	0V output for analog voltage or feedback
12	GND	0V output for analog voltage or feedback
13	----	
14	----	
15	Chassis	Shield

The cable shield must be connected to the metallic hood at each end.

Connector X5 For electronic handwheel connection

15-pin high density SUB-D type female connector.



Pin	Signal and Function	
1	A1	Feedback signals of first handwheel
2	/ A1	
3	B1	
4	/ B1	
5	A2	Feedback signals of second handwheel
6	/ A2	
7	B2	
8	/ B2	
9	+5 V	Supply output
10	+5 V	Supply output
11	GND	Supply output
12	GND	Supply output
13	100P	Push button of Fagor 100P handwheel
14	----	
15	Chassis	Shield

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

The cable shield must be connected to the metallic hood at each end.

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

For further information on the characteristics of the feedback inputs (square and sinusoidal signals), see appendix ***Technical specifications of the CNC***

When using a FAGOR 100P model handwheel, connect it as first handwheel and connect the axis selecting signal (button) to pin 13.

Connector X6 For Operator Panel connection

26-pin high density SUB-D type female connector.

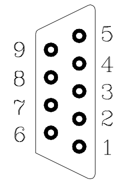
FAGOR AUTOMATION provides the cable necessary for this connection. This cable has two 26-pin male connectors of the high density SUB-D type.

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

It is a straight connection, 1 to 1, 2 to 2, 3 to 3 and so on. The cable hose shield is soldered to the metal hoods covering both connectors.

Connector X8 — For axis analog voltage connection

9-pin normal density SUB-D type female connector.



Pin	Signal and Function	
1	Chassis	Shield
2	Analog 1	Analog output for the first axis
3	Analog 2	Analog output for the second axis
4	Analog 3	Analog output for the third axis
5	Analog 4	Analog output for the fourth axis
6	GND	Analog voltage reference signals
7	GND	
8	GND	
9	GND	

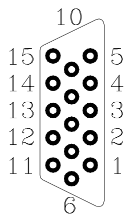
The cable shield must be connected to the metallic hood at each end.

The axis nomenclature is set when setting g.m.p. AXIS1 (P0) to AXIS4 (P3).

Connectors X10, X11, X12, X13 — Feedback inputs for the axes

X10. For feedback connection of the first axis
X11 For feedback connection of the second axis
X12 For feedback connection of the third axis
X13 For feedback connection of the fourth axis

15-pin high density SUB-D type female connectors.



Pin	Signal and Function	
1	A	Feedback signals
2	/ A	
3	B	
4	/ B	
5	I0	
6	/I0	
7	AL	
8	/ AL	
9	+5 V	Supply of the feedback system
10	+5 V	
11	GND	
12	GND	
13	----	
14	----	
15	Chassis	Shield

The cable shield must be connected to the metallic hood at each end.

CAN connector

Digital I/O expansion

The CAN connection allows connecting up to 4 remote modules to the the Central Unit to expand the number of digital inputs and outputs

Depending on the elements used, each remote module may have:

- 24 inputs and 16 outputs
- 48 inputs and 32 outputs
- 72 inputs and 48 outputs

The 16-position rotary switch (0-15) indicates the address (device select) occupied by each element integrated in the CAN bus.

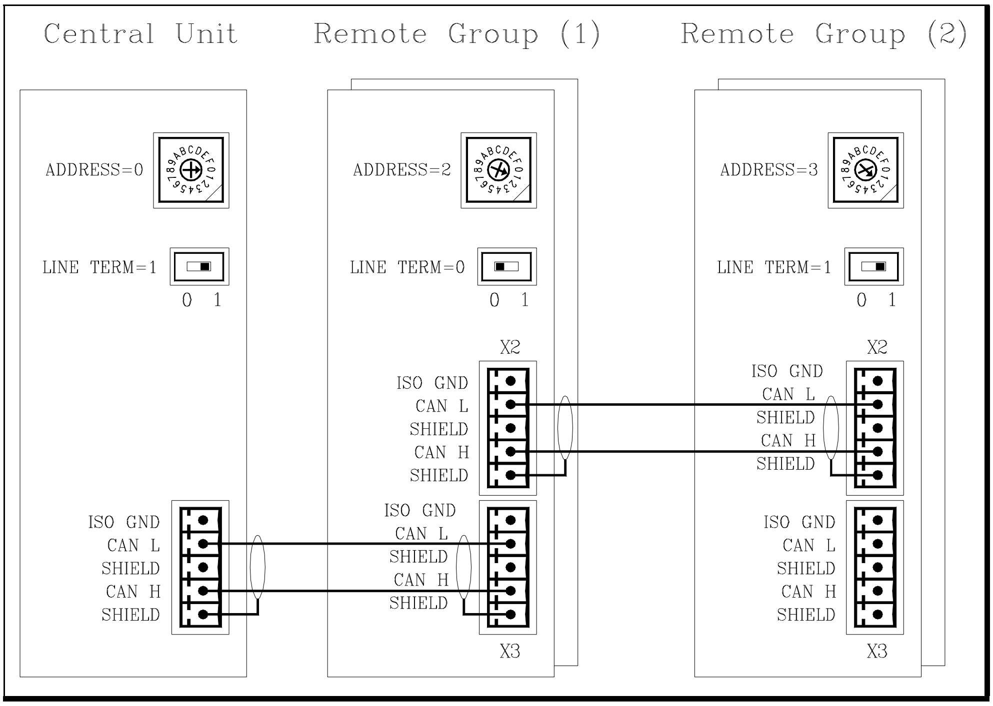
The CNC must always occupy the “0” position and the rest will occupy the consecutive positions starting from “1”.

The corresponding module must be turned off and back on (or press the Reset button) for the address change to be assumed.

They are connected in series using any of the two connectors. Use a specific CAN cable: twisted pair with a section of 0.22 mm with overall shield and an impedance of 120 Ω.

The elements at the end must have the “line termination” switch in position “1”, the rest in position “0”.

The figure shows the CAN connection between the Central Unit and 2 groups of remote modules.



The transmission speed depends on the length of the cable or total CAN connection distance. PLC parameter IOCANSPE (P88) must be set.

1.1.2 Remote modules

They are laid out in groups. Each one may have 1 or 2 elements. The available elements are:

Power Supply

It must be part of all the configurations.

It must be supplied with 24 Vdc and connected to the system's CAN bus.

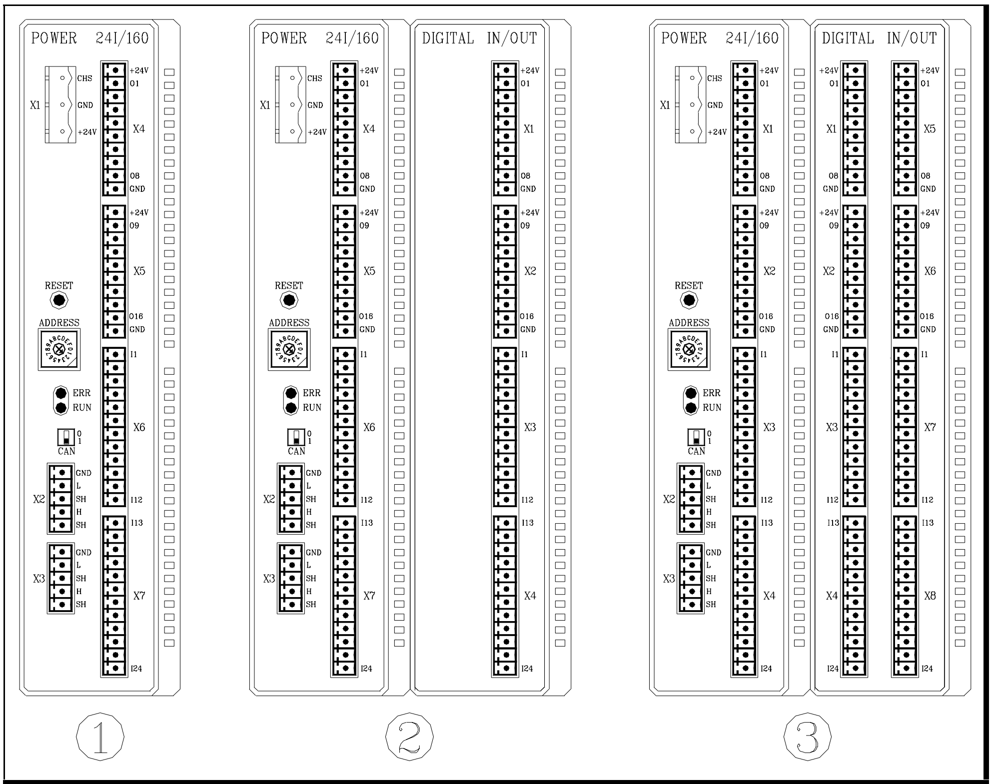
Single Input / Output element (digital IN/OUT)

It is used as I/O expansion. It has 24 digital inputs and 16 digital outputs

Double Input / Output element (digital IN/OUT)

It is used as I/O expansion. It has 48 digital inputs and 32 digital outputs

Mounting possibilities:



1. Power Supply
   24 inputs and 16 outputs
2. Power supply + single I/O
   48 inputs and 32 outputs
3. Power supply + double I/O
   72 inputs and 48 outputs

Electrical characteristics

All digital inputs (I) are protected with galvanic isolation using opto-couplers.

The electrical characteristics of the inputs are:

Nominal voltage:	+24 Vdc (between +18 Vdc and +30 Vdc)
High threshold (1)	from +18 Vdc up
Low threshold (0)	Below +9 Vdc
Typical consumption for each input	5 mA
Maximum consumption for each input	7 mA

All digital outputs (O) are protected with galvanic isolation using opto-couplers.

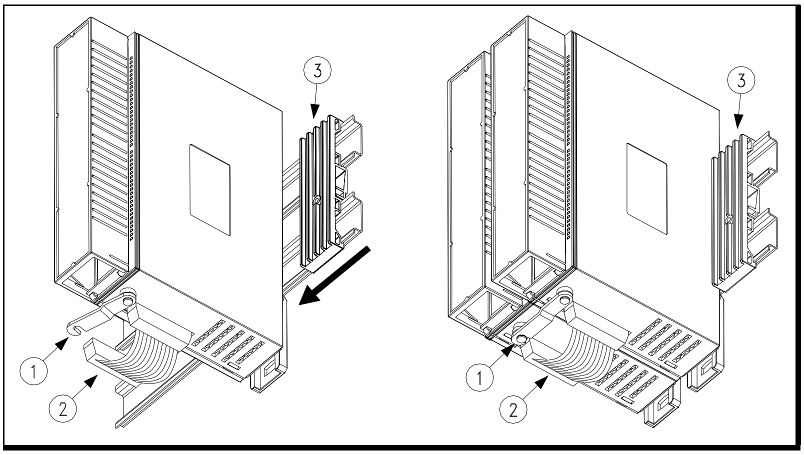
The electrical characteristics of the outputs are:

Nominal voltage:	+24 Vdc (between +18 Vdc and +30 Vdc)
Output voltage	2V less than supply Vdc
Maximum output current	500 mA per output

Digital outputs have a fuse inside to protect them against overvoltage (greater than 33 Vdc) and against reverse connection of the power supply.

Connection

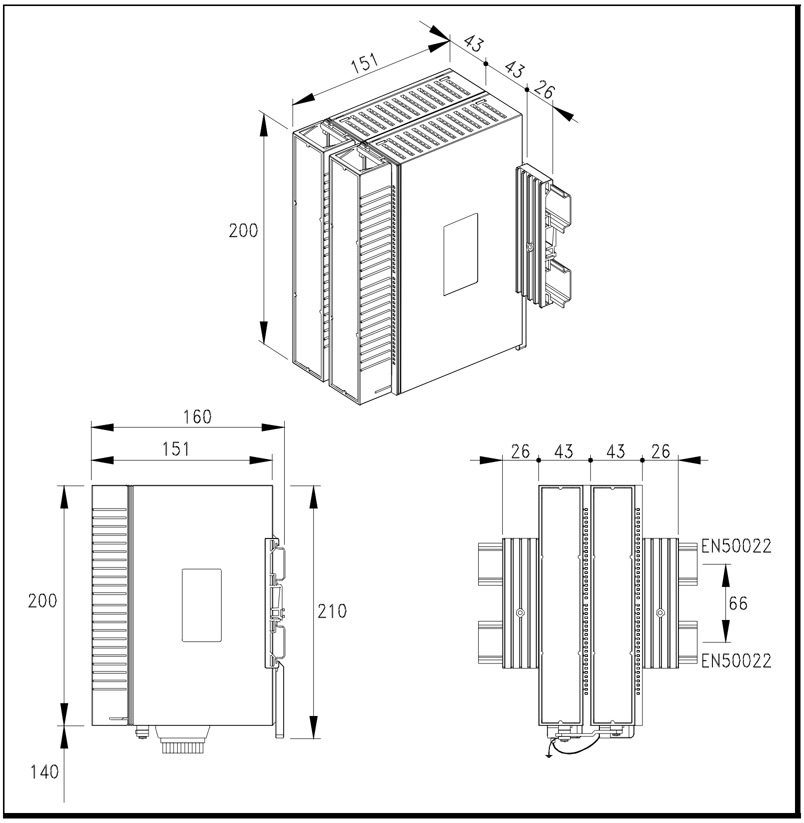
Place the modules on 2 profiles, according to UNE 50022 standard with two securing studs, one at each end of the group that besides maintaining the proper gap between profiles help securing the modules.



1. For ground connection
2. Ribbon cable for module interconnection.
3. Securing studs

Each group will be connected to the system (CPU, Keyboard, etc) through the CAN bus as described later on.

Dimensions



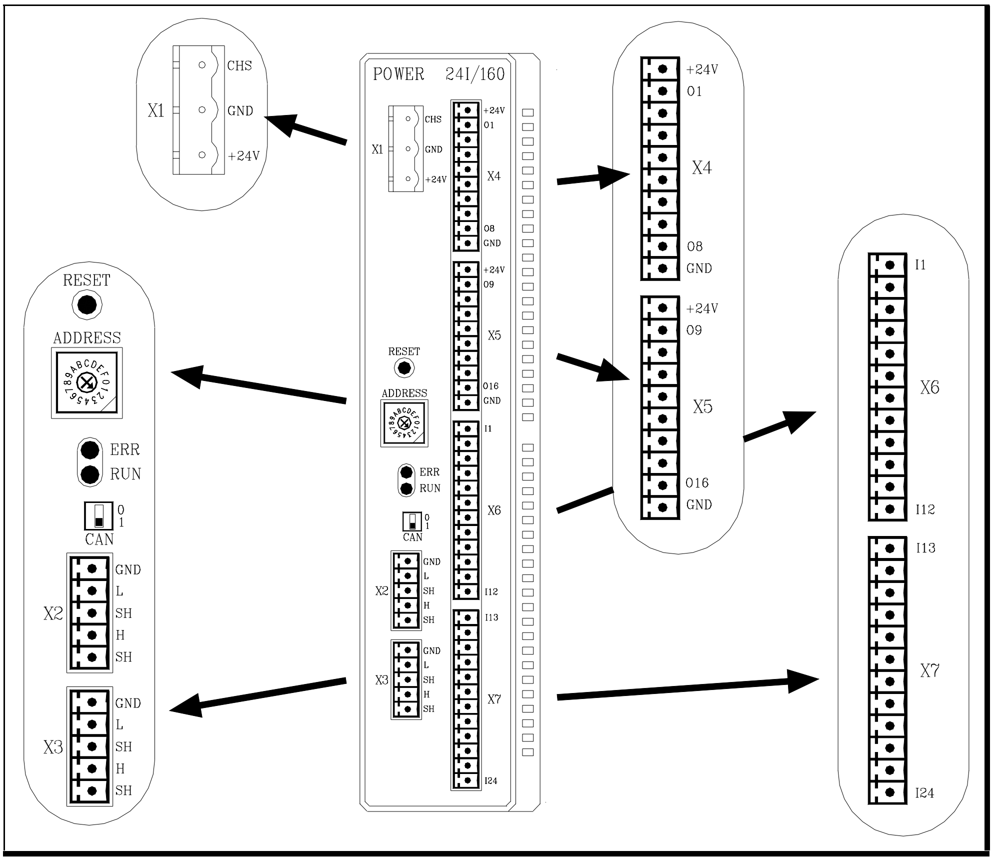
Always leave a 140 mm gap below the module for ventilation and future maintenance.

Power Supply

It must be part of all the configurations (1 per group).

It must be supplied with 24 Vdc and connected to the system's CAN bus.

It has 24 digital inputs and 16 digital outputs



X1	+24V / GND	Supplied at 24Vdc
X1	Chassis	Ground connection

Reset — It must be pressed when changing the logic address.

Address — 16-position switch (0-15) that indicates the CAN bus address ("0" not allowed).

ERR — Red lamp.

Off	The module works fine
Fast blinking	Module configuration stage
Simple Blinking	Warning. Poor transmission
Double blinking	No communication with the Central Unit
On	Error. Too many errors

RUN — Green lamp.

On	The module works fine
Simple Blinking	Module stopped
Fast blinking	Module configuration stage
Continuous blinking	Start-up stage or Error

CAN — Line termination switch The elements at the end of the CAN bus must have the "line termination" switch in position "1", the rest in position "0".

X2, X3 — Connectors for CAN bus connection.

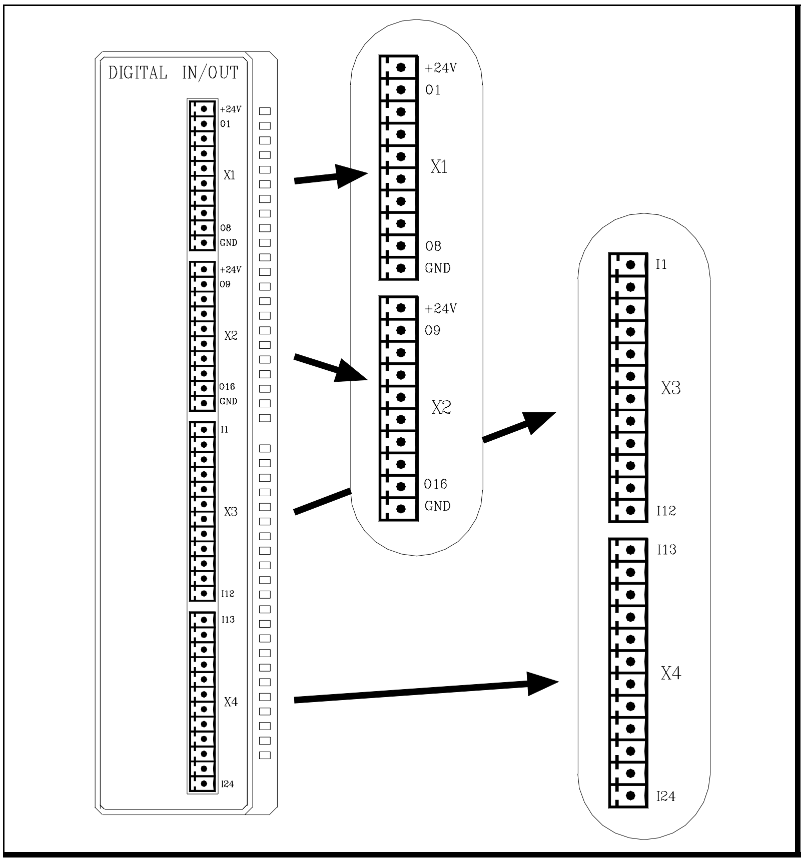
X4, X5 — Connectors for connecting 16 digital outputs. Both connectors must be supplied with +24V/GND

X6, X7 — Connectors for connecting 24 digital inputs.

Single I/O element

It is used together with the Power Supply element to expand inputs and outputs.

It has 24 digital inputs and 16 digital outputs

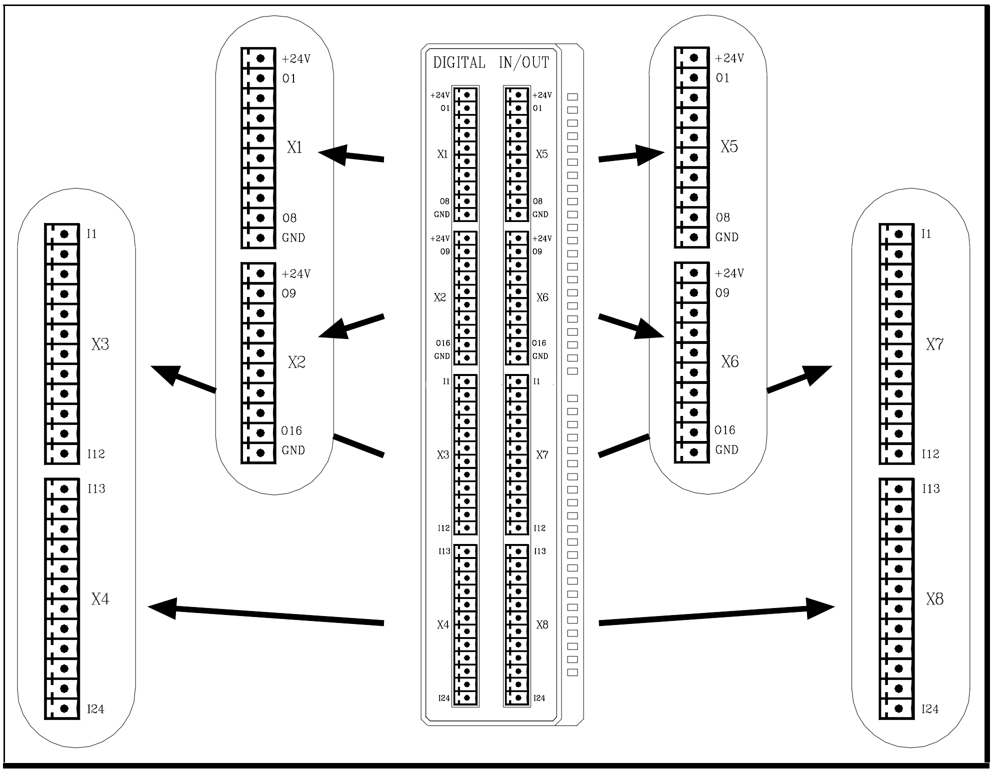


X1, X2	Connectors for connecting 16 digital outputs. Both connectors must be supplied with +24V/GND
X3, X4	Connectors for connecting 24 digital inputs.

Double I/O element

It is used together with the Power Supply element to expand inputs and outputs.

It has 48 digital inputs and 32 digital outputs



X1, X2, X5, X6	Connectors for connecting 16 digital outputs. All connectors must be supplied with +24V/GND
X3, X4, X7, X8	Connectors for connecting 24 digital inputs.

Remote modules setting

The following PLC machine parameters identify each one of the 4 possible remote modules.

IOCANID1 (P89)
IOCANID2 (P90)
IOCANID3 (P91)
IOCANID4 (P92)

They indicate which module each machine parameter group refers to.

Assign the CAN bus address that occupies the node (the one indicated by the address selector switch)

ICAN1 (P93)
OCAN1 (P94)
ICAN2 (P95)
OCAN2 (P96)
ICAN3 (P97)
OCAN3 (P98)
ICAN4 (P99)
OCAN4 (P100)

They indicate the configuration of each remote module, number of inputs (ICAN*) and outputs (OCAN*).

Example for a remote module located in node 1, with 48 inputs and 32 outputs:

IOCANID1=1 ICAN1=48 OCAN1=32

NUICAN1 (P101)
NUOCAN1 (P102)
NUICAN2 (P103)
NUOCAN2 (P104)
NUICAN3 (P105)
NUOCAN3 (P106)
NUICAN4 (P107)
NUOCAN4 (P108)

They are used to set the remote modules

NUICAN* indicates the number of the first input and NUOCAN* that of the first output of the group.

The Central Unit reserves local inputs I1 through I64 and local outputs O1 through O32. When having an expansion board that includes I/Os, it also reserves local inputs I65 through I128 and local output O33 through O64.

On remote module the inputs and outputs of the different elements are numbered sequentially. The inputs and outputs are defined in groups of 8 and the possible values of NUICAN* and NUOCAN* must be multiple of 8 plus 1 (8n +1).

Possible NUICAN values without the expansion board that has I/Os:
65, 73, 81, 89, 97, 105, 113, 121, 129, 137, 145, 153....

Possible NUICAN values with the expansion board that has I/Os:
129, 137, 145, 153, 161, 169, 177,185, 193, 201, 209....

Possible NUOCAN values without the expansion board that has I/Os:
33, 41, 49, 57, 65, 73, 81, 89, 97, 105, 113, 121, 129....

Possible NUOCAN values without the expansion board that has I/Os:
65, 73, 81, 89, 97, 105, 113, 121, 129, 137, 145, 153....

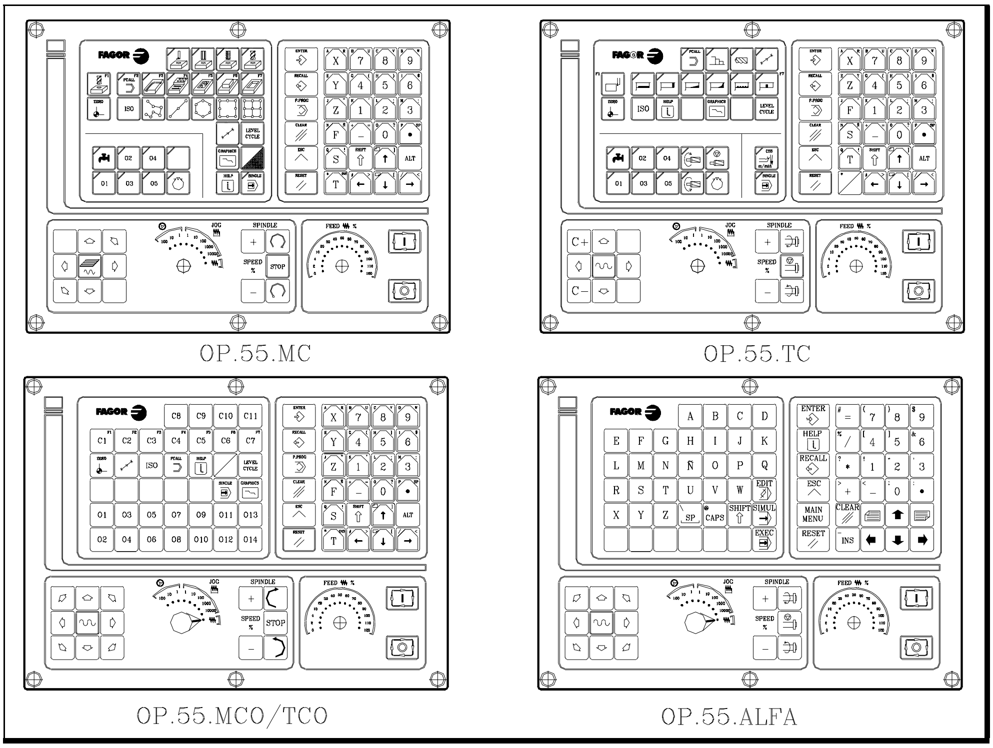
If NUICAN=0 or NUOCAN=0, the group following the one assigned to the previous node is assigned to the corresponding node.

See the examples of remote module setting in *3.7 PLC Parameters*

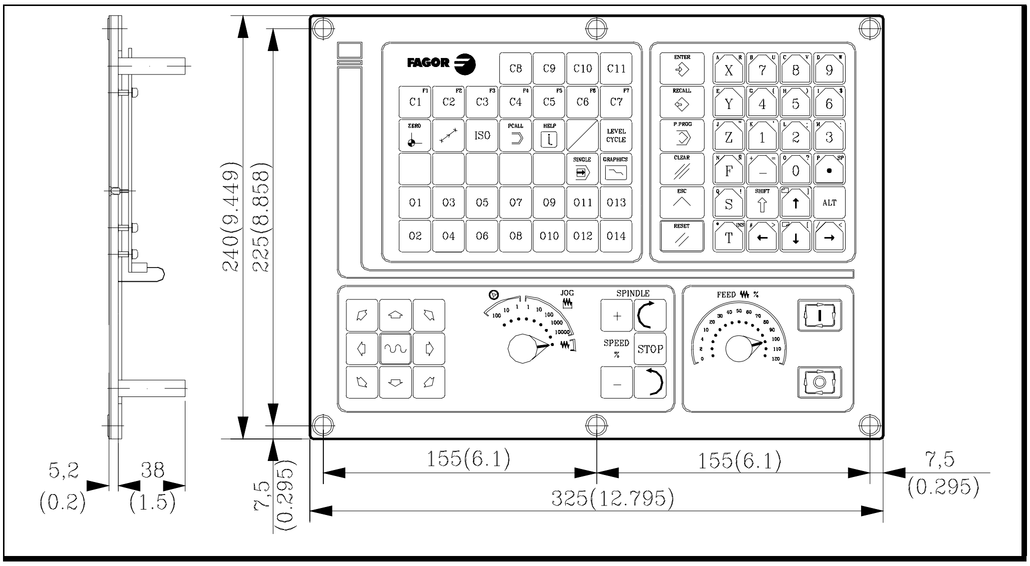
1.2 Operator panel

The "Operator Panel" depends on the CNC model

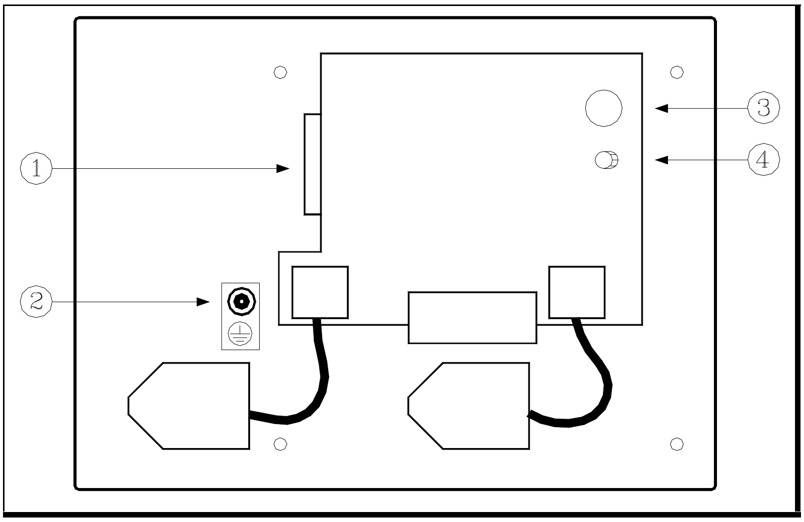
M model	Operator panel: OP.8040/55.ALFA
T model	Operator panel: OP.8040/55.ALFA
MC model	Operator panel: OP.8040/55.MC
TC model	Operator panel: OP.8040/55.TC
MCO/TCO model	Operator panel: OP.8040/55.MCO/TCO



Dimensions:

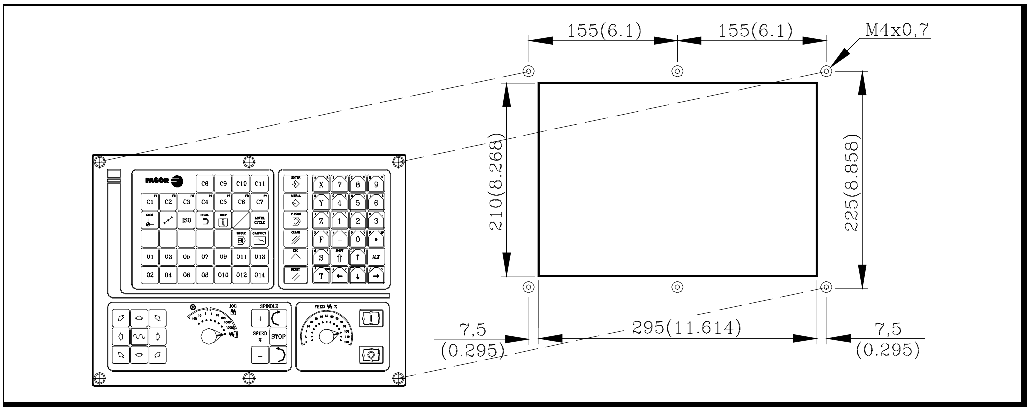


Elements:



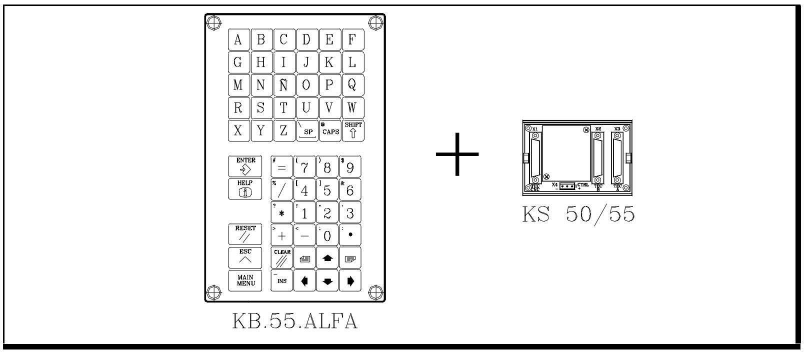
1. SUB-D type 25-pin female connector to connect the keyboard with the central unit or with a switching board.
2. Ground terminal.
3. Buzzer.
4. Potentiomenter for adjusting the buzzer volume

Enclosure:

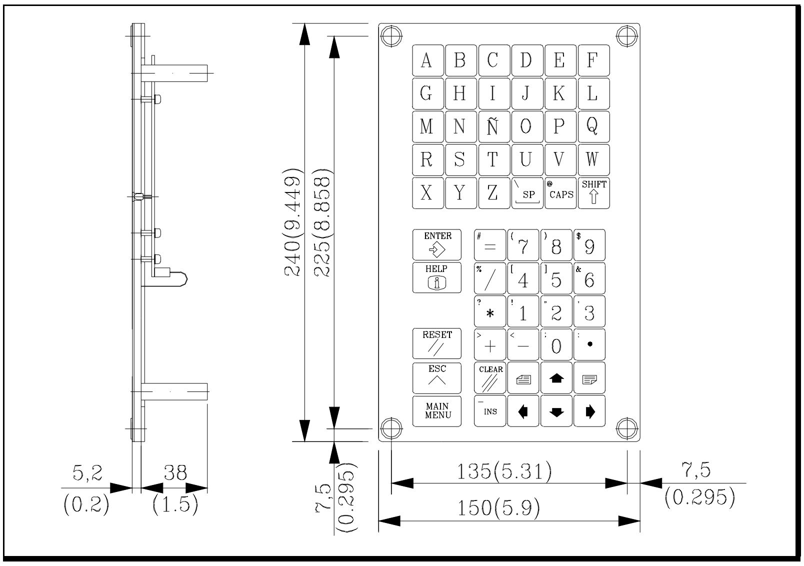


1.2.1 Alphanumeric keyboard (optional)

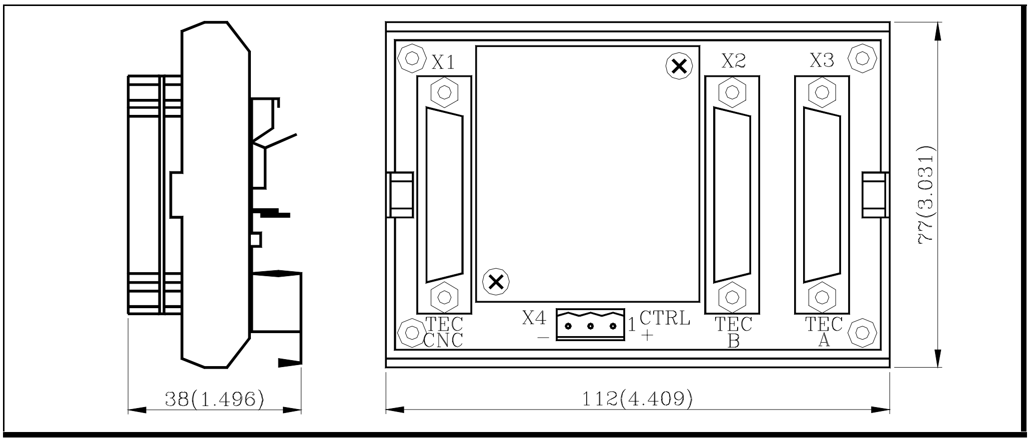
MC, TC and MCO/TCO can have an optional alphanumeric keyboard where each key has a single letter or number assigned to it (KB55.ALFA). This keyboard is connected to the Operator Panel through the KS50/55 adapter.



Dimensions:



Dimensions and elements of the KS50/55:



X1 SUB-D type 25-pin female connector to connect with the “Central Unit + Monitor”.

X2 SUB-D type 25-pin female connector to connect with the “Operator Panel”.

X3 SUB-D type 25-pin female connector to connect with the “Alphanumeric panel”.

X4 3-prong male Phoenix connector, 7.62 mm pitch, to select the keyboard that attends to the Central Unit.

Pin	Value	Meaning
1	0V 24V	The CNC attends to the alphanumeric keyboard The CNC attends to the operator panel
2 3	---- GND	Not being used at this time. External power supply

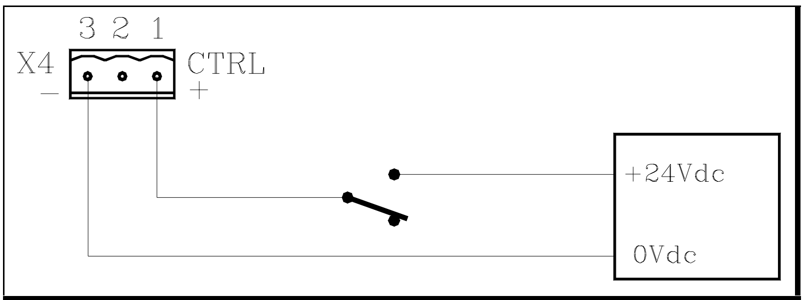
If there is no voltage at connector X4, the CNC attends to the alphanumeric keyboard.

The maximum cable length allowed between the “Central Unit + Monitor” and the “Operator panel” or “Alphanumeric keyboard” is 25 m (82 ft).

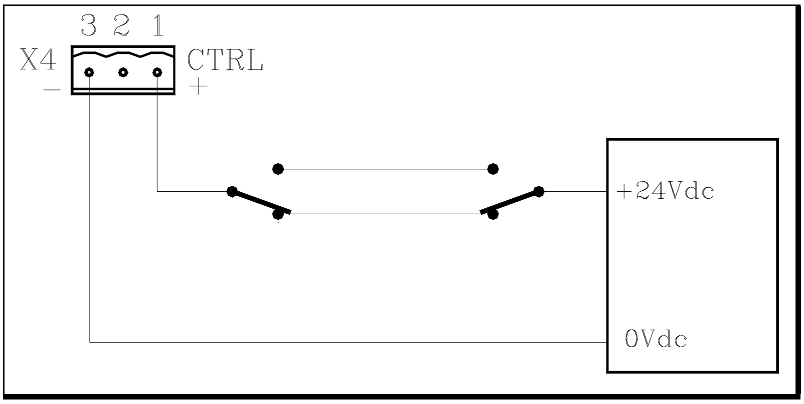
When using 2 keyboards, a keyboard switching board must be installed.

Here are some examples for keyboard selection.

With a switcher



With two switchers



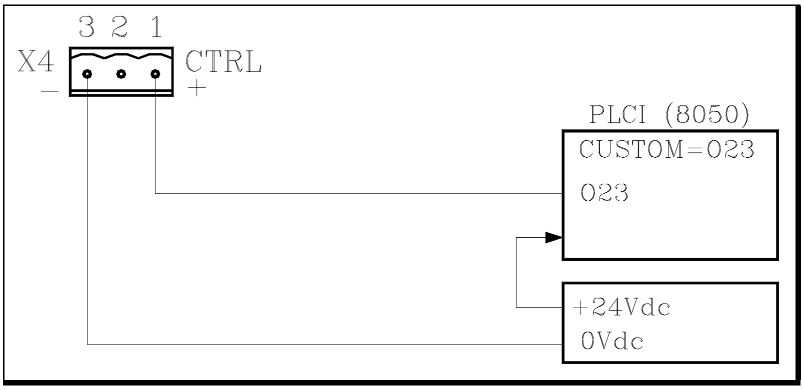
With a PLC

CNC general logic output CUSTOM (M5512) indicates to the PLC the currently selected work mode.

CUSTOM (M5512) = 0	M or T work mode
CUSTOM (M5512) = 1	MC, TC, MCO or TCO work mode

If CUSTOM=O23 is programmed at the PLC, output O23 indicates the work mode selected at the CNC.

Therefore, when doing the connection of the figure, every time the work mode is changed, the corresponding keyboard will be selected.



User notes:

User notes:

8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

MACHINE AND POWER CONNECTION

2

2. MACHINE AND POWER CONNECTION

Warning:

Power switch

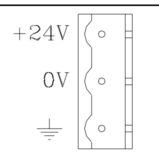
- *This power switch must be mounted in such a way that it is easily accessed and at a distance between 0.7 meters (27.5 inches) and 1.7 meters (5.5ft) off the floor.*

Install this unit in the proper place

- *It is recommended to install the CNC away from coolants, chemical products, possible blows etc. which could damage it.*

Mains connection of the Central Unit

The "Central Unit + Monitor" has a three-prong male Phoenix connector with a 7.62 mm pitch



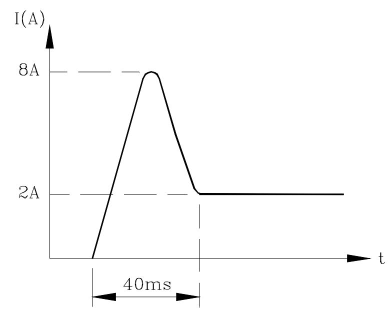
Pin	Signal and Function	
1	+24V	Power supply
2	0V	Power supply
3	Chassis	Shield

Use an independent external power supply with the following specifications:

Nominal voltage:	20 V minimum 30V maximum
Ripple:	4V
Nominal current:	2A
Current peak on power-up:	8A

The central unit has a protection against overvoltage that activates at 36 V.

The supply current has the following shape on power-up



Machine connection

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

D.C. relay coils.

Diode type 1N4000.

A.C. relay coils.

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

R 220 Ohmios/1W C 0.2 µF/600V

A.C. motors.

RC connected between phases with values:

R 300 Ohmios/6W C 0.47µF/600V

Ground connection.

It is imperative to carry out a proper ground connection in order to achieve:

- Protection of anybody against electrical shocks caused by a malfunction.
- Protection of the electronic equipment against interference generated by the proper machine or by other electronic equipment near by which could cause erratic equipment behavior.

Thus, it is essential to connect all metallic parts to a point and it to ground in order to achieve this. Therefore, it is crucial to install one or two ground points where the above mentioned elements must be connected.

Use large section cables for this purpose in order to obtain low impedance and efficiently avoid any interference. This way all parts of the installation will have the same voltage reference.

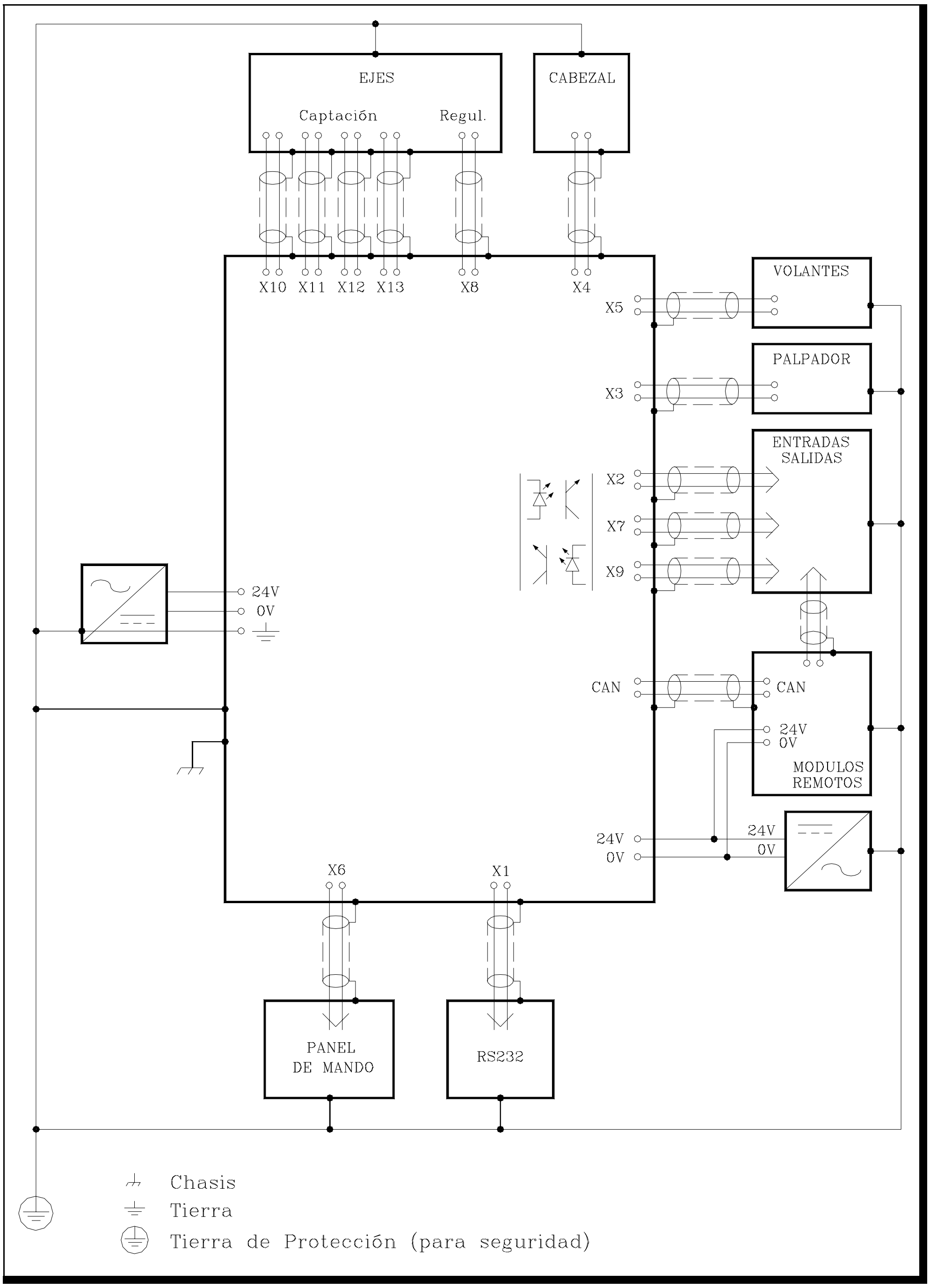
Proper ground installation reduces the effects of electrical interference. But, signal cables also require additional protections. This is generally achieved by using twisted-pair cables that are also covered with antistatic shielding mesh-wire. This shield must be connected to a specific point avoiding ground loops that could cause undesired effects. This connection is usually done at one of CNC's ground point.

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

When a second point is necessary, it is recommended to join both points with a cable whose section is no smaller than 8 mm².

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

Ground connection diagram.



2.1 Digital inputs and outputs.

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

The electrical characteristics of these outputs are:.

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

All outputs are protected by means of:

- Galvanic isolation by optocouplers.
- Has protection against short-circuits, overvoltage of the external power supply (over 33 Vdc) and against reverse connection of the power supply (up to –30 Vdc).

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

The electrical characteristics of these inputs are:

Nominal voltage value	+24 Vdc
Maximum voltage value	+30 Vdc
Minimum voltage value	+18 Vdc
High threshold voltage (logic level 1)	from +18V on.
Low threshold voltage (logic level 0)	under +5V.
Typical consumption for each input	5 mA.
Maximum consumption for each input	7 mA.

All inputs are protected by means of:

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

Warning:

The external 24V power supply. Used for the PLC's inputs and outputs MUST be regulated.

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

2.2 Digital inputs and outputs.

Analog outputs They may be used with axis and spindle drives.

The electrical characteristics of these outputs are:

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

Shielded cables should be used connecting the shield at each connector as shown here. See chapter *1. 8040 CNC configuration*

Warning:

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

2.3 Setup

General considerations

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

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

The power supply used for the digital inputs and outputs must be regulated and its zero volts must be connected to the main ground point.

Check the connection of the cables and connectors. DO NOT connect or disconnect these cables to/from the CNC when the CNC is on.

Without powering the electrical cabinet on, check all the pins of the connectors for short-circuits.

Precautions

It is recommended to reduce the axis travel installing the limit switches closer to each other or detaching the motor from the axis until they are under control.

Verify that there is no power going from the servo drives to the motors.

Verify that the connectors for the digital inputs and outputs are disconnected.

Verify that the E-STOP button is pressed.

Connection

Verify that the A.C. power is correct.

With the CNC completely disconnected from the electrical cabinet, power the electrical cabinet and verify that it responds properly:

Verify that there is proper voltage between the pins corresponding to 0V and 24V of the connectors for the digital inputs and outputs.

Apply 24V to each one of the terminals of the electrical cabinet being used that correspond to the digital outputs of the CNC and verify their correct performance. Check that the electrical cabinet responds properly.

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

Connect the A.C. power to the CNC. If there is any problem, the CNC will display the corresponding error.

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

With power turned off, connect the I/O and feedback connectors to the CNC.

Connect the CNC and the electrical cabinet to A.C. power and confirm the counting direction of each axis.

Introduction to machine parameters

The machine parameters relate the CNC to the particular machine. The values that the CNC assigns to each one of them by default are described in the chapter *3. Machine parameters*.

These values, shown in the Parameter Tables, may be modified manually from the CNC's keyboard or from a peripheral (cassette reader, floppy disk reader, computer, etc.) via the two serial communication ports RS 232C and RS 422.

Some characters appear next to certain parameters indicating when the CNC assumes the new value assigned to that parameter.

// It is necessary to press "Shift-Reset" or turn the CNC off and back on.

/ Simply press Reset.

The rest of parameters (the unmarked ones) are updated automatically just by changing them.

Setting of the machine parameters for the axes

Once the active axes have been assigned by means of g.m.p. "AXIS1" (P0) thru "AXIS8" (P7), the CNC will enable the relevant axes parameter tables.

The values to be assigned to the parameters of each of these tables will depend on the results obtained when adjusting each machine axis.

Before making this adjustment, position the axes near the middle of their travel and place the hard stops (monitored by the electrical cabinet) near these mid-travel points in order to prevent any possible damage to the machine.

Verify that the PLC Mark "LATCHM" is OFF. Then, after selecting the parameters of the desired axes, go on to adjusting them following these advises:

- Adjust the axes one by one.
- Connect the power output of the drive corresponding to the axis being adjusted.
- Selecting the Jog mode at the CNC, jog the axis to be adjusted.

  In case of runaway, the CNC will display the relevant following error and the machine parameter labelled LOOPCHG (corresponding to the sign of the analog output of the CNC) will have to be changed.
- If the axis does not run away; but the direction of the move is not the desired one, parameters labelled AXISCHG (P13) (axis feedback counting direction) and LOOPCHG (P26) (sign of the analog output) will have to be changed.

Machine reference point (home) adjustment for each axis

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

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

- This adjustment should be done one axis at a time.
- Indicate in the a.m.p. REFPULSE (P32) the type of marker pulse Io being used for Home Search.

- Set a.m.p. REFDIREC (P33) to indicate the direction of the axis when searching Home.
- Set g.m.p. REFEED1 (P34) and REFEED2 (P35) to indicate the feedrates for Home search.
- a.m.p. REFVALUE (P36) will be set to “0”.
- Once in the JOG mode and after positioning the axis in the right area, start homing the axis. When done, the CNC will assign a "0" value to this point.
- If the Machine Reference Zero desired is in a different physical location from the Machine Reference Point (location of the marker pulse), proceed as follows:

  After moving the axis to a known position (with respect to Machine Reference Zero), observe the position reading of the CNC for that point.

  This will be the distance away from the machine reference point; thus, the value to be assigned to a.m.p. REFVALUE (P36) will be:

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

Example:

If the point whose known position is located 230 mm from Machine Reference Zero and the CNC reads -123.5 mm as the coordinate value for this point, the coordinate of the Machine Reference Point with respect to Machine Reference Zero will be:

REFVALUE = 230 - (-123.5) = 353.5 mm.

Assign this new value and press [RESET] so it is assumed by the CNC.

It is also necessary to search Home once again in order for this axis to assume the correct reference values.

Axis travel limits (software limits)

Once all the axes have been referenced, their software limits must be measured and set.

This is achieved a single axis at a time as follows:

- Move the axis in the positive direction towards the end of the axis travel stopping at a safe distance from the mechanical end-of-travel stop.
- Assign the coordinate shown by the CNC for that point to a.m.p. LIMIT+ (P5).
- Repeat these steps in the negative direction assigning the resulting coordinate to a.m.p. LIMIT- (P6).
- Once this process is completed, hit SHIFT RESET or turn the CNC off and back on in order for it to assume the new values.

Adjustment of the drift (offset) and maximum feedrate (G00)

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

Offset (drift) adjustment

Disconnect the analog input and short-circuit it with a wire jumper.

Turn the offset potentiometer of the drive until the voltage on the tach terminals is 0mVdc. Check this with a volt meter set at a range of 200 mV. DC.

Remove take the wire jumper that short-circuited the analog input.

Maximum feedrate adjustment

It is recommended to adjust the drives so the maximum feedrate is obtained with an analog signal of 9.5V. If they are adjusted to a different voltage, it must be indicated in the a.m.p. or s.m.p. MAXVOLT (P37).

Also, the maximum feedrate must be indicated in the a.m.p. G00FEED (P38).

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

Example:

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

The maximum feedrate will be:

3000 r.p.m. x 5mm/rev. = 15000 mm/minute

This will be the value to be assigned to a.m.p. G00FEED (P38).

Once these values are assigned to the relevant parameters, the drives must be adjusted.

To do so, a CNC program can be executed which will move the axis back and forth continuously at G00 feedrate. One such program could be:

```
N10 G00 G90 X200
X-200
```

(GOTO N10)

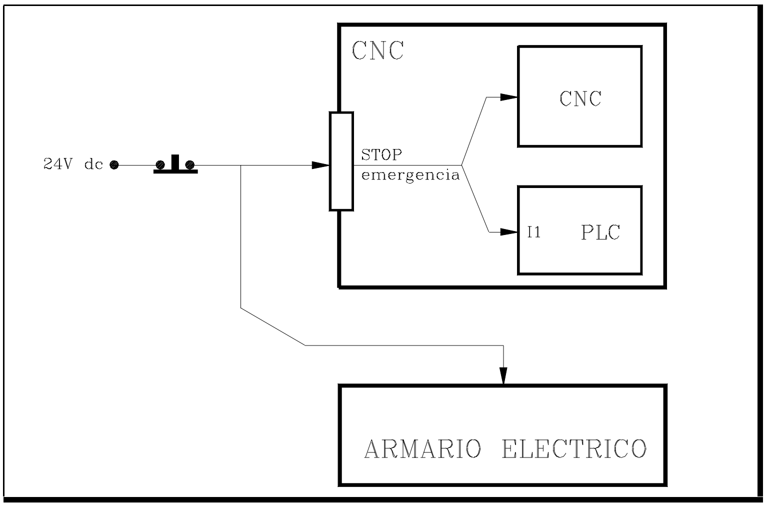
If the Tach in use provides 20V per 1000 rpms, its voltage should be:

(20 V / 1000 r.p.m.) x 3000 r.p.m. = 60 V.

2.4 Connection of the emergency input and output

The EMERGENCY INPUT of the CNC corresponds with the PLC input I1 (pin 10 of connector X2) and must be supplied with 24V.

Since the CNC also processes this signal directly, if the 24V disappear, the CNC will display EXTERNAL EMERGENCY ERROR and will deactivate all axes enables and will cancel all analog outputs.



During the initializing process carried out by the CNC on power-up, the EMERGENCY OUTPUT of the CNC (pin 2 of connector X10) remains at low (at "0") in order to avoid a premature activation of the electrical cabinet.

If this process is successful, the CNC will set the real value of PLC output O1 to "1". Otherwise, it will keep the /EMERGENCY OUTPUT signal active (low) and it will display the corresponding error message.

Once the initialization process is over, the PLC will execute the PLC program stored in memory. If none is available, it wait for one to be entered and executed.

When the execution of the first cycle (CY1) (or the first program scan) is finished the PLC will assign the value of output O1 to physical output "/EMERGENCY OUTPUT".

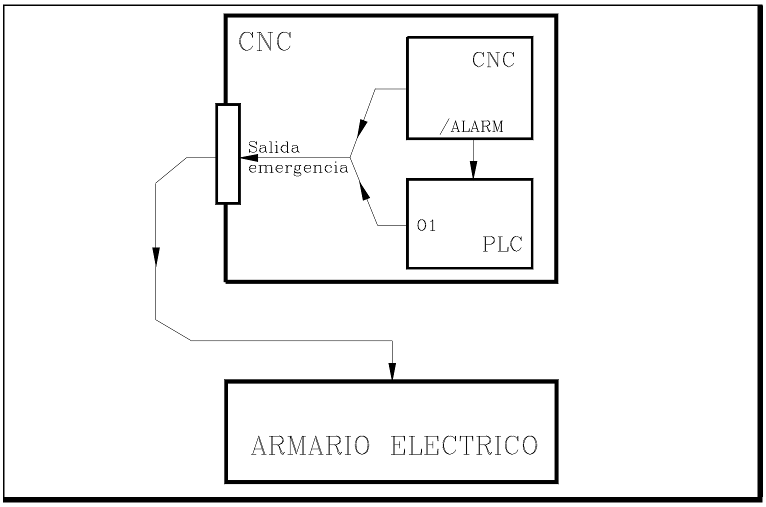
It is recommended to program the CY1 cycle of the PLC program assigning a value of 1 to O1 when everything checks out fine and a value of 0 when there is an error.

The interface of the electrical cabinet will take into account all the elements that could cause this type of error. Among such elements are:

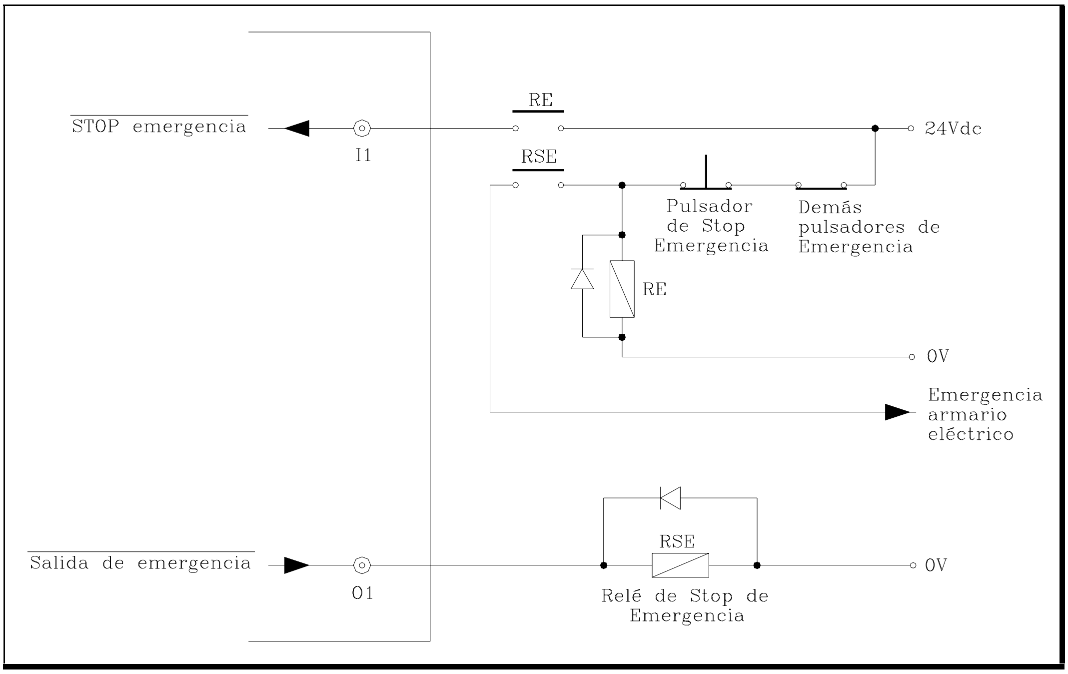
- E-stop has been pressed.
- The travel limit of any axis has be exceeded.
- There is a malfunction on a drive or it is locked without analog signal.

When the CNC detects an error, it will indicate it to the PLC with the general logic output "/ALARM" and it will set the emergency output low (pin 2 of connector X2).

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



The recommended connection diagram is the following:



User notes:

User notes:

8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

MACHINE PARAMETERS

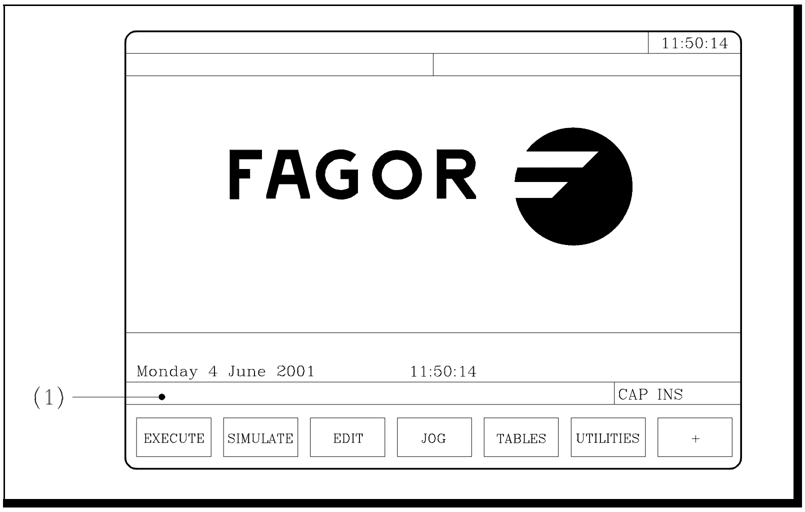
3

3. MACHINE PARAMETERS

Warning:

It is recommended to save the CNC machine parameters into the "Memkey Card" (CARD A) or in a peripheral or PC to avoid losing them.

On power-up, the CNC performs a system autotest and when this is over, it displays the following screen:



The CNC allows the display of a previously defined screen instead of the Fagor logo. See operating manual.

During the autotest, if any error occurs, its relevant message will be displayed in the report window (1).

The main menu for the various operating modes will appear at the bottom of the CRT. These options will be selected using the softkeys F1 through F7.

Since it is possible to have more than 7 options to choose from at one time, use the "+" softkey to display the rest of them.

Once the "Machine Parameters" operating mode has been selected, the CNC shows the machine parameter tables that are saved in the MemkeyCard (CARD A).

The available machine parameter tables are:

- General machine parameters
- Machine parameters for the Axes (one table per axis)
- Machine parameters for the main, second and auxiliary spindles
- Drive parameters (access to the parameters of each drive)
- Parameters for the serial ports and Ethernet.
- PLC Parameters.
- M Miscellaneous Functions (Auxiliary functions).
- Leadscrew backlash Compensation (one table per axis)
- Cross compensation

To access each one of them, use the softkeys shown at the bottom of the screen.

Some characters appear next to certain parameters indicating when the CNC assumes the new value assigned to that parameter.

//	It is necessary to press “Shift-Reset” or turn the CNC off and back on.
/	Simply press Reset.
	The rest of parameters (the unmarked ones) are updated automatically just by changing them.

On each table, it is possible to move the cursor line by line using the arrow keys or page by page using the Page-up and Page-down keys.

Operation with parameter tables

Once one of the table lines has been selected, the user can move the cursor over this line by means of the right and left arrow keys .

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

CL	deletes characters.
INS	switches between insert and replace writing modes.
CAP	switches between upper case and lower case letters; when the CRT shows CAP, it will indicate that the upper case mode has been selected. Make sure this mode is selected since all characters entered in these tables must be upper case.
ESC	quits the line editing
ENTER	Assumes the edited line and ends the editing of the line.

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

EDIT	a parameter. The CNC will indicate the proper format by means of the softkeys.
MODIFY	a parameter. Position the cursor on the desired parameter and press the Modify softkey. Once the modification is done, press ENTER for the CNC to assume the new value.
FIND	a parameter. The cursor will be positioned over the indicated parameter. With this function it is also possible to “find” the beginning or the end of the table.
INITIALIZE	the table assuming the default values.
LOAD	into memory the tables saved in the "Memkey Card" (CARD A), a peripheral device or a PC
SAVE	the tables into the "Memkey Card" (CARD A), a peripheral device or a PC
MM/INCHES	to see the parameter values in the desired units. Only those parameters affected by this conversion will be altered. It will not change the g.m.p. INCHES (P8) that indicates machine units.

Machine parameter setting

In order for the machine-tool to be able to properly execute the programmed instructions as well as interpret the different elements connected to it, the CNC must “know” the specific data of the machine, such as: feedrates, accelerations, feedback, automatic tool change, etc..

This data is determined by the machine builder and can be introduced either from the CNC's keyboard or via the CNC's two serial ports.

The CNC offers the following machine parameter groups:

- General machine parameters
- Axis parameters.
- Spindle parameters.
- Drive parameters
- Parameters for the two communication channels, RS-422 and RS-232-C.
- Ethernet configuration parameters
- PLC Parameters.
- M Miscellaneous Functions (Auxiliary functions).
- Leadscrew backlash compensation.
- Cross Compensation.

First, the general machine parameters must be set since they determine the machine axes.

There are some parameters to indicate whether the machine has cross compensation or not. These compensation tables will be generated by the CNC from the values assigned to those parameters.

The general machine parameters also determine the number of elements at the tables for tools, tool magazine, tool offsets and M functions (miscellaneous).

The axes parameters will define the Leadscrew Compensation Tables and they will only be generated for those axes which require them.

Note: When selecting the drive parameters at the CNC, it is possible to display and modify the parameters stored at each drive.

The CNC does not have parameters of the drive although their copies may be stored in the "Memkey Card" (Card A).

3.1 General parameters

AXIS1 (P0) AXIS2 (P1) AXIS3 (P2) AXIS4 (P3)
AXIS5 (P4) AXIS6 (P5) AXIS7 (P6) AXIS8 (P7)

They permit associating axes, handwheels, spindles or live tools with each feedback input and analog output according to the following code:

0	Free. Not associated with any axis.	1	X axis
2	Y axis	3	Z axis
4	U axis	5	V axis
6	W axis	7	A axis
8	B axis	9	C axis
10	Main spindle	11	Handwheel
12	Handwheel with axis selector button	13	Auxiliary spindle / Live tool
14	Second main spindle	21	Handwheel associated with X
22	Handwheel associated with Y	23	Handwheel associated with Z
24	Handwheel associated with U	25	Handwheel associated with V
26	Handwheel associated with W	27	Handwheel associated with A
28	Handwheel associated with B	29	Handwheel associated with C

Handwheels, the choices are:

General handwheel It can be used to jog any axis one by one.
Select the axis and turn the handwheel to move it.

Individual handwheel: It replaces the mechanical handwheels.
Up to 2 handwheels can be used (one per axis).
It only moves the axis it is associated with.

When using a Fagor 100P handwheel, no other handwheels may be used and it must be connected as first handwheel.

For further information, see section ***4.1.2 Movement with an electronic handwheel***

The following table show the feedback input, the analog voltage output and the default values associated with each parameter.

Parameter		Feedback	Command	Default value	
				Mill model	Lathe
AXIS1 (P0)	1st axis	Connector X10	Pin 2 (X8)	1 (X axis)	1 (X axis)
AXIS2 (P1)	2nd axis	Connector X11	Pin 3 (X8)	2 (Y axis)	3 (Z axis)
AXIS3 (P2)	3rd axis	Connector X12	Pin 4 (X8)	3 (Z axis)	0 (free)
AXIS4 (P3)	4th axis	Connector X13	Pin 5 (X8)	4 (U axis)	0 (free)
AXIS5 (P4)	Spindle	Connector X4	Connector X4	10 (spindle)	10 (spindle)
AXIS6 (P5)	1st handwheel	Connector X5	----	11 (Handwheel)	11 (Handwheel)
AXIS7 (P6)	2nd handwheel	Connector X5	----	0 (free)	0 (free)
AXIS8 (P7)	Not being used at this time.	----	----	0 (free)	0 (free)

INCHES (P8) It defines the measuring units assumed by the CNC for machine parameters, tool tables and programming on power-up and after executing M02,M30, EMERGENCY or RESET. The code is:

0 millimeters (G71) *Default value*
1 inches (G70)

IMOVE (P9) Indicates which function G00 (rapid traverse) or G01 (linear interpolation) is assumed on power-up, after executing M02,M30, EMERGENCY or RESET. The code is:

0	G00 (rapid traverse)	*Default value*to
1	G01 (linear interpolation)	

ICORNER (P10) Indicates which function, G05 (round corner) or G07 (square corner) is assumed on power-up, after executing M02,M30, EMERGENCY or RESET. The code is:

0	G07 (square corner)	*Default value*
1	G05 (round corner)	

IPLANE (P11) Indicates which function: G17 (XY plane) or G18 (ZX plane) is assumed on power-up, after executing M02,M30, EMERGENCY or RESET. The code is:

0	G17 (XY plane)	*Default value (M)*
1	G18 (ZX plane)	*Default value (T)*

ILCOMP (P12) It is only used in the Mill model CNC and indicates which function: G43 (tool length compensation ON) or G44 (tool length compensation OFF) is assumed on power-up, after executing M02,M30, EMERGENCY or RESET; the code is:

0	G44 (compensation OFF)	*Default value*
1	G43 (tool length compensation ON)	

ISYSTEM (P13) Indicates which function: G90 (absolute programming) or G91 (incremental programming) is assumed on power-up, after executing M02,M30, EMERGENCY or RESET.

0	G90 (absolute programming)	*Default value*
1	G91 (incremental programming)	

IFEED (P14) Indicates which function: G94 (feedrate in mm/min or inch/min) or G95 (mm/rev or inch/rev) is assumed on power-up, after executing M02,M30, EMERGENCY or RESET.

0	G94 (mm/min or inches/min)	*Default value*
1	G95 (mm/rev or inches/rev.)	

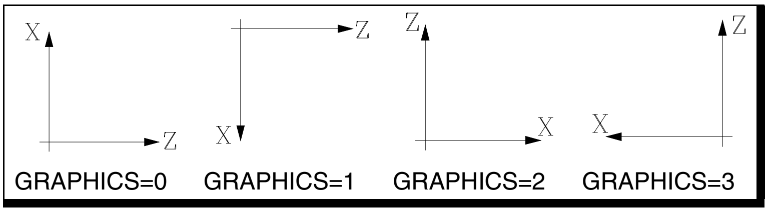
THEODPLY (P15) Indicates whether the CNC will display real or theoretical position values according to the following code:

0	real position values	
1	theoretical position values	*Default value*

GRAPHICS (P16) On M, MC, and MCO models, this parameter indicates the axis system being used for the graphic representation as well as the motion possibilities for the W axis added to those of the Z axis in the graphic representation (W additive).

- 0 Mill graphics
- 1 Mill graphics with added W axis
- 2 Boring Mill graphics
- 3 Boring Mill graphics with added W axis

On T, TC and TCO models, this parameter indicates the axes coordinates system to be used for the graphic representation.



RAPIDOVR (P17) Indicates whether it is possible to vary the feedrate override between 0% and 100% when working in G00.

YES	modifying allowed	*Default value*
NO	not allowed, fixed at 100%	

The feedrate override % may be changed from the operator panel switch, from the PLC, via DNC or by program.

The feedrate % can always be changed in JOG movements.

MAXFOVR (P18) Indicates the maximum value of the Feedrate override % applicable to the programmed feedrate.

Possible values Integers 0 through 255.

By default 120

From the operator panel switch, it may be varied between 0% and 120% and from the PLC, DNC or by program between 0% and 255%

CIRINLIM (P19) Indicates the maximum angular feedrate value for circular interpolations.

This limitation prevents circular interpolations resulting in polygons instead of arcs when the radius is very small. The CNC adjusts the angular feedrate in order not to exceed the selected maximum angular feedrate..

Possible values Integers between 0 and 65535.

By default 0 (unlimited)

Example If "CIRINLIN" = 1500 and an arc of a radius = 0.5mm at F=10000mm/min.

The theoretical angular speed is:

10000 mm/min / 0.5 mm = 20000 min^{-1}

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

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

CIRINERR (P20) Indicates the maximum error allowed when calculating the end point of an arc.

From the programmed path, the CNC will calculate the radius for both the starting point and end point of the arc. Although both of them should be “exactly” the same, This parameter allows a certain calculation tolerance by establishing the maximum difference between these two radii.

Possible values: 0.0001.... 99999.9999 mm
0.00001.... 3937.00787"

By default 0.01 mm.

PORGMOVE (P21) Indicates whether the CNC assumes or not as the new polar coordinate origin the center of the last G02 or G03 programmed.

YES	It assumes the arc center.	
NO	It is not affected by G02 and G03.	*Default value*

BLOCKDLY (P22) It indicates the delay between motion blocks when operating in G7 (square corner).

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

Possible values Integers between 0 and 65535 ms.

By default 0 (no delay)

NTOOL (P23) Indicates the number of tools in the tool magazine. On the other hand, the CNC adjusts the length of the tool table to that value.

Possible values Integers 0 through 255.

By default 100.

NPOCKET (P24) Indicates the number of pockets in the tool magazine. On the other hand, the CNC adjusts the length of the tool magazine table to that value.

Possible values Integers 0 through 255.

By default 100 (M) and 0 (T).

RANDOMTC (P25) Indicates whether the tool magazine is RANDOM or not.

On a RANDOM magazine, the tool may occupy any position (pocket).

On a NON-RANDOM magazine, the tool always occupies its own pocket. The magazine position number is the same as the tool number.

YES	It is a RANDOM tool magazine.	*Default value*
NO	It is Not a RANDOM tool magazine.	

By default, the CNC sets this parameter to “NO”.

If this machine parameter is set for RANDOM magazine, g.m.p. TOFFM06 (P28) must be set for machining center.

TOOLMONI (P26) Selects the display units of the tool’s nominal and real lives.

0	In minutes.	*Default value*
1	In number of operations.	

NTOFFSET (P27) Indicates the number of tool offsets available in the tool offset table. On the other hand, the CNC adjusts the length of the tool offset magazine table to that value.

Possible values Integers 0 through 255.

By default 100

TOFFM06 (P28) Indicates whether the machine is a machining center or not.

If it is, the CNC will select, at the tool magazine, the tool indicated when executing the "T" function and it will be necessary to execute M06 afterwards in order to carry out the tool change.

YES	YES, it is a machining center.	
NO	It is not a machining center.	*Default value*

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

NMISCFUN (P29) Indicates the number of M functions available in the M function table.

Possible values Integers 0 through 255.

By default 32

MINAENDW (P30) Indicates the minimum time period that the AUX END signal must remain activated so the CNC will interpret it as a valid signal.

AUX END is a PLC signal which indicates to the CNC that functions M,S or T have been executed.

If the corresponding M function has been set in the M table not to wait for the AUX END signal, the time period indicated in this parameter will be the duration of the MSTROBE signal.

Possible values Integers between 0 and 65535 ms.

By default 100

See section ***4.6 Auxiliary M, S, T function transfer***

NPCROSS (P31) Indicates the number of points available in the first cross compensation table.

This compensation is used when the movement of one axis causes a position change on another axis. The CNC offers a table where one could enter the position variations of one axis for the particular positions of the other axis.

Possible values Integers 0 through 255.

By default 0 (not available)

MOVAXIS (P32) Used in the first cross compensation table, it indicates the axis causing position variations on another axis. The definition code is:

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

By default 0 (none)

COMPAXIS (P33) Used in the first cross compensation table, it indicates the axis suffering the position variations caused by another axis. The compensation is applied onto this axis. The definition code is:.

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

By default 0 (none)

Example: If NPCROSS=20, MOVAXIS=X and COMPAXIS=W, the CNC will allow access to the cross compensation table.

Each one of these 20 points (NPCROSS) of this table will indicate the X position value and the error suffered by the W axis when the X axis is positioned at this point.

This way, the CNC will apply the compensation of the X axis table onto the W axis.

REFPSUB (P34) Indicates the number of the subroutine associated with function G74 (machine reference zero or home search). This subroutine will be executed automatically when G74 is programmed alone in a block or, also, when searching home in the JOG mode by pressing the softkey “ALL AXES”.

Possible values: Integers between 0 and 9999.

By default 0 (there is no associated subroutine)

INT1SUB (P35)
INT2SUB (P36)
INT3SUB (P37)
INT4SUB (P38)

They indicate the number of the subroutine associated with the corresponding general logic input: "INT1" (M5024), "INT2" (M5025), "INT3" (M5026)", "INT4" (M5027).

When one of these inputs is activated, the program currently being executed is interrupted and the CNC jumps to execute the associated subroutine whose number is indicated in the corresponding parameter.

These interruption subroutines do not change the nesting level of local parameters, thus only global parameters must be used in them.

Once the CNC completes the execution of the subroutine, it will continue running the original program.

Possible values: Integers between 0 and 9999.

By default 0 (there is no associated subroutine)

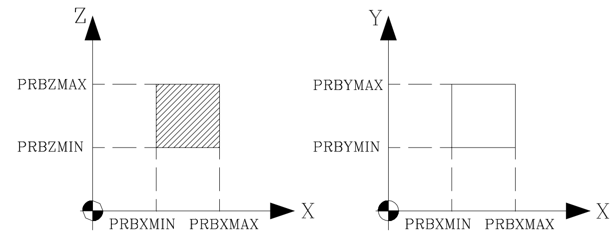
PRBPULSE (P39) Indicates whether the probe functions of the CNC react to the up-flank (leading edge) or down-flank (trailing edge) of the probe signal. This probe is connected to the connector X7 of the AXES module.

+ sign	Positive pulse (leading edge 24V. or 5V .)	*Default value*
- sign	Negative pulse (0V).	

PRBXMIN (P40)
PRBXMAX (P41)
PRBYMIN (P42)
PRBYMAX (P43)
PRBZMIN (P44)
PRBZMAX (P45)

Indicate the position of the tabletop probe used for tool calibration.

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



PRBXMIN	probe's minimum X coordinate.
PRBXMAX	probe's minimum X coordinate.
PRBYMIN	probe's minimum Y coordinate.
PRBYMAX	probe's minimum Y coordinate.
PRBZMIN	probe's minimum Z coordinate.
PRBZMAX	probe's minimum Z coordinate.

Possible values: ±99999.9999 mm or ±3937.00787 inches.

By default 0

PRBMOVE (P46) Indicates the maximum distance the tool can travel when calibrating it with a probe in JOG mode.

Possible values: 0.0001.... 99999.9999 mm
0.00001.... 3937.00787"

By default 50 mm.

USERDPLY (P47) Indicates the number of the USER display program associated to the EXECUTE mode. This program will be executed via the user channel when pressing the softkey USER in the EXECUTE mode.

Possible values: Integers between 0 and 65535.

By default 0 (not available)

USEREDIT (P48) Indicates the number of USER display program associated to the EDIT mode. This program will be executed via the user channel when pressing the softkey USER in the EDIT mode.

Possible values: Integers between 0 and 65535.

By default 0 (not available)

USERMAN (P49) Indicates the number of the USER display program associated to the JOG mode. This program will be executed via the user channel when pressing the softkey USER in the JOG mode.

Possible values: Integers between 0 and 65535.

By default 0 (not available)

USERDIAG (P50) Indicates the number of the USER display program associated to the DIAGNOSIS mode. This program will be executed via the user channel when pressing the softkey USER in the DIAGNOSIS mode.

Possible values: Integers between 0 and 65535.

By default 0 (not available)

ROPARMIN (P51)
ROPARMAX (P52)

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

Possible values Integers 0 through 9999.

(internally between 100 and 299)

By default 0 (unprotected)

Note: The parameters write-protected from the CNC may be modified from the PLC.

PAGESMEM (P53) Not being used at this time.

NPCROSS2 (P54) Indicates the number of points available in the second cross compensation table.

This compensation is used when the movement of one axis causes a position change on another axis. The CNC offers a table where one could enter the position variations of one axis for the particular positions of the other axis.

Possible values Integers 0 through 255.

By default 0 (not available)

MOVAXIS2 (P55) Used in the second cross compensation table, it indicates the axis causing position variations on another axis. The definition code is:

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

By default 0 (none)

COMAXIS2 (P56) Used in the second cross compensation table, it indicates the axis suffering the position variations caused by another axis. The compensation is applied onto this axis. The definition code is:

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

By default 0 (none)

Example: If NPCROSS2=15, MOVAXIS2=2 and COMAXIS2=8, the CNC will allow access to the second cross compensation table.

Each one of these 15 points (NPCROSS2) of this table will indicate the X position value and the error suffered by the B axis when the Y axis is positioned at this point.

This way, the CNC will apply the compensation of the Y axis table on to the B axis.

NPCROSS3 (P57) Indicates the number of points available in the third cross compensation table.

This compensation is used when the movement of one axis causes a position change on another axis. The CNC offers a third table where one could enter the position variations of one axis for the particular positions of the other axis.

Possible values Integers 0 through 255.

By default 0 (not available)

MOVAXIS3 (P58) Used in the third cross compensation table, it indicates the axis causing position variations on another axis. The definition code is:

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

By default 0 (none)

COMAXIS3 (P59) Used in the third cross compensation table, it indicates the axis suffering the position variations caused by another axis. The compensation is applied onto this axis. The definition code is:

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

By default 0 (none)

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

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

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

TOOLSUB (P60) Indicates the number of the subroutine associated with the tools. This subroutine will be executed automatically every time a T function is executed.

Possible values: Integers between 0 and 9999.

By default 0 (not available)

CYCATC (P61) This parameter must be used when having a machining center, g.m.p. TOFFM06 (P28) = YES.

Indicates whether a cyclic tool changer is being used or not.

A "cyclic tool changer" is an automatic tool changer which requires an M06 command (tool change) after searching for a tool and before searching for the next one.

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

NO It is not a cyclic changer

YES It is a cyclic changer. *Default value*

TRMULT (P62)
TRPROG (P63)
TRDERG (P64)
MAXDEFLE (P65)
MINDEFLE (P66)
TRFBAKAL (P67)

Not being used at this time.

TIPDPLY (P68) Indicates whether the CNC displays the position of the tool tip or that of the tool base when working with tool length compensation.

0	The one corresponding to the base	*Default value (M)*
1	The one corresponding to the tip	*Default value (T)*

On the Mill model, it is necessary to execute G43 in order to work with tool length compensation. When not working with tool length compensation (G44), the CNC displays the tool base position.

On the Lathe model, it always works with tool length compensation. Therefore, by default, the CNC always displays the tool tip position.

ANTIME (P69) It is used on punch presses having an eccentric cam as a punching system.

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

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

Possible values Integers between 0 and 65535 ms.

By default 0

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

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

PERCAX (P70) Not used.

TAFTERS (P71) g.m.p. TOOLSUB (P60) indicates the number of the subroutine associated with the tool.

The TAFTERS parameter determines whether the tool selection is carried out before or after executing that subroutine.

YES	After executing the subroutine.	
NO	Before executing the subroutine.	*Default value*

LOOPTIME (P72) It sets the sampling period used by the CNC and, consequently, it affects its block processing time.

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

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

Also, the CNC configuration limits the sampling period. The shorter it is, the less time the CPU will have to process the data. Thus, remember that:

- *Sinewave feedback requires more calculation time.*
- *More axes means more calculation time.*
- *if the user channel is active, more calculation time is required.*

IPOTIME (P73) It sets the interpolation period used by the CNC and, consequently it affects its block processing time.

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

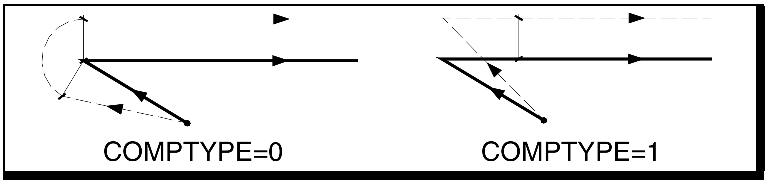
Possible values:

- 0 IPOTIME = LOOPTIME
- 1 IPOTIME = Double LOOPTIME value

COMPTYPE (P74) It set the type of beginning/end of tool radius compensation applied by the CNC.

- 0 it approaches the starting point going around the corner
- 1 it goes directly to the perpendicular of the point (it does not go around the corner)

By default 0



FPRMAN (P75) It is only used on Lathe model CNCs and it indicates whether feedrate per revolution is permitted or not.

- NO Not admitted *Default value*
- YES Admitted

MPGAXIS (P76) It is only used on the Lathe model CNCs and it indicates which axis the handwheel is assigned to. It is set according to the following codes:

0	Shared	1	X axis	2	Y axis	3	Z axis	4	U axis
5	V axis	6	W axis	7	A axis	8	B axis	9	C axis

By default 0 (shared)

DIRESET (P77) It is only used on the Lathe model CNC and it indicates whether it is effective with or without prior CYCLE STOP.

- NO Only if the STOP condition occurs *Default value*
- YES The CNC accepts the RESET any time.

If DIRESET=YES, the CNC first carries out an internal CYCLE STOP to interrupt program execution and, then, executes the RESET.

Obviously, if it is performing a threadcutting or similar operation, not admitting a CYCLE STOP, it will wait for the operation to be concluded before interrupting the program.

PLACOMP (P78) It is used on the lathe model to indicate whether there is tool compensation in all planes or just in the ZX plane.

- 0 Only in the ZX plane *Default value*
- 1 In all planes.

When "PLACOM = 1", the CNC interprets the tool table as follows:

	ZX Plane	WX Plane
The Z and K parameters, with the abscissa axis	Z axis	W axis
The X and I parameters, with the ordinate axis	X axis	X axis

MACELOOK (P79) When using "Look-Ahead" the operator sets the percentage of acceleration being applied in Look-Ahead by means of function G51.

With g.m.p. MACELOOK (P79) the OEM can limit the maximum percentage of acceleration that the user may set with G51.

Possible values Integers 0 through 255.

By default 0 (unlimited)

MPGCHG (P80)
MPGRES (P81)
MPGNPUL (P82)

These parameters must be used when having an electronic handwheel to jog the axes.

Parameter MPGCHG (P80) indicates the turning direction of the electronic handwheel. If correct, leave it as is. Otherwise, select YES is there was a NO before or vice versa.

Possible values: NO and YES

By default, NO

Parameter MPGRES (P81) indicates the counting resolution of the electronic handwheel and depends on the display format selected for the corresponding a.m.p. DFORMAT (P1).

Possible values: 0, 1 and 2.

By default 0

FORMAT	RESOLUTION		
	MPGRES=0	MPGRES=1	MPGRES=2
5.3 mm 4.4"	0.001mm 0.0001"	0.010mm 0.0010"	0.100mm 0.0100"
4.4 mm 3.5"	0.0001mm 0.00001"	0.0010mm 0.00010"	0.0100mm 0.00100"
6.2 mm 5.3"	0.01mm 0.001"	0.10mm 0.010"	1.00mm 0.100"

Parameter MPGNPUL (P82) indicates the number of pulses per turn of the electronic handwheel.

Possible values: Integers between 0 and 65535.

By default, 0 (same as 25)

Example : Having a Fagor electronic handwheel (25 pulses per turn) we would like to move 1 mm per handwheel turn.

Set the a.m.p. for the feedback input of the electronic handwheel AXIS1 (P0) through AXIS7 (P6), to a value of 12 (Fagor 100P handwheel).

Also set g.m.p. MPGAXIS (P76) to indicate which axis has been assigned this handwheel.

Set parameter MPGNPUL=25 or 0 meaning 25 pulses per turn of the Fagor handwheel.

Since the handwheel outputs square signals and the CNC applies a x4 multiplying factor to them, we get 100 pulses per turn.

The value to be assigned to parameter MPGRES depends on the axis resolution format.

Format		MPGRES=0	MPGRES=1	MPGRES=2
5.3 mm	Resolution	0.001mm	0.010mm	0.100mm
	Pulses / turn	0.100mm	1.00mm	10.000mm
4.4 mm	Resolution	0.0001mm	0.0010mm	0.0100mm
	Pulses / turn	0.0100mm	0.1000mm	1.0000mm
6.2 mm	Resolution	0.01mm	0.10mm	1.00mm
	Pulses / turn	1.00mm	10.00mm	100.00mm

With 5.3mm type display format, set MPGRES=1
With 4.4mm type display format, set MPGRES=2

With 6.2mm type display format, set MPGRES=0

MPG1CHG (P83)
MPG1RES (P84)
MPG1NPUL (P85)
MPG2CHG (P86)
MPG2RES (P87)
MPG2NPUL (P88)
MPG3CHG (P89)
MPG3RES (P90)
MPG3NPUL (P91)

These parameters must be used when the machine has several electronic handwheels, one per axis and up to 3 handwheels.

Set the a.m.p. for the feedback input of the electronic handwheel AXIS1 (P0) through AXIS7 (P6), to one of the following values:

21	handwheel associated with the X axis
22	handwheel associated with the Y axis
23	handwheel associated with the Z axis
24	handwheel associated with the U axis
25	handwheel associated with the V axis
26	handwheel associated with the W axis
27	handwheel associated with the A axis
28	handwheel associated with the B axis
29	handwheel associated with the C axis

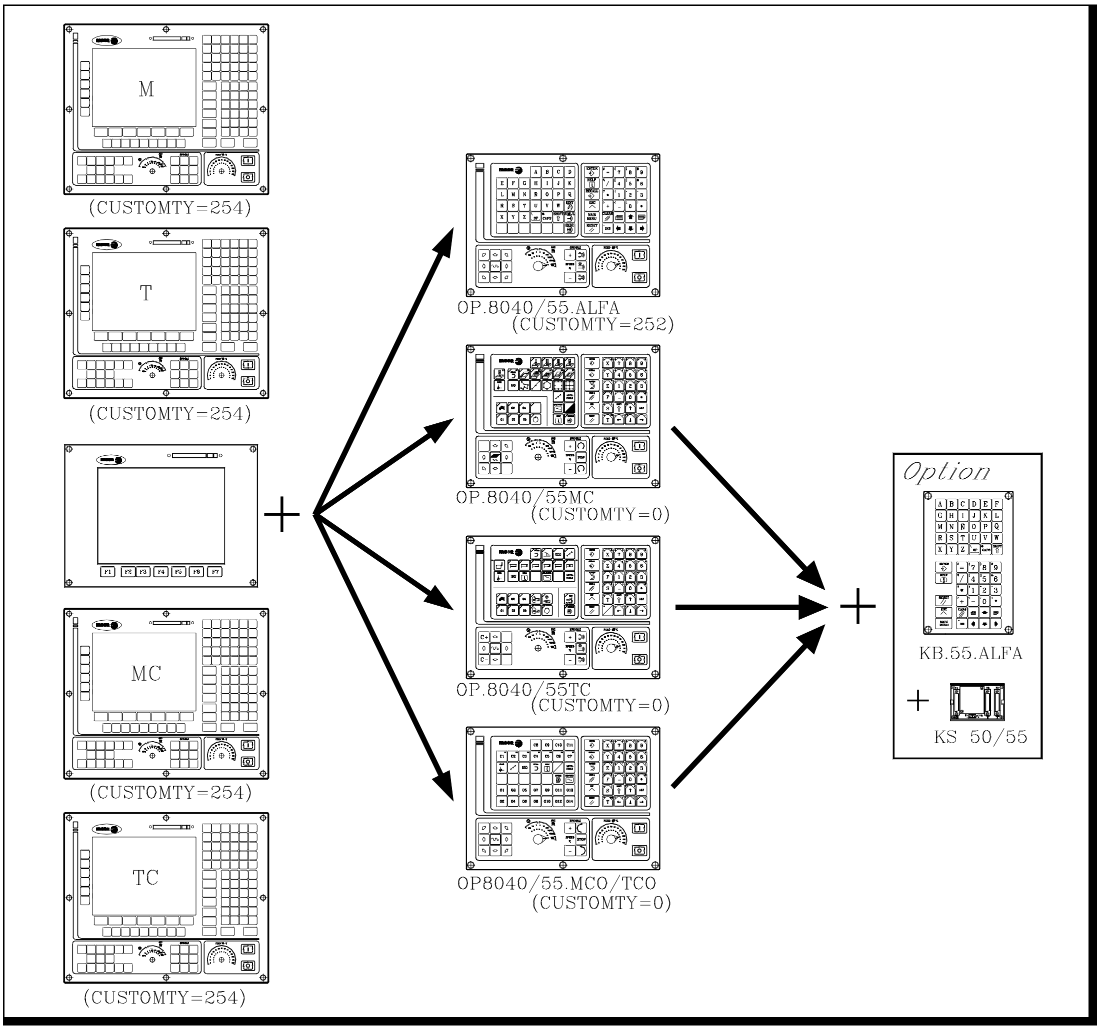
Parameters "MPG1***" correspond to the first handwheel, "MPG2***" to the second one and "MPG3***" to the third one.

The CNC uses the following order to know which one is the first, second and third handwheel: X, Y, Z, U, V, W, A, B, C

The meaning of parameters MPG*CHG, MPG*RES and MPG*NPUL is similar to the meaning of parameters MPGCHG (P80), MPGRES (P81) and MPGNPUL (P82).

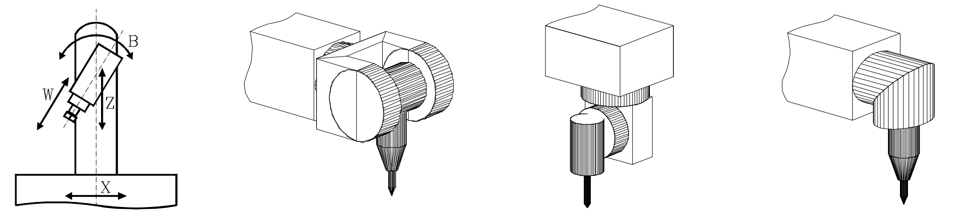
CUSTOMTY (P92) It indicates the configuration being used.

The possible configurations are:



XFORM (P93) Indicates the spindle type.

0 Standard spindle
1 Reserved
2 45º spindle head, spherical or swinging
3 Angled spindle head

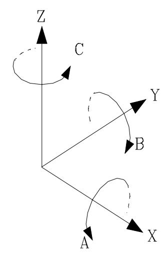


Swinging	45º	Spherical	Angular
XFORM=2	XFORM=2	XFORM=2	XFORM=3

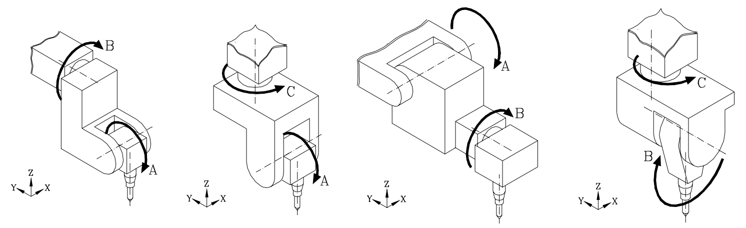
XFORM1 (P94) Sets the spindle axes and their order.

The rotary axes are called A, B or C depending on whether the rotation axis is X, Y or Z respectively.

When having a dual swivel spindle "XFORM=2", parameter "XFORM1" indicates which one is the main axis (carrier) and which is the secondary or the one being carried.

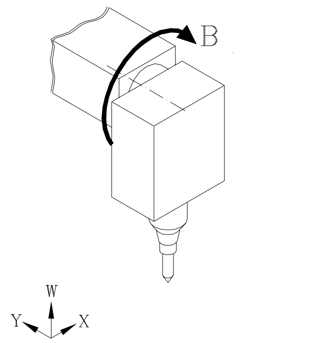


XFORM1=0	B is the main axis and A is the secondary one.
XFORM1=1	C is the main axis and A is the secondary axis
XFORM1=2	A is the main axis and B is the secondary one.
XFORM1=3	C is the main axis and B is the secondary axis



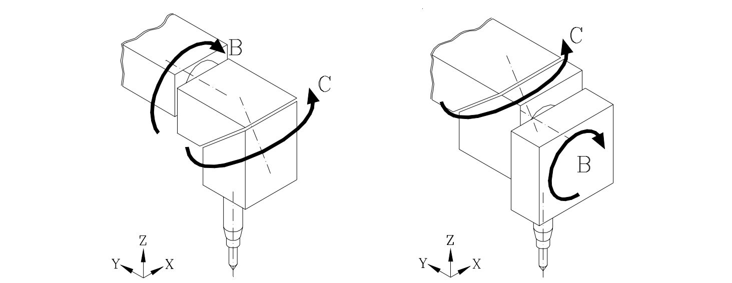
XFORM=2	XFORM=2	XFORM=2	XFORM=2
XFORM1=0	XFORM1=1	XFORM1=2	XFORM1=3

The swinging spindle head “XFORM=2” does not have a secondary or “dragged” axis but parameter “XFORM1” is defined like in the case of the 45º and spherical one.



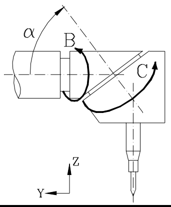
XFORM=2
XFORM1=0

When having an angled spindle "XFORM=3", the main axis must be parallel to one of the axes X, Y, Z and the secondary axis will be at an angle with respect to it.



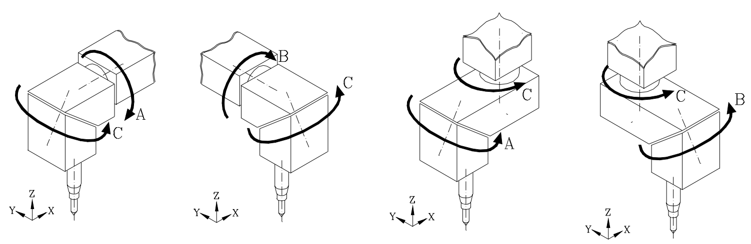
Possible   Not possible

In the example, B is the main axis associated with the Y axis and C is the secondary axis associated with the Z axis.



Parameter "XFORM1" indicates which is the main rotary axis (carrier) and which is the secondary or carried axis.

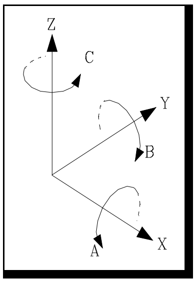
XFORM1=0	A is the main axis and C is the secondary axis
XFORM1=1	B is the main axis and C is the secondary axis
XFORM1=2	C is the main axis and A is the secondary axis
XFORM1=3	C is the main axis and B is the secondary axis



XFORM=3 XFORM1=0   XFORM=3 XFORM1=1   XFORM=3 XFORM1=2   XFORM=3 XFORM1=3

XFORM2 (P95) Defines the rotating direction of the rotary axes.

0 The one indicated by the DIN 66217 standard (see figure)
1 Changes the rotating direction of the main axis
2 Changes the rotating direction of the secondary axis.
3 Changes the rotating direction of both axes (main and secondary).



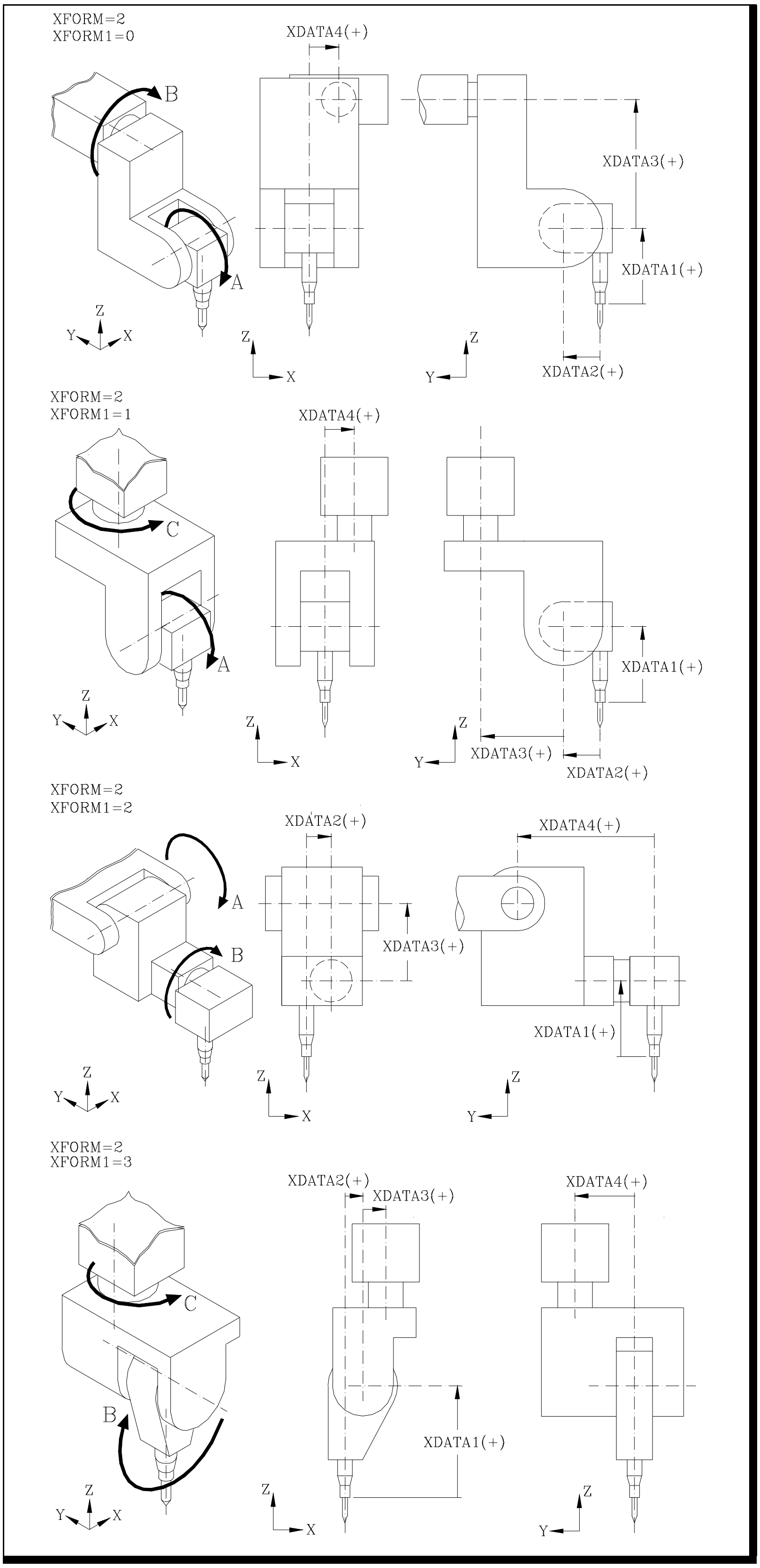
XDATA0 (P96)
XDATA1 (P97)
XDATA2 (P98)
XDATA3 (P99)
XDATA4 (P100)
XDATA5 (P101)
XDATA6 (P102)
XDATA7 (P103)
XDATA8 (P104)
XDATA9 (P105)

These parameters are used to define the dimensions of the spindle.

All of them need not be defined. Next a description of which parameters must be set for each spindle model and their meanings.

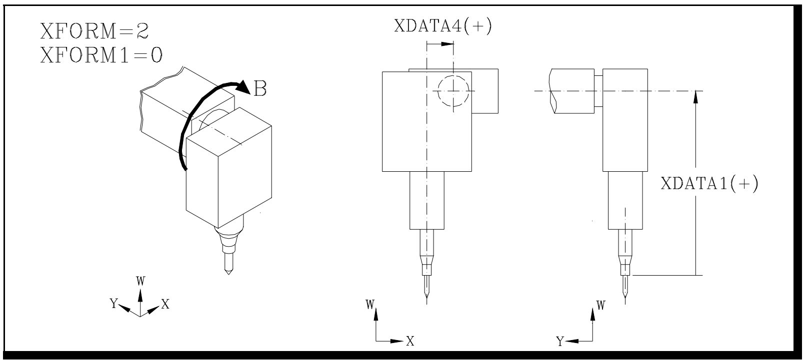
Dual swivel spindle head

XDATA1 (P97)	Distance between the spindle nose and the carried rotary axis (secondary)
XDATA2 (P98)	Distance between the tool axis and the carried rotary axis.
XDATA3 (P99)	Distance between both rotary axes.
XDATA4 (P100)	Distance between the tool axis and the main (carrier) rotary axis. This distance must be measured in the direction of the carried rotary axis.



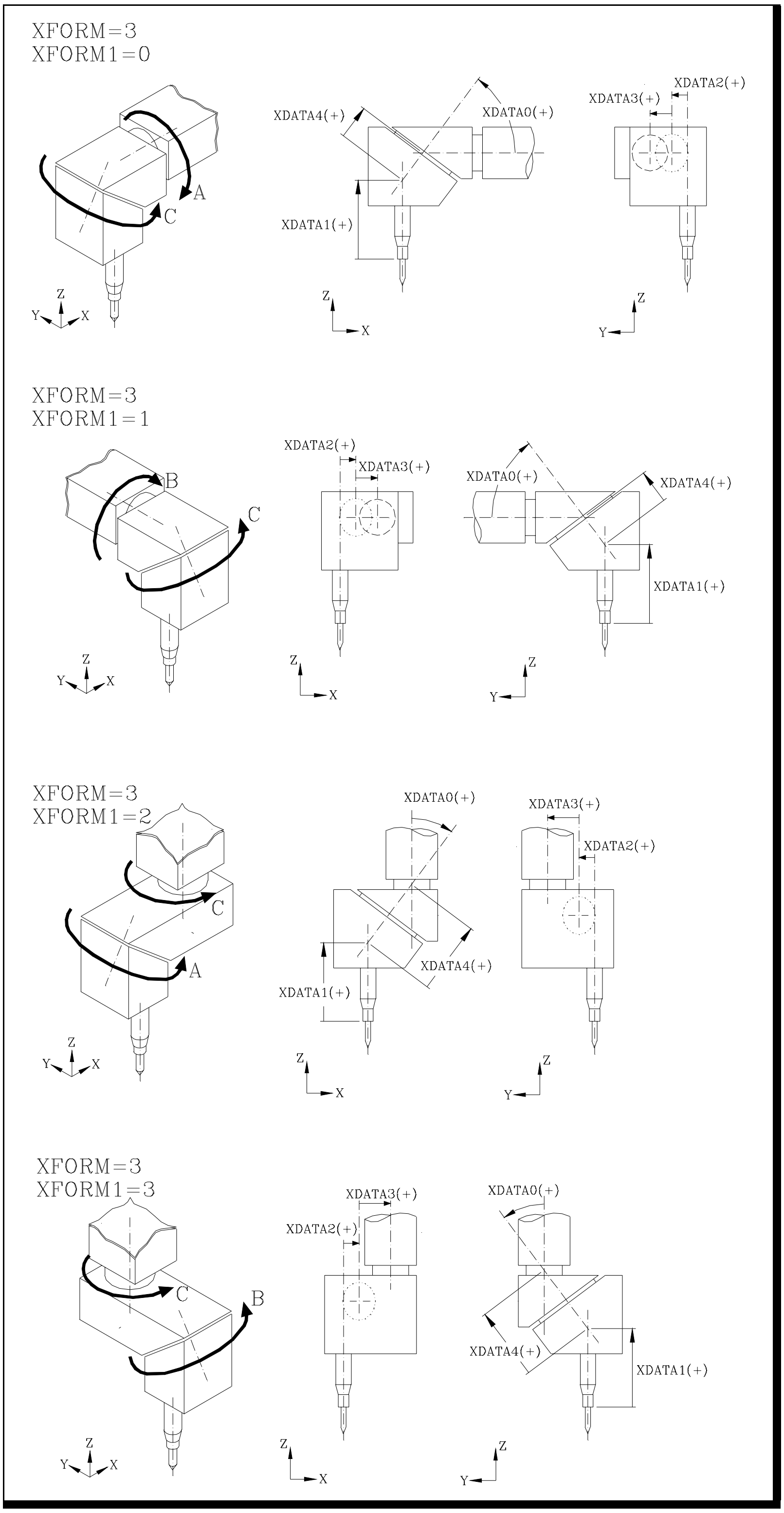
Swinging spindle

XDATA1 (P97)	Distance, when the quill is retracted, between the nose of the quill and the rotary axis along the tool axis (W).
XDATA2 (P98=0)	Distance between the tool axis and the carried rotary axis (there is no secondary).
XDATA3 (P99=0)	Distance between both rotary axes (there is no secondary).
XDATA4 (P100)	Distance between the tool axis and the main (carrier) rotary axis.



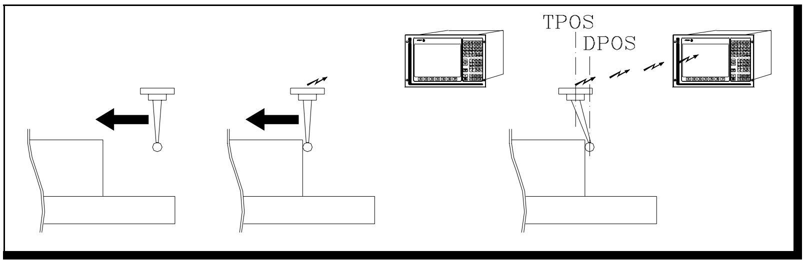
Angled spindle head

XDATA0 (P96)	Angle in degrees between both rotary axes.
XDATA1 (P97)	Distance between the spindle nose and the carried rotary axis (secondary)
XDATA2 (P98)	Distance between the tool axis and the carried rotary axis.
XDATA3 (P99)	Distance between both rotary axes.
XDATA4 (P100)	Distance between the tool axis and the main (carrier) rotary axis. This distance must be measured in the direction of the carried rotary axis.



PRODEL (P106) The CNC takes this parameter into account when probing, functions G75, G76 and Probe cycles.

When the digital probe communicates with the CNC via infrared beams, there could be some delay (milliseconds) from the time the probe touches the part to the instant the CNC receives the probe signal.



The probe keeps moving until the CNC receives the probe signal.

Parameter PRODEL indicates, in milliseconds, the delay mentioned earlier.

Possible values Integers 0 through 255.

By default 0

While probing, the CNC always takes into account the value assigned to parameter PRODEL and provides the following information (variables associated with the coordinates):

TPOS Actual position of the probe when the CNC receives the probe signal.

DPOS Theoretical position of the probe when the probe touched the part.

With "PRODEL=0", the DPOS variable has the same value as the TPOS variable.

To set this parameter, the PROBE2 probe calibration cycle can be used. After it is executed, global parameter P299 returns the best value to be assigned to parameter PRODEL.

MAINOFFS (P107) Indicates whether the CNC maintains the tool offset number (D) on power-up and after an EMERGENCY or RESET.

0 It does not maintain it. It always assumes offset D0.

1 It maintains it.

By default 0 (not maintained)

ACTGAIN2 (P108) The axes and the spindle can have 2 ranges of gains and accelerations.

By default, it always assumes the one indicated by the a.m.p. and s.m.p. ACCTIME (P18), PROGAIN (P23), DERGAIN (P24) and FFGAIN (P25)

Parameter ACTGAIN2 indicates when it assumes the gains and accelerations indicated by a.m.p. . ACCTIME2 (P59), PROGAIN2 (P60), DERGAIN2 (P61) and FFGAIN2 (P62) and s.m.p. ACCTIME2 (P47), PROGAIN2 (P48), DERGAIN2 (P49) and FFGAIN2 (P50)

Parameter ACTGAIN2 has 16 bits starting from left to right. Each bit has a function or operating mode assigned to it.

bit	Function	bit	Function
1	G0	9	G75/G76
2	G1	10	G95
3	G33	11	Rigid tapping
4		12	JOG
5		13	
6		14	
7	G50	15	
8	G51	16	

Every time one of this functions or operating mode is activated, the CNC checks the value set for the corresponding bit and acts as follows:

bit = 0 Applies the first range "ACCTIME, PROGAIN ..."
bit = 1 Applies the second range "ACCTIME2, PROGAIN2 ..."

When that function or operating mode is deactivated, the CNC applies the first range "ACCTIME, PROGAIN ..."

Example: When setting ACTGAIN2 = 1000 0000 0001 0000, the CNC applies the second range to all the axes and the spindle whenever function G1 or the JOG mode is selected.

The change of gains and accelerations is always made at the beginning of the block. When working in round corner (G5), the change does not take place until G07 is programmed.

G2 X10 Y10 I10 J0	*Range 1*	*G05 G2 X10 Y10 I10 J0*	*Range 1*
G1 X20	*Range 2*	*G1 X20*	*Range 1*
G3 X30 Y20 I0	*Range 1*	*G3 X30 Y20 I0 J10 G7 G1*	*Range 1*
J10G1 Y30	*Range 2*	*Y30*	*Range 2*

The gain and accelerations may also be changed from the PLC. To do that, there is a general logic CNC input ACTGAIN2 (M5013).

Every time this input is activated, the CNC selects the second gain and acceleration range regardless of the active operating mode or function.

TRASTA (P109) Not used.

DIPLCOF (P110) With variable PLCOF(X-C), it is possible to set an additive zero offset for each CNC axis from the PLC.

Parameter "DIPLCOF" indicates whether the CNC takes into consideration or not this value when displaying the coordinates of the axes on the screen and when accessing the POS(X-C) and TPOS(X-C) variables.

0 When displaying the position of the axes referred to home, it only takes into account the additive zero offset set by PLC.

The coordinate returned by the POS(X-C) and TPOS(X-C) variables takes into account the additive offset set by PLC.

1 When displaying the position of the axes, it ignores the additive offset set by PLC.

The coordinate returned by the POS(X-C) and TPOS(X-C) variables ignores the additive offset set by PLC.

2 When displaying the position of the axes, the CNC takes into account the additive offset set by the PLC except when showing the COMMAND - ACTUAL -TO GO coordinates.

The coordinate returned by the POS(X-C) and TPOS(X-C) variables takes into account the additive offset set by PLC.

By default 0

HANDWIN (P111)
HANDWHE1 (P112)
HANDWHE2 (P113)
HANDWHE3 (P114)
HANDWHE4 (P115)

The CNC has specific connectors to connect the handwheels.

Optionally, specific digital PLC inputs (I) may also be used to connect handwheels (signals A and B). In these cases, the handwheels must operate at 24V.

g.m.p. HANDWIN (P111) indicates which input group the electronic handwheels are associated with. Possible values: 0, 17, 33, 49, 65, 81, 97, 113, 129, 145, 161, 177, 193, 209, 225, 241,.....

HANDWIN = 0	There is no handwheel connected to the PLC inputs
HANDWIN = 17	Handwheels connected to the input group I17 through I25
HANDWIN = 33	Handwheels connected to the input group I33 through I41
HANDWIN = 225	Handwheels connected to the input group I225 through I240
HANDWIN = 241	Handwheels connected to the input group I241 through I256

The meaning of these inputs is the following:

I17	I33...	I225	I241	button signal from the handwheel with selector button (only the first one)
I18	I34...	I226	I242	A signal from the first handwheel.
I19	I35...	I227	I243	B signal from the first handwheel.
I20	I36...	I228	I244	A signal from the second handwheel.
I21	I37...	I229	I245	B signal from the second handwheel.
I22	I38...	I230	I246	A signal from the third handwheel.
I23	I39...	I231	I247	B signal from the third handwheel.
I24	I40...	I232	I248	A signal from the fourth handwheel.
I25	I41...	I233	I249	B signal from the fourth handwheel.

To define the type of handwheel and its associated axis, use the following g.m.p.:

HANDWHE1 (P112)	for the first handwheel
HANDWHE2 (P113)	for the second handwheel
HANDWHE3 (P114)	for the third handwheel
HANDWHE4 (P115)	for the fourth handwheel

The values to be assigned to these parameters are:

11	Handwheel	12	Handwheel with axis selector button
21	Handwheel associated with X	22	Handwheel associated with Y
23	Handwheel associated with Z	24	Handwheel associated with U
25	Handwheel associated with V	26	Handwheel associated with W
27	Handwheel associated with A	28	Handwheel associated with B
29	Handwheel associated with C		

Either one general handwheel can be used (11 or 12) or 3 handwheels associated with the axes. In other words, it is not possible to use 2 general handwheels or to combine a general handwheel with any other/s associated with the axes.

STOPTAP (P116) Indicates whether the general inputs /STOP (M5001), /FEEDHOL (M5002) and /XFERINH (M5003) are enabled (P116=YES) or not (P116=NO) while executing function G84, regular tapping or rigid tapping.

INSFEED (P117) Sets the tool inspection feedrate.

When accessing tool inspection, the CNC assumes this feedrate as the new one, and it resumes the execution of the program at the previous feedrate (the one used in the program or set via MDI while in tool inspection) when tool inspection is over.

Possible values:

0.0001.... 199999.9999 degrees/min or mm/min.

0.00001.... 7874.01574 inches/min.

If set to "0" (by default), tool inspection will be carried out at the feedrate currently used for machining.

DISTYPE (P118) Only to be used by Fagor Automation technical personnel.

PROBERR (P119) Indicates whether the CNC issues an error message when the axes reach the programmed position without having received the probe signal while executing function G75 or G76.

YES	It issues the error message.	
NO	It does NOT issue the error message.	*Default value*

SERSPEED (P120) Sets the Sercos communications speed (baudrate).

Possible values:

0	4 Mbit/s	*Default value*
1	2 Mbit/s	
80	Sercos test. Continuous signal mode.	
81	Sercos test. Zero bit stream mode at 2 Mbit/s	
91	Sercos test. Zero bit stream mode at 4 Mbit/s	

SERPOWSE (P121) Set the power or intensity of the sercos light traveling through the optic fiber.

Recommended values:

2 for cables of up to 7 meters (23 ft)
4 for cables between 7 and 15 meters (23 ft and 49 ft)
6 for cables longer than 15 meters (23 ft)

By default 0

Assigning these values, for example a value of 4 for a length of 3m, causes communication errors due to signal distortion in the optical fiber.

LANGUAGE (P122) Sets the operating language

Possible values:

0 English	1 Spanish	2 Portuguese
3 Italian	4 German	5 Dutch
6 Portuguese	7 Czech	8 Polish
9 Mainland Chinese		*By default 0*

GEOMTYPE (P123) It indicates whether the cutter geometry is associated with the tool (T) or with the tool offset (D).

The "T" function, tool number, indicates the magazine position it occupies.

The "D" function, offset, indicates the tool dimensions.

0 It is associated with the tool *Default value*
1 It is associated with the tool offset

When using a tool holding turret, the same turret position is usually used by several tools. In those cases, the "T" function refers to the turret position and the "D" function to the dimensions and geometry of the tool occupying that position. Thus, "GEOMTYPE=YES".

SPOSTYPE (P124) Not used.

AUXSTYPE (P125) It indicates whether the live tool is handled with the M45 function or like a second spindle (G28 function)

0 Using function M45.
1 As second spindle (function G28).

By default 0

When a live tool uses several work ranges, it must be used like a second spindle. To do that:

- Set "AUXSTYPE (P125)=1"
- Define the machine parameters of the second spindle to set the live tool.
- Use the G28 function to select the live tool.

In the Mill model, with "AUXTYPE (P125)=1" and "STOPTAP (P116)=YES" it is possible to interrupt the execution of the drilling and tapping canned cycles by means of the general inputs /STOP (M5001), /FEEDHOL (M5002) and /XFERINH (M5003).

FOVRG75 (P126) It indicates whether function G75 ignores the Feedrate Override switch of the front panel or not.

NO It ignores the setting of the switch. Always at 100%
YES It is affected by the % of the switch.

By default, NO

CFGFILE (P127) Number of the file to configure the windows that may be customized.

STEODISP (P128) It indicates whethe the CNC displays the real or theoretical RPM (affected by the %) of the main spindle.

0 it displays the real RPM. *Default value*

1 it displays the theoretical RPM

When not having spindle encoder (NPULSES=0), it is recommended to set P128=1 so it displays theoretical value.

HDIFFBAC (P129) This parameter has 16 bits from left to right.

bit	Function	bit	Function
1	It limits the movement.	9	
2		10	
3		11	
4		12	
5		13	4th handwheel
6		14	3rd handwheel
7		15	2nd handwheel
8		16	1st handwheel

Bit 1 indicates how the CNC acts when requesting a feedrate greater than the maximum allowed depending on the handwheel turning speed and the position of the switch.

(0) It limits the feedrate to the maximum allowed but it moves the indicated distance.

(1) It limits the feedrate and the distance to the maximum allowed. The movement stops when the handwheel stops. It does not move the indicated distance.

The individual handwheels, those associated with each axis, always limit the feedrate and the distance.

bits 13, 14, 15 and 16 indicate whether the handwheels output differential signals (1) or not (0).

By default, all the bits will have a value of 0.

RAPIDEN (P130) It indicates whether the rapid key affects the execution and simulation or not.

0 It has no effect *Default value*

1 It does affect them

The rapid key is treated in execution and simulation as follows:

- The movements are carried out in rapid traverse (G00) while the rapid key is pressed.
- The rapid key is ignored while threading, while look-ahead is active.
- If G95 is active, it switches to G94 mode. When releasing the rapid key, it goes back to G95 mode.
- It only affects the main channel. It is ignored in the PLC channel.

MSGFILE (P131) Number of the program that contains the OEM texts in several languages.

By default, the CNC sets this parameter to “0” (there is no program).

If programmed with a value of “0”, the texts defined by the OEM are in a single language and stored in several programs:

PLCMSG	Texts for PLC messages
PLCERR	Texts for PLC error messages
P999995	Texts and titles used by all the OEM screens.
P999994	Help texts of the OEM screens or cycles.

The MSGFILE program may be in user memory or in the Memkey Card. If it is in both places, it takes the one in user memory.

FLWEDIFA (P132) Not being used at this time.

RETRACAC (P133) It indicates whether Retracing is allowed or not

0 It is not allowed *Default value*

1 It is desired.

Retracing is activated and deactivated with the RETRACE (M5051) signal.

If while executing a part program, the PLC sets this signal high, the CNC interrupts the execution of the program and starts executing backwards what has executed so far.

When the PLC sets the Retrace signal back low and retracing is canceled. The CNC starts executing forward what was done backwards and it will go on to execute the part of the program that was not machined.

G15SUB (P134) It is used on the Lathe model CNC. Indicates the number of the subroutine associated with function G15.

Possible values: Integers between 0 and 9999.

By default 0 (there is no associated subroutine)

When there is an associated subroutine, the CNC acts as follows:

- If there is no other G15 inside the associated subroutine, the G15 will be executed after the subroutine.
- If there is another G15 inside the associated subroutine, it will execute this G15 without calling the subroutine and after executing the associated subroutine, it will not execute the G15 again.

This feature may be used to change the set of drive parameters when switching from spindle mode to C axis mode.

TYPCROSS (P135) It indicates whether cross compensation is carried out using theoretical or real coordinates.

0 Using real coordinates *Default value*

1 Using theoretical coordinates

AXIS9 (P136)
PAXIS9 (P137) With g.m.p. AXIS1 (P0) through AXIS8 (P7), it is possible to associate axes, handwheels, spindles or live tools with each feedback input and analog outputs of the CNC.

When using parameters AXIS1 through AXIS8, it is possible to have an auxiliary spindle controlled by the PLC with parameters AXIS9 and PAXIS9.

Setting: AXIS9 = 13 PAXIS9 = 0

It is possible to work as if it had an auxiliary spindle, in other words, no error will be generated when executing M45 and it will be possible to execute the cycles that involve the auxiliary spindle.

There will be no machine parameter table associated with the auxiliary spindle.

The analog output associated with the auxiliary spindle will be managed through the PLC.

AXIS10 (P138)
PAXIS10 (P139)
AXIS11 (P140)
PAXIS11 (P141)
AXIS12 (P142)
PAXIS12 (P143)

Not being used at this time.

ACTBACKL (P144)

It is related to a.m.p. BACKLASH (P14), leadscrew backlash compensation due to change of direction.

It has 16 bits from left to right. Bit 3 indicates whether the compensation is applied only on circular G2/G3 paths (1) or on any kind of movement (0).

bit	Function	bit	Function
1		9	
2		10	
3	G2/G3	11	
4		12	
5		13	
6		14	
7		15	
8		16	

By default, all the bits will have a value of 0.

ACTBAKAN (P145)

It is related to a.m.p. BAKANOUT (P29) and BAKTIME (P30), additional analog pulse to recover the possible leadscrew backlash when reversing the movement.

It has 16 bits from left to right. Bit 3 indicates whether the additional pulse is applied only on circular G2/G3 paths (1) or on any kind of movement (0).

bit	Function	bit	Function
1		9	
2		10	
3	G2/G3	11	
4		12	
5		13	
6		14	
7		15	
8		16	

By default, all the bits will have a value of 0.

3.2 Axis parameters

AXISTYPE (P0) Defines the axis type and whether it is commanded from the CNC or the PLC

0 Normal linear axis.
1 Rapid positioning linear axis (G00).
2 Normal rotary axis.
3 Rapid positioning rotary axis (G00).
4 Rotary axis with HIRTH toothing (positioning in whole degrees).
5 Normal linear axis commanded from the PLC.
6 Rapid positioning linear axis (G00) commanded from the PLC.
7 Normal rotary axis commanded from the PLC.
8 Rapid positioning rotary axis (G00) commanded from the PLC.
9 Rotary axis with HIRTH toothing (positioning in whole degrees) commanded from the PLC.

By default 0

By default, rotary axes are Rollover and are displayed between 0º and 359.9999º.

If rollover is not desired, set a.m.p. ROLLOVER (P55)=NO. The axis position will be displayed in degrees.

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

See section ***4.1 Axes and coordinate systems***

DFORMAT (P1) Indicates the work units (radius or diameter) and the display format used for the axis.

Value	Work units	Format in degrees	Format in mm	Format in inches
0	mm	5.3	5.3	4.4
1	mm	4.4	4.4	3.5
2	mm	6.2	6.2	5.3
3	mm	It is not displayed		
4	inches	5.3	5.3	4.4
5	inches	4.4	4.4	3.5
6	inches	6.2	6.2	5.3

GANTRY (P2) Indicates, if it is a GANTRY axis, which axis is this one associated with. This parameter is to be set only on the slaved axis according to the following code.

0	No Gantry	1	to X	2	to Y	3	to Z	4	to U
5	to V	6	to W	7	to A	8	to B	9	to C

By default 0 (not Gantry)

Several Gantry pairs are allowed.

The position of the Gantry axis is displayed next to its associated axis unless machine parameter "DFORMAT(P1)=3".

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

Parameter GANTRY for X axis = 0

Parameter GANTRY for U axis = 1 (associated with X axis)

This way, When programming an X axis move, the U axis will also move the same distance.

SYNCHRO (P3) It is possible to couple or decouple each one of the axes by PLC program using the logic inputs of the CNC: “SYNCHRO1” through “SYNCHRO6”.

Each axis will be synchronized to the axis indicated in its machine parameter “SYNCHRO”. Indicates which axis will this one (slave) be coupled to when it so requested by the PLC.

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

By default 0

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

SYNCHRO of the X axis = 0

SYNCHRO of the V axis = X

When the PLC activates the logic input “SYNCHRO” of the CNC corresponding to the V axis, this axis will be electronically coupled to the X axis.

DROAXIS (P4) Indicates whether it is a normal axis or it only works as a Digital Read Out

NO It is a normal axis.

YES It only works as a Digital Read Out.

By default, NO

LIMIT+ (P5)
LIMIT - (P6)

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

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

By default "LIMIT+" = 8000 mm. and "LIMIT-" = -8000 mm.

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

On rotary axes:

- *When both parameters are set to "0", the axis may be moved indefinitely in any direction (rotary tables, indexers, etc.)*
- *For example, the "C" axis with P5=0, P6=720 and the axis positioned at 700 (the screen displays 340) if G90 C10 is programmed, this axis tries to go via the shortest path (701, 702,...) but it issues an error message since it exceeds the travel limit. For example, C axis with P5=0, P6=720 and the axis position at 700 (340 on the screen), G90 C10 is programmed and the CNC tries to move it via the shortest path (701, 702, etc.), but it issues an error message because it exceeds the travel limits.*
- *if on positioning-only axes and Hirth axes their travel is limited to less than a revolution, the movement cannot be carried out via the shortest path.*
- *When the travel is limited to less than a revolution and a positive and negative display is desired, for example P5=-120, P6=120, it is possible to program G90 with positive and negative values.*

PITCH (P7)

Defines the pitch of the ballscrew or the linear feedback device (scale) being used. When using a Fagor linear encoder, the signal pitch to be entered here will be either 20 µm or 100 µm.

When dealing with a rotary axis, it must indicate the number of degrees per encoder turn. For example, if the encoder is mounted onto a motor with a 1/10 gear reduction, this parameter must be set to 360º/10 = 36.

Possible values: 0.0001.... 99999.9999 mm
0.00001.... 3937.00787"

By default 5 mm.

NPULSES (P8)

Indicates the number or pulses/rev provided by the rotary encoder. Enter a value of 0 when using linear encoders. When using gear ratios, the whole assembly must be taken into account when defining the number of pulses per turn.

Possible values: Integers between 0 and 65535.

By default 1250

DIFFBACK (P9)

Indicates whether the spindle encoder uses differential signals (double ended) or not.

NO	It does not use them	
YES	It uses them	*Default value*

SINMAGNI (P10) Indicates the multiplying factor (x1, x4, x20, etc.) that the CNC must apply only to sinewave feedback signals.

Set this parameter to "0" when using squarewave feedback signals and the CNC will always apply a x4 multiplying factor.

Possible values: Integers between 0 and 255.

By default 0

The counting resolution of the CNC is determined using a.m.p. PITCH (P7), NPULSES (P8) and SINMAGNI (P10) as shown in the following table:

	PITCH	NPULSES	SINMAGNI
Square signal encoder	Ballscrew pitch	# of Lines	0
Sinusoidal signal encoder	Ballscrew pitch	# of Lines	multiplying factor
Square signal linear encoder	linear encoder pitch	0	0
Sinusoidal signal linear encoder	linear encoder pitch	0	multiplying factor

FBACKAL (P11) This parameter is to be used only when the feedback signals are sinusoidal or differential (double ended).

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

OFF Not desired, it is canceled.

ON Alarm on.

By default, ON

FBALTIME (P12) It indicates the maximum time period given to the axis to respond to the analog voltage output by the CNC.

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

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

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

Possible values Integers between 0 and 65535 ms.

By default 0 (unchecked)

AXISCHG (P13) Indicates the counting direction. If correct, leave it as is; if not, change it from YES to NO or viceversa. If this parameter is changed, a.m.p. LOOPCHG (P26) must also be changed so the spindle does not “run away”.

Possible values: YES and NO.

By default, NO

BACKLASH (P14) Indicates the amount of backlash. Enter 0 when using linear encoders.

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

By default 0

LSCRWCOM (P15) Indicates whether the CNC should apply leadscrew error compensation or not.

OFF It is not desired.

ON Leadscrew compensation being used.

By default, OFF

NPOINTS (P16) Indicates the number of leadscrew error compensation points available in the table. The values in this table will be applied if a.m.p. "LSCRWCOM" (P15) is ON.

Possible values: Integers between 0 and 255.

By default 30

DWELL (P17) Indicates the dwell from the moment the "ENABLE" signal is activated until the analog voltage is sent out.

Possible values Integers between 0 and 65535 ms.

By default 0 (not available)

ACCTIME (P18) Defines the acceleration stage or the time it takes the axis to reach the feedrate selected with a.m.p. GOFFED (P38). This value also represents the deceleration time.

Possible values Integers between 0 and 65535 ms.

By default 0 (not available)

INPOSW (P19) Indicates the width of the IN POSITION zone (dead band) where the CNC considers the axis to be in position.

Possible values: 0.... 99999.9999 degrees or mm.
0.... 3937.00787 inches.

By default 0.01 mm

INPOTIME (P20) Indicates the time period that the axis must remain in the "IN POSITION" zone in order to consider it to be in position.

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

Possible values Integers between 0 and 65535 ms.

By default 0

MAXFLWE1 (P21) Indicates the maximum following error allowed when this axis moves.

Possible values: 0.... 99999.9999 degrees or mm.
0.... 3937.00787 inches.

By default 30 mm.

MAXFLWE2 (P22) Indicates the maximum following error allowed when this axis is stopped.

Possible values: 0.... 99999.9999 degrees or mm.
0.... 3937.00787 inches.

By default 0.1 mm.

PROGAIN (P23) Indicates the value of the Proportional Gain. It indicates the mV of analog voltage for a following error (axis lag) of 1 mm.

Analog voltage (mV) = Following error (mm) x PROGAIN

Possible values Integers between 0 and 65535 mV/mm.

By default, 1000mV/mm

Example:

Setting a.m.p. G00FEED (P38) = 20000 mm/min and the feedrate for the desired following error of 1mm (0.040") is F=1000mm/min.

Drive analog: 9.5 V for a feedrate of 20,000 mm/min

Analog corresponding to F = 1000 mm/min:

Analog = (9.5/20000) x 1000 = 475mV

Therefore, “PROGAIN” = 475

DERGAIN (P24) Indicates the value of the Derivative Gain. Its value represents the analog voltage (in millivolts) corresponding to a change in following error of 1mm (0.03937 inches) in 10 milliseconds.

This analog voltage will be added to the one calculated for the Proportional Gain.

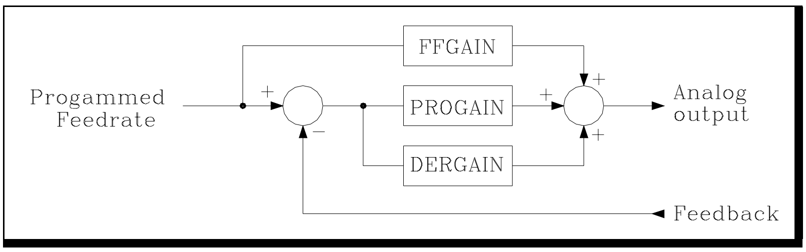
$$\text{Command} = \left(\xi \cdot \mathrm{PROGAIN} + \frac{\xi \cdot \mathrm{DERGAIN}}{10 \cdot t}\right)$$

It is a good idea to also use the acc./dec. a.m.p. ACCTIME2 (P18) for this axis (with a value other than “0”) if this gain is to be applied.

Possible values: Integer value between 0 and 65535.

By default “0”, (no derivative gain applied)

FFGAIN (P25) Indicates the % of the analog voltage due to the programmed feedrate. The rest will depend upon the following error. Both the Proportional and Derivative gains will be applied onto this following error..



Command

$$= \left(\xi \cdot \mathrm{PROGAIN} + \frac{\xi \cdot \mathrm{DERGAIN}}{10 \cdot t} + \frac{\mathrm{FFGAIN} \times \mathrm{Fprog} \times \mathrm{MAXVOLT}}{100 \cdot \mathrm{G00FEED}}\right)$$

The Feed-Forward Gain lets improve the positioning loop minimizing the following error and it should be used when the “ACCTIME” machine parameter for this axis is active (acc/dec. being applied).

Possible values: Integers between 0 and 100.

By default “0”, (no feed-forward gain applied)

Usually, a value between 40% and 80% is assigned depending mainly on the type of machine and its characteristics.

LOOPCHG (P26) Indicates the sign of the analog output. If correct, leave it as is; if not, change it from YES to NO or viceversa.

Possible values: YES and NO.

By default, NO

MINANOUT (P27) Indicates the minimum analog output for this axis.

It is given in D/A converter units and it admits integer values between 0 and 32767 which corresponds to an analog voltage of 10V. By default 0.

MINANOUT	Minimum analog voltage
1	0.3 mV
-----	-----
3277	1 V
-----	-----
32767	10 V

SERVOFF (P28) Indicates the analog offset value for the spindle drive.

It is given in D/A converter units and it admits integer values between 0 and +32767 which corresponds to an analog voltage of +10V. By default 0 (not applied).

SERVOFF	**Command**
-32767	-10 V
-----	-----
-3277	-1 V
-----	-----
1	0.3 mV
-----	-----
3277	1 V
-----	-----
32767	10 V

BAKANOUT (P29) Additional analog pulse to compensate for backlash when changing movement direction.

It is given in D/A converter units and it admits integer values between 0 and 32767 which corresponds to an analog voltage of 10V. By default 0 (not applied).

BAKANOUT	Additional analog voltage
1	0.3 mV
-----	-----
3277	1 V
-----	-----
32767	10 V

Every time the axis changes direction, the CNC will apply the analog voltage corresponding to the move plus the one corresponding to The value selected in this machine parameter. This additional analog voltage will be applied for a period of time indicated in the a.m.p. BAKTIME (P30).

BAKTIME (P30) Indicates the duration of the additional analog pulse to make up for the backlash when reversing the movement.

Possible values Integers between 0 and 65535 ms.

By default 0

DECINPUT (P31) Indicates whether or not this axis has a home switch for machine reference search.

NO It doesn't have one.
YES it has one.

By default, YES

REFPULSE (P32) Indicates the type of flank of the marker pulse Io used for searching home.

\+ Up flank (change from 0V to 5V).

\- Down flank (change from 5V to 0V).

By default "+"

REFDIREC (P33) Indicates the direction of the home search in this axis.

\+ Positive direction. *Default value*

\- Negative direction.

REFEED1 (P34) Indicates the axis feedrate when searching home until it hits the home switch.

Possible values:

0.0001.... 199999.9999 degrees/min or mm/min.

0.00001.... 7874.01574 inches/min.

By default, 1000 mm/min

REFEED2 (P35) Indicates the axis feedrate when searching home after hitting the home switch until it finds the marker pulse (Io).

Possible values:

0.0001.... 99999.9999 degrees/min or mm/min.

0.00001.... 3937.00787 inches/min.

By default, 100 mm/min

REFVALUE (P36) Indicates the position value of the machine reference point (physical location of the marker pulse) with respect to machine reference zero.

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

By default 0

The machine reference point is set by the manufacturer to synchronize the coordinate system. The machine positions the axis at this point instead of moving it to the machine reference zero point.

When the machine uses semi-absolute scales (with coded marker pulses), the axis may be homed anywhere within its travel. Thus, this parameter must only be set when applying leadscrew error compensation. The amount of leadscrew error to be assigned to this point is "0".

With Sercos connection, when the drive uses absolute feedback, instead of parameter "REFVALUE", it takes into account its equivalent "SERCOS 177" of the drive.

MAXVOLT (P37) Indicates the maximum analog voltage corresponding to the maximum feedrate of the axis indicated by a.m.p. G00FEED (P38).

Possible values: Integer value between 0 and 9999 mV.

By default, 9500 (9.5 V).

G00FEED (P38) Indicates the maximum feedrate G00 (rapid traverse) of this axis.

Possible values:

0.0001.... 199999.9999 degrees/min or mm/min.

0.00001.... 7874.01574 inches/min.

By default, 10000 mm/min

UNIDIR (P39) Indicates the direction of the unidirectional approach in G00 moves.

\+ Positive direction. *Default value*

\- Negative direction.

OVERRUN (P40) Indicates the distance to be kept between the approach point and the programmed point. If it is a Lathe model, this distance must be in radius.

Possible values:

0.0001.... 99999.9999 degrees or millimeters.

0.00001.... 3937.00787 inches

By default 0 (no unidirectional approach)

UNIFEED (P41) Indicates the feedrate to be used from the approach point to the programmed point.

Possible values:

0.0001.... 99999.9999 degrees/min or mm/min.

0.00001.... 3937.00787 inches/min.

By default 0

MAXFEED (P42) Indicates the maximum programmable feedrate (F0).

Possible values:

0.0001....199999.9999 degrees/min or mm/min.

0.00001.... 7874.01574 inches/min.

By default, 5000 mm/min

JOGFEED (P43) Indicates the feedrate F assumed in the JOG mode if no feedrate is active.

Possible values:

0.0001.... 199999.9999 degrees/min or mm/min.

0.00001.... 7874.01574 inches/min.

By default, 1000 mm/min

PRBFEED (P44) Indicates the probing feedrate when calibrating a tool in "JOG" mode.

Possible values:

0.0001.... 99,999.9999 mm/min.

0.00001.... 3937.00787 inches/min.

By default, 100 mm/min

MAXCOUPE (P45) Indicates the maximum difference allowed between the following errors of the axes electronically coupled (by program, PLC or as GANTRY axes).

This value is only assigned to the slave axis.

Possible values:

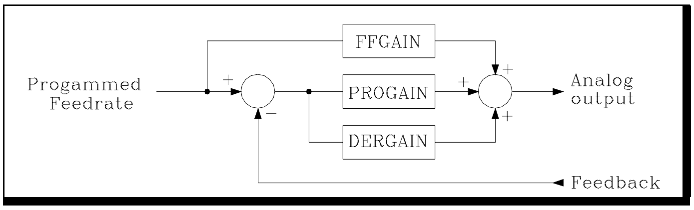
0.0001.... 99999.9999 millimeters.

0.00001.... 3937.00787 inches.

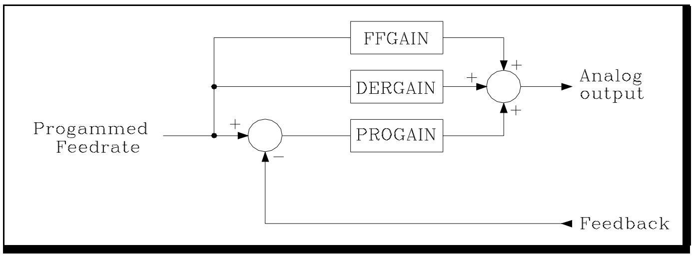
By default 1 mm

ACFGAIN (P46) Indicates whether or not the value assigned to a.m.p. DERGAIN (P24) is applied onto the variations of the programmed feedrate (AC-forward).

NO It is applied onto the variations of following error (derivative gain).



YES It is applied onto the variations of programmed feedrate due to ACC/DEC (AC-forward).



By default, NO

REFSHIFT (P47) This parameter is used when once the machine has been all set up, it is necessary to reinstall the feedback system and the new machine reference point (home) no longer coincides physically with the previous one.

It indicates the difference (shift) between those two reference points (previous and current)

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

By default 0

If this parameter has a value other than “0” , when searching home, the axis will move this additional distance ("REFSHIFT (P47)" value) after finding the new marker pulse. This way, the machine reference point (home) will still be the same.

This movement is carried out at the feedrate indicated by a.m.p. REFEED2 (P35).

STOPTIME (P48)
STOPMOVE (P49) These parameters are used in conjunction with a.m.p. "STOPAOUT (P50)" with function G52 (move to hardstop).

The CNC considers that the hardstop has been run into when a certain time period elapses without the axis moving. This time period is indicated, in thousands of a second, by parameter STOPTIME (P48).

Possible values: Integers between 0 and 65535 ms.

By default 0.

The CNC considers the axis to be stopped when its movements do not exceed the value set by STOPMOVE (P49) during the time period set by STOPTIME (P48).

Possible values: 0.0001.... 99999.9999 mm.
0.00001.... 3937.00787 inches.

By default 0

STOPAOUT (P50) This parameter is used with function G52 (move to hardstop) and it indicates the residual analog voltage supplied by the CNC to exert pressure once contact has been detected.

It is given in D/A converter units and it admits integer values between 0 and 32767 which corresponds to an analog voltage of 10V. By default 0.

STOPAOUT	Minimum analog voltage
1	0.3 mV
-----	-----
3277	1 V
-----	-----
32767	10 V

Note: This parameter is especially designed for hydraulic devices.
When using servo motors, first reduce the maximum torque of the drive by means of an "M" function in order to prevent the motor from overheating.

INPOSW2 (P51) This parameter is used when function G50 (controlled round corner) is active.

It defines the area before the programmed coordinate where the CNC considers the axis to be in position and goes on to execute the next block.

Possible values:

0.... 99999.9999 degrees or millimeters.

0.... 3937.00787 inches.

By default 0.01 mm

It should be assigned a value 10 times the value of "INPOSW"

I0TYPE (P52) It indicates the type of Io signal (marker pulse) provided by the feedback device.

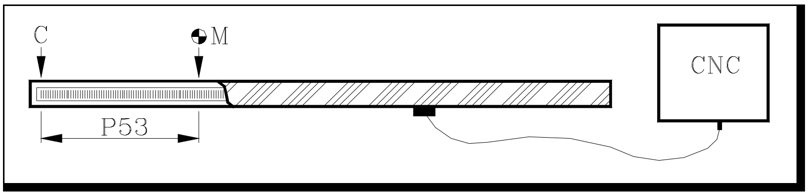
0 normal I0.
1 A type of distance-coded Io
2 B type distance-coded Io

When using linear encoders with distance-coded reference marks (I0), set a.m.p. I0CODI1 (P68) and I0CODI2 (P68).

ABSOFF (P53) The CNC takes this parameter into consideration when a.m.p. I0TYPE (P52) is set with a value other than "0".

Linear encoders having a distance-coded reference mark indicate the machine position with respect to the "zero" of the linear encoder.

In order for the CNC to show the position of the axes with respect to the machine reference zero (home), this parameter must be assigned the position value (coordinate) of the machine reference zero (point "M") with respect to the "zero" of the linear encoder (C).



Possible values: ±99999.9999 millimeters.
±3937.00787 inches.

By default 0

MINMOVE (P54) This parameter has to do with the axis logic outputs "ANT1" through "ANT6".

If the axis move is smaller than the value indicated by this a.m.p. MINMOVE (P54), the corresponding axis logic output "ANT1 through "ANT6" goes high.

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

By default 0

ROLLOVER (P55) This machine parameter is taken into account when the axis has been set as rotary, "AXISTYPE (P0) = 2 or 3". It indicates whether the rotary axis is also rollover or not.

NO It is NOT Rollover.
YES It is Rollover.

By default, YES

SERCOSID (P56) Indicates the sercos address (device select code) associated with the axis.

Possible values: 0 Analog axis
1-8 Sercos address (device select code)

By default 0

These addresses for the various axes and spindles must be sequential and starting from "1". That is, with 3 sercos axes and one sercos spindle, the values for this parameter must be 1, 2, 3, 4.

EXTMULT (P57) This parameter is to be used when utilizing a distance-coded feedback system.

It indicates the relationship between the mechanical pitch or period of the glass or steel tape graduation and the electrical pitch or the period of the feedback signal supplied to the CNC.

$$\text{EXTMULT} = \frac{\text{Glass graduation pitch (mechanical pitch)}}{\text{feedback signal period (electrical pitch)}}$$

For instance, the FAGOR "FOT" linear encoder has a glass graduation pitch of 100 µm while its output signals have an electrical pitch of 20 µm.

EXTMULT = 100 / 20 = 5

Values to be allocated for semi-absolute Fagor linear encoders (with coded Io):

SOP SVOP	GOP	MOT MOC MOP	COT COC COP			EXTMULT = 20/20	= 1
SOX SVOX	GOX	MOX	COX			EXTMULT = 4/20	= 5
		MOY	COY			EXTMULT = 2/20	= 10
				LOP		EXTMULT = 40/40	= 1
				LOX		EXTMULT = 40/4	= 10
					FOT	EXTMULT = 100/20	= 5
					FOX	EXTMULT = 100/4	= 25
					FOP	EXTMULT = 100/100	= 1
						By default 0	

SMOTIME (P58) Sometimes the spindle does not respond as desired on particular movements. Handwheel jog, etc.

In these cases, the response of the axis may be smoothed by applying a filter to the speed changes.

This filter is defined by means of parameter SMOTIME which indicates the duration of the filter in milliseconds which in turn is set by g.m.p. LOOPTIME (P72).

Possible values:

Integers between 0 and 64 times the value assigned to g.m.p. LOOPTIME(P72)

If LOOPTIME=0 (4ms) the maximum value for SMOTIME will be 64 x 4 = 256 ms.

By default 0

In order to obtain a better response, the SMOTIME parameter of all axes interpolating together should be set to the same value.

ACCTIME2 (P59)
PROGAIN2 (P60)
DERGAIN2 (P61)
FFGAIN2 (P62)

These parameters define the second range of gains and accelerations. They must be set like those defining the first range.

First range		Second range	
ACCTIME	(P18)	ACCTIME2	(P59)
PROGAIN	(P23)	PROGAIN2	(P60)
DERGAIN	(P24)	DERGAIN2	(P61)
FFGAIN	(P25)	FFGAIN2	(P62)

To select the second range of gains and accelerations, g.m.p. ACTGAIN2 (P108) must be properly set or the general CNC input ACTGAIN2 (M5013) must be activated.

SERCOSLE (P63) The CNC takes this parameter into account when the axis has been assigned a Sercos address, a.m.p. SERCOSID (P56) other than "0".

Even when the data exchange between the CNC and the drive is done via sercos, one must define whether the feedback is also handled via sercos or through the corresponding connector for the axis or spindle.

SERCOSLE =0 The position loop is controlled at the CNC.
The axis feedback is input into the CNC through a connector.
The velocity command is sent to the drive via Sercos.

SERCOSLE =1 The position loop is controlled at the CNC.
The axis feedback is sent to the CNC via Sercos. First feedback (motor feedback).
The velocity command is sent to the drive via Sercos.

SERCOSLE =2 The position loop is controlled at the CNC.
The axis feedback is sent to the CNC via Sercos. Second feedback (direct feedback).
The velocity command is sent to the drive via Sercos.

POSINREF (P64) Usually when working with sercos feedback, the motor-drive system has an absolute encoder. Thanks to this, the system knows at all times, the relative position of the axis within a revolution of the motor.

In these cases, when referencing the axis (homing), the CNC knows the position of the axis as soon as the home switch is pressed. Thus, not being necessary to move to the machine reference point (or marker pulse).

Parameter POSINREF indicates whether the axis has to move to a marker pulse or not after hitting the home switch.

NO	it does not move.	
YES	it moves	*Default value*

The movement to the reference point is carried out at the feedrate indicated by a.m.p. REFVALUE(P36). If P36 = 0, it moves at F0.

SWITCHAX (P65) When having 2 axes controlled by a single servo drive, machine parameter SWITCHAX of the secondary axis indicates which one is the main axis it is associated with.

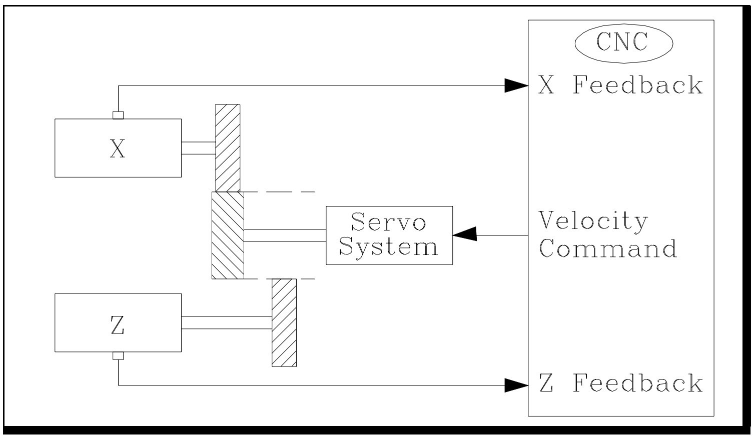
0 None	1 to X	2 to Y	3 to Z	4 to U
5 to V	6 to W	7 to A	8 to B	9 to C

By default 0

For further information, see section ***4.11 Axes (2) controlled by a single drive***

Example.

On a machine where the X and Z axes cannot move at the same time, the X axis is the main axis and the Z axis is the secondary (associated with the X axis).



SWITCHAX for X = 0
SWITCHAX for Z = 1

SWINBACK (P66) When having 2 axes controlled by a single servo drive, machine parameter SWINBACK of the secondary axis indicates whether it has its own feedback device or it uses that of the main axis it is associated with.

0	It uses that of the main axis.
1	It has its own feedback device (external)

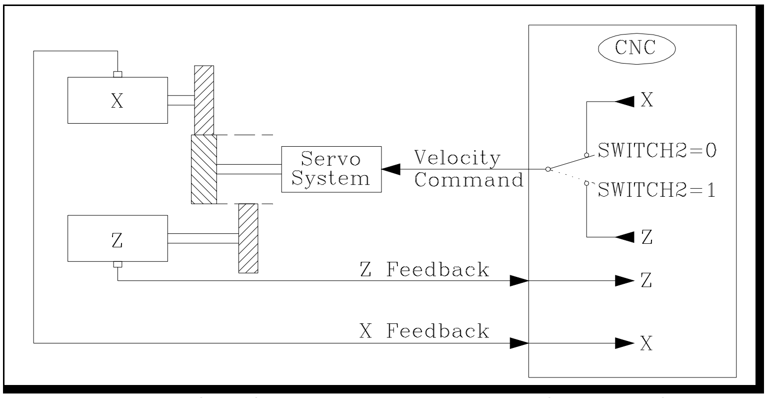
By default 0

For further information, see section ***4.11 Axes (2) controlled by a single drive***

The following examples show several possibilities.

In all of them, the toggling of the analog voltage must be done from the PLC using the SWTCH2 mark.

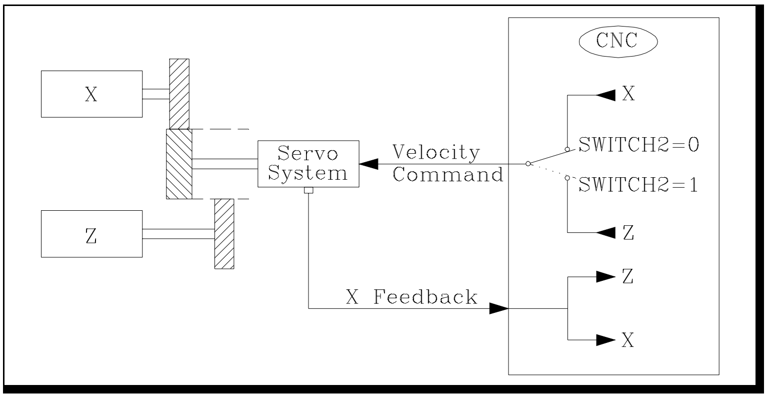
a) Each axis has its own feedback device.



X axis (main)
SWINBACK for X = 0

Z axis (secondary)
SWINBACK for Z = 1

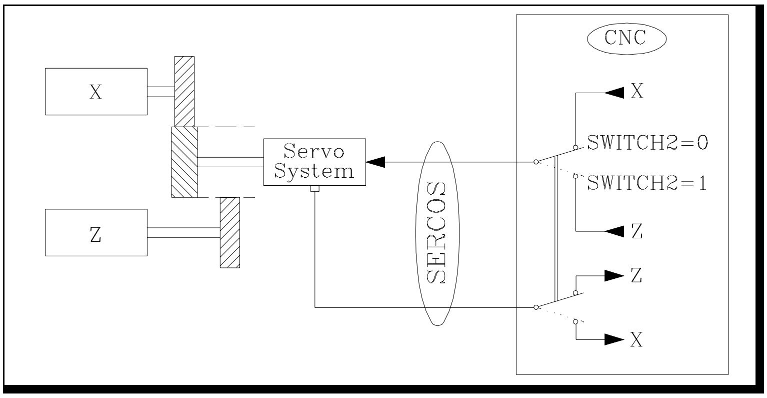
b) The two axes share the same feedback device. It must be connected to the feedback connector of the main axis.



X axis (main)
SWINBACK for X = 0

Z axis (secondary)
SWINBACK for Z = 0

c) The communication with the drive is done through Sercos, feedback included.



X axis (main)
SWINBACK for X = 0

Z axis (secondary)
SWINBACK for Z = 1

The CNC internally switches the feedback it receives via Sercos and it supplies it to either axis depending on the status of the SWITCH2 mark.

JERKLIM (P67) It defines the derivative of the acceleration. It allows limiting the changes in acceleration so the machine moves more smoothly on increments or decrements of small feedrates and with FFGAIN values close to 100%.

The smaller the value of JERKLIM the smoother will the machine response be, but it will increase the acc/dec time.

When increasing the value of JERKLIM, it decreases the acc/dec time but the machine response worsens.

Possible values

0.... 99999.9999 m/s^3

By default 0

Recommended values:

in mm	JERKLIM = 33.33 G00FEED / ACCTIME2
in inches	JERKLIM = 846.582 G00FEED / ACCTIME2

The CNC ignores this parameter when moving with handwheels, look ahead, threading (G33) and rigid tapping.

I0CODI1 (P68) pitch between two distance-coded fixed reference marks
I0CODI2 (P69) pitch between two distance-coded variable reference marks

The CNC takes this parameter into consideration when a.m.p. I0TYPE (P52) is set with a value other than "0".

It is defined in number of waves.

Possible values:	between 0 and 65535
By default:	I0CODD1=1000 and I0CODD2=1001

Example with Fagor linear encoder COC

Pitch between fixed I_0	20 000 µm
Pitch between variable I_0	20 020µm
Sinusoidal signal period	20 µm
Number of waves fixed I_0	20000/20 = 1000
Number of waves variable I_0	20020/20 = 1001

Values to be allocated for semi-absolute Fagor linear encoders (with coded Io):

SOP SVOP	GOP	MOT MOC MOP	COT COC COP			IOCODI1 = 20000/20 IOCODI2 = 20020/20	= 1000 = 1001
SOX SVOX	GOX	MOX	COX			IOCODI1 = 20000/4 IOCODI2 = 20020/4	= 5000 = 5005
		MOY	COY			IOCODI1 = 20000/2 IOCODI2 = 20020/2	= 10000 = 10010
				LOP		IOCODI1 = 80000/40 IOCODI2 = 80040/40	= 2000 = 2001
				LOX		IOCODI1 = 80000/4 IOCODI2 = 80040/4	= 20000 = 20010
					FOT	IOCODI1 = 100000/20 IOCODI2 = 100100/20	= 5000 = 5005
					FOX	IOCODI1 = 100000/4 IOCODI2 = 100100/4	= 25000 = 25025
					FOP	IOCODI1 = 100000/100 IOCODI2 = 100100/100	= 1000 = 1001
						By default 0	

3.3 Spindle parameters.

This CNC can control the main spindle, a second spindle and an auxiliary spindle

They all have their own setup parameters. The main and secondary spindle have identical parameter tables.

In order to synchronize the main and secondary spindles, they both must have a feedback device, their s.m.p. M19TYPE (P43) must be set to "1" and their parameters defining the third range of gains and accelerations must be set for a similar behavior of both spindles.

The G77 function synchronizes the spindles in speed, so the secondary spindle turns at the same speed as the main spindle.

The G30 function synchronizes the spindles in position and it sets an angular offset between them so the secondary spindle follows the main spindle maintaining that offset.

3.3.1 Machine parameters for main and 2nd spindles

SPDLTYPE (P0) Indicates the type of spindle output being used.

0	Analog output ±10V.	*Default value*
1	2-digit BCD coded “S” output.	
2	8-digit BCD coded “S” output.	

DFORMAT (P1) Indicates the display format for the spindle. It is not used for the second spindle.

0	In 4 digits.	*Default value*
1	In 5 digits.	
2	In 4.3 format	
3	In 5.3 format	
4	It is not displayed.	

MAXGEAR1 (P2)
MAXGEAR2 (P3)
MAXGEAR3 (P4)
MAXGEAR4 (P5)

They indicate the maximum spindle speed assigned to each range (gear).

Possible values: Integers between 0 and 65535 rpm

Default values

MAXGAR1 (P2)	Gear 1 (M41)	1000 rpm
MAXGAR2 (P3)	Gear 2 (M42)	2000 rpm
MAXGAR3 (P4)	Gear 3 (M43)	3000 rpm
MAXGAR4 (P5)	Gear 4 (M44)	4000 rpm

When not using all 4 gears, use the lower ones and set the unused ones to the same value as the highest one used.

AUTOGEAR (P6) Indicates whether the change of range is generated automatically or not by the CNC activating the M functions M41, M42, M43 and M44.

NO	There is no automatic gear change.
YES	There is automatic gear change.

By default, NO

POLARM3 (P7)
POLARM4 (P8)

Indicates the sign of the spindle analog for M03 and M04.

+	Positive analog.
-	Negative analog.

By default “POLARM3=+” and “POLARM4=-”

If the same value is assigned to both parameters, the CNC will output a single polarity (0V to 10V) signal with the indicated sign.

SREVM05 (P9) This parameter is used with a Mill model CNC. It is not used for the second spindle.

Indicates whether it is necessary or not to stop the spindle (M05) when reversing rotation direction during a tapping canned cycle (G84).

NO	It is not necessary.	
YES	It is necessary.	*Default value*

MINSOVR (P10)
MAXSOVR (P11)

Indicate the minimum and maximum % applicable to the programmed spindle speed. It is not used for the second spindle.

Possible values: Integers between 0 and 255.

By default "MINSOVR=50" and "MAXSOVR=120"

The resulting speed will be limited to the value indicated by s.m.p. MAXVOLT1 (P37), MAXVOLT2 (P38), MAXVOLT3 (P39) or MAXVOLT4 (P40) corresponding to the selected gear (range)

SOVRSTEP (P12)

Indicates the incremental step of the programmed spindle speed every time the override keys at the operator panel are pressed. It is not used for the second spindle.

Possible values: Integers between 0 and 255.

By default 5

NPULSES (P13)

Indicates the number of pulses per revolution provided by the spindle encoder. 0 means that there is no spindle encoder.

When the main spindle does not have an encoder (NPULSES=0), the CNC shows its theoretical rpm (affected by the %).

Possible values: Integers between 0 and 65535.

By default 1000

DIFFBACK (P14)

Indicates whether the spindle encoder uses differential signals (double ended) or not.

NO = It does NOT use differential signals.
YES = It uses differential signals.

By default, YES

FBACKAL (P15)

Indicates whether the feedback alarm is OFF or ON.

OFF = Not desired, it is canceled.
ON = Alarm on.

By default, ON

AXISCHG (P16)

Indicates the counting direction. If correct, leave it as is; if not, change it from YES to NO or viceversa. If this parameter is changed, s.m.p. LOOPCHG (P26) must also be changed so the spindle does not "run away".

Possible values: YES and NO.

By default, NO

DWELL (P17)

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

Possible values: Integers between 0 and 65535 ms.

By default 0 (no delay)

ACCTIME (P18) This parameter is used when working with the spindle in closed loop and it indicates the acceleration time given to reach the maximum speed set by s.m.p. MAXVOLT1 (P37) thru MAXVOLT4 (P40) in each range. This value also represents the deceleration time.

Possible values: Integers between 0 and 65535 ms.

By default 0 (no control)

INPOSW (P19) Indicates the width of the IN POSITION zone where the CNC considers the spindle to be in position when working in closed loop (M19).

Possible values: 0.... 99999.99999 degrees.

By default, 0.01 degrees

INPOTIME (P20) Indicates the time period that the spindle must remain in the "IN POSITION" zone in order to consider it to be in position.

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

Possible values: Integers between 0 and 65535 ms.

By default 0

MAXFLWE1 (P21) Indicates the maximum following error allowed for the spindle when moving in closed loop (M19).

Possible values: 0.... 99999.99999 degrees.

By default, 30 degrees

MAXFLWE2 (P22) Indicates the maximum following error allowed for the spindle when stopped in closed loop (M19).

Possible values: 0.... 99999.99999 degrees.

By default, 0.1 degrees

PROGAIN (P23) The CNC takes this parameter into account when operating in closed loop (M19).

Indicates the value of the Proportional Gain. Its value represents the analog voltage corresponding to a following error of 1 degree.

Analog voltage (mV) = Following error (degrees) x PROGAIN

Possible values: Integers between 0 and 65535 mV/degree

By default, 1000mV/degree.

This value is taken for the first spindle gear and the CNC calculates the values for the rest of the gears.

Example:

s.m.p. MAXGEAR1 (P2) = 500 rev/min. The desired speed for a 1 degree of following error is S = 1000°/min (2.778 rev/rpm).

Drive analog: 9.5V for 500 rpm

Analog output corresponding to S = 1000 º/min. (2.778 rpm)

Analog = (9.5/500) x 2,778 = 52,778mV

Therefore, "PROGAIN" = 53

DERGAIN (P24) The CNC takes this parameter into account when operating in closed loop (M19).

Indicates the value of the Derivative Gain. Its value represents the analog voltage (in millivolts) corresponding to a change in following error of 1 degree in 10 milliseconds.

This analog voltage will be added to the one calculated for the Proportional Gain.

$$\text{Command} = \left(\xi \cdot \text{PROGAIN} + \frac{\xi \cdot \text{DERGAIN}}{10 \cdot t}\right)$$

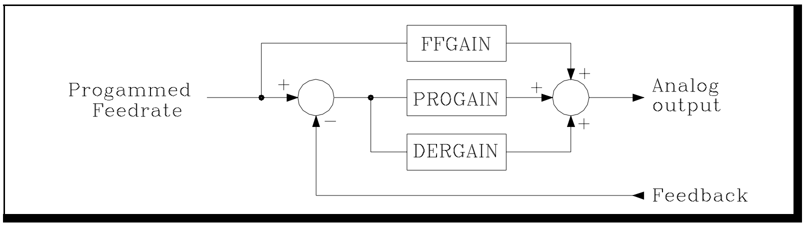
It is a good idea to also use the acc./dec. s.m.p. ACCTIME2 (P18) for this axis (with a value other than “0”) if this gain is to be applied.

Possible values: Integer value between 0 and 65535.

By default “0”, (no derivative gain applied)

FFGAIN (P25) The CNC takes this parameter into account when operating in closed loop (M19).

Indicates the % of the analog voltage due to the programmed speed. The rest will depend upon the following error. Both the Proportional and Derivative gains will be applied onto this following error..



Command

$$= \left(\xi \cdot \text{PROGAIN} + \frac{\xi \cdot \text{DERGAIN}}{10 \cdot t} + \frac{\text{FFGAIN} \times \text{Fprog} \times \text{MAXVOLT}}{100 \cdot \text{G00FEED}}\right)$$

The Feed-Forward Gain lets improve the positioning loop minimizing the following error and it should be used when the “ACCTIME” machine parameter for this axis is active (acc/dec. being applied).

Possible values: Integers between 0 and 100.

By default “0”, (no feed-forward gain applied)

Usually, a value between 40% and 80% is assigned depending mainly on the type of machine and its characteristics.

LOOPCHG (P26) Indicates the sign of the analog output. If correct, leave it as is; if not, change it from YES to NO or viceversa.

Possible values: YES and NO.

By default, NO

MINANOUT (P27) Indicates the minimum value for the spindle analog output.

It is given in D/A converter units and it admits integer values between 0 and 32767 which corresponds to an analog voltage of 10V. By default 0.

MINANOUT	Minimum analog voltage
1	0.3 mV
-----	-----
3277	1 V
-----	-----
32767	10 V

SERVOFF (P28) Indicates the analog offset value for the spindle drive.

It is given in D/A converter units and it admits integer values between 0 and +32767 which corresponds to an analog voltage of +10V. By default 0 (not applied).

SERVOFF	Command
-32767	-10 V
-----	-----
-3277	-1 V
-----	-----
1	0.3 mV
-----	-----
3277	1 V
-----	-----
32767	10 V

LOSPDLIM (P29)
UPSPDLIM (P30) Indicate the upper and lower limits of the actual spindle speed so the CNC can "notify" the PLC (by means of the "REVOK" signal) that the actual spindle rpms are the same as the programmed ones.

Possible values: Integer value between 0 and 255.

By default "LOSPDLIM=50" (50%) and "UPSPDLIM=150" (150%).

DECINPUT (P31) Indicates whether or not the spindle has a home switch to synchronize the spindle when working in M19.

NO It doesn't have one.

YES it has one. *Default value*

REFPULSE (P32) Indicates the type of marker pulse Io to synchronize the spindle when working in M19.

\+ Positive pulse (5V). *Default value*

\- Negative pulse (0V).

REFDIREC (P33) Indicates the rotating direction when synchronizing the spindle during M19.

\+ Positive direction. *Default value*

\- Negative direction.

REFEED1 (P34) Indicates the spindle's positioning speed when in M19 and the synchronizing speed until it finds the home switch.

Possible values: 0.00001.... 199999.99999 degrees/min.

By default 9000 degrees/min

REFEED2 (P35) Indicates the synchronizing speed of the spindle after hitting the home switch and until it finds the marker pulse.

Possible values: 0.00001.... 199999.99999 degrees/min.

By default 360 degrees/min

REFVALUE (P36) Indicates the position value assigned to the reference point of the spindle (home or marker pulse).

Possible values: ±99999.99999 degrees.

By default 0

MAXVOLT 1 (P37)
MAXVOLT 2 (P38)
MAXVOLT 3 (P39)
MAXVOLT 4 (P40) Indicates the analog voltage corresponding to the maximum speed of range 1, 2, 3 and 4.

Possible values: Integer value between 0 and 9999 mV.

By default, 9500 (9.5 V).

GAINUNIT (P41) The CNC takes this parameter into account when operating in closed loop (M19).

Defines the units for s.m.p. PROGAIN (P23) and DERGAIN (P24).

0 millivolts/degree

1 millivolts/0.01 degree

By default, 0 (mV/degree).

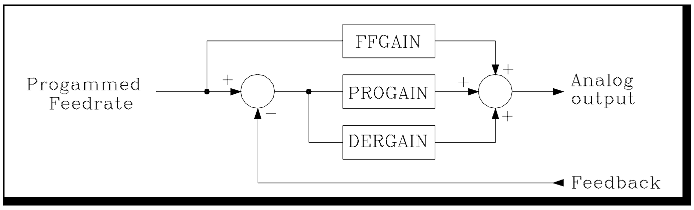
This parameter is used when working with the spindle in closed loop.

A value of "1" will be assigned when the analog voltage corresponding to a following error of 1 degree is very small. This offers greater sensitivity for adjusting s.m.p. PROGAIN (P23) and DERGAIN (P24)

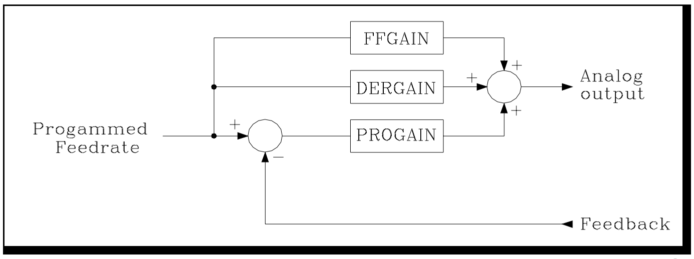
ACFGAIN (P42) The CNC takes this parameter into account when operating in closed loop (M19).

Indicates whether or not the value assigned to s.m.p. DERGAIN (P24) is applied onto the variations of the programmed speed (AC-forward).

NO It is applied onto the variations of following error (derivative gain).



YES It is applied onto the variations of programmed feedrate due to ACC/DEC (AC-forward).



By default, NO

By default, the CNC sets this parameter to "NO".

M19TYPE (P43) This parameter sets the type of spindle orient (M19) available.

It indicates whether the spindle must be homed when switching from open to closed loop or it is enough to home it once on power-up.

0 When switching from open loop to closed loop.

1 Once after power-up.

By default 0

SERCOSID (P44) Indicates the sercos address (device select code) associated with the spindle.

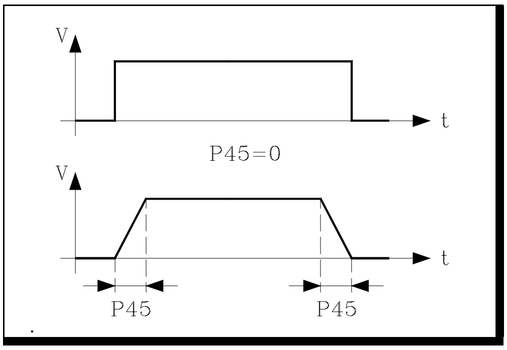
Possible values:	0	Analog spindle
	1-8	Sercos address (device select code)

By default 0

These addresses for the various axes and spindles must be sequential and starting from "1". That is, with 3 sercos axes and one sercos spindle, the values for this parameter must be 1, 2, 3, 4.

OPLACETI (P45) When working in open loop (M3, M4) spindle speed variations may be in a step or in a ramp.

This parameter indicates the duration of the ramp in milliseconds for the maximum "S". If OPLACETI=0, it will be in a step.



Possible values: Integer value between 0 and 65535.

By default 0 (in a step)

SMOTIME (P46) Sometimes the spindle does not respond as desired on particular movements. Handwheel jog, etc.

In these cases, the response of the spindle may be smoothed by applying a filter to the speed changes.

This filter is defined by means of parameter SMOTIME which indicates the duration of the filter in milliseconds which in turn is set by g.m.p. LOOPTIME (P72).

Possible values:

Integers between 0 and 64 times the value assigned to g.m.p. LOOPTIME(P72)

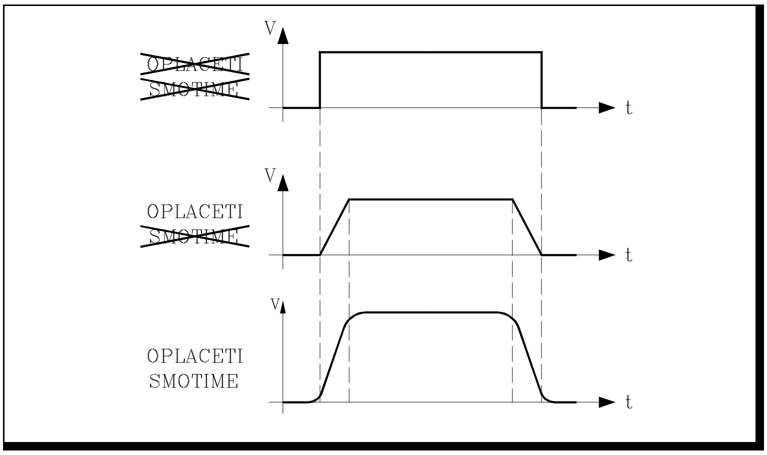
By default 0 (not applied)

If LOOPTIME=0 (4ms) the maximum value for SMOTIME will be 64 x 4 = 256 ms.

The default value for this parameter is "0".

In order to obtain a better response, the SMOTIME parameter of all axes interpolating together should be set to the same value.

The spindle's response can also be smoothened when working in open loop (M3, M4). In this case, s.m.p. OPLACETI (P45) and SOMTIME (P46) must be used.



ACCTIME2 (P47)
PROGAIN2 (P48)
DERGAIN2 (P49)
FFGAIN2 (P50)

These parameters define the second range of gains and accelerations. They must be set like those defining the first range.

First range		Second range	
ACCTIME	(P18)	ACCTIME2	(P47)
PROGAIN	(P23)	PROGAIN2	(P48)
DERGAIN	(P24)	DERGAIN2	(P49)
FFGAIN	(P25)	FFGAIN2	(P50)

To select the second range of gains and accelerations, g.m.p. ACTGAIN2 (P108) must be properly set or the general CNC input ACTGAIN2 (M5013) must be activated.

SERCOSLE (P51)

The CNC takes this parameter into account when the spindle has been assigned a Sercos address, s.m.p. SERCOSID (P56) other than "0".

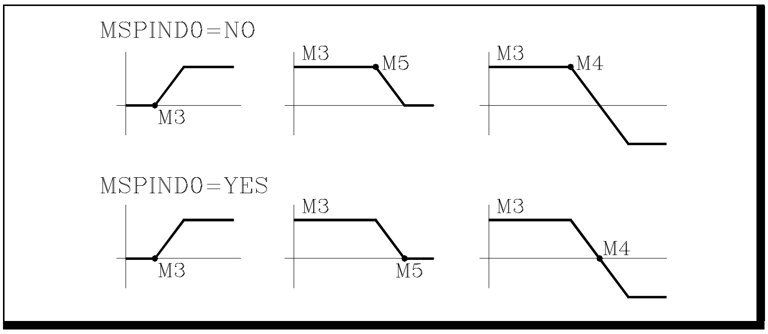
Even when the data exchange between the CNC and the drive is done via sercos, one must define whether the feedback is also handled via sercos or through the corresponding connector for the axis or spindle.

SERCOSLE =0 The position loop is controlled at the CNC.
The axis feedback is input into the CNC through a connector.
The velocity command is sent to the drive via Sercos.

SERCOSLE =1 The position loop is controlled at the CNC.
The axis feedback is sent to the CNC via Sercos. First feedback (motor feedback).
The velocity command is sent to the drive via Sercos.

SERCOSLE =2 The position loop is controlled at the CNC.
The axis feedback is sent to the CNC via Sercos. Second feedback (direct feedback).
The velocity command is sent to the drive via Sercos.

MSPIND0 (P52) Indicates when functions M3, M4, M5 are to be sent out. While the spindle is accelerating and decelerating.



SYNPOSOF (P53) When both spindles are synchronized in position, the second spindle must follow the main spindle maintaining the offset set by function G30.

The parameter of the main spindle sets the maximum error allowed. If this value is exceeded, no error message is displayed and the movement is not stopped. It only sets general output SYNCPOSI (M5559) low.

Possible values: 0.... 99999.99999 degrees.

By default, 2 degrees

SYNSPEOF (P54) When both spindles are synchronized in speed, the second spindle must turn at the same speed as the main spindle.

The parameter of the main spindle sets the maximum error allowed. If this value is exceeded, no error message is displayed and the movement is not stopped. It only sets general output SYNSPEED (M5560) low.

Possible values: Integers between 0 and 65535 rpm

By default 1 rpm

ACCTIME3 (P55)
PROGAIN3 (P56)
DERGAIN3 (P57)
FFGAIN3 (P58)

These parameters define the third range of gains and accelerations. They must be set like those defining the first range.

First range		Second range		Third gear	
ACCTIME	(P18)	ACCTIME2	(P47)	ACCTIME3	(P55)
PROGAIN	(P23)	PROGAIN2	(P48)	PROGAIN3	(P56)
DERGAIN	(P24)	DERGAIN2	(P49)	DERGAIN3	(P57)
FFGAIN	(P25)	FFGAIN2	(P50)	FFGAIN3	(P58)

The CNC uses the third range when working with synchronized spindles (G77).

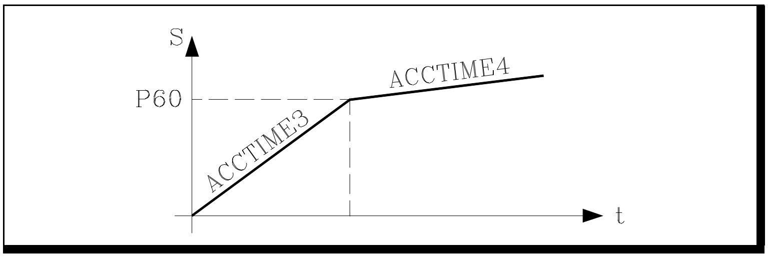
The spindles (main and second) must have their own feedback devices and their parameters must be set in such a way that their behaviors are similar.

Default values		
	ACCTIME3 (P55)	4000
	PROGAIN3 (P56)	50
	DERGAIN3 (P57)	0
	FFGAIN3 (P58)	100.

When working with FFGAIN3 (P58) = 100, set the MAXGEAR and MAXVOLT parameters properly.

ACCTIME4 (P59)
SECACESP (P60)

In order to compensate for the lack of a linear response on some spindles, it is possible to use two accelerations: ACCTIME3 for low speeds [up to the one set by SECACESP (P60)] and ACCTIME4 for the rest of higher speeds.



ACCTIME4 is set like ACCTIME3.

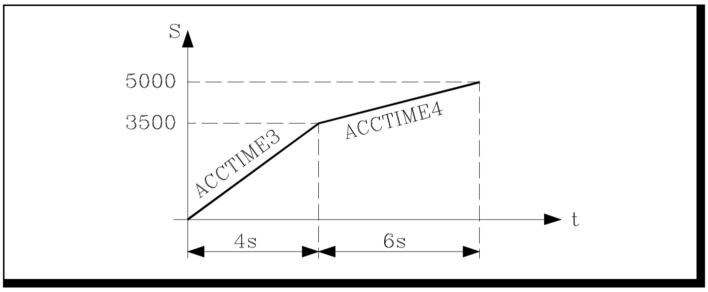
Parameter SECACESP (P60) indicates at which speed the acceleration is changed. It is given in rpm with an integer between 0 and 65535. If P60=0, the CNC always applies ACCTIME3.

Default values ACCTIME4 (P59) = 8000
SECACESP (P60) = 700

Once the spindles are in synchronism, the CNC applies to both spindles the accelerations defined for the main spindle.

Example:

Being the maximum speed for the selected range (gear) MAXGEAR = 6000 rpm



Maximum synch speed: 5000 rpm
SYNMAXSP (P63) = 5000

Acceleration changing speed: 3500 rpm
SECACESP (P60) = 3500

ACCTIME3(P55)= 6000 x 4 / 3500 = 6857 ms

ACCTIME4(P59)= 6000 x 6 / 1500 = 24000 ms

SYNCPOLA (P61)

It is defined at the second spindle. It indicates whether the spindles being synchronized are facing each other or not (opposite turning directions in M3 or M4) for the CNC to take it into consideration when synchronizing them.

NO They are NOT facing each other. They both turn in the same direction.

YES They are facing each other. They turn in opposite directions.

By default, NO

CONCLOOP (P62) It indicates whether the spindle operates in closed positioning loop (as if it were an axis) or not.

NO	It operates in open loop
YES	It operates in closed position loop (as if it were an axis).

By default, NO

In order to operate in closed positioning loop, the spindle must have an encoder and a good servo system for the full speed range.

When working with M19, the first two ranges of gains and accelerations are used regardless of the value given to this parameter.

When working in closed positioning loop (M3, M4, M5) the third range of gains and accelerations is used: ACCTIME3, PROGAIN3, DERGAIN3 and FFGAIN3.

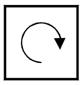
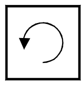
When working with synchronized spindles (G77), third range of gains and accelerations is used. Therefore, the CONCLOOP parameter of the spindle to be synchronized should be set to "YES"

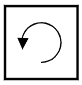
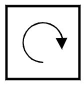
SYNMAXSP (P63) It is set for the main spindle. It indicates the maximum turning speed when the spindle are synchronized (G77).

Possible values: Integers between 0 and 65535 rpm
A value of "0" means that it is not limited.

By default 1000 rpm

M3M4SIM (P64) In TC mode, it indicates the turning direction of the corresponding spindle with each key for turning direction.

M3M4SIM=0  with M3 and  with M4

M3M4SIM=1  with M3 and  with M4

By default 0

For example in a live tool cycle, it will take into account the value assigned to the spindle that has been defined as live tool whereas in a Deep Hole Drilling cycle, it will consider the one assigned to the main spindle.

SINMAGNI (P65) Indicates the multiplying factor (x1, x4, x20, etc.) that the CNC must apply only to sinewave feedback signals of the spindle.

Set this parameter to "0" when using squarewave feedback signals and the CNC will always apply a x4 multiplying factor.

Possible values: Integers between 0 and 255.

By default 0

Spindle feedback resolution is set by s.m.p. NPULSES (P13) and SINMAGNI (P65).

Example:

We would like to obtain a 0.001º resolution by using a 3600-line sinewave encoder.

We must calculate the multiplying factor "SINMAGNI" to be applied by the CNC to the pulses provided by the encoder in order to obtain the desired resolution.

SINMAGNI = degrees per turn / (number of pulses x Resolution)

SINMAGNI = 360 / (3600 x 0.001) = 100

Therefore: NPULSES =3600 SINMAGNI=100

3.3.2 Machine parameters for auxiliary spindle

MAXSPEED (P0) Indicates the maximum speed of the auxiliary spindle.

Possible values: Integers between 0 and 65535 rpm

By default 1000 rpm

SPDLOVR (P1) It indicates whether the spindle override keys of the operator panel modify or not the speed of the auxiliary spindle when it is active.

NO They do not alter it.

YES They do alter it. The CNC will apply the values set for main s.m.p. "MINSOVR" (P10), "MAXOVR" (P11) and "SOVRSTEP" (P12).

By default, NO

MINANOUT (P2) It sets the minimum analog voltage value.

It is given in D/A converter units and it admits integer values between 0 and 32767 which correspond to an analog voltage of 10V. By default 0.

MINANOUT	Minimum analog voltage
1	0.3 mV
-----	-----
3277	1 V
-----	-----
32767	10 V

SERVOFF (P3) Indicates the analog offset value for the spindle drive.

It is given in D/A converter units and it admits integer values between 0 and +32767 which corresponds to an analog voltage of +10V. By default 0 (not applied).

SERVOFF	**Command**
-32767	-10 V
-----	-----
-3277	-1 V
-----	-----
1	0.3 mV
-----	-----
3277	1 V
-----	-----
32767	10 V

MAXVOLT (P4) Indicates the analog voltage corresponding to the maximum speed defined by s.m.p. MAXSPEED (P0).

Possible values: Integer value between 0 and 9999 mV.

By default, 9500 (9.5 V).

SERCOSID (P5) Indicates the sercos address (device select code) associated with the auxiliary spindle.

Possible values: 0 Analog auxiliary spindle

1-8 Sercos address (device select code)

By default 0

These addresses for the various axes and spindles must be sequential and starting from "1". That is, with 3 sercos axes and one sercos spindle, the values for this parameter must be 1, 2, 3, 4.

3.4 Drive parameters

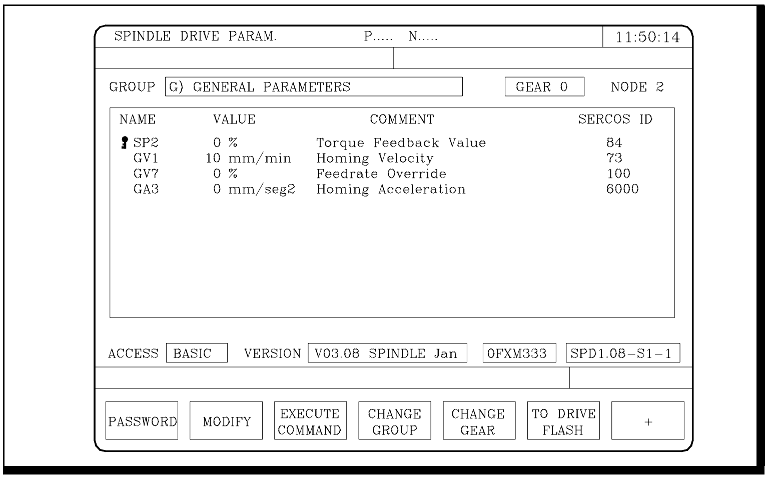
This option is available when the drives are connected to the CNC via Sercos.

It displays the tables of the drive parameters that are stored in the "Memkey Card" (Card A) and the sofkeys of the sercos axes. Press one of those softkeys for editing the drive parameters for that particular axis.

Note: When selecting the drive parameters at the CNC, it will display the ones stored in each drive and if any is modified, it is modified at the drive. The CNC does not have parameters of the drive although their copies may be stored in the "Memkey Card" (Card A).

Refer to the Fagor drive manual to know the commands, variables, names, values, passwords, etc.

When accessing the parameters of a drive, the CNC shows a screen like this:



In the GROUP window, one must select the group of parameters or variables to be displayed. To change the group, press the [CHANGE GROUP] softkey, select the new group using the up/down arrow keys and press [ENTER].

In the RANGE window, one must select the range of parameters or variables to be displayed. To select another range, press the [CHANGE RANGE] softkey, select the new range using the up/down arrow keys and press [ENTER].

The NODE window shows the node number identifying that drive in the SERCOS ring.

In other words, the position of the Sercos switch. The main window shows the variables or parameters of the selected group or range indicating their Fagor name in each variable, its value, its meaning and its Sercos identifier.

If the variable does not have a write permission, a key will appear before the Fagor name.

This information is updated when selecting a new information (group or range), when modifying a variable or parameter or when pressing page/up page-down. It is not refreshed continuously.

The ACCESS window shows the permitted access level. There are 3 access levels at the drive: basic level, OEM level and Fagor level.

To change the level, press the [Password] softkey, key in the relevant code and press [ENTER].

The VERSION window shows the software version installed at the drive, the name of the motor associated with the drive and the drive model.

The softkeys available in this mode are:

Password Modifies the access level selected in the Access window.

Modify To modify the variables that are not protected (those without a key next to them).

After selecting the variable with the up/down arrow keys and pressing the "Modify" softkey, two windows are displayed.

The first one shows the range of possible values and the second one the current value. Enter the new value and press [ENTER].

The drive assumes that value and refreshes the screen.

Execute Command Shows the list of commands that can be executed by the drive. Select one with the up/down arrow keys and press [ENTER].

Change Group Selects the group of parameters or variables to be displayed.

Change Range Selects the number of parameter or variable range to be displayed.

To Drive Flash: The drive stores all its parameters in its flash memory and it then executes a soft-reset command. This command interrupts the communication through Sercos. Press [ENTER] to restore it.

Save to Card A: It makes a copy of the parameters stored in the RAM memory of the drive into the "Memkey Card" (Card A) of the CNC . It stores them with the name of the axis they are associated with (for example: X axis parameters).

Load from Card A: It copies the parameters stored in the "Memkey Card" (Card A) into the RAM memory of the drive with the name of the axis being edited.

Drive Errors It displays a window with the warnings and errors of the drive. If all of them do not fit in the window, use the up/down arrow keys to scroll them.

Options It shows a screen where one can select to display either all the parameters and variables or just the ones than can be modified.

Press the [Modify Option] softkey to change it and [ENTER] to validate it. This option is common to all the axes.

This option works fine with drive version V3.9 or newer. When using older versions, it might not be possible to access all variables and parameters and some data might not be shown such as the name of the associated motor.

If the communication through the Sercos ring is interrupted, a screen is displayed. Press [ENTER] to restore it.

3.4.1 Friction compensation

From version V3.14 on, the drive offers parameters TP10, TP11, TP12, TP13, TP14 and TV4 for friction compensation. See drive manual.

It also has 2 more general purpose variables: XV10 and XV11 (ID SERCOS 34800 and 38401). These variables may be accessed from the CNC via Sercos.

The following example shows how to use variable XV10 to monitor or show on the oscilloscope the X axis following error using WinDDS.

Assign the value of PLC register R800 to the “ID SERCOS 34800” of the drive that occupies the “Sercos 1” address.

Set plc.m.p. SWR800 = 1.34800"

The PLC program must set register R800 to the value of the X axis following error (FLWEX variable). A periodic module should be used to refresh that value at every position loop. (Note: In order for the Sercos variable to be actually updated, the NWR instruction must be written in the periodic module).

```
PE 4
()= CNCRD (FLWEX, R800, M1)
MWR
END
```

3.5 Serial line parameters

BAUDRATE (P0) Indicates the communication speed, in baud, between the CNC and the peripherals.

It is given in Baud and it is selected with the following code:

0	110 baud	5	2.400 baud	10	57.600 baud
1	150 baud	6	4.800 baud	11	115.200 baud
2	300 baud	7	9.600 baud	12	reserved
3	600 baud	8	19.200 baud		
4	1.200 baud	9	38.400 baud		

By default, 7 (9600 baud).

NBITSCHR (P1) Indicates the number of data bits per transmitted character.

0 Uses the 7 least significant bits of an 8-bit character. It is used when transmitting ASCII characters (standard)

1 Uses all 8 bits of the transmitting character. Used when transmitting special characters whose codes are greater than 127.

By default 1

PARITY (P2) Indicates the type of parity check used.

0 No parity.
1 Odd parity.
2 Even parity.

By default 0

STOPBITS (P3) Indicates the number of stop bits at the end of each transmitted word.

0 1 STOP bit.
1 2 STOP bits.

By default 0

PROTOCOL (P4) Indicates the type of communications protocol to be used.

0 Communications protocol for general device.
1 DNC protocol.
2 Communications protocol for Fagor floppy disk unit.

By default 1 (DNC)

PWONDNC (P5) Indicates whether the DNC feature will be active on power-up or not.

NO Not active on power-up.
YES Active on power-up

By default, NO

DNCDEBUG (P6) Indicates whether the debugging feature for DNC communications is active or not.

It is advisable to use this safety feature in all DNC communications. It could be deactivated in the debugging process.

NO Debug NOT active. Communication aborted.
YES Debug active. Communication not aborted.

By default, NO

ABORTCHR (P7) Indicates the character used to abort communications with general peripheral device.

0 CAN *Default value*
1 EOT

EOLCHR (P8) Indicates the character used to indicate "end of line" when communicating with general peripheral device.

0	LF	*Default value*
1	CR	
2	LF-CR	
3	CR-LF	

EOFCHR (P9) Indicates the character used to indicate "end of text" (end of file) when communicating with a general peripheral device.

0	EOT	*Default value*
1	ESC	
2	SUB	
3	ETX	

XONXOFF (P10) Indicates whether the XON-XOFF communications protocol is active or not when operating with a generic peripheral.

ON	It is active.	*Default value*
OFF	It is NOT active.	

3.6 Ethernet parameters

With these parameters, the CNC may be configured as a node within the computer network. Doing that requires the Ethernet option.

If the CNC is configured as a node on the computer network, the following operations are possible from any PC of that network:

- Access the part-program directory of the Hard Disk.
- Edit, modify, delete, rename, etc. the programs stored on the hard disk.
- Copy programs from the hard disk to the PC and vice versa.

HDDIR (P0)

Not being used at this time.

CNMODE (P1)
CNID (P2)
CNGROUP (P3)
CNDOMAIN (P4)

These parameters configure the CNC as a node within the computer network.

CNMODE indicates the type of computer network being used.

0 work group
1 in domain

CNID indicates the name assigned to the node on the network.

Up to a 15 characters may be used.

By default, FAGORCNC

CNGROUP indicates the name of the group the node belongs to on the network.

Up to a 15 characters may be used.

For example: PRODUCTION

CNDOMAIN indicates the name of the domain the node belongs to on the network.

Up to a 15 characters may be used.

For example: FAGOR

EXTNAME1 (P5)
CNHDDIR1 (P6)
CNHDPAS1 (P7)

This parameters allow sharing the hard disk (HD) with the rest of the devices of the computer network.

CNHDDIR1 Hard Disk directory to be shared.

Up to a 22 characters may be used.

The whole HD must be shared because directories cannot be created, P6 = \CNC\USER

EXTNAME1 name used for the shared directory.

Up to a 12 characters may be used.

CNHDPAS1 password for accessing the hard disk from the computer network.

Up to a 14 characters may be used.

EXTNAME2 (P8)
....
SERUNI2 (P21)

Not being used at this time.

Instructions to connect a CNC with Ethernet to a Local Area Network

Considerations

It uses the NetBEUI protocol (from Microsoft).

The network the CNC will be connected to may work in domain mode or work group mode.

Point-to-point connection between a PC and the CNC.

In a point-to-point connection, there are the following cabling options:

- Coax cable, using the BNC connection (it is not the most common one)
- Standard twisted-pair cable using a "hub" between the CNC and the PC
- Modified twisted-pair cable with crossed lines. It is sold as a commercialized product.

Machine parameters for Ethernet (CNC)

CNMODE	0
CNID	Name with which the CNC will be known by the rest of the network nodes.
CNGROUP	Name of the work group the CNC will belong to.
CNDOMAIN	Leave it blank.

At the PC (Windows95):

Access the properties menu of the network environment by doing:

Start => Configuration => Control Panel => Network.

On the Configuration screen:

The NetBEUI protocol must be displayed.

If the pages displays "Clients for Microsoft networks", select it and enter into properties. The line "Initiate session in the Windows NT domain" Must NOT be selected.

In the Identification screen:

In the field for work group, it must display the same group that was assigned to CNC parameter CNGROUP.

Reset both units and the connection will then be established.

CNC connection to a multi-point network.

Machine parameters for Ethernet (CNC)

CNMODE	"0" if it is not a domain network, /1 if it is a work group network
CNID	Name with which the CNC will be known by the rest of the network nodes.
CNGROUP	Name of the work group the CNC will belong to.
CNDOMAIN	If it is a domain network, name of the domain the CNC will be integrated into.

On the network server, a new network node will have to be designated with the name allocated to CNID, belonging to the work group allocated to CNGROUP and to the domain assigned to CNDOMAIN.

Instructions for setting up a user PC to access CNC directories

Recommended configuration:

- Open the «Windows Explorer»
- On the «Tools» menu, select the «Connect to Network Drives» option.
- Select the Drive. For example: «D»
- Indicate the path: CNC name followed by the name of the shared directory. For example: \\FAGOR8040\CNCHD
- When selecting the option: «Connect again when initiating the session», the selected CNC will appear on each power-up as another path of the «Windows Explorer» without having to define it again.

3.7 PLC Parameters

WDGPRG (P0) Indicates the Watchdog time-out period for the main PLC program.

Possible values: Integers between 0 and 65535 ms.

By default 0

WDGPER (P1) Indicates the Watch-Dog time-out period for the periodic module of the PLC.

Possible values: Integers between 0 and 65535 ms.

By default 0

USER0 (P2) --------- USER23 (P25) Parameters “USER0” through “USER23” do not mean anything to the CNC.

They could contain the type of information that the OEM may find necessary to customize this machine, such as: Information about the type of machine, PLC program version, etc.

This information can be accessed from the PLC program by means of the “CNCRD” high-level instruction.

Possible values:

USER0(P2) through USER7(P9)
Integers 0 through 255.
USER8(P10) through USER15(P17)
Integers 0 through 65535.
USER16(P18) through USER23(P25)
±99999.9999 mm or ±3937.00787 inches.

By default 0

CPUTIME (P26) This parameter indicates the time the system CPU dedicates to the PLC. Possible values

With CPU Turbo:

0	1 ms every 8 samplings	
1	1 ms every 4 samplings	
2	1 ms every 2 samplings	
3	1 ms every sampling	
4	2 ms every sampling.	Only with Looptime = 3, 4, 5 or 6
5	3 ms every sampling.	Only with Looptime = 4, 5 or 6
6	4 ms every sampling.	Only with Looptime = 5 or 6
7	5 ms every sampling.	with Looptime = 6

Without CPU Turbo:

0	1 ms every 8 samplings	
1	1 ms every 4 samplings	
2	1 ms every 2 samplings	
3	1 ms every sampling.	Only with Looptime = 4, 5 or 6 and with Looptime = 3 if /C
4	2 ms every sampling.	Only with Looptime = 4, 5 or 6
5	3 ms every sampling.	Only with Looptime = 5 or 6
6	4 ms every sampling.	with Looptime = 6
7	4 ms every sampling.	with Looptime = 6

By default 0

The sampling period is determined by the g.m.p. LOOPTIME (P72).

Hence, for a sampling period of 4 msec. and a CPUTIME=0, the system CPU dedicates 1 millisecond every 8 samplings (thus, 32 milliseconds) to the PLC.

The Status window of the PLC statistics screen indicates the time the system CPU dedicates to the PLC. See operating manual, section 9.9

Same as with sinewave feedback, number of axes and the user channel active, the PLC demands calculation time from the system CPU.

The more time the CPU dedicates to the PLC, the greater the sampling time will be, g.m.p. LOOPTIME (P72).

PLCMEM (P27) Not being used at this time.

SRR700 (P28)

SRR739 (P67)

They are used in the data exchange via Sercos between the CNC and the drives.

They indicate which drive and what type of information will be put in CNC registers R700 through R739.

P28 ⇒ R700	P29 ⇒ R701	P30 ⇒ R702
P31 ⇒ R703	P32 ⇒ R704	etc.

The setting format for plc.m.p. "P28" through "P67" is 1.5

The units digit identifies the Sercos node number to get information from.

The decimal part indicates the Sercos identifier number.

Example:

P32=1.00040 Indicates that PLC register R704 contains the "VelocityFeedback" supplied by the drive located in Sercos node 1.

Notes: To identify the units of the variables, see the drive manual.
Read-only registers R700 through R739 are updated at the beginning of the PLC scan, unless the MRD instruction is used.

SWR800 (P68)

SWR819 (P87)

They are used in the data exchange via Sercos between the CNC and the drives.

They indicate what type of information is put in registers R800 through R819 and which drive will be assigned that value.

P68 ⇒ R800	P69 ⇒ R801	P70 ⇒ R802
P71 ⇒ R803	P72 ⇒ R804	etc.

The setting format for plc.m.p. "P68" through "P87" is 1.5

The units digit identifies the Sercos node number to send information to.

The decimal part indicates the Sercos identifier number.

Example:

P70=2.34178 Indicates that the value of PLC register R802 will be assigned to "DigitalOutputsValues" of the drive located in Sercos node 2.

Note: To identify the units of the variables, see the drive manual.

IOCANSPE (P88)

When using CAN connection, the transmission speed depends on the length of the cable or total CAN connection distance. Possible values:

0	1Mbit/sec	maximum distance 20 m
1	800Kbit/sec	maximum distance 45 m
2	500Kbit/sec	maximum distance 95 m
3	250Kbit/sec	maximum distance 495 m
4	125Kbit/sec	maximum distance 1000 m

Fagor modules that do not have a baudrate selector can only work at 500 Kbit/sec.

IOCAGEN (P89)

Not being used at this time.

IOCANID1 (P90)
IOCANID2 (P91)
IOCANID3 (P92)
IOCANID4 (P93)

They are used to set the remote modules

They indicate which remote module each plc.m.p. (ICAN*, OCAN*, NUICAN*, NUOCAN*) refer to.

Assign the CAN bus address that occupies the node (the one indicated by the address selector switch)

ICAN1 (P94)
OCAN1 (P95)
ICAN2 (P96)
OCAN2 (P97)
ICAN3 (P98)
OCAN3 (P99)
ICAN4 (P100)
OCAN4 (P101)

They are used to set the remote modules

They indicate the configuration of each remote module, number of inputs (ICAN*) and outputs (OCAN*).

Example for a remote module located in node 1, with 48 inputs and 32 outputs:

IOCANID1=1 ICAN1=48 OCAN1=32

NUICAN1 (P102)
NUOCAN1 (P103)
NUICAN2 (P104)
NUOCAN2 (P105)
NUICAN3 (P106)
UOCAN3 (P107)
NUICAN4 (P108)
NUOCAN4 (P109)

They are used to set the remote modules

NUICAN* indicates the number of the first input and NUOCAN* that of the first output of the group.

The Central Unit reserves local inputs I1 through I64 and local outputs O1 through O32. When having an expansion board that includes I/Os, it also reserves local inputs I65 through I128 and local output O33 through O64.

On remote module the inputs and outputs of the different elements are numbered sequentially. The inputs and outputs are defined in groups of 8 and the possible values of NUICAN* and NUOCAN* must be multiple of 8 plus 1 (8n +1).

Possible NUICAN values without the expansion board that has I/Os:

65, 73, 81, 89, 97, 105, 113, 121, 129, 137, 145, 153....

Possible NUICAN values with the expansion board that has I/Os:

129, 137, 145, 153, 161, 169, 177,185, 193, 201, 209....

Possible NUOCAN values without the expansion board that has I/Os:

33, 41, 49, 57, 65, 73, 81, 89, 97, 105, 113, 121, 129....

Possible NUOCAN values without the expansion board that has I/Os:

65, 73, 81, 89, 97, 105, 113, 121, 129, 137, 145, 153....

If NUICAN=0 or NUOCAN=0, the group following the one assigned to the previous node is assigned to the corresponding node.

Examples of remote module setting

The Central Unit does not have an I/O expansion board
Remote module in node 1, with 48 inputs and 32 outputs,
Remote module in node 2, with 24 inputs and 16 outputs,

Case 1: We want the inputs and outputs to be sequential starting with the first ones available.

IOCANID1=1	ICAN1=48	OCAN1=32	NUICAN1=0	NUOCAN1=0
IOCANID1=2	ICAN1=24	OCAN1=16	NUICAN1=0	NUOCAN1=0

Node 1 is assigned inputs I65 through I112 and outputs O33 through O64
Node 2 is assigned inputs I113 through I136 and outputs O65 through O80

Case 2: We want the inputs and outputs to be sequential; but starting with I129 and O65, getting it ready for the I/O expansion board.

IOCANID1=1	ICAN1=48	OCAN1=32	NUICAN1=129	NUOCAN1=65
IOCANID1=2	ICAN1=24	OCAN1=16	NUICAN1=0	NUOCAN1=0

Node 1 is assigned inputs I129 through I176 and outputs O65 through O96
Node 2 is assigned inputs I177 through I200 and outputs O97 through O112

Case 3: We expect an I/O and element expansion in node 1 (up to 72 inputs and 48 outputs).

IOCANID1=1	ICAN1=48	OCAN1=32	NUICAN1=129	NUOCAN1=65
IOCANID1=2	ICAN1=24	OCAN1=16	NUICAN1=201	NUOCAN1=113

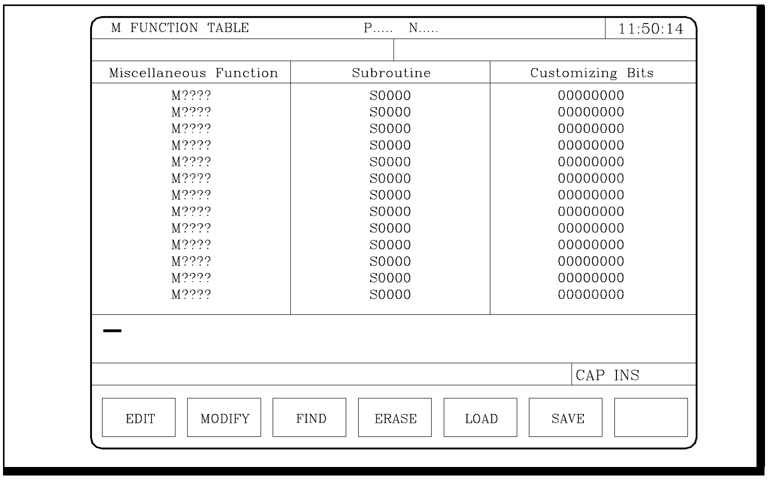
Node 1 is assigned inputs I129 through I176 and outputs O65 through O96
Node 2 is assigned inputs I201 through I224 and outputs O113 through O128

3.8 Tables

3.8.1 Miscellaneous (M) function table

The number of M functions in this table is determined by the g.m.p. NMISCFUN (P29), being possible to define up to 255 M functions.

It must borne in mind that functions: M00, M01, M02, M03, M04, M05, M06, M8, M9, M19, M30, M41, M42, M43 and M44, besides what is indicated in this table, have specific meanings when programming the CNC.



Each miscellaneous function will be called by its M number

Possible values: Integers between 0 and 9999.
The table elements that are not defined will be displayed as M????.

A subroutine can be associated with each M function and it will be indicated by the letter S.

Possible values: Integers between 0 and 9999. If 0 is assigned to this field, it means that the M function has no subroutine associated with it.

The third field consists of 8 customizing bits called bit 0 through bit 7:

*	*	*	*	*	*	*	*
7	6	5	4	3	2	1	0

bit 0 Indicates whether the CNC must wait or not for the AUX END signal (M done) to consider it executed and go on to the next program block

0 It waits for the AUX END signal.

1 It does NOT wait for AUX END.

bit 1 Indicates whether the M function is executed before or after the movement block where it is programmed.

0 It is executed before the move.

1 It is executed after the move.

bit 2 Indicates whether the M function interrupts the block preparation or not.

0 It does NOT interrupt the block preparation.

1 It interrupts the block preparation.

bit 3 Indicates whether the M function is executed or not after the associated subroutine is executed.

0 It is executed after the associated subroutine.

1 ONLY the associated subroutine is executed.

bit 4 When bit "2" has been set to "1", it indicates whether block preparation is to be interrupted until the execution of the M function begins or until it ends (until the M-done signal is received).

0 It interrupts block preparation until the execution of the "M" function begins.

1 It interrupts block preparation until the "M-done" signal (AUXEND) is received.

bit 5 Not being used at this time.

bit 6 Not being used at this time.

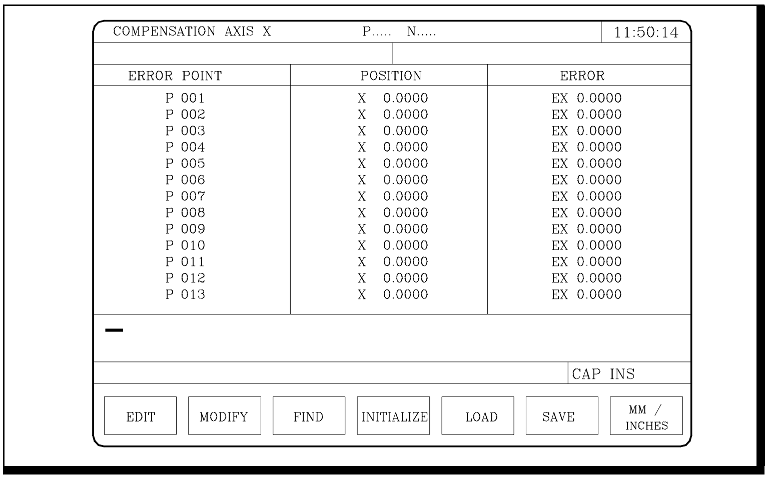
bit 7 Not being used at this time.

When executing an M function which has not been defined in the M table, the programmed function will be executed at the beginning of the block and the CNC will “wait” for the “AUXEND” signal to continue the execution of the program.

3.8.2 Leadscrew error compensation table

The CNC will provide a table for each one of the axes having leadscrew compensation. This type of compensation is selected by setting g.m.p. LSCRWCOM (P15).

The number of elements of the table is determined by the g.m.p. NPOINTS (P16), being possible to define up to 255 points per axis.



Each parameter of the table represents a profile point to be compensated defining for each one:

The axis position of that profile point with respect to Machine Reference ZERO.

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

The leadscrew error in this point.

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

When defining the profile points in the table, the following requirements must be met:

- The axis points must be in sequential order starting from the most negative (least positive) point to be compensated.
- For those points outside the compensation zone, the CNC will apply the compensation value corresponding to the table point closest to them.
- The machine reference point must have no error (zero).
- The error difference between two consecutive points must not be greater than the distance between them (maximum slope= 100%).

On rotary axes, although the display is limited between 0 and 360º, the internal count is accumulative. When using leadscrew error compensation, set positions 0° and 360°, first and last point of the table, with the same amount of error. This way, the CNC will apply the same compensation in all the revolutions.

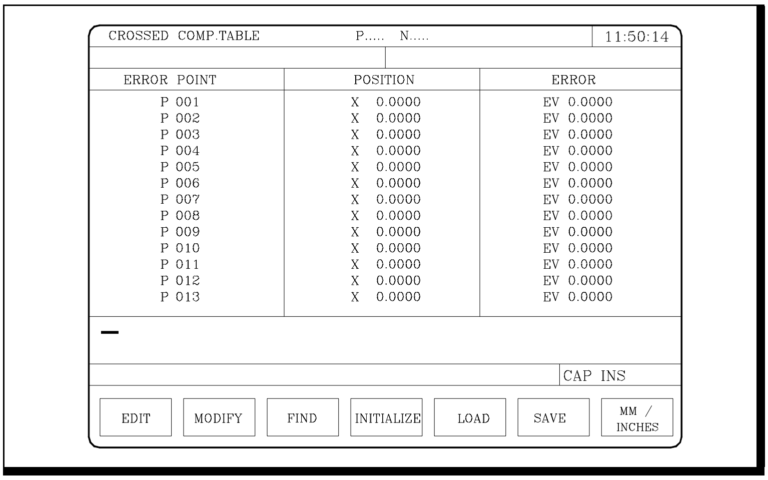
Otherwise, the compensation will be limited to the indicated field.

3.8.3 Cross compensation parameter table

Up to 3 cross compensation tables may be used. To enable each one, set g.m.p.:

MOVAXIS	(P32)	COMPAXIS	(P33)	NPCROSS	(P31)
MOVAXIS2	(P55)	COMPAXIS2	(P56)	NPCROSS2	(P54)
MOVAXIS3	(P58)	COMPAXIS3	(P59)	NPCROSS3	(P57)

NPCROS indicates the number of points of the table, MOVAXIS indicates the axis that moves and COMPAXIS the axis affected by the movement of the "movaxis" and, consequently, to be compensated.



The table must set the amount of error to be compensated in specific positions of the moving axis.

The position is defined in home coordinates (referred to machine reference zero). Depending on g.m.p. TYPCROSS (P135), the CNC will take into account either the theoretical or real (actual) coordinates.

Possible values for the Position and Error fields:

±99999.9999 millimeters. ±3937.00787 inches.

When defining the profile points in the table, the following requirements must be met:

- The axis points must be in sequential order starting from the most negative (least positive) point to be compensated.
- For those points outside the compensation zone, the CNC will apply the compensation value corresponding to the table point closest to them.
- The machine reference point must have no error (zero).

When both leadscrew and cross compensations are applied on the same axis, the CNC will apply the sum of the two.

User notes:

8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3X)
(SOFT T: 6.3X)

CONCEPTS

4

4. CONCEPTS

Warning:

It is recommended to save the machine parameters as well as the PLC program and files into the "Memkey Card" (CARD A) or in a peripheral or PC to avoid losing them.

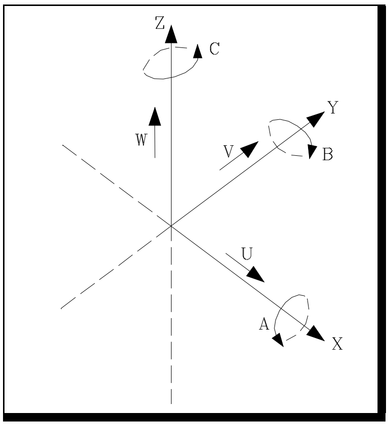
4.1 Axes and coordinate systems

Given that the objective of the CNC is to control the movement and positioning of axes, it is necessary to determine the position of the point to be reached through the coordinates.

The CNC allows you to use absolute, relative or incremental coordinates throughout the same program.

Axis nomenclature

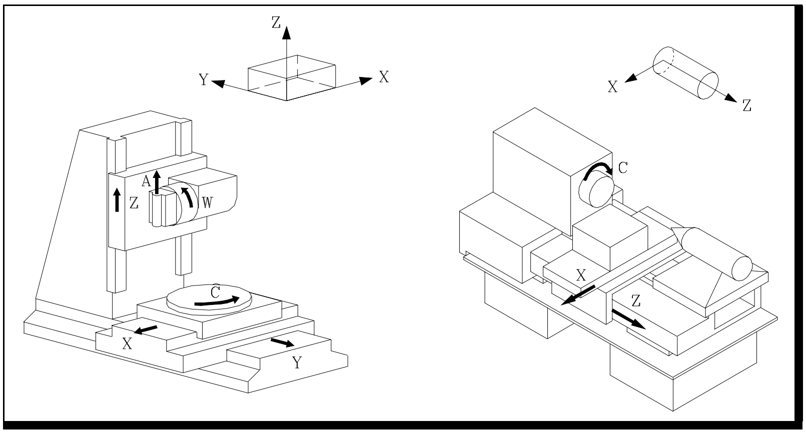
The axes are named according to DIN 66217.



Characteristics of the system of axes:

X,Y	main movements on the main work plane of the machine.
Z	parallel to the main axis of the machine, perpendicular to the main XY plane.
U,V,W	auxiliary axes parallel to X,Y, Z respectively
A,B,C	rotary axes on each of the X,Y, Z axes.

In the figure (below) an example of the nomenclature of the axes on a milling-profiling machine with a tilted table.



Axis selection

Of the 9 possible axes that may exist, the CNC allows the manufacturer to select up to 7 of them.

Moreover, all the axes should be suitably defined as linear/rotary, etc. through the machine parameters of axes which appear in the Installation and Start-up Manual.

There is no limitation to the programming of the axes, and interpolations can be made simultaneously with up to 7 axes.

Example of milling: The machine has three regular linear axes: X, Y and Z, one linear U axis controlled by the PLC, an analog Spindle (S) and an electronic handwheel.

Setting of g.m.p. AXIS1(P0) through AXIS8 (P7)

AXIS1 (P0) = 1	X axis	associated with feedback X1 and output O1
AXIS2 (P1) = 2	Y axis	associated with feedback X2 and output O2
AXIS3 (P2) = 3	Z axis	associated with feedback X3 and output O3
AXIS4 (P3) = 4	U axis	associated with feedback X4 and output O4
AXIS5 (P4) = 10	Spindle (S)	associated with feedback X5(1-6) & output O5
AXIS6 (P5) = 0		
AXIS7 (P6) = 11	Handwheel	associated with feedback input X6(1-6)
AXIS8 (P7) = 0		

The CNC activates a machine parameter table for each axis (X, Y, Z, U) and another one for the spindle (S).

a.m.p. AXISTYPE (P0) must be set as follows:

X axis	AXISTYPE (P0) = 0	Regular linear axis
Y axis	AXISTYPE (P0) = 0	Regular linear axis
Z axis	AXISTYPE (P0) = 0	Regular linear axis
U axis	AXISTYPE (P0) = 5	Regular linear axis controlled by the PLC

a.m.p. SPDLTYPE (P0) must be set as follows:

SPDLTYPE (P0) = 0 ±10V analog output.

Likewise, a.m.p DFORMAT (P1) and s.m.p. DOFORMAT (P1) must be properly set to indicate their display formats.

Example of lathe: The machine has two regular linear axes: X and Z, an analog spindle (S) and an auxiliary spindle (live tool).

Setting of g.m.p. AXIS1 (P0) through AXIS8 (P7)

AXIS1 (P0) = 1	X axis	associated with feedback X1 and output O1
AXIS2 (P1) = 3	Z axis	associated with feedback X2 and output O2
AXIS3 (P2) = 10	Spindle (S)	associated with feedback X3 and output O3
AXIS4 (P3) = 13	Auxiliary spindle	associated with feedback X5(1-6) & output O5
AXIS5 (P4) = 0		
AXIS6 (P5) = 0		
AXIS7 (P6) = 0		
AXIS8 (P7) = 0		

The CNC activates a machine parameter table for each axis (X, Z), one for the main spindle (S) and another one for the auxiliary spindle.

a.m.p. AXISTYPE (P0) must be set as follows:

X axis	AXISTYPE (P0) = 0	Regular linear axis
Z axis	AXISTYPE (P0) = 0	Regular linear axis

a.m.p. SPDLTYPE (P0) must be set as follows:

SPDLTYPE (P0) = 0 ±10V analog output.

Likewise, a.m.p DFORMAT (P0) and s.m.p. DOFORMAT (P0) must be properly set to indicate their display formats.

Rotary axes

With this CNC, it is possible to select the type of rotary axis by means of a.m.p. AXISTYPE(P0) which may be:

Normal rotary axis	AXISTYPE (P0) = 2
Positioning-only axis	AXISTYPE (P0) = 3
Rotary HIRTH axis.	AXISTYPE (P0) = 4

By default, their position is always displayed between 0 and 360º (Rollover axis).

If these limits are not to be set, modify a.m.p. ROLLOVER (P55).

ROLLOVER = YES	rotary axis display between 0 and 360º
ROLLOVER = NO	No display limits.

Although the display is limited between 0 and 360º, the internal count is accumulative. Therefore, a.m.p. "LIMIT+(P5)" and "LIMIT-(P6)" should be set to limit the maximum number of turns in each direction.

When both parameters are set to "0", the axis can move indefinitely in either direction (rotary tables, indexers, etc.). See section ***3.2 Axis parameters***

When using leadscrew error compensation, set positions 0° and 360°, first and last point of the table, with the same amount of error. This way, the CNC will apply the same compensation in all the revolutions. See section ***4.3.7 Leadscrew error compensation***

Normal rotary axes

- They can interpolate with linear axes.
- G00 and G01 movement
- Absolute coordinate programming (G90):
  The sign indicates the turning direction and the end coordinate the position (between 0 and 359.9999).
- Incremental coordinate programming (G91):
- The sign indicates the turning direction. If the programmed movement exceeds 360º, the axis will turn more than once before positioning at the desired point.

Normal rotary axis AXISTYPE=2		
LIMIT+ = 8000 LIMIT- =-8000	ROLLOVER=YES	Position reading between 0 and 360º G90 The sign indicates the turning direction G91 The sign indicates the turning direction
	ROLLOVER=NO	Position reading between 7999.9999° and -7999.9999° G90, G91 like a linear axis
LIMIT+ = 0 LIMIT- =0	ROLLOVER=YES	Position reading between 0 and 360º G90 The sign indicates the turning direction G91 The sign indicates the turning direction
	ROLLOVER=NO	Strange position reading; there are 2 loops, one between 0 and 360º and the other one between 0 and –360º It is possible to switch from one to the other. G90, G91 like a linear axis
LIMIT+ = 350 LIMIT- =10	ROLLOVER=YES/ NO	It can only move between 10º and 350º G90 and G91 as with limits of 8000 and –8000, but it will issue an error message if the target position is beyond limits.

Positioning-only axis

- It cannot interpolate with linear axes.
- Movement always in G00, and they do not admit tool radius compensation (G41, G42).
- Absolute coordinate programming (G90):
  Always positive and in the shortest direction. End coordinate between 0 and 359.9999.
- Incremental coordinate programming (G91):
- The sign indicates the turning direction. If the programmed movement exceeds 360º, the axis will turn more than once before positioning at the desired point.

Positioning-only rotary axis AXISTYPE=3		
LIMIT+ = 8000 LIMIT- =-8000	ROLLOVER=YES	Position reading between 0 and 360º G90 does not admit a negative value. Always via the shortest path G91 The sign indicates the turning direction
	ROLLOVER=NO	Position reading between 7999.9999° and -7999.9999° G90, G91 like a linear axis
LIMIT+ = 0 LIMIT- =0	ROLLOVER=YES	Position reading between 0 and 360º G90 does not admit a negative value. Always via the shortest path G91 The sign indicates the turning direction
	ROLLOVER=NO	Strange position reading; there are 2 loops, one between 0 and 360º and the other one between 0 and –360º It is possible to switch from one to the other. G90, G91 like a linear axis
LIMIT+ = 350 LIMIT- =10	ROLLOVER=YES/ NO	It can only move between 10º and 350º G90 and G91 as with limits of 8000 and –8000, but it will issue an error message if the target position is beyond limits.

Rotary Hirth axis

- It is a positioning-only axis which cannot take decimal coordinates. All positioning movements must be in whole degrees.
- More than one hirth axis may be used, but they can only move one at a time.

Rotary Hirth axis (whole degrees) AXISTYPE=3		
LIMIT+ = 8000 LIMIT- =-8000	ROLLOVER=YES	Position reading between 0 and 360º G90 does not admit a negative value. G91 The sign indicates the turning direction
	ROLLOVER=NO	Position reading between 7999.9999° and -7999.9999° G90, G91 like a linear axis
LIMIT+ = 0 LIMIT- =0	ROLLOVER=YES	Position reading between 0 and 360º G90 does not admit a negative value. Always via the shortest path G91 The sign indicates the turning direction
	ROLLOVER=NO	Strange position reading; there are 2 loops, one between 0 and 360º and the other one between 0 and –360º It is possible to switch from one to the other. G90, G91 like a linear axis
LIMIT+ = 350 LIMIT- =10	ROLLOVER=YES/ NO	It can only move between 10º and 350º G90 and G91 as with limits of 8000 and –8000, but it will issue an error message if the target position is beyond limits.

GANTRY axes

Gantry axes are any two axes that, due to the way the machine is built, must move together in synchronism. For example: bridge type mills.

Only the movements of one of those axes must be programmed and it is called the main axis. The other axis is referred to as "slave axis".

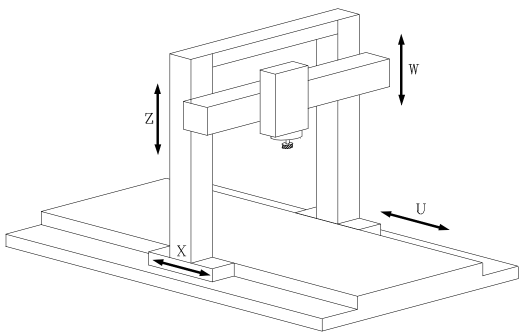
In order to operate this way, it is necessary to have the a.m.p. GANTRY (P2) corresponding to both axes set as follows:

- Parameter "GANTRY" of the main axis set to "0".
- Parameter "GANTRY" of the slave axis must indicate which axis is its "master" (or main axis).

Also, a.m.p. MAXCOUPE (P45) of the slave axis must indicate the maximum allowed difference between the following errors of both axes.

It is possible to have more than one pair of gantry axes.

Example of a bridge type milling machine with two Gantry axes (X-U, Z-W).



Machine parameters:
X axis: GANTRY=0
U axis: GANTRY=1
Z axis: GANTRY=0
W axis: GANTRY=3

Slaved axes and synchronized axes

Coupled or synchronized axes are two or more axes which are normally independent, but, sometimes need to be moved at the same time and in synchronism (temporarily slaved, versus permanently as by machine parameter). For example on multi-spindle milling machines.

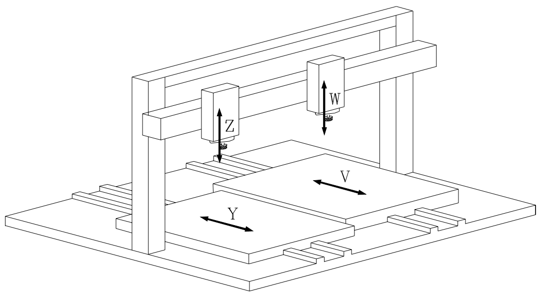
Slaved axes:

- With function G77 it is possible to define which axes are to be coupled (temporarily slaved) by indicating the main axis and its subordinates or slave axes.
- It is possible to slave more than two axes to each other, to have several different electronic couplings (slaving), to add a new slave to the ones previously slaved, etc.
- With function G78, it is possible to decouple (unslave) one or all of the axes slaved temporarily; that is by means of G77, and not by machine parameter GANTRY (which would be "permanent" slaving).

Synchronized axes:

- The axes are synchronized by the PLC, by activating the CNC input "SYNCHRO" of the axis to become the slave.
- To be able to do this, a.m.p. SYNCHRO (P3) of that axis must be set indicating which axis will be its master.
- It is possible to couple (slave) more than two axes to each other, to have several other axes slaved to each other, to add a new slave to existing ones, etc; but, they will always be slaved to the axes determined by the corresponding SYNCHRO parameters:
- To decouple (unslave) one of the slaved axes, the corresponding "SYNCRO" input of the CNC must be deactivated.

Example of a multi-spindle bridge type milling machine with two pairs of slaved axes (Y-V, Z-W).



Slaving
(by program):
G77 Y V
G77 Z W

Synchronism
(by external signal):
Y axis: SYNCHRO=0
V axis: SYNCHRO=2
Z axis: SYNCHRO=0
W axis:
SYNCHRO=3

If the machine has the X, Y, Z, V, W axes, the following signals must be activated (logic state "1") at the PLC:

SYNCHRO4 to slave the V axis to the Y axis.
SYNCHRO5 to slave the W axis to the Z axis.

4.1.1 Relationship between the axes and the JOG keys

The Mill model CNC has 5 pairs of JOG keys and the Lathe model has 4 pairs of keys to jog the axes of the machine.

Mill Model

Lathe Model

The X, Y and Z axes always use their own nomenclature and the rest of the axes depend on the chosen name.

The logical order is: X Y Z U V W A B C.

Examples: A milling machine has the X Y Z U B axes.

The X axis keys are:	[X+]	[X-]
The Y axis keys are:	[Y+]	[Y-]
The Z axis keys are:	[Z+]	[Z-]
The U axis keys are:	[4+]	[4-]
The B axis keys are:	[5+]	[5-]

A laser machine has the X Y A B axes.

The X axis keys are:	[X+]	[X-]
The Y axis keys are:	[Y+]	[Y-]
The A axis keys are:	[Z+]	[Z-]
The B axis keys are:	[4+]	[4-]

A punch press has the X Y C axes

The X axis keys are:	[X+]	[X-]
The Y axis keys are:	[Y+]	[Y-]
The C axis keys are:	[Z+]	[Z-]

A lathe has the X Z U A axes

The X axis keys are:	[X+]	[X-]
The Z axis keys are:	[Z+]	[Z-]
The U axis keys are:	[3+]	[3-]
The A axis keys are:	[4+]	[4-]

4.1.2 Movement with an electronic handwheel

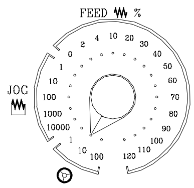
The various handwheel configurations are:

General handwheel

It can be used to jog any axis one by one.
Select the axis and turn the handwheel to move it.

Individual handwheel:

It replaces the mechanical handwheels.
Up to 3 handwheels can be used (one per axis).
It only moves the axis it is associated with.



To move any of them, turn the switch to any of the handwheel positions.

Positions 1, 10 and 100 indicate the multiplying factor being applied besides the internal x4 to the feedback pulses supplied by the electronic handwheel.

For example, if the manufacturer has set a distance of 0.100 mm or 0.0100 inches per handwheel turn, thus:

Switch position	Distance per turn	
1	0.100 mm	or 0.0100 inch
10	1.000 mm	or 0.1000 inch
100	10.000 mm	or 1.0000 inch

There are 3 operating modes with handwheels:

Standard handwheel:

- With the general handwheel, select the axis to be moved and turn the handwheel.
- With individual handwheels, turn the handwheel associated with the axis to be moved.

Path handwheel:

- For chamfering and rounding corners.
- 2 axes are moved along a selected path (chamfer or rounding) by moving a single handwheel.
- This feature must be handled by the PLC.
- The general handwheel is assumed as the "path handwheel" or the individual handwheel associated with the X axis (Mill) or Z (lathe).

Feed handwheel

- To control the feedrate of the machine.
- This feature must be handled by the PLC.

Warning:

Depending on the turning speed of the handwheel and the position of the selector switch, when requesting a movement at a faster feedrate than the maximum allowed.

- *With individual handwheels, the movement stops when stopping the handwheel. It does not move the indicated distance.*
- *With general handwheels, g.m.p. HDIFFBAC (P129) indicates whether the movement is stopped or it moves the indicated distance.*

Standard handwheel

With the general handwheel proceed as follows:

1. Select the axis to be jogged.

   Press one of the JOG keys of the axis to be jogged. The selected axis will be highlighted.

   When using a Fagor handwheel with an axis selector button, the axis may be selected as follows:

      Push the button on the back of the handwheel. The CNC select the first axis and it highlights it.

      When pressing the button again, the CNC selects the next axis and so on in a rotary fashion.

      To deselect the axis, hold the button pressed for more than 2 seconds.

2. Jog the axis

   Once the axis has been selected, it will move as the handwheel is being turned and in the direction indicated by it.

With individual handwheels:

Each axis will move as the corresponding handwheel is being turned according to the switch position and in the direction indicated by it.

Simultaneous handwheels

The machine may have a general handwheel and up to 3 individual handwheels associated with each axis.

The individual handwheels have priority over the general handwheel. So, if an individual handwheel is moving, the general handwheel will be ignored.

Path handwheel:

With this feature, it is possible to jog two axes at the same time along a linear path (chamfer) or circular path (rounding) with a single handwheel.

The CNC assumes as the path handwheel the general handwheel or, when this one is missing, the one associated with the X axis (Mill) or Z axis (lathe).

This feature must be handled by the PLC.

To activate or cancle the "Path handwheel" work mode, act upon the logic CNC input "MASTRHND" M5054,

M5054 = 0 Normal handwheels.

M5054 = 1 Path handwheel mode ON.

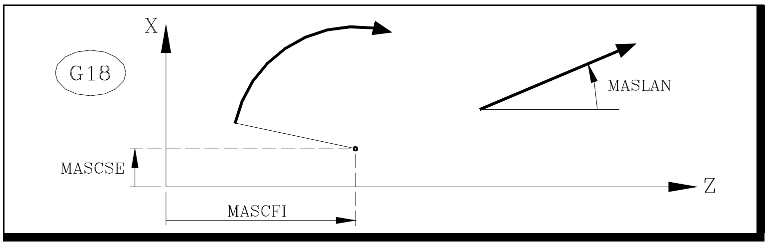
To indicate the type of movement, act upon the logic CNC input "HNLINARC" M5053,

M5053 = 0 Along a linear path.

M5053 = 1 Along an arc.

For a linear path, indicate the angle of the path at the MASLAN variable (value in degrees between the linear path and the firs axis of the plane).

For an arc, indicate the coordinates of the arc center at the MASCFI and MASCSE variables (for the first and second axes of the main plane).



The MASLAN, MASCFI and MASCSE variables can read or written from the CNC, DNC and the PLC .

__Simultaneous handwheels__

When selecting the Path Handwheel mode, the CNC behaves as follows:

- If there is a General Handwheel, it will be the one working in Path handwheel mode. The individual handwheels, if any, will remain associated with the corresponding axes.
- If there is no General Handwheel, one of the individual handwheel starts working in Path handwheel mode. The one associated with the X axis if Mill model or the one associated with the Z if lathe model.

Feed handwheel mode

Usually, when making a part for the first time, the machine feedrate is controlled by means of the feedrate override switch.

From this version on, it is also possible to use the machine handwheels to control that feedrate. This way, the machining feedrate will depend on how fast the handwheel is turned.

To do this, proceed as follows:

- Inhibit all the feedrate override switch positions from the PLC.
- Detect how far the handwheel is turned (reading of pulses received)
- Set the corresponding feedrate override from the PLC depending on the pulses received from the handwheel.

The following CNC variables return the number of pulses the handwheel has turned.

HANPF	shows the number of pulses of the 1st handwheel.
HANPS	shows the number of pulses of the 2nd handwheel.
HANPT	shows the number of pulses of the 3rd handwheel.
HANPFO	shows the number of pulses of the 4th handwheel.

Example: The machine has a button to activate and deactivate this feature (feed handwheel) and the feedrate control is carried out with the second handwheel.

```
CY1
R101=0
END
```

Resets the register containing the previous handwheel reading

```
PRG
DFU I71 = CPL M1000
```

Every time the button is pressed, mark M1000 is inverted

```
M1000 = MSG1
```

If the feature is active, a message is displayed.

```
NOT M1000
= AND KEYDIS4 $FF800000 KEYDIS4
= JMP L101
```

If this feature is not active, it enables all the positions of the Feedrate Override Switch and resumes program execution

```
DFU M2009
= CNCRD(HANPS,R100,M1)
= SBS R101 R100 R102
= MOV R100 R101
= MLS R102 3 R103
= OR KEYDIS4 $7FFFFF KEYDIS4
```

If this feature is activated and an upflank occurs at the clock mark M2009, it reads in R100 the handwheel pulses (HANPS), calculates in R102 the number of pulses received from the last reading, updates R101 for the next reading, calculates in R103 the value of the right feedrate % and inhibits all the positions of the Feedrate Override Switch (KEYDIS4).

```
CPS R103 LT 0 = SBS 0 R103 R103
```

CPS R103 GT 120 = MOV 120 R103

It adjusts the value of R103 (feedrate %). It ignores the handwheel turning direction (sign) and limits the value to 120%

DFU M2009
= CNCWR(R103,PLCFRO,M1)

With the up flank at the clock mark M2009, set the calculated feedrate override (PLCFRO=R103)

L101

END

4.2 feedback system

The various feedback inputs available at the CNC admit sinewave and squarewave differential signals from feedback systems.

The following axis machine parameters indicate the type of feedback system and the resolution utilized for each axis.

When using linear feedback devices

PITCH (P7)	Ballscrew pitch or that of the linear encoder being used
NPULSES (P8)	= 0
DIFFBACK (P9)	Indicates whether the feedback device uses differential signals (double ended) or not.
SINMAGNI (P10)	Feedback multiplying factor applied by the CNC.
FBACKAL (P11)	Feedback alarm (only with differential signals).

When using rotary encoders

PITCH (P7)	Number of degrees per encoder turn.
NPULSES (P8)	Number of pulses (lines) per encoder turn.
DIFFBACK (P9)	Indicates whether the feedback device uses differential signals (double ended) or not.
SINMAGNI (P10)	Feedback multiplying factor applied by the CNC.
FBACKAL (P11)	Feedback alarm (only with differential signals).

Next, the feedback counting speed (frequency) limitation is described as well as how to set these machine parameters for the axes.

4.2.1 Counting speed limitation

Sinewave signals

The maximum counting speed (frequency) for sinewave feedback is 250KHz.

The maximum feedrate for each axis will depend upon the selected resolution and the signal pitch (distance per pulse) in use while with rotary encoders it will depend on the number of pulses per revolution.

Example 1: When using a Fagor linear encoder, the signal pitch is 20 µm. Therefore, with a counting resolution of 1 µm, the maximum feedrate will be:

20 µm/pulse x 250.000 pulses/sec = 5 m/sec = 300 m/min.

When using Fagor linear encoder, the maximum feedrate is limited by their own characteristics to 60 m/min.

Example 2: Using an indexer with a sinewave Fagor encoder of 3600 lines per turn. Therefore, for a feedback resolution of 1 µm, the maximum axis feedrate will be:

(360 degrees/turn / 3600 pulses/turn) x 250,000 pulses/s. =
= 25.000 degrees/sec.= 1.500.000 degrees/min.

Since Fagor sine-wave encoders admit a frequency of up to 200 KHz, the maximum feedrate will be:

(360 degrees/turn / 3600 pulses/turn) x 200.000 pulses/s. =
= 20,000 degrees/sec.= 1,200,000 degrees/min.

Squarewave signals

The maximum frequency (speed) for squarewave differential feedback is 425 KHz. with a separation of 450 ns between A and B flanks. Which is equivalent to 90º ±20º.

The maximum feedrate for each axis will depend upon the selected resolution and the signal pitch (distance per pulse) in use.

When using Fagor linear encoder, the maximum feedrate is limited by their own characteristics to 60 m/min.

When using FAGOR rotary encoders, their intrinsic output frequency limit is (200Kz).

4.2.2 Resolution

The CNC provides a number of machine parameters for the axes and for the spindle in order to establish the counting resolution of each one of the axes and the spindle.

PITCH (P7) Defines the pitch of the ballscrew or the linear feedback device (scale) being used. When using a Fagor linear encoder, the signal pitch to be entered here will be either 20 µm or 100 µm.

When dealing with a rotary encoder, it must indicate the number of degrees per encoder turn. For example, if the encoder is mounted onto a motor with a 1/10 gear reduction, this parameter must be set to 360º/10 = 36.

NPULSES (P8) Indicates the number or pulses/rev provided by the rotary encoder. Enter a value of 0 when using linear encoders. When using gear ratios, the whole assembly must be taken into account when defining the number of pulses per turn.

SINMAGNI (P10) Indicates the multiplying factor (x1, x4, x20, etc.) that the CNC must apply only to sinewave feedback signals.

Set this parameter to "0" when using squarewave feedback signals and the CNC will always apply a x4 multiplying factor.

The counting resolution for each axis will be defined by means of the combination of these parameters as shown in the following table:

	PITCH	NPULSES	SINMAGNI
Square signal encoder	Ballscrew pitch	# of Lines	0
Sinusoidal signal encoder	Ballscrew pitch	# of Lines	multiplying factor
Square signal linear encoder	linear encoder pitch	0	0
Sinusoidal signal linear encoder	linear encoder pitch	0	multiplying factor

Example 1: Resolution in "mm" with squarewave encoder

We would like to obtain a 2µm resolution by using a squarewave encoder mounted on 5 mm pitch ballscrew.

Since the CNC applies a x4 multiplying factor to squarewave signals, we would require an encoder which provides the following number of pulses (lines) per turn.

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

Nr pulses = 5000 µm / (4 x 2 µm) = 625 pulses/turn

Therefore:

INCHES = 0   PITCH=5.0000   NPULSES = 625   SINMAGNI=0

Although the CNC accepts a maximum squarewave frequency of 400 KHz, when using Fagor squarewave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

Max. Feed = 200,000 pulses/sec. / 625 pulses./turn) x 5 mm/turn

Max. feed = 1600 mm/sec. = 96 m/min.

Example 2: Resolution in "mm" with sinewave encoder

We would like to obtain a 2µm resolution by using a 250-line sinewave encoder mounted on 5 mm-pitch ballscrew.

We must calculate the multiplying factor "SINMAGNI" to be applied by the CNC to the pulses provided by the encoder in order to obtain the desired resolution.

SINMAGNI = ballscrew pitch / (Nr pulses x Resolution)

SINMAGNI = 5000 µm / (250 x 2 µm) = 10

Therefore:

INCHES = 0	PITCH=5.0000	NPULSES = 250	SINMAGNI=10

Although the CNC accepts a maximum squarewave frequency of 250 KHz, when using Fagor sine-wave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

Max. Feed = 200,000 pulses/sec. / 250 pulses./turn) x 5 mm/turn

Max. feed = 4.000 mm/sec. = 240 m/min.

Example 3: Resolution in "mm" with squarewave linear encoder

Since the CNC applies a x4 multiplying factor to squarewave signals, we must select a linear encoder whose grading pitch is 4 times the desired resolution.

FAGOR linear encoders use a grading pitch of either 20 µm or 100 µm. Therefore, the resolution that can be obtained with them are: 5 µm (20/4) or 25 µm (100/4).

Therefore:

INCHES = 0	PITCH=0.0200 PITCH=0.1000	NPULSES = 0	SINMAGNI=0

The CNC's maximum squarewave feedback input frequency is 400 KHz which means that the maximum feedrate obtainable with a 20 µm pitch linear encoder is:

Max. Feed = 20 µm/pulse x 400000 pulses/sec.

Max. feed = 8000 mm/sec. = 480 m/min.

When using Fagor linear encoder, the maximum feedrate is limited by their own characteristics to 60 m/min.

Example 4: Resolution in "mm" with sinewave linear encoder

We have a sinewave linear encoder with a 20 µm pitch and we would like to obtain 1 µm resolution.

We must calculate the multiplying factor "SINMAGNI" to be applied by the CNC to the pulses provided by the linear encoder in order to obtain the desired resolution.

SINMAGNI = linear encoder pitch / resolution = 20 µm / 1 µm = 20

Therefore:

INCHES = 0   PITCH=0.0200   NPULSES = 0   SINMAGNI=20

The CNC's maximum sinewave feedback input frequency is 250 KHz which means that the maximum feedrate for this axis will be:

Max. Feed = 20 µm/pulse x 250,000 pulses/sec.

Max. feedrate = 5,000 mm/s. = 300 m/min.

When using Fagor linear encoder, the maximum feedrate is limited by their own characteristics to 60 m/min.

Example 5: Resolution in "inches" with squarewave encoder

Calculate the necessary squarewave encoder line count and parameter settings to obtain a 0.0001 inch counting resolution on a 4 pitch ballscrew (4 turns/inch = 0.25 inch/rev.).

Since the CNC applies a x4 multiplying factor to squarewave signals, we would require an encoder which provides the following number of pulses (lines) per turn.

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

Nr pulses = 0.25 / (4 x 0.0001) = 625 pulses/turn

Therefore:

INCHES = 1   PITCH=0.25000   NPULSES = 625   SINMAGNI=0

Although the CNC accepts a maximum squarewave frequency of 400 KHz, when using Fagor squarewave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

Max. Feed = 200,000 pulses/sec. / 625 pulses./turn) x 0.255 inch/turn

Max. Feed = 80 inches/sec. = 4800 inches/min

Example 6: Resolution in "inches" with sinewave encoder

We would like to obtain a0.0001 inch resolution by using a 250-line sinewave encoder mounted on a ballscrew with a 5 turns/inch pitch.

We must calculate the multiplying factor "SINMAGNI" to be applied by the CNC to the pulses provided by the encoder in order to obtain the desired resolution.

SINMAGNI = ballscrew pitch / (Nr pulses x Resolution)

SINMAGNI = 0.2 inch/turn / (250 x 0.0001) = 8

Therefore:

INCHES = 1 PITCH=0.20000 NPULSES = 250 SINMAGNI=8

Although the CNC accepts a maximum squarewave frequency of 250 KHz, when using Fagor sine-wave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

Max. feedrate = (200.000 pulses/sec. / 250 pulses/turn) x 0.2 inch/turn

Max. feedrate = 160 inch/sec. = 9.600 inch/min.

Example 7: Resolution in "degrees" with squarewave encoder

We would like to obtain a 0.0005º resolution by using a squarewave encoder mounted on a x10 reduction gear.

Since the CNC applies a x4 multiplying factor to squarewave signals, we would require an encoder which provides the following number of pulses (lines) per turn.

Nr of pulses = º/turn / (multiplying factor x gear ratio x Resolution)

Nr of pulses = 360 / (4 x 10 x 0.0005) = 18,000 pulses/turn

Therefore:

INCHES = 0 PITCH=36.0000 NPULSES = 18000 SINMAGNI=0

Although the CNC accepts a maximum squarewave frequency of 400 KHz, when using Fagor squarewave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

Max. Feed = 200,000 pulses/sec. / 18,000 pulses/turn)

Max. Feed = 11.111 turns/sec. = 666.666 rpm

Example 8: Resolution in "degrees" with sinewave encoder

We would like to obtain a 0.001º resolution by using a 3600-line sinewave encoder.

We must calculate the multiplying factor "SINMAGNI" to be applied by the CNC to the pulses provided by the encoder in order to obtain the desired resolution.

SINMAGNI = degrees per turn / (Nr. of pulses x Resolution)

SINMAGNI = 360 / (3600 x 0.001) = 100

Therefore:

INCHES = 0 PITCH=360.0000 NPULSES = 3600 SINMAGNI=100

Although the CNC accepts a maximum squarewave frequency of 250 KHz, when using Fagor sine-wave rotary encoders their output frequency is limited to 200KHz; thus, the maximum possible feedrate (F) will be:

Max. Feed = 200,000 pulses/sec. / 3,600 pulses/turn)

Max. Feed = 55,5556 turns/sec. = 3333,33 rpm

4.3 Axis adjustment

In order to be able to set the axes, their corresponding feedback devices must be previously connected to the CNC.

Before making this adjustment, position the axes near the middle of their travel and place the hard stops (monitored by the electrical cabinet) near these mid-travel points in order to prevent any possible damage to the machine.

The axis adjustment is carried out in two steps. First, the servo drive loop is adjusted and, then, the CNC loop.

Drive loop setting.

1. Verify that the power output of the drives is OFF. Set all a.m.p. FBALTIME (P12) to a value other than "0"; for example, FBALTIME=1000.
2. Turn the CNC OFF.
3. Turn the drive power output ON.
4. Turn the CNC ON.
5. If the axis runs away, the CNC will issue the Following Error message for this axis. Turn the CNC off and swap the tacho wires at the drive.
6. Repeat steps 4 and 5 until the CNC stops issuing errors.

Loop setting of the CNC.

The axes are set one at a time.

1. Select the JOG operating mode at the CNC.
2. Jog the axis to be adjusted.

   If the axis runs away, the CNC issues the corresponding following error message. In this case, the a.m.p. LOOPCHG (P26) must be changed.

   If the axis does not run away, but it does not move in the desired direction, Change both a.m.p. AXISCHG (P13) and LOOPCHG (P26).

4.3.1 Drive setting

Offset (drift) adjustment

This adjustment is made on one axis at a time:

- Select the JOG mode at the CNC and press the softkey sequence: [Display] [Following Error]. The CNC shows the current following Error (axis lag) of the axes.
- Adjust the offset by turning the offset potentiometer at the drive (NOT AT THE CNC) until a "0" following error is obtained.

Maximum feedrate adjustment

The drives should be adjusted so they provide maximum axis feedrate when receiving an analog voltage (velocity command) of 9.5 V.

Set each a.m.p. MAXVOLT (P37) = 9500 so the CNC outputs a maximum analog voltage of 9.5 V.

The maximum axis feedrate, a.m.p. MAXFEED (P42), depends on the motor rpm as well as on the gear reduction and type of ballscrew being used.

Example for the X axis:

The maximum motor rpm is 3,000 and the ballscrew pitch is 5mm/rev. Thus:

Maximum rapid traverse feedrate (G00) = ballscrew rpm. x ballscrew pitch
"MAXFEED" (P42) = 3,000 rpm. x 5mm/rev. = 15000 mm/min.

In order to adjust the drive, a.m.p. G00FEED (P38) should be set to the same value as a.m.p. MAXFEED (P42).

Also, a small CNC program must be executed which will move the axis back and forth a short distance in order to verify that the amount of following error in both directions is the same. One such program could be:

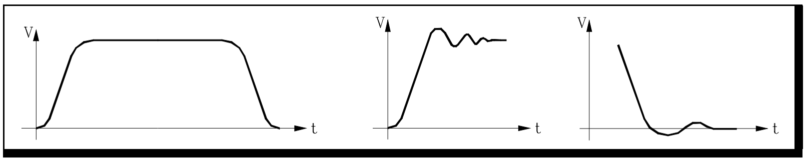
```
N10   G00 G90 X200
N20   X-200
      (RPT N10, N20)
```

While the axis is moving back and forth, measure the analog voltage provided by the CNC to the drive and adjust the feed potentiometer at the drive (NOT AT THE CNC) until reaching 9.5 V.

4.3.2 Gain setting

The various types of gains must be adjusted for each axis in order to optimize the system's performance for the programmed movements.

An oscilloscope is highly recommended to make this critical adjustment by monitoring the tacho signals. The illustration below shows the optimum shape for this signal (on the left) and the instabilities to be avoided during start-up and brake down:



There are three gain types for each axis. They are adjusted by means of axis machine parameters and following the sequence indicated next.

Proportional Gain

It defines the analog output corresponding to a feedrate resulting in 1º of following error.

It is defined with a.m.p. PROGAIN (P23)

Feed Forward Gain

It sets the percentage of analog output dependent of the programmed feedrate.

To use it, acc/dec must be active ACCTIME (P18)

It is defined with a.m.p. FFGAIN (P25)

Derivative Gain or AC- Forward Gain.

The "Derivative Gain" sets the percentage of analog output applied depending on the fluctuations of following error.

The "AC-Forward Gain" sets the percentage of analog output proportional to the feedrate increments (acceleration and deceleration stages).

To use it, acc/dec must be active ACCTIME (P18)

It is defined with a.m.p. DERGAIN (P24) and ACFGAIN (P46).

If ACFGAIN = No	it applies Derivative Gain.
If ACFGAIN = Yes	it applies AC-Forward Gain.

4.3.3 Proportional gain setting

In a "pure" proportional positional loop, the analog output of the CNC to control an axis is, at all times, proportional to the following error (axis lag) which is the difference between its theoretical and actual (real) position.

Analog output = Proportional Gain x Following Error

a.m.p. PROGRAIN (P23) sets the value of the Proportional Gain. It is given in millivolts/mm, it takes any integer between 0 and 65535.

Its value indicates the analog output corresponding to a feedrate resulting in 1 millimeter (0.03937 inch) of following error.

Example: The maximum feedrate for a particular axis (rapid traverse G00) is 15m/min, but we would like to limit its maximum programmable machining feedrate (F) to 3 m/min with a gain of 1 mm lag at a feedrate of 1m/min. (Gain of 1 in metric)

a.m.p. G00FEED (P38) must be set to 15,000 (15 m/min).

a.m.p. MAXVOLT (P37) must be set to 9500 and the servo drive adjusted so as to provide 15m/min with an analog voltage of 9.5 V.

a.m.p. MAXFEED (P42) must be set to 3,000 (3 m/min).

Analog output corresponding to F 1000 mm/min:

Analog voltage = (F x 9.5V) / "G00FEED"
Analog voltage = (1000 mm/min x 9.5V) / 15000 mm/min = 0.633V
Analog voltage = 633mV

Therefore, "PROGAIN" (P23) = 633

Bear in mind When setting the proportional gain that:

- The maximum amount of following error allowed by the CNC for the axis is the value indicated by a.m.p. MAXFLWE1 (P21). When exceeded, the CNC issues the corresponding following error message.
- The amount of following error decreases as the gain increases, but it tends to make the system unstable.
- In practice, the great majority of machines show an excellent behavior with a unitary gain (gain of 1, as shown in the previous examples).

Warning:

Once the axes have been adjusted separately, the ones being interpolated together should be further adjusted so their following errors are as identical as possible.

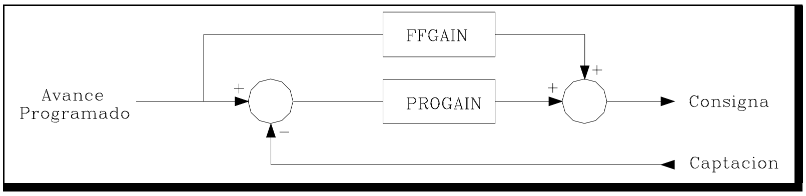
The more identical their following errors, the more "round" the programmed circles will turn out.

4.3.4 Feed-forward gain setting

With the Feed-Forward gain, it is possible to reduce the following error without increasing the gain, thus keeping the system stable.

It set the percentage of analog output due to the programmed feedrate. The rest depends on the proportional and Derivative/AC-forward gains.

This gain is only to be used when operating with acceleration / deceleration.



For example, if a.m.p. FFGAIN (P25) has been set to "80", the axis analog voltage will be:

- 80% of it will depend on the programmed feedrate (feed-forward gain)
- 20% of it will depend on the axis following error (proportional gain)

Setting the Feed-Forward gain involves a critical adjustment of a.m.p. MAXVOLT (P37).

1. Move the axis in G00 and at 10%.
2. Measure the actual analog voltage at the drive.
3. Set parameter MAXVOLT (P37) to a value 10 times the measured value.

   For example, If the measured voltage was 0,945V, then set this parameter to 9.45V, in other words: P37=9450.

Next, set a.m.p. FFGAIN (P25) to the desired value.

As an example, the following values may be used:

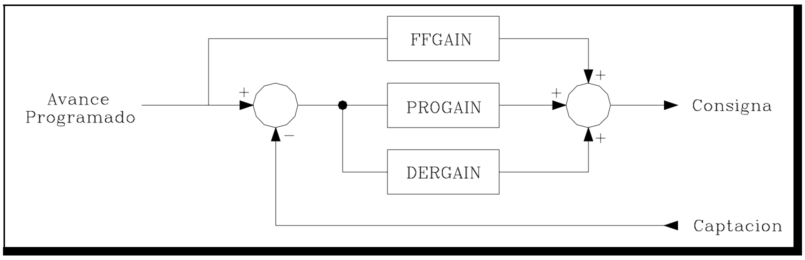
For slow machining
between 40 and 60%
For regular feed machining
between 60 and 80%
For fast machining (laser, plasma)
between 80 and 100%

4.3.5 Derivative / AC-forward setting

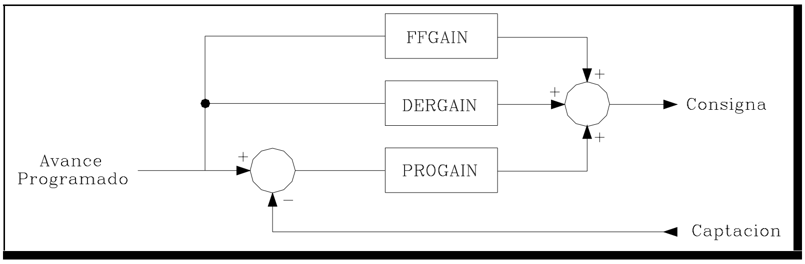
With the Derivative gain, it is possible to reduce the following error during the acc./dec. stages.

Its value is given by a.m.p. DERGAIN (P24).

When this additional analog voltage is due to fluctuations of following error, "ACFGAIN" (P46) = NO, it is called: "Derivative Gain".



When it is due to variations of the programmed feedrate, "ACFGAIN" (P46) = YES, it is called AC-forward Gain" since it is due to acc./dec.



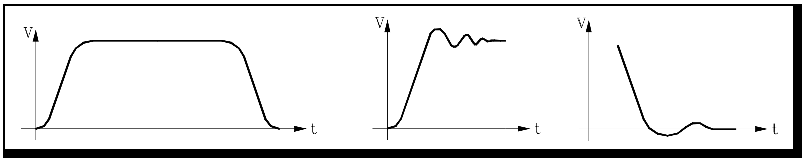
Best results are usually obtained when using it as AC-forward Gain, "ACFGAIN" (P46) = YES together with Feed-Forward Gain.

This gain is only to be used when operating with acceleration / deceleration.

A practical value between 2 to 3 times the Proportional Gain, "PROGAIN" (P23), may be used.

To perform a critical adjustment, proceed as follows:

- Verify that there is no oscillations on following error, In other words, that it is not unstable.
- Check, with an oscilloscope, the tacho voltage or the analog voltage at the drive (velocity command), verify that it is stable (left graph) and that there are neither instabilities when starting up (center graph) nor when braking down (right graph).



4.3.6 Leadscrew backlash compensation.

On this CNC, the leadscrew backlash may be compensated for when reversing the direction of movement.

Leadscrew backlash is defined with a.m.p. BACKLASH (P14).

Sometimes, an additional analog pulse may also be needed to recover the possible backlash when reversing the axis movement.

a.m.p. BACKNOUT (P29) sets the value of the additional analog voltage and a.m.p. BACKTIME (P30) indicates the duration of this additional analog pulse.

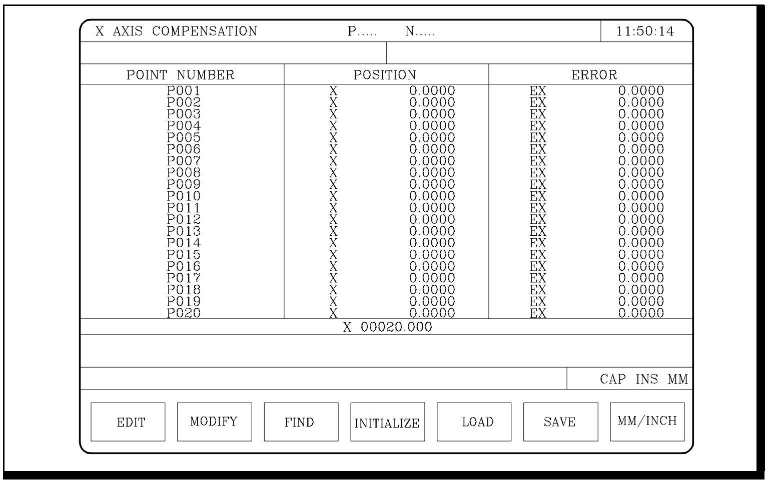
4.3.7 Leadscrew error compensation

The CNC provides a table for each one of the axes requiring leadscrew compensation.

This type of compensation is activated by setting a.m.p. LSCRWCOM P15)=ON or those axes.

The number of elements of the table is determined by the g.m.p. NPOINTS (P16), being possible to define up to 255 points per axis.

Each table parameter represents one leadscrew point to be compensated.



Each parameter of the table represents a profile point to be compensated defining for each one:

The axis position of that profile point with respect to Machine Reference ZERO. Possible values:

±99999.9999 mm or ±3937.00787 inches.

The leadscrew error in this point. Possible values:

±99999.9999 mm or ±3937.00787 inches.

When defining the profile points in the table, the following requirements must be met:

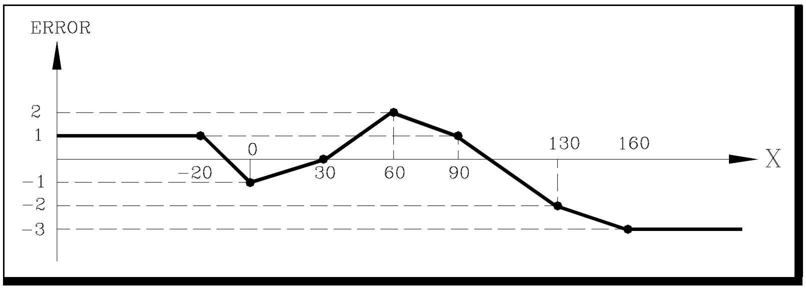
- The axis points must be in sequential order starting from the most negative (least positive) point to be compensated.
- For those points outside the compensation zone, the CNC will apply the compensation value corresponding to the table point closest to them.
- The machine reference point must have no error (zero).
- The error difference between two consecutive points must not be greater than the distance between them (maximum slope= 100%).

On rotary axes, although the display is limited between 0 and 360º, the internal count is accumulative. When using leadscrew error compensation, set positions 0° and 360°, first and last point of the table, with the same amount of error. This way, the CNC will apply the same compensation in all the revolutions.

Otherwise, the compensation will be limited to the indicated field.

Programming example:

The X axis ballscrew must be compensated for between X-20 and X160 according to the leadscrew error graph below:



Set a.m.p. LSCRWCOM (P15) = ON and NPOINTS (P16) = 7

Considering that the Machine Reference Point (physical location of the marker pulse) is located 30 mm from HOME (Machine Reference Zero), at X30. The leadscrew error compensation parameters must be set as follows:

P001	X	-20,000	EX	0,001
P002	X	0,000	EX	-0,001
P003	X	30,000	EX	0,000
P004	X	60,000	EX	0,002
P005	X	90,000	EX	0,001
P006	X	130,000	EX	-0,002
P007	X	160,000	EX	-0,003

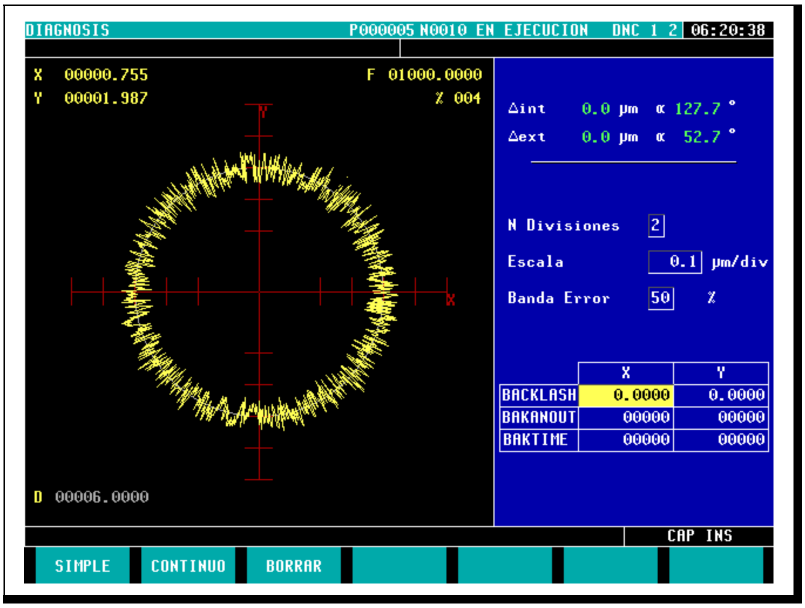
4.3.8 Circle geometry test

This adjustment improves the axis reversal peaks. It consists in machining a circle (without compensation) and verifying it on the graph displayed at the CNC.

The following example shows a program that machines repetitive circles.

```
        X0 Y0
        G5 G1 F1000
N10     G2 X0 Y0 I10 J0
        (RPT N10, N10) N50
        M30
```

After selecting this program in the Execution mode and starting it, access the "Diagnosis, Adjustments, Circle Geometry test" mode and the CNC will display the following screen:



If the machine parameters are protected, it will request the access password because the bottom right of the screen shows some of them. When not knowing the password, those values cannot be changed, but it will be possible to access the screen and the circle geometry test.

The left side of the CNC screen shows the result of the test.

The data at the top right is refreshed by the CNC after the test is completed.

The data at the center right must be defined before running the test

The bottom right side of the screen shows the parameters associated with the plane axes and the values used to set them.

The graph on the left must be defined before running the test. To do that, define the data of the center right:

- Number of divisions to the left and to the right of the theoretical circle.
- Scale or value in microns of each division.
- Error margin or % of circle radius occupied by the error margin (divisions area).

Knowing the password, the values shown at the bottom right may be modified. The assigns the new values to the relevant machine parameters; therefore, it is recommended to jot the initial values down.

One the graphic display area and the machine parameters have been defined, capture the data by pressing the following softkeys:

SINGLE It deletes the current graph and draws, over the theoretical circle, the machining error enlarged according to the defined scale until a full circle is drawn or until the STOP softkey or ESC key is pressed.

CONTINUOUS It deletes the current graph and draws, over the theoretical circle, a series of circles with the machining error enlarged according to the defined scale until the STOP softkey or ESC key is pressed.

DELETE It may be pressed at any time, even while drawing the graph. It deletes the screen and resets the statistics shown at its right.

During continuous graphics, it is possible to modify the machine parameters and view the new graph over the previous one, or press the DELETE softkey to only display the new one.

The data shown by the CNC at the top right is updated while capturing data.

Inside ∆	Maximum negative value of the error over the theoretical value, in microns or ten-thousandths of an inch and its angular position.
Outside∆	Maximum positive value of the error over the theoretical value, in microns or ten-thousandths of an inch and its angular position.

Once data capture is done, it draws two lines indicating the angular positions of both errors on the graph. They appear in dashed lines when the error exceeds the value assigned to the display area in its quadrant and it goes on to the opposite quadrant.

Note: While capturing points for the geometry test, the execution graphics stops drawing.

4.4 Reference systems

A CNC machine needs the following origin and reference points defined :

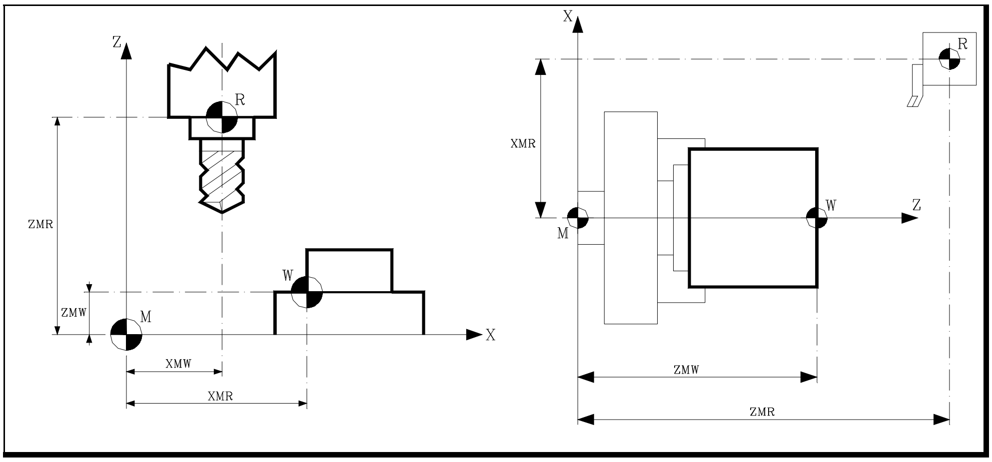
Machine zero Or machine's origin point. This is set by the manufacturer as the origin of the system of coordinates of the machine.

Part zero Or part's origin point. This is the point of origin which is set for programming the measurements of the part. It can be freely selected by the programmer, and its zero machine reference can be set by the zero offset.

Reference point This is a point on the machine established by the manufacturer (physical location of the marker pulse from the feedback device).

When the feedback system is semi-absolute (with coded marker pulse, Io), this point is only used when leadscrew error compensation must be applied onto the axis. The amount of leadscrew error to be assigned to this point is "0".

When the feedback is a regular incremental system (without coded marker pulse, Io), besides using this point in the leadscrew error compensation, the system is synchronized at this point instead of having to move the axis all the way to the Machine Reference Zero (home).



M	Machine zero
W	Part zero
R	Machine reference point
XMW, YMW, ZMW, etc	Coordinates of part zero
XMR, YMR, ZMR, etc	Coordinates of machine reference point

4.4.1 Machine reference (home) search

With this CNC, home search may be performed in jog mode or by program. Home search may be carried out on one axis at a time or on several axes at the same time.

On axes with no distance-coded feedback system.

The CNC will move all selected axes which have a home switch and in the direction indicated by a.m.p. REFDIREC (P33) for each axis.

This movement will be carried out at the feedrate established by a.m.p. REFEED1 (P34) for each axis until the home switch is hit.

Once all the axes have reached their respective home switches, the machine reference search (marker pulse) will be performed moving the selected axes one by one and in the selected sequence.

This second movement will be carried out at the feedrate established by a.m.p. REFEED2 (P35) for each axis until the marker pulse is found.

On axes with distance-coded feedback system:

Home switches are no longer necessary since the axes may be homed anywhere along its travel. However, a.m.p. REFVALUE (P36) must be set when operating with leadscrew error compensation. The amount of leadscrew error to be assigned to this point is "0".

The home search will be performed on one axis at a time and in the selected sequence.

The axes will move a maximum of 20 mm or 100 mm in the direction set by a.m.p. REFDIREC (P33) at the feedrate set by a.m.p. REFEED2 (P35) for each axis until the marker pulse is found.

If, during the home search, the home switch is pressed (if any), the CNC will reverse the homing direction.

When this search (with or without distance-coded Io) is carried out in JOG mode, the active zero offset will be cancelled and the CNC will display the position values indicated by a.m.p. REFVALUE (P36).

In all other cases, the active zero offset will be maintained and the CNC will display the position value with respect to the zero offset (or part zero) active before the home search.

Warning:

If after the machine is all set up it is necessary to remove the feedback system, it may happen that when it is reinstalled, its marker pulse is no longer at the same physical location as it was before.

In that case, the distance (shift) between the previous marker pulse location and the current one must be assigned to a.m.p. REFSHIFT (P47) of the affected axis in order for the machine reference point (home) to remain the same.

This way, when searching home, the axis will move this additional distance, indicated by a.m.p. REFSHIFT (P47) value, after finding the new marker pulse. This movement is carried out at the feedrate indicated by a.m.p. REFEED2 (P35).

GANTRY axes

Home search on Gantry axes may be carried out in JOG mode or by program. It will be carried out as follows:

On axes with no distance-coded feedback system.

The CNC starts the movements of both axes in the direction indicated by a.m.p. REFDIREC (P33) of the main axis.

These movements will be performed at the feedrate indicated by a.m.p. REFEED1 (P34) for the main axis until the home switch for this axis is hit.

Then, the home search will start on both axis at the feedrate indicated by a.m.p. REFEED2 (P35) of the main axis.

The CNC will wait until the marker pulse (home) of the slaved axis is found and then, it will look for the marker pulse from the main axis.

On axes with distance-coded feedback system:

The CNC starts moving both axes in the direction indicated by a.m.p. REFDIREC (P33) for the main axis at the feedrate indicated by a.m.p. REFEED2 (P35) of the main axis.

The CNC will wait until the marker pulse (home) of the slaved axis is found and then, it will look for the marker pulse from the main axis.

If the difference obtained between both reference positions is not the same as the one indicated by a.m.p. REFVALUE (P36) for both axes, the CNC will correct the position of the slaved axis. This will end the home search operation.

When this search is carried out in the JOG mode, the active zero offset will be cancelled and the CNC will display the position value indicated by a.m.p. REFVALUE (P36) for the main axis. In all other cases, the displayed position value will be referred to the zero offset (or part zero) active before the home search.

Warning:

If the a.m.p. REFDIREC (P33) of the main axis has been set for a positive direction, the a.m.p. REFVALUE (P36) of the slaved axis must be set to a value lower than that assigned to the main axis.

Likewise, if the a.m.p. REFDIREC (P33) of the main axis has been set for a negative direction, the a.m.p. REFVALUE (P36) of the slaved axis must be set to a value greater than that assigned to the main axis. They must never have the same value.

When encoders are used for feedback, the difference between the values assigned to a.m.p. REFVALUE (P36) of both axes must be smaller than the pitch of the ballscrew.

It is recommended that the distance between the marker pulses of both encoders be half the leadscrew pitch.

4.4.2 Setting on systems without distance-coded feedback

Machine reference point

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

- Indicate in the a.m.p. REFPULSE (P32) the type of marker pulse Io being used for Home Search.
- Likewise, set a.m.p. REFDIREC (P33) to indicate the direction of the axis when searching Home.
- On the other hand, set a.m.p. REFEED1 (P34) that defines the approach feedrate of the axis until the home switch is pressed and a.m.p. REFEED2 (P35) that indicates the homing feedrate until the reference mark (marker pulse) is detected.
- The machine reference point will be set to "0". a.m.p. REFVALUE (P36).
- Once in the JOG mode and after positioning the axis in the right area, start homing the axis. When done, the CNC will assign a "0" value to this point.
- After moving the axis to the Machine Reference Zero or up to a known position (with respect to Machine Reference Zero), observe the position reading of the CNC for that point.

  This will be distance from the Machine Reference Zero to that point. Therefore, the value to be assigned to a.m.p. REFVALUE (P36), which defines the coordinate corresponding to the Machine Reference Point (physical location of the marker pulse).

  REFVALUE = Machine coordinate – CNC reading.

  Example:

  If the point whose known position is located 230 mm from Machine Reference Zero and the CNC reads -123.5 mm as the coordinate value for this point, the coordinate of the Machine Reference Point with respect to Machine Reference Zero will be:

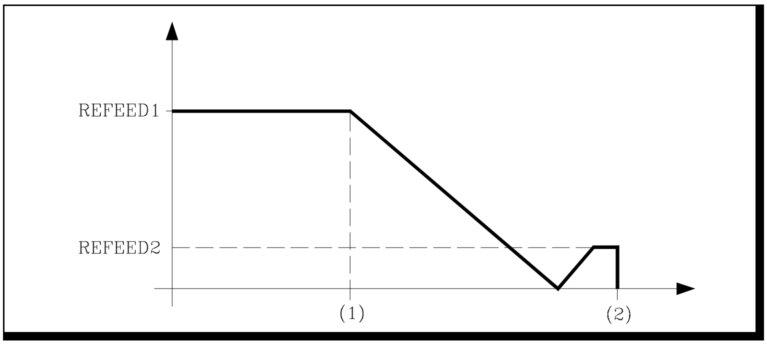
  "REFVALUE" = 230 - (-123.5) = 353.5 mm.

- After allocating this new value, press SHIFT + RESET or turn the CNC off and back on in order for the CNC to assume this new value.
- The axis must be homed again in order for it to assume its right reference values.

Considerations If at the time when the home search is requested, the axis is sitting on the home switch, the axis will back up (in the direction opposite to the one indicated by “REFDIREC (P33) ”) until it is off the switch and then, it will go on to searching home.

If the axis is positioned beyond the software limits "LIMIT+" (P5) and "LIMIT-" (P6), it must be brought back into the work area (within those limits) and on the proper side for referencing (home searching).

Care must be taken when placing the home switch and when setting feedrates “REFEED1 (P34)” and “REFEED2 (P35)”.



The home switch (1) will be installed so the marker pulse (2) will be found in the zone corresponding to feedrate “REFEED2” (P35).

If there is no room for it, reduce the value of “REFEED1 (P34)”. For example, for encoders whose consecutive marker pulses are very close to each other.

When the selected axis does not have a machine reference (home) switch (a.m.p. DECINPUT (P31) = NO), the CNC will move the spindle at the feedrate set by a.m.p. REFEED2 (P35) until the first marker pulse from the current position is found, thus ending the home search.

FAGOR linear encoders (scales) provide a negative marker (reference) pulse Io every 50mm (about 2 inches) and the FAGOR rotary encoders provide one positive reference pulse per revolution.

Do not mistake the type of pulse provided by the feedback system with the value to be assigned to a.m.p. REFPULSE (P32).

This parameter must indicate the type of active flank (leading or trailing edge), positive or negative of the reference mark (Io) used by the CNC.

4.4.3 Setting on systems with distance-coded feedback

Offset adjustment

The offset of the linear encoder must be adjusted on one axis at a time, preferably, following this procedure:

1. Set the following a.m.p:

   REFDIREC (P33) Homing direction.

   "REFEED2" (P35) Homing feedrate.

2. Verify that the value allocated to a.m.p. REFPULSE (P32) (type of marker pulse of the feedback system) is correct.

   To do this, set a.m.p. DECINPUT (P31) = NO and a.m.p. I0TYPE (P52) = 0 Then perform a home search.

   If assumed immediately, change a.m.p. REFPULSE (P32) and check again.

3. Set a.m.p. I0TYPE (P52) = 1 and ABSOFF (P53) = 0.

4. Once in JOG mode and after positioning the axis in the proper area, home the axis. The new position value displayed by the CNC is the distance from the current point to the origin of the linear encoder.

5. Perform several consecutive home searches and observe the CNC display during the whole process.

   The counting must be continuous. If it is not, if jerky, set a.m.p. I0TYPE (P52) = 2 and repeat steps 4 and 5.

6. Move the axis up to the Machine Reference Zero or up to a point whose position with respect to Machine Reference Zero is already known and observe the position value displayed by the CNC. This value is the distance from the current point to the origin of the linear encoder.

7. The value to be assigned to a.m.p. ABSOFF (P53) must be calculated with the following formula:

   ABSOFF (P53) = CNC reading - Machine coordinate.

   Example:

   If the point whose position is already known is located 230 mm from Machine Reference Zero and the CNC shows -423.5 mm as the position for this point, the linear encoder offset will be:

   ABSOFF (P53) = -423,5 - 230 = -653.5 mm.

8. After allocating this new value, press SHIFT + RESET or turn the CNC off and back on in order for the CNC to assume this new value.

9. Home the axis again in order for it to assume the new correct reference values.

Considerations

If the axis is positioned beyond the software limits "LIMIT+" (P5) and "LIMIT-" (P6), it must be brought back into the work area (within those limits) and on the proper side for referencing (home searching).

When using distance-coded linear encoders, home switches are no longer necessary.

However, home switches may be used as travel limits during home search.

If while homing, the home switch is pressed, the axis will reverse its movement and it will keep searching home in the opposite direction.

Distance-coded Fagor linear encoders have negative coded marker pulses (Io).

Do not mistake the type of pulse provided by the feedback system with the value to be assigned to a.m.p. REFPULSE (P32).

This parameter must indicate the type of active flank (leading or trailing edge), positive or negative of the reference mark (Io) used by the CNC.

If while homing an axis, its corresponding DECEL* signal is set high, the axis will reverse movement and the home search will be carried out in the opposite direction.

4.4.4 Axis travel limits (software limits)

Once all the axes have been referenced, their software limits must be measured and set.

This operation must be carried out one axis at a time and it could be done as follows:

- Move the axis in the positive direction towards the end of the axis travel stopping at a safe distance from the mechanical end-of-travel stop.
- Assign the coordinate shown by the CNC for that point to a.m.p. LIMIT+ (P5).
- Repeat these steps in the negative direction assigning the resulting coordinate to a.m.p. LIMIT- (P6).
- Once both travel limits have been set for all the axes, press SHIFT + RESET or turn the CNC OFF and back ON in order for these new values to be assumed by the CNC.

4.5 Unidirectional approach

The FAGOR 8055 CNC provides a number of machine parameters to help improve the repetitiveness when positioning the axes in rapid (G00) by always approaching the end point in the same direction.

"UNIDIR" (P39)

Indicates the direction of unidirectional approach.

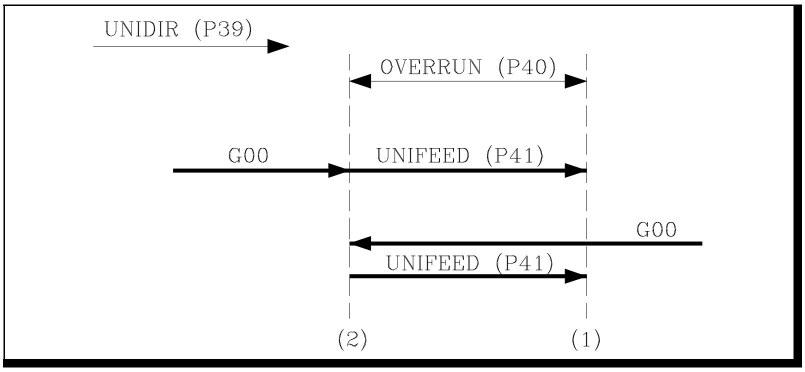
OVERRUN

Indicates the distance to be kept between the approach point and the programmed point. If this parameter is set to 0, the CNC will not perform the unidirectional approach.

"UNIFEED" (P41)

Indicates the feedrate to be used from the approach point to the programmed point.

The CNC will calculate the approach point (2) based on the programmed target point (1) and the a.m.p. UNIDIR (P39) and OVERRUN (P40).



The positioning will be carried out in two stages:

1. Rapid positioning (G00) up to the calculated approach point (2). If the axis is moving in the direction opposite to that indicated by "UNIDIR", it will overshoot the programmed point.
2. Positioning at feedrate UNIFEED (P41) from this point to the programmed point (1).

4.6 Auxiliary M, S, T function transfer

Every time a block is executed in the CNC, information is passed to the PLC about the M, S, and T functions which are active.

Auxiliary M function

The CNC uses logic outputs "MBCD1" thru "MBCD7" (R550 thru R556) to "tell" the PLC which M functions it must execute. One function per logic output.

It also activates the general logic output "MSTROBE" to "tell" the PLC to start executing them.

Every time the CNC detects an M function, it analyzes the M function table to find out when to pass it along to the PLC (either before or after the movement) and whether it must wait for the "AUXEND" signal or not before resuming program execution.

If the programmed function is not defined in that table, it will be executed at the beginning of the block and the CNC will wait for the "AUXEND" signal to resume program execution.

See sections ***8.1 Auxiliary M, S, T functions***, ***9.7 General logic outputs*** and ***3.8 Tables***

Example 1:

Execution of a motion block containing 7 M functions 4 of which are executed before the axes move (M51, M52, M53, M54) and 3 afterwards (M61, M62, M63).

1. It sends out to the PLC the 4 M functions programmed to be executed before the move

   It sets logic outputs “MBCD1=51”, “MBCD2=52” “MBCD3=53” “MBCD4=54” and it activates the general logic output "MSTROBE to “tell” the PLC to go ahead with their execution.

   Should any of them need the AUXEND activated, the CNC will “wait” for this signal to be activated before going on to executing the rest of the block.

   If none of them need the AUXEND signal activated, the CNC will maintain the “MSTROBE” signal activated for a period of time set by the general machine parameter “MINAENDW (P30)”. This output stays active for the time indicated by g.m.p. MINAENDW (P30).

2. The programmed axis move will be executed.

3. It sends out to the PLC the 3 M functions programmed to be executed after the move.

   It sets logic outputs “MBCD1=61”, “MBCD2=62”, “MBCD3=63” and it activates the general logic output "MSTROBE to “tell” the PLC to go ahead with their execution.

   Should any of them need the AUXEND activated, the CNC will “wait” for this signal to be activated before going on to executing the rest of the block.

   If none of them need the AUXEND signal activated, the CNC will maintain the “MSTROBE” signal activated for a period of time set by the general machine parameter “MINAENDW (P30)”. This output stays active for the time indicated by g.m.p. MINAENDW (P30).

Example 2:

Execution of a motion block containing 7 M functions 4 of which are executed before the axes move (M51, M52, M53, M54) and 3 afterwards (M61, M62, M63).

1. It sends out to the PLC the 4 M functions programmed to be executed before the move

   It sets logic outputs "MBCD1=51", "MBCD2=52" "MBCD3=53" "MBCD4=54" and it activates the general logic output "MSTROBE to "tell" the PLC to go ahead with their execution.

   Should any of them need the AUXEND activated, the CNC will "wait" for this signal to be activated before going on to executing the rest of the block.

   If none of them need the AUXEND signal activated, the CNC will maintain the "MSTROBE" signal activated for a period of time set by the general machine parameter "MINAENDW (P30)". This output stays active for the time indicated by g.m.p. MINAENDW (P30).

2. It sends out to the PLC the 3 M functions programmed to be executed after the move.

   It sets logic outputs "MBCD1=61", "MBCD2=62", "MBCD3=63" and it activates the general logic output "MSTROBE to "tell" the PLC to go ahead with their execution.

   Should any of them need the AUXEND activated, the CNC will "wait" for this signal to be activated before going on to executing the rest of the block.

   If none of them need the AUXEND signal activated, the CNC will maintain the "MSTROBE" signal activated for a period of time set by the general machine parameter "MINAENDW (P30)". This output stays active for the time indicated by g.m.p. MINAENDW (P30).

S function

The CNC transfers the "S function" out to the PLC only when using the BCD-coded "S" output. s.m.p. SPDLTYPE (P0) set to other than "0".

The CNC sends the programmed "S" value via logic output "SBCD" (R557) and activates the general logic output "SSTROBE" to indicate to the PLC to go ahead with its execution.

This transmission is carried out at the beginning of the block execution and the CNC will wait for the "AUXEND" general input to be activated and then consider its execution completed.

T function

The CNC will indicate via the variable "TBCD" (R558) the T function which has been programmed in the block and activates the general logic output "TSTROBE" to tell the PLC to go ahead with its execution.

This transmission is made at the beginning of the block execution and the CNC will wait for the general input "AUXEND" to be activated to consider the execution completed.

Second T function

The CNC transfers the "second T function" to the PLC in the following cases:

- When having a machining center with non-random tool magazine. g.m.p. TOFFM06 (P28) = YES and RANDOMTC (P25) = NO
- When using a random tool magazine, g.m.p. RANDOMTC (P25) = YES and a special tool change takes place. See chapter 6 of the Operating Manual: tool table, status.

On executing the M06 function, a the CNC indicates the position of the magazine (empty pocket) where the tool being in the spindle must be placed.

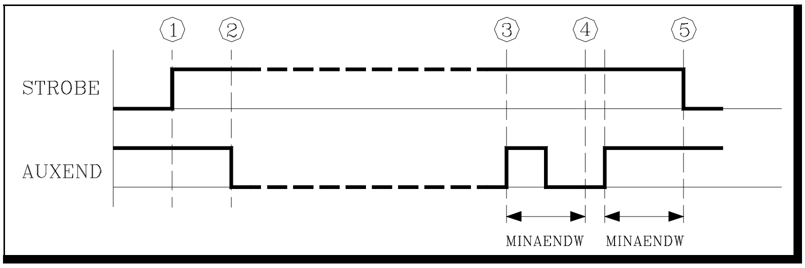
This indication will be made by means of the variable "T2BCD" (R559) and by activating the general logic output "T2STROBE" to tell the PLC that it must execute this. The CNC will wait for the general input AUXEND to be activated to consider the execution completed.

Warning:

It must be borne in mind that at the beginning of the execution of the block, the CNC can tell the PLC the execution of the M, S, T and T2 functions by activating their STROBE signals together and waiting for a single signal "AUXEND" for all of them.

4.6.1 Transferring M, S, T using the AUXEND signal

1. Once the block has been analyzed and after sending the corresponding values in the “MBCD1-7”, “SBCD”, “TBCD” and “T2BCD” variables, the CNC will tell the PLC by means of the general logic outputs “MSTROBE”, “SSTROBE”, “TSTROBE” and “T2STROBE” that the required auxiliary functions must be executed.



2. When the PLC detects the activation of one of the STROBE signals, it must deactivate the general CNC logic input “AUXEND” to tell the CNC that the execution of the corresponding function or functions has begun.
3. The PLC will execute all the auxiliary functions required, it being necessary to analyze the general CNC logic outputs:

   "MBCD1" through "MBCD7" and "MSTROBE"

      to execute the M functions.

   “SBCD” and “SSTROBE”

      to execute the S function

   “TBCD” and “TSTROBE”

      to execute the T function

   “T2BCD and “T2STROBE"

      to execute the second T function

   Once this has been executed the PLC must activate the general logic input “AUXEND” to indicate to the CNC that the processing of the required functions was completed.
4. Once the general input “AUXEND” is active, the CNC will require that this signal be kept active for a period of time greater than that defined by means of the g.m.p. MINAENDW (P30).

   This way, erroneous interpretations of this signal by the CNC due to an improper PLC program logic are avoided .
5. Once the period of time “MINAENDW (P30)” has elapsed with the general input “AUXEND” at a high logic level, the CNC will deactivate the general logic outputs “MSTROBE”, “SSTROBE”, “TSTROBE”, “T2STROBE” to tell the PLC that the execution of the required auxiliary function or functions has been completed.

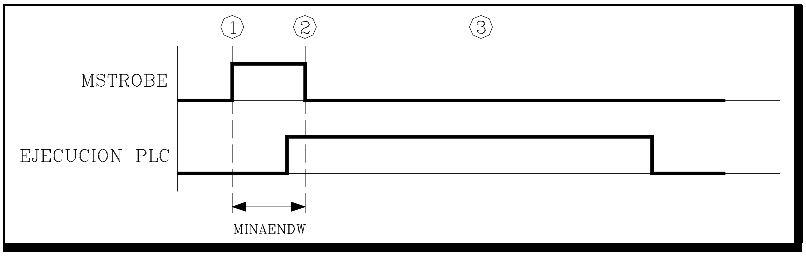
When executing 2 consective blocks which send information to the PLC and after finishing the execution of the first block, the CNC waits a MINAENDW period of time before starting to execute the second block.

This way, it assures that a MINAENDW delay takes place between the STROBE off (end of first block) and STROBE on (beginning of the second block).

It is advisable for the “MINAENDW (P30)” value to be equal to or greater than the duration of a PLC cycle, in order to ensure the detection of this signal by the PLC.

4.6.2 Transferring the auxiliary (miscellaneous) M functions without the AUXEND signal

1. Once the block has been analyzed and after passing the corresponding values in variables “MBCD1-7”, the CNC will tell the PLC through the general logic output “MSTROBE” that the required auxiliary function or functions must be executed.



2. The CNC will keep the general logic output “MSTROBE” active during the time indicated by means of g.m.p. MINAENDW (P30).

   Once this period of time has elapsed the CNC will continue to execute the program.

   It is advisable for the “MINAENDW (P30)” value to be equal to or greater than the duration of a PLC cycle, in order to ensure the detection of this signal by the PLC.

3. When the PLC detects the activation of the general logic signal “MSTROBE” it will execute the required auxiliary “M” functions at the CNC logic outputs “MBCD1 thru 7”.

4.7 Main and second spindle

This CNC can handle 2 spindles: a main spindle and a second spindle. They both can be operative simultaneously, but only one can be controlled at a time. This selection is made using functions G28 and G29. See programming manual.

Next, the steps to be followed when using two spindles are described.

Set-up

Set g.m.p. AXIS1 (P0) through AXIS8 (P7) with the desired values. A value of "10" for the Main Spindle and 14 for the Second Spindle.

Set the corresponding machine parameters for each spindle.

Spindle Selection

On power-up, the CNC always selects the main spindle.

All the keyboard actions and by spindle related functions affect the main spindle.

Example: S1000 M3

Main spindle clockwise at 1000 rpm

To select the second spindle, execute function G28.

From then on, All the keyboard actions and spindle related functions affect the second spindle.

The main spindle remains in its previous status.

Example: S1500 M4

Second spindle turns counterclockwise at 1500 rpm.

The main spindle keeps turning at 1000 rpm

To select the main spindle again, execute function G29.

From then on, all the keyboard actions and spindle related functions affect the main spindle.

The second spindle stays in its previous status.

Example: S2000

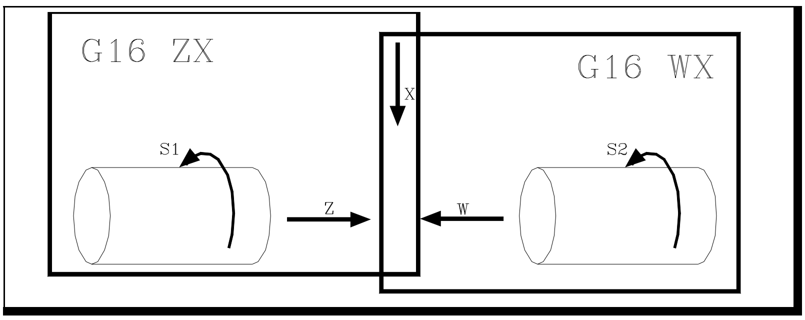
The main spindle keeps turning clockwise but at 2000 rpm.

The second spindle keeps turning at 1500 rpm.

Work plane Selection

Use function G16 to select the work plane. See programming manual.

Example:



Machining canned cycles

When working in a plane other than ZX, for example G16 WX, the CNC interprets the canned cycle parameters as follows:

Parameter Z and all those related to it, with the abscissa axis, W in the example.

Parameter X and all those related to it, with the ordinate axis, X in the example.

Tool Compensation

When working in a plane other than ZX, for example G16 WX, the CNC allows associating the tool offset table to the work plane.

To do that, set g.m.p. PLACOMP (P78) to "1". See section ***3.1 General parameters***

When g.m.p. PLACOM = 1, the CNC interprets the tool table as follows:

	ZX Plane	WX Plane
The Z and K parameters, with the abscissa axis	Z axis	W axis
The X and I parameters, with the ordinate axis	X axis	X axis

4.7.1 Spindle types

The setting of s.m.p. SPDLTYPE (P0) allows the following possibilities:

SPDLTYPE = 0 Analog spindle output..

SPDLTYPE = 1 2-digit BCD coded spindle output (S)

SPDLTYPE = 2 8-digit BCD coded spindle output (S)

When using BCD coded output, the spindle will operate in open loop and it will be controlled by means of functions M3, M4 and M5.

When using analog output, the spindle can operate:

- In open loop, controlled by means of functions M3, M4 and M5.
- In closed loop, by means of function M19. This requires an encoder mounted on the spindle and s.m.p. NPULSES (P13) must be set to a value other than "0".
- Controlled via PLC. With this feature, the PLC may take control of the spindle for a certain period of time.

  A typical application of this feature is the control of the spindle oscillation during spindle speed range (gear) change.

Regardless of the type of spindle output being used, the CNC admits up to 4 spindle speed ranges.

The spindle speed range change may be made either manually or automatically by the CNC.

To change spindle ranges, functions M41, M42, M43 and M44 are used to let the PLC know which one is to be selected.

4.7.2 Spindle speed (S) control

BCD output

When using BCD coded output, the spindle will operate in open loop and it will be controlled by means of functions M3, M4 and M5.

To do that, set s.m.p. SPDLTYPE (P0) with the right value.

SPDLTYPE = 1 2-digit BCD coded spindle output (S)

SPDLTYPE = 2 8-digit BCD coded spindle output (S)

Whenever a new spindle speed is selected, the CNC will transfer the programmed S value into register "SBCD" (R557) and it will activate general logic output "SSTROBE" (M5533) to "tell" the PLC to go ahead with its execution.

This transmission is carried out at the beginning of the block execution and the CNC will wait for the "AUXEND" general input to be activated and then consider its execution completed.

If it uses 2-bit BCD code, the CNC will indicate the S value to the PLC by means of this register and according to the following conversion table:

Programmed S	S BCD	Programmed S	S BCD	Programmed S	S BCD
0	S 00	50-55	S 54	800-899	S 78
1	S 20	56-62	S 55	900-999	S 79
2	S 26	63-70	S 56	1000-1119	S 80
3	S 29	71-79	S 57	1120-1249	S 81
4	S 32	80-89	S 58	1250-1399	S 82
5	S 34	90-99	S 59	1400-1599	S 83
6	S 35	100-111	S 60	1600-1799	S 84
7	S 36	112-124	S 61	1800-1999	S 85
8	S 38	125-139	S 62	2000-2239	S 86
9	S 39	140-159	S 63	2240-2499	S 87
10-11	S 40	160-179	S 64	2500-2799	S 88
12	S 41	180-199	S 65	2800-3149	S 89
13	S 42	200-223	S 66	3150-3549	S 90
14-15	S 43	224-249	S 67	3550-3999	S 91
16-17	S 44	250-279	S 68	4000-4499	S 92
18-19	S 45	280-314	S 69	4500-4999	S 93
20-22	S 46	315-354	S 70	5000-5599	S 94
23-24	S 47	355-399	S 71	5600-6299	S 95
25-27	S 48	400-449	S 72	6300-7099	S 96
28-31	S 49	450-499	S 73	7100-7999	S 97
32-35	S 50	500-559	S 74	8000-8999	S 98
36-39	S 51	560-629	S 75	9000-9999	S 99
40-44	S 52	630-709	S 76		
45-49	S 53	710-799	S 77		

If a value over 9999 is programmed the CNC will tell the PLC the spindle speed corresponding to value 9999.

If S output in 8-digit BCD is used the CNC will indicate the programmed spindle speed to the PLC by means of this register.

This value will be coded in BCD format (8 digits) in thousandths of a revolution per minute.

S 12345.678 0001 0010 0011 0100 0101 0110 0111 1000

Analog voltage

In order for the CNC to provide an analog output to control the spindle speed, it is necessary to set s.m.p. SPDLTYPE (P0) = 0.

The CNC will generate the analog output (within +10V.) corresponding to the programmed rotation speed or a unipolar analog output voltage if the s.m.p. POLARM3 (P7) and POLARM4 (P8) have been assigned the same value.

The Closed Loop mode of operation (with M19) is described later on in this manual.

PLC controlled spindle

With this feature, the PLC may take control of the spindle for a certain period of time.

To do that, follow these steps:

1. Have the PLC place the "S" value at CNC logic input "SANALOG" (R504). This "S" value corresponds to the analog voltage to be applied to the spindle drive.

   Also, set CNC logic input "PLCCNTL" (M5465) high to let the CNC know that from this moment on, the PLC is the one setting the analog voltage for the spindle.

2. From this instant on, the CNC outputs the spindle analog voltage indicated by the PLC at CNC logic input "SANALOG" (R504).

   If the PLC changes the value of the "SANALOG" input, the CNC will update the analog voltage accordingly.

3. Once the operation has concluded, the CNC must recover the control of the spindle back from the PLC. To do this, CNC logic input "PLCCNTL" (M5465) must be set low again.

A typical application of this feature is the control of the spindle oscillation during spindle speed range (gear) change.

4.7.3 Spindle speed range change

With this CNC, the machine can use a gear box for adjusting the best spindle speed and torque for the particular machining needs at any time.

The CNC admits up to 4 spindle speed ranges (gears) that are determined by s.m.p. “MAXGEAR1 (P2)”, “MAXGEAR2 (P3)”, MAXGEAR3 (P4)" and “MAXGEAR4 (P5)”. They indicate the maximum speed (in rpm) for each range.

The value assigned to “MAXGEAR1 (P2)” will be the one corresponding to the lowest range and the one assigned to “MAXGEAR4 (P5)” will be the one corresponding to the highest range.

When not using all 4 ranges (gears), use the lower parameters starting with MAXGEAR1 (P2). Set the unused gears with the same value as the highest of the ones used.

The auxiliary functions M41, M42, M43 and M44 are used to “tell” the PLC that spindle range 1, 2, 3 or 4 must be selected.

In turn, the PLC must “tell” the CNC the speed range being selected. This will be indicated by means of the logic inputs for the spindle: “GEAR1 (M5458)”, “GEAR2 (M5459)”, “GEAR3 (M5460)” and “GEAR4 (M5461)”.

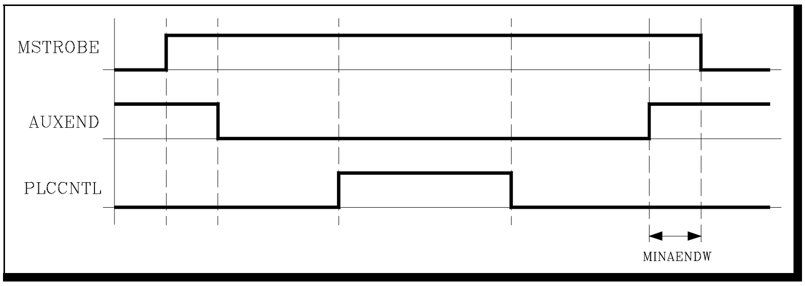
Since to each "S" speed corresponds a spindle range, before selecting a new "S" one must:

1. Analyze whether the new "S" involves a range change.
2. If it does, execute the M function corresponding to the new range (M41 thru M44) in order for the PLC to select it.
3. Wait for the PLC to select the new range. Check spindle logic inputs "GEAR1" (M5458), "GEAR2" (M5459), "GEAR3" (M5460) and "GEAR4" (M5461).
4. Select the new speed "S".

To have the CNC perform all these operations automatically, set s.m.p. AUTOGEAR (P6) =YES to indicate that the range change is to be generated by the CNC.

When selecting an automatic range change, the CNC will inform the PLC of the new range (M41 thru M44; but it will not execute any subroutine associated with them.

Automatic gear change controlled by the PLC



When the CNC detects a range change, it sends out to the PLC the corresponding M code (M41 thru M44) via one of the logic outputs "MBCD1-7" (R550 thru R556).

It also activates general logic output "MSTROBE" (M5532) to "tell" the PLC to go ahead with the execution.

The PLC deactivates CNC general logic input "AUXEND" (M5016) to indicate to the CNC that it began processing the "M" function.

When requiring spindle oscillation control during a range change, follow these steps:

1. Indicate, from the PLC, at CNC logic input "SANALOG" (R504) the value of the residual S voltage to be applied to the spindle drive.

   Also, set CNC logic input "PLCCNTL" (M5465) high to let the CNC know that from this moment on, the PLC is the one setting the analog voltage for the spindle.

2. From this instant on, the CNC outputs the spindle analog voltage indicated by the PLC at CNC logic input "SANALOG" (R504).

   If the PLC changes the value of the "SANALOG" input, the CNC will update the analog voltage accordingly.

3. Once the operation has concluded, the CNC must recover the control of the spindle back from the PLC. To do this, CNC logic input "PLCCNTL" (M5465) must be set low again.

Once the requested range change is completed, the PLC must set the corresponding CNC logic input "GEAR1" (M5458), "GEAR2" (M5459), "GEAR3" (M5460) or "GEAR4" (M5461) high.

Finally, the PLC will reactivate CNC general logic input "AUXEND" (M5016) indicating to the CNC that it has finished executing the auxiliary function.

Automatic gear change when working with M19

Every time M19 is programmed, it is recommended that the corresponding spindle range be selected.

If no range is already selected, the CNC proceeds as follows:

It converts the speed indicated in degrees per minute ats.m.p. REFEED1 (P34) into rpm.

It selects the speed range corresponding to those rpm.

The spindle range cannot be changed when operating in M19. It must be selected beforehand.

4.7.4 Spindle in closed loop

In order for the spindle to operate in closed loop by means of "spindle orientation (M19)", the following conditions must be met:

- The velocity command for the spindle must be analog (±10V). s.m.p. SPDLTYPE (P0) = 0.
- An encoder must be mounted onto the spindle. s.m.p. NPULSES (P13) must indicate the number of square pulses supplied by the spindle encoder.

Also, when switching from open to closed loop, either an "M19" or an "M19 S±5.5" must be executed.

The S±5.5 code indicates the spindle position, in degrees, from the spindle reference point (marker pulse).

When switching form open to closed loop, the CNC behaves as follows:

- If the spindle has a home switch, it performs a home-switch search at the turning speed set by s.m.p. REFEED1 (P34).

  It then searches for actual marker pulse (Io) of the spindle encoder at the turning speed set by s.m.p. REFEED2 (P35).

  And, finally, it positions the spindle at the programmed S±5.5 point.
- If the spindle does not have a home switch, it searches the encoder marker pulse at the turning speed set by s.m.p. REFEED2 (P35).

  And, then, it positions the spindle at the programmed S±5.5 point.

Calculating spindle resolution

The CNC assumes that one encoder revolution represents 360º. Therefore, the feedback (counting) resolution depends on the number of lines of the spindle encoder.

Resolution = 360° / (4 x number of encoder lines per revolution)

Hence, to obtain a resolution of 0.001º, a 90,000 line encoder is required and a 180,000 line encoder to obtain a resolution of 0.0005º.

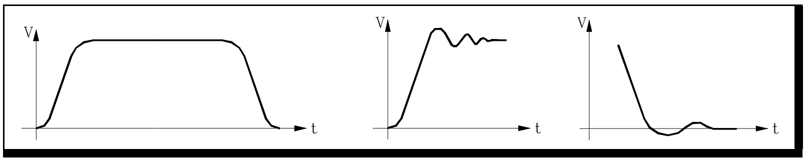
s.m.p. NPULSES (P13) must indicate the number of square pulses supplied by the spindle encoder.

In order to be able to use feedback alarm on the spindle encoder, "FBACKAL" (P15), the pulses provided by the encoder must be differential (double ended) squarewave "DIFFBACK (P14) = YES".

Gain setting

The various types of gains must be adjusted in order to optimize the system's performance for the programmed movements.

An oscilloscope is highly recommended to make this critical adjustment by monitoring the tacho signals. The illustration below shows the optimum shape for this signal (on the left) and the instabilities to be avoided during start-up and brake down:



There are three types of gain. They are adjusted by means of machine parameters and following the sequence indicated next.

Proportional Gain

It defines the analog output corresponding to a feedrate resulting in 1º of following error.

It is defined with s.m.p. PROGAIN (P23)

Feed Forward Gain

It sets the percentage of analog output dependent of the programmed feedrate.

To use it, acc/dec must be active, s.m.p. ACCTIME (P18)

It is defined with s.m.p. FFGAIN (P25)

Derivative Gain or AC- Forward Gain.

The "Derivative Gain" sets the percentage of analog output applied depending on the fluctuations of following error.

The "AC-Forward Gain" sets the percentage of analog output proportional to the feedrate increments (acceleration and deceleration stages).

To use it, acc/dec must be active, s.m.p. ACCTIME (P18)

It is defined with s.m.p. DERGAIN (P24) and ACFGAIN (P46).

If ACFGAIN = No	it applies Derivative Gain.
If ACFGAIN = Yes	it applies AC-Forward Gain.

Proportional gain setting

In a "pure" proportional positional loop, the analog output of the CNC to control the spindle is, at all times, proportional to the following error (axis lag) which is the difference between its theoretical and actual (real) position.

Analog output = Proportional Gain x Following Error

s.m.p. PROGRAIN (P23) sets the value of the Proportional Gain. Expressed in millivolts/degree, it takes any integer between 0 and 65535.

Its value indicates the analog output corresponding to a feedrate resulting in 1º of following error.

This value is taken for the first spindle gear and the CNC calculates the values for the rest of the gears.

Example: The maximum speed for the 1st range (rapid traverse G00) is 500 rpm and we would like to obtain 1º at a speed of 1000 º/min. (2.778 rpm)

Drive analog: 9.5V for 500 rpm

Analog output corresponding to S = 1000 º/min. (2.778 rpm)

Analog voltage = (F x 9.5V) / "G00FEED"

Analog voltage = (9.5 V / 500 rev/min) *2.778 rev/min =52.778 mV.

Therefore, “PROGAIN” = 53

Bear in mind When setting the proportional gain that:

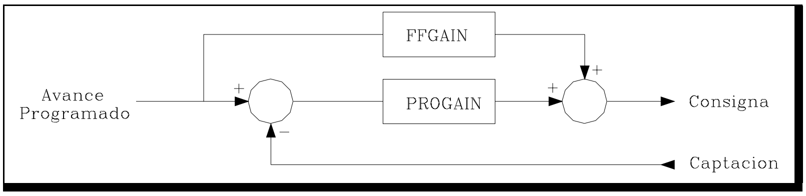
- The maximum amount of following error allowed by the CNC for the spindle is the value indicated by s.m.p. MAXFLWE1 (P21). When exceeded, the CNC issues the corresponding following error message.
- The amount of following error decreases as the gain increases, but it tends to make the system unstable.

Feed-forward gain setting.

With the Feed-Forward gain, it is possible to reduce the following error without increasing the gain, thus keeping the system stable.

It set the percentage of analog output due to the programmed feedrate. The rest depends on the proportional and Derivative/AC-forward gains.

This gain is only to be used when operating with acceleration / deceleration.



For example, if s.m.p. FFGAIN (P25) has been set to "80", the spindle analog voltage will be:

- 80% of it will depend on the programmed feedrate (feed-forward gain)
- 20% of it will depend on the spindle following error (proportional gain)

Setting the Feed-Forward gain involves a critical adjustment of s.m.p. MAXVOLT (P37).

1. Move the spindle in G00 and at 10%.
2. Measure the actual analog voltage at the drive.
3. Set parameter MAXVOLT (P37) to a value 10 times the measured value.
   For example, If the measured voltage was 0,945V, then set this parameter to 9.45V, in other words: P37=9450.

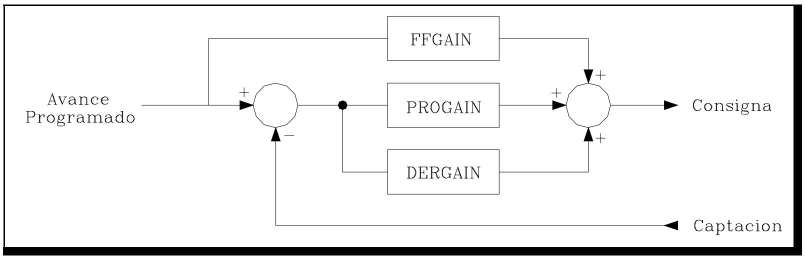
Next, set s.m.p. FFGAIN (P25) to the desired value.

Derivative / AC-forward setting

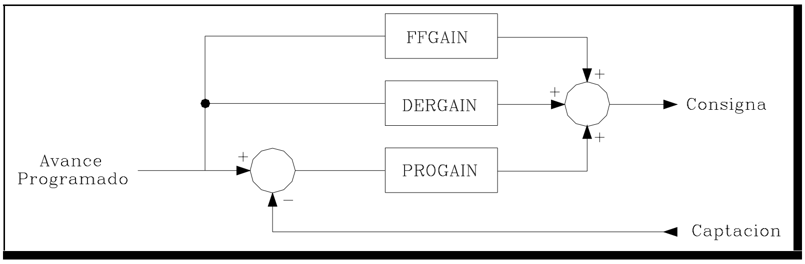
With the Derivative gain, it is possible to reduce the following error during the acc./dec. stages.

Its value is given by s.m.p. DERGAIN (P24).

When this additional analog voltage is due to fluctuations of following error, "ACFGAIN" (P46) = NO, it is called: "Derivative Gain".



When it is due to variations of the programmed feedrate, "ACFGAIN" (P42) = YES, it is called AC-forward Gain" since it is due to acc./dec.



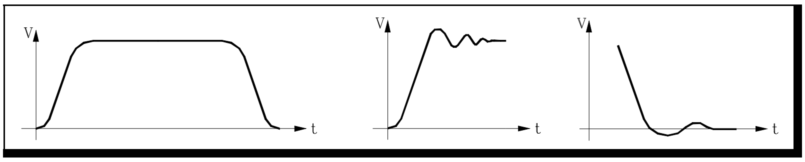
Best results are usually obtained when using it as AC-forward Gain, "ACFGAIN" (P42) = YES together with Feed-Forward Gain.

This gain is only to be used when operating with acceleration / deceleration.

A practical value between 2 to 3 times the Proportional Gain, "PROGAIN" (P23), may be used.

To perform a critical adjustment, proceed as follows:

- Verify that there is no oscillations on following error, In other words, that it is not unstable.
- Check, with an oscilloscope, the tacho voltage or the analog voltage at the drive (velocity command), verify that it is stable (left graph) and that there are neither instabilities when starting up (center graph) nor when braking down (right graph).



Machine reference point setting

To set the machine reference point proceed as follows:

- Indicate in the s.m.p. REFPULSE (P32) the type of marker pulse Io being used for Home Search.
- Likewise, set s.m.p. REFDIREC (P33) to indicate the direction of the axis when searching Home.
- On the other hand, set s.m.p. REFEED1 (P34) that defines the approach feedrate of the spindle until the home switch is pressed and s.m.p. REFEED2 (P35) that indicates the homing feedrate until the reference mark (marker pulse) is detected.
- The machine reference point will be set to "0". s.m.p. REFVALUE (P36).
- Once in the JOG mode and after positioning the spindle in the right area, start homing the spindle. When done, the CNC will assign a "0" value to this point.
- After moving the spindle to the Machine Reference Zero or up to a known position (with respect to Machine Reference Zero), observe the position reading of the CNC for that point.

  This will be distance from the Machine Reference Zero to that point. Therefore, the value to be assigned to s.m.p. REFVALUE (P36), which defines the coordinate corresponding to the Machine Reference Point (physical location of the marker pulse).

  REFVALUE (P36) = Machine coordinate – CNC reading.

  Example:

  If the point whose known position is located 230 mm from Machine Reference Zero and the CNC reads -123.5º as the coordinate value for this point, the coordinate of the Machine Reference Point with respect to Machine Reference Zero will be:

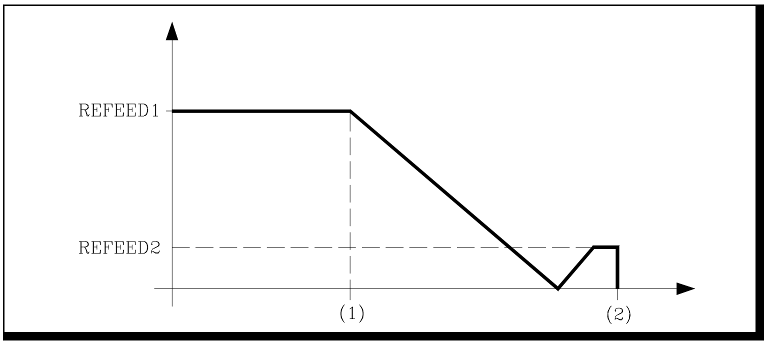
  "REFVALUE" P36 = 12 - (-123.5) = 135.5º

- After allocating this new value, press SHIFT + RESET or turn the CNC off and back on in order for the CNC to assume this new value.
- The spindle must be homed again in order for it to assume its right reference values.

Considerations

If at the time when the home search is requested, the spindle is sitting on the home switch, the spindle will back up (in the direction opposite to the one indicated by "REFDIREC (P33) ") until it is off the switch and then, it will go on to searching home.

Care must be taken when placing the home switch and when setting feedrates "REFEED1 (P34)" and "REFEED2 (P35)".



The home switch (1) will be installed so the marker pulse (2) will be found in the zone corresponding to feedrate "REFEED2" (P35).

If there is no room for it, reduce the value of "REFEED1 (P34)". For example, for encoders whose consecutive marker pulses are very close to each other.

When the spindle does not have a machine reference (home) switch (s.m.p. DECINPUT (P31) = NO), the CNC will move the spindle at the feedrate set by s.m.p. REFEED2 (P35) until the first marker pulse from the current position is found, thus ending the home search.

Fagor rotary encoders provide one positive reference pulse per revolution.

Do not mistake the type of pulse provided by the feedback system with the value to be assigned to s.m.p. REFPULSE (P32).

This parameter must indicate the type of active flank (leading or trailing edge), positive or negative of the reference mark (Io) used by the CNC.

4.8 Auxiliary spindle controlled by PLC

With this feature, the PLC can temporarily control the auxiliary spindle.

To do that, follow these steps:

1. Indicate from the PLC at the logic CNC input "SANALOAS" (R509) the amount of analog voltage to be applied to the drive for the auxiliary spindle.

   On the other hand, set logic CNC input "PLCCNTAS" (M5056) high to indicate to the CNC that from then on, it is going to be up to the PLC to control the analog voltage output for the auxiliary spindle.

2. From then on, the CNC outputs the analog voltage indicated by the PLC for the auxiliary spindle as indicated at the CNC logic input "SANALAS" (R509).

   If the PLC changes the value of the "SANALOAS" input, the CNC will update its analog voltage output.

3. Once the operation has concluded, the control of the auxiliary spindle must be returned to the CNC. To do that, the logic CNC input "PLCCNTAS" (M5056) must be set low.

4.9 Treatment of emergency signals

The CNC provides the following emergency signals:

/EMERGENCY STOP

Physical emergency input.

It is generated from the outside and corresponds to the physical emergency input.

This signal is active low (0 V).

/EMERGENCY OUTPUT

Physical emergency output.

It is generated internally when an error is detEcted at the CNC or at the PLC.

This signal is active low (0 V).

/EMERGEN (M5000)

Logic input of the CNC, generated by the PLC.

When the PLC activates this signal, the CNC stops the axes feed and the rotation of the spindle and it displays the corresponding error message.

This signal is active low (0 V).

/ALARM (M5507)

Logic input of the PLC, generated by the CNC.

The CNC activates this signal to let the PLC “know” that an alarm or emergency condition has occurred.

This signal is active low (0 V).

CNC Treatment of emergency signals

The emergency inputs of the CNC are:

/EMERGEN (M5000)

Physical input coming from the PLC.

/EMERGENCY STOP

Physical input from the outside, pin 10 of connector X2

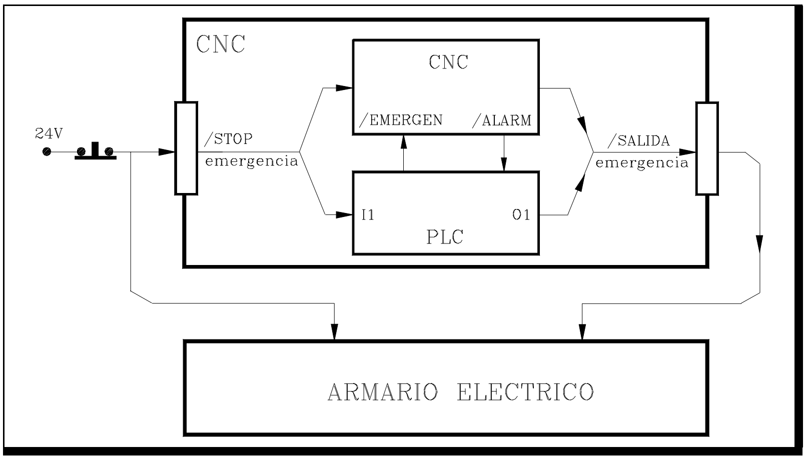
The emergency outputs of the CNC are:

/ALARM (M5507)

Physical output to the PLC.

/EMERGENCY OUTPUT

Physical output to the outside, pin 2 of connector X2.



There are to ways to cause an emergency at the CNC, by activating the physical input /EMERGENCY STOP or the general logic input "/ EMERGEN" from the PLC.

Whenever any of these signals is activated, the CNC stops the axes feed and the spindle rotation and it displays the corresponding error message.

By the same token, when the CNC detects an internal malfunction or at an external device, it stops the axes feed and the spindle rotation displaying at the same time the corresponding error message.

In both cases, the CNC will activate the /EMERGENCY OUTPUT and /ALARM signals to indicate to the PLC and to the outside world that an emergency has occurred at the CNC.

Once the cause of the emergency has disappeared, the CNC will deactivate these signals to indicate to the PLC and to the outside world that everything is back to normal.

PLC Treatment of emergency signals

The emergency inputs of the PLC are:

/EMERGENCY STOP

Physical input coming from the outside.

/ALARM (M5507)

Physical input coming from the CNC.

The emergency outputs of the PLC are:

/EMERGENCY OUTPUT

Physical output to the outside.

/EMERGEN (M5000)

Physical output to the CNC.

There are two ways to “tell” the PLC that an emergency condition must be treated, by activating the physical input EMERGENCY STOP of the PLC (which is I1) or the general logic input “/ALARM” of the PLC which is mark M5507.

In both cases, the treatment of these signals will be up to the PLC programmer. The PLC program must have the necessary instructions to properly attend to these emergency inputs and act accordingly.

By the same token, the PLC program must have the necessary instructions to properly activate the emergency outputs when required.

These emergency signals are the physical output /EMERGENCY OUTPUT (output O1 of the PLC) and the general logic output / EMERGEN” which is mark M5000 of the PLC.

It must be born in mind that every time a new PLC program cycle is initiated, the real inputs are updated with the physical inputs. Therefore, input I1 will have the value of the physical input / EMERGENCY STOP.

Also, before executing the PLC program cycle, the values of the M and R resources corresponding to the CNC logic outputs (internal variables) are updated as well as mark M5507 corresponding to the /ALARM signal.

After the execution of each cycle, the PLC updates the physical outputs with the values of the real outputs except the physical output /EMERGENCY OUTPUT which will be activated whenever the real output O1 or mark M5507 (/ALARM signal coming from the CNC) is active.

4.10 Sercos

g.m.p. SERSPEED (P120) and SERPOWSE (P121) allow setting the communications speed and power for Sercos.

4.10.1 Data exchange via Sercos

In order to be able to use these features, the drive's software version must be 3.1 or later.

The data exchange between the CNC and the drives is carried out at each position loop.

The more data to be transmitted, the more overloaded the Sercos transmission will be. It is recommended to limit these registers and to leave only the ones absolutely necessary after the setup.

On the other hand, there is data that MUST be transmitted at each position loop (velocity commands, feedback, etc.) and other information that could be transmitted in various loops (monitoring, etc.). Since the CNC must know the priority for those transmissions, from now on, we will use the mnemonics "Cyclic channel" and "Service Channel" to refer to each of them.

Cyclic channel: Data transmitted at each position loop (velocity commands, feedback, etc.)

The type of data to be transmitted must be indicated. The data to be sent to the drives must be placed in certain PLC registers and the data to be read from the drives is received in other PLC registers.

Service channel: Data to be transmitted in several position loops (monitoring, etc.).

The Service Channel can only be accessed through a high-level block of a part-program, a PLC channel or a user channel.

Cyclic channel. Read-only Sercos variables for the CNC-PLC:

The plc.m.p. SRR700 (P28) through SRR739 (P67) indicate which drive and what type of information will be placed in CNC registers R700 through R739.

P28=>R700   P29=>R701   P30=>R702   P31=>R703   etc.

The type of information available and its associated identifiers are:

	Sercos identifier
Class2Diagnostics (Warnings)	00012
Class3Diagnostics (OperationStatus)	00013
VelocityFeedback	00040
PositionFeedbackValue1	00051
TorqueFeedback	00084
CurrentFeedback	33079
FagorDiagnostics	33172
AnalogInputValue	33673
AuxiliaryAnalogInputValue	33674
DigitalInputsValues	33675
PowerFeedback	34468
PowerFeedbackPercentage	34469

The bits of identifier 33172 "FagorDiagnostics" contain the following information:

bits	Meaning	Id. Sercos at the drive
0,1,2,3	GV25 ActualGearRatio	000255
4,5,6,7	GV21 ActualParameterSet	000254
8	SV4	000330
9	SV5	000331
10	SV3	000332
11	TV10 TGreaterEqualTx	000333
12	TV60 PGreaterEqualPx	000337

The setting format for plc.m.p. SRR700 (P28) through SRR739 (P67) is 1.5

The units digit identifies the Sercos node number to get information from.

The decimal part indicates the Sercos identifier number.

Example: P32=1.00040

Indicates that PLC register R704 contains the "VelocityFeedback" supplied by the drive located in Sercos node 1.

Notes:

To identify the units of the variables, see the drive manual.

Read-only registers R700 through R739 are updated at the beginning of the PLC scan, unless the MRD instruction is used.

Cyclic channel. Write Sercos variables for the CNC-PLC:

plc.m.p. SWR800 (P68) through SWR819 (P87) indicate which type of information has been put in registers R800 through R819 and which drive will be assigned that value.

P68=>R800 P69=>R801 P70=>R802 P71=>R803 etc.

The type of information available and its associated identifiers are:

	Sercos identifier
DA1Value	34176
DA2Value	34177
DigitalOutputsValues	34178
VelocityCommand	00036

The "VelocityCommand" variable can be modified for the axes that have been selected as DRO axes, by a.m.p. DROAXIS (P4) or via PLC by activating the logic CNC axis input "DRO1,2,3,..."

The setting format for plc.m.p. SWR800 (P68) through SWR819 (P87) is 1.5

The units digit identifies the Sercos node number to send information to.

The decimal part indicates the Sercos identifier number.

Example: P70=2.34178

Indicates that the value of PLC register R802 will be assigned to "DigitalOutputsValues" of the drive located in Sercos node 2.

Note:

To identify the units of the variables, see the drive manual.

Service channel.

The Service Channel can only be accessed through a high-level block of a part-program, a PLC channel or a user channel. All variables can be accessed except the string type appearing in the Drive manual.

Reading and writing from a part-program or from a user channel:

Read: (P*** = SVARaxis ***)
Write: (SVARaxis** = P***)

Example: (P110=SVARX 40)

assigns to parameter P110 the Sercos value corresponding to the identifier 40 of the X axis which, in turn corresponds to "VelocityFeedback"

Reading and writing from the PLC channel:

Read: ... = CNCEX ((P*** = SVARaxis ***), M1)
Write: ... = CNCEX ((SVARaxis** = P***), M1)

Example.... = CNCEX ((SVARX 100=P120), M1)

assigns the value of parameter P120 to the sercos variable corresponding to the identifier 100 of the X axis which in turn corresponds to "VelocityLoopProportionalGain".

Service channel. Changing parameter sets and gear ratios

It is recommended to use this feature when the feedback is handled via Sercos "SERCOSLE=1 or 2"

The drive may have up to 8 gear ratios (0 through 7). Sercos identifier 218, GearRatioPreselection.

It may also have up to 8 parameter sets (0 through 7). Sercos identifier 217, ParameterSetPreselection.

To select these sets from the CNC, the new write variables must be used:

SETGEX, SETGY, SETGZ	for the axes
SETGES	for the main spindle
SSETGS	for the second spindle

The 4 least significant bits of these variables indicate the gear ratio and the other 4 the parameter set to be selected.

To send this information to the drive, a high-level block must be executed in a part-program, PLC channel or user channel as mentioned earlier.

It takes time to the drive to change the parameter set and the gear ratios. That is why a new PLC mark has been defined SERPLCAC (M5562). This mark will be active from when the change is requested until the drive assumes the new values.

No other SETGE* change may be requested while this mark is active because the command would be lost.

4.10.2 Drive's absolute feedback

If the drive has version V4.02 or later, absolute feedback is treated at the drive's first feedback.

The CNC checks the "RV5" variable of the drive (drive set with absolute encoder) and drive parameter PP177 (Absolute distance1) that indicates the distance between machine zero (home) and the encoder's absolute zero.

4.11 Axes (2) controlled by a single drive

To control 2 axes through a single servo drive:

- Set a.m.p. SWITCHAX (P65) and SWINBACK (P66)

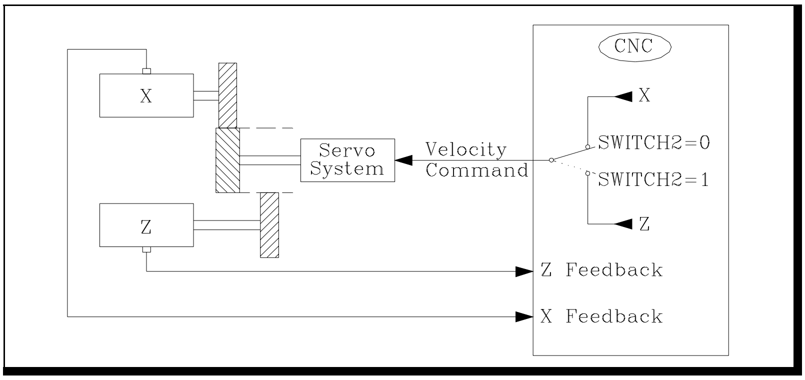
main axis	SWITCHAX = 0 and SWINBACK = 0
associated axis	SWITCHAX = code of the main axis SWINBACK = 1 it its own feedback SWINBACK = 0 if taken from the main one

- Act upon marks SWITCH1 through 7 corresponding to the secondary axis for selecting the axis to be governed. "0" for the main axis and "1" for the secondary.

When communication is via sercos, proceed as follows:

- The a.m.p. SERCOSID for both axes must be set with the same value (same Sercos address).
- To govern one of the axes, enable the SERVOON, SPENA and DRENA signals of that axis and activate the DRO signal of the other axis so it works as a DRO (not controlled).

X and Z paraxial axes (moving one at a time) and independent feedback.



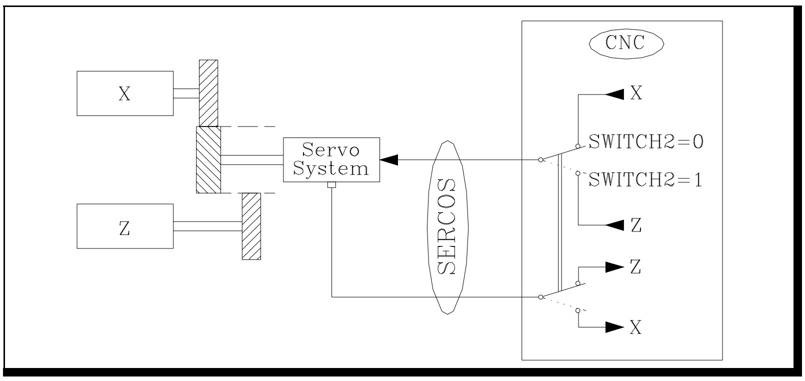
X axis (main)	Z axis (secondary)
SWITCHAX for X = 0	SWITCHAX for Z axis = 1 (X axis)
SWINBACK for X axis = 0	SWINBACK for Z axis = 1

The analog voltage is always output through the X axis connector (main).

The mark for the secondary axis is SWITCH2 (M5155)

With SWITCH2=0 , analog voltage of the X axis and with SWITCH2=1 that of the Z axis.

X and Z paraxial axes (moving one at a time) and communication via Sercos, feedback included.



X axis (main)	Z axis (secondary)
SWITCHAX for X = 0	SWITCHAX for Z axis = 1 (X axis)
SWINBACK for X axis = 0	SWINBACK for Z axis = 0

Using the mark for the secondary axis, SWITCH2 (M5155), one can select which axis the analog voltage and feedback data transmitted via SERCOS correspond to

With SWITCH2=0, analog voltage and feedback data for the X axis.

With SWITCH2=1, analog voltage and feedback data for the Z axis.

Cylindrical grinder (X and Z axes).

To make the reciprocating movement (back-and-forth table swing - Z axis) independent from the movement of the other axis (X), that movement should be controlled through the PLC execution channel.

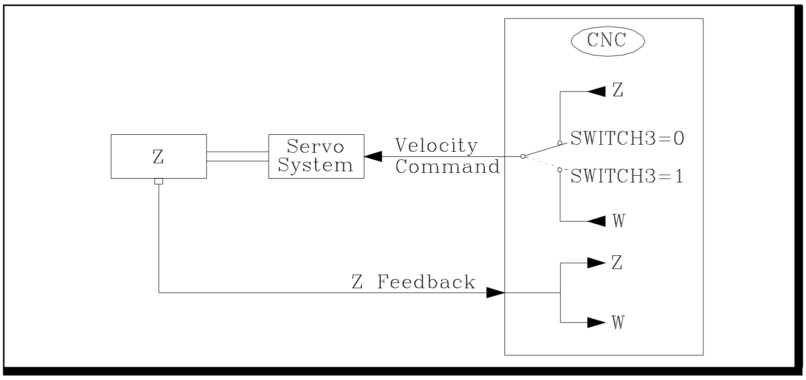
When a cycle controls both axes or to move the Z axis manually (jog or handwheel), the Z axis must be controlled by the CNC.

Since an axis cannot be controlled through 2 execution channels, the CNC must be "cheated" by calling the axis with two different names.

Z Main axis. Controlled by CNC

W Secondary axis. Controlled by PLC

Although both axes may be displayed, only the Z axis (main) will displayed in this example.



Z axis (main)	W axis (secondary)
	DFORMAT for W =0 (not displayed).
SWITCHAX for Z = 0	SWITCHAX for W = 3 (Z axis)
SWINBACK for Z axis = 0	SWINBACK for W = 0

Connect the Z axis feedback (main axis).

Since the two axes share the same feedback device, set the ungoverned axis as DRO so it does not trigger the following error alarm.

The velocity command is always output through the Z axis connector (main axis).

The mark for the secondary axis is SWITCH3 (M5205)

With SWITCH3=0 velocity command of the Z axis and with SWITCH3=1 that of the W axis.

PLC Program

The M40 mark indicates that there is no external emergency (I1) and that the positioning loop of the axes are closed (NOT LOPEN).

I1 AND NOT LOPEN = M40

An external switch (I12) turns the reciprocating movement off, PLC execution channel, and to switch to the main execution channel (M41=1).

To switch from the PLC execution channel to the CNC channel, the PLC channel must be interrupted (PLCABORT) and one must make sure that the axis has stopped (INPOS3)

I12 AND (other conditions) = SET PLCABORT = SET M44

M44 AND INPOS3 = M41

With CNC channel selected (M41=1)

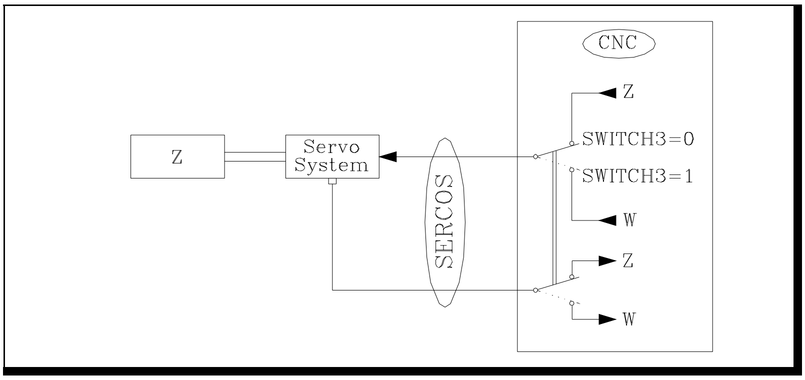
M40 AND M41	= DRO3	W axis as DRO
	= SERVO2ON	Z axis normal
	= RES SWITCH3	Z axis velocity command

With the PLC channel selected (M41=0)

M40 AND NOT M41	= DRO2	Z axis as DRO
	= SERVO3ON	W axis normal
	= SET SWITCH3	Z axis velocity command

Sercos

When using SERCOS communication with the servo drive, the axis being applied the velocity command and feedback data is selected by the mark of the secondary axis SWITCH3 (M5205).



a.m.p. SWINBACK (P66) of the secondary axis must be set to "0".

Z axis (main)	W axis (secondary)
	DFORMAT for W =0 (not displayed).
SWITCHAX for Z = 0	SWITCHAX for W = 3 (Z axis)
SWINBACK for Z axis = 0	SWINBACK for W = 0

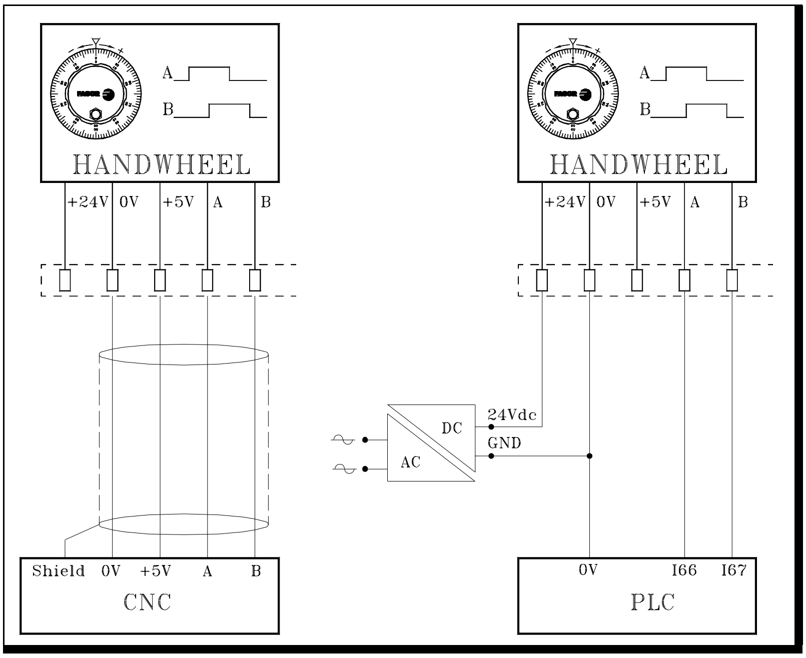
4.12 Fagor handwheels: HBA, HBE and LGB

Fagor handwheels HBA, HBE and LGB have:

- a pulse generator (encoder).
- an emergency output.
- One or two enable buttons.
- An axis selector switch.
- A resolution selector switch.

The encoder signals must be taken to the specific connectors of the CNC.

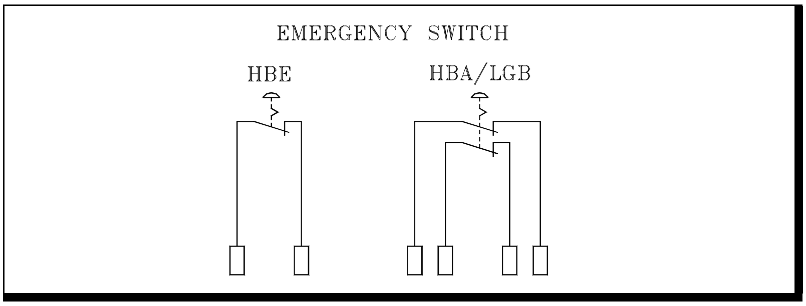
The signals from the 24V HBA and HBE models may also be taken to the PLC digital inputs located at the Central Unit (not remote).



In the example on the left, the handwheel signals are taken to the feedback input (connector). Set the corresponding g.m.p. AXIS , for example: AXIS4 (P3) = 11

In the example on the right, the handwheel signals are taken to the digital PLC input. Set g.m.p. HANDWIN (P111) = 65 and HANDWHE1 (P112) = 11.

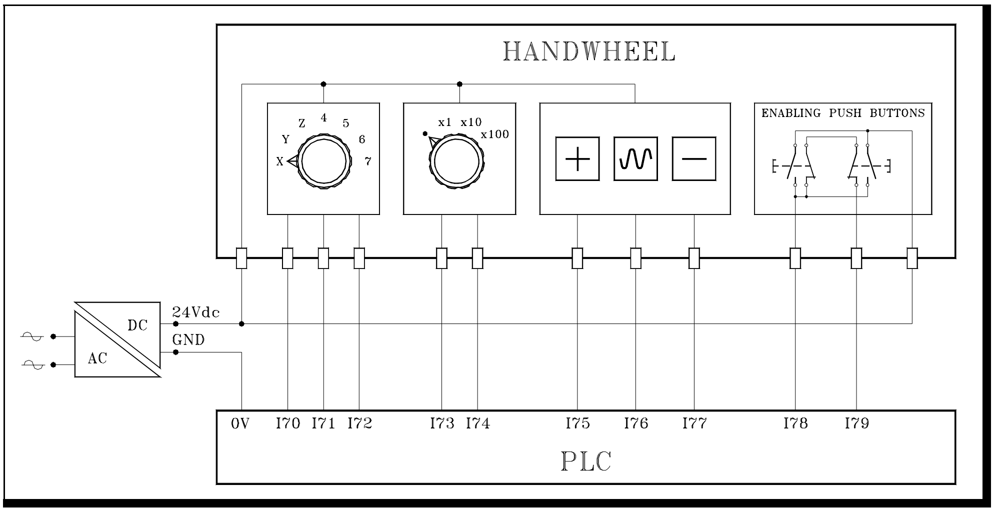
The emergency button must be used in the safety chain of the electrical cabinet.



The HBE handwheel has one contact and the HBA and LGB models have a dual safety contact.

The enable push button (or buttons), the axis selector and resolution selector switches are always handled by the PLC.

Connection example and PLC program for the HBA-072914 handwheel.



There are 2 ways to use the "Enable Push Button".

I78	Just press one of the buttons
I79	Both buttons must be pressed

The examples uses input I79, making it necessary to push both buttons in order to use the handwheel.

Definition of symbols (mnemonics)

```
DEF    HDWON    M600    Handwheel jog
DEF    JOGON    M601    JOG
DEF    XSEL     M602    X axis selected
DEF    YSEL     M603    Y axis selected
DEF    ZSEL     M604    Z  axis selected
DEF    4SEL     M605    4th axis selected
DEF    5SEL     M606    5th axis selected
DEF    6SEL     M607    6th axis selected
DEF    7SEL     M608    7th axis selected
PRG
REA
```

If the handwheel enable (I79) and the switch is at handwheel position (x1, x10 or x100)

I79 AND (I73 OR I74) = HDWON

	I73	I74
JOG	0	0
x1	0	1
x10	1	1
x100	1	0

To move the axes in JOG proceed as follows ....

... enable handwheel "I79" ...

... turn the switch to the (·) position: "NOT I73 AND NOT I74"

... position the CNC panel selector in the JOG area (not handwheel, not incremental) "SELECTOR > 7"

I79 AND NOT I73 AND NOT I74 AND CPS SELECTOR GE 8

= JOGON

Axis selection. Inputs I70, I71, I72

NOT I70 AND NOT I71 AND NOT I72 = XSEL
NOT I70 AND NOT I71 AND I72 = YSEL
NOT I70 AND I71 AND I72 = ZSEL
NOT I70 AND I71 AND NOT I72 = 4SEL
I70 AND I71 AND NOT I72 = 5SEL
I70 AND I71 AND I72 = 6SEL
I70 AND NOT I71 AND I72 = 7SEL

	I70	I71	I72
XSEL	0	0	0
YSEL	0	0	1
ZSEL	0	1	1
4SEL	0	1	0
5SEL	1	1	0
6SEL	1	1	1
7SEL	1	0	1

If handwheel jog (HDWON), R60 must be ready to store what will be written into the HBEVAR variable. The "a, b, c" bits indicate the x1, x10, x100 factor for each axis and bit 30 (*) must be set to "1" in order for the CNC to read the handwheel pulses.

					C			B			A			W			V			U			Z			Y			X		
	*	^			c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a

() = MOV 0 R60 Delete its contents

Sets the bit (a) of the selected axis to "1". x1 multiplying factor.

HDWON AND XSEL = MOV 1 R60
HDWON AND YSEL = MOV 8 R60
HDWON AND ZSEL = MOV R60 R61
HDWON AND 4SEL = MOV R60 R61
HDWON AND 5SEL = MOV R60 R61
HDWON AND 6SEL = MOV R60 R61
HDWON AND 7SEL = MOV R60 R61

It then analyzes the multiplying factor indicated at the switch (x1, x10, x100)

I73 AND I74 = RL1 R60 1 R60
I73 AND NOT I74 = RL1 R60 2 R60

	I73	I74	c	b	a
x1	0	1	0	0	1
x10	1	1	0	1	0
x100	1	0	1	0	0

And finally, the bit 30 (*) of HBEVAR=1 is enabled, for the CNC to read the handwheel pulses.

( )= OR R60 $40000000 R60

When enabling the handwheel or changing the position of one of the switches, HBEVAR and its image register (R61) are updated (refreshed).

DFU HDWON OR CPS R60 NE R61 = MOV R60 R61
= CNCWR(R61,HBEVAR,M201)

When disabling the handwheel, HBEVAR=0 and its image register (R61) are initialized.

DFD HDWON = MOV 0 R61 = CNCWR(R61,HBEVAR,M201)

If JOG movement (JOGON) and [+] key pressed: "I75", then axis movement in the positive direction

JOGON AND I75 AND XSEL = AXIS+1
JOGON AND I75 AND YSEL = AXIS+2
JOGON AND I75 AND ZSEL = AXIS+3
JOGON AND I75 AND 4SEL = AXIS+4
JOGON AND I75 AND 5SEL = AXIS+5
JOGON AND I75 AND 6SEL = AXIS+6
JOGON AND I75 AND 7SEL = AXIS+7

If JOG movement (JOGON) and [-] key pressed: "I77", then axis movement in the negative direction.

JOGON AND I77 AND XSEL = AXIS-1

JOGON AND I77 AND YSEL = AXIS-2

JOGON AND I77 AND ZSEL = AXIS-3

JOGON AND I77 AND 4SEL = AXIS-4

JOGON AND I77 AND 5SEL = AXIS-5

JOGON AND I77 AND 6SEL = AXIS-6

JOGON AND I77 AND 7SEL = AXIS-7

If JOG movement (JOGON) and [Rapid] key pressed: "I76", axis movement in rapid.

JOGON AND I76 = MANRAPID

Safety. When releasing the "Enable Push Button", the STOP command is sent out to the CNC (100 ms pulse) to stop the possible movement active at the time (for example: 10 mm in incremental). Only if the JOG mode is selected and NOT MDI

DFD I79 = TG1 17 100

MANUAL AND NOT MDI AND T17 = NOT /STOP

END

In order to comply with the EN-61000-4-4 (IEC 1000-4-4) regulation on “immunity against rapid transients and blasts” use a PVC cable 7x1x0.14 with the external shield connected at both ends for the feedback cable.

User notes:

8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

INTRODUCTION TO THE PLC

5

5. INTRODUCTION TO THE PLC

Warning:

It is recommended to save the PLC program and files into the "Memkey Card" (CARD A) or in a peripheral or PC to avoid losing them.

The PLC program (PLC_PRG) may be edited at the front panel or copied from the "Memkey Card" (CARD A) or from a peripheral device or PC.

The PLC program (PLC_PRG) is stored in the internal CNC memory with the part-programs and it is displayed in the program directory (utilities) together with the part-programs.

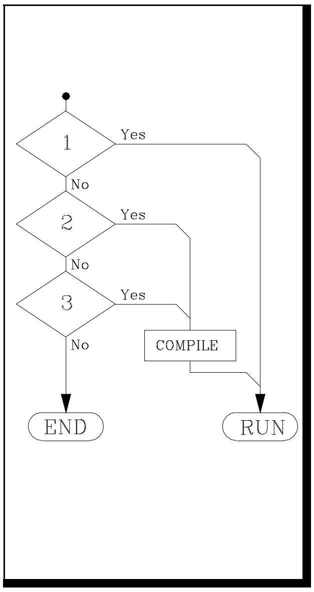
Before executing the PLC_PRG program, it must be compiled.

Once it is done compiling, the CNC requests whether the PLC should be started or not.

To make the operator life easier and avoid new compilations, the source code generated at each compilation is stored in memory.

After power-up, the CNC acts as follows:

1. If there is an executable program stored in memory, it executes it (RUN).
2. If there is no executable program, but there is a PLC_PRG in memory, it compiles it (COMPILE) and executes it (RUN).
3. If there is no PLC_PRG in memory, it looks for it in the "Memkey Card" (CARD A)
   - If it is there, it compiles it (COMPILE) and executes it (RUN).
   - If it is not there, it does nothing. Later on, when accessing the Jog mode, Execution mode, etc. the CNC will issue the corresponding error message.



Once the program has been compiled, it is not necessary to keep the source program (PLC_PRG) in memory because the PLC always executes the executable program.

The PLC has 512 inputs and 512 outputs. Some of them, depending on the CNC configuration, communicate with external devices.

There is an exchange of information between the CNC and the PLC which is done automatically and the system has a series of commands which allow the following to be done quickly and simply:

- The control of Logic CNC inputs and outputs by means of an exchange of information between both systems.
- The transfer from the CNC to the PLC of M, S and T auxiliary functions.

* To display a screen previously defined by the user, as well as generating messages and errors in the CNC.
* Reading and writing internal CNC variables from the PLC.
* Access to all PLC variables from any part program.
* Monitoring on the CNC screen of PLC variables.
* Access to all PLC variables from a computer, via DNC and by means of the RS 232 C and RS 422 serial ports.

5.1 PLC Resources

Inputs (I)

These are elements which supply information to the PLC from signals received from the outside world. They are represented by the letter I and there are 512 inputs available.

Outputs (O)

These are elements which allow the PLC to activate or deactivate the different devices in the electrical cabinet. These are represented by the letter O and there are 512 outputs available.

Marks (M)

These are elements capable of memorizing in one bit (as if it were an internal relay) the status of the different internal variables of the CNC (information of the logic outputs received in the communication between the CNC and the PLC of the CNC) and the status of the different variables of the PLC, whether these are internal or established by the user. They are represented by the letter M, and there are 2000 user marks and other special marks.

Registers (R)

These are elements which allow a numerical value to be stored in 32 bits or facilitate CNC-PLC communication with the Logic CNC inputs-outputs. They are represented by the letter R and there are 256 user registers and other special registers.

Timers (T)

These are elements which, once activated, alter the status of their output for a specific time (time constant). They are represented by the letter T, and there are 256 timers.

Counters (C)

These are elements capable of counting up or down a specific amount of events. They are represented by the letter C and there are 256 counters.

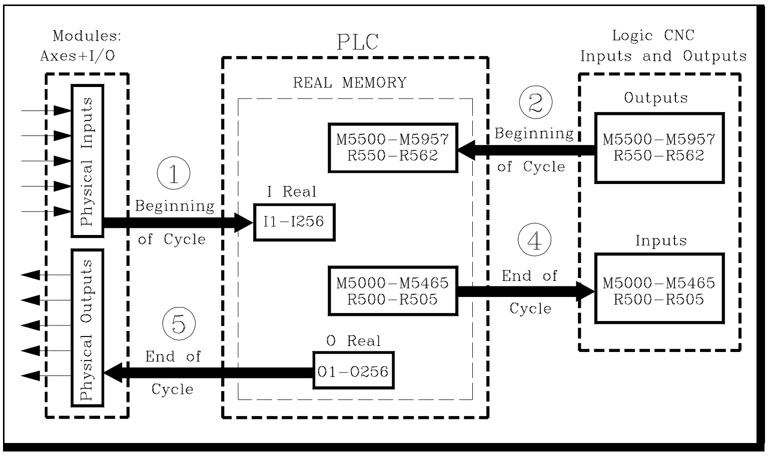
5.2 PLC program execution

The PLC executes the user program cyclically. In other words, once it executes the complete program, it restarts running this program from the first instruction.

This cyclic processing of the program is done as follows:

1. At the beginning of the cycle, PLC's "I" resources are assigned the current values of the physical inputs (connectors).

   For example, if the phisical input I10 is at 24V, the PLC sets the I10 resource to "1"



2. It allocates the current values of the logic CNC outputs (CNCREADY, START, FHOUT, ...) to PLC resources M5500 thru M5957 and R550 thru R562 .
3. It runs the program cycle.

   Section ***5.4 Modular structure of the program*** indicates how the PLC program is structured and which are its execution modules.
4. After executing the cycle, it updates the Logic CNC inputs (/EMERGEN, /STOP, /FEEDHOL, ...) with the current values of PLC resources M5000 thru M5465 and R500 thru R505.
5. It assigns the current values of the PLC's "O" resources to the physical outputs (connectors)

   For example, if the "O5" resource is at "1", the PLC sets physical output O5 (connector) to 24V.
6. It concludes this cycle scan and it gets ready for the next one.

Bear in mind that all the actions of the program executed by the PLC alter the status of its resources.

Example: I10 AND I20 = O5

When this condition is met [resource I10 is "1" and I20 is also "1"], the PLC sets resource "O5" to "1". If this condition is not met, the PLC sets resource "O5" to "0".

Therefore, the status of a resource may change during the execution of the PLC program.

Example, assuming that the initial status of resource M100 is "0":

M100 AND I7 = O3

Resource M100 = "0"

I10 = M100

M100 takes the value of resource I10

M100 AND I8 = M101

The value of M100 depends on the previous instruction.

This type of problems may be prevented by careful programming or by using "Image" resource values (instead of "Real" values).

The PLC has 2 memories to store the status of the various registers: Real and Image memory.

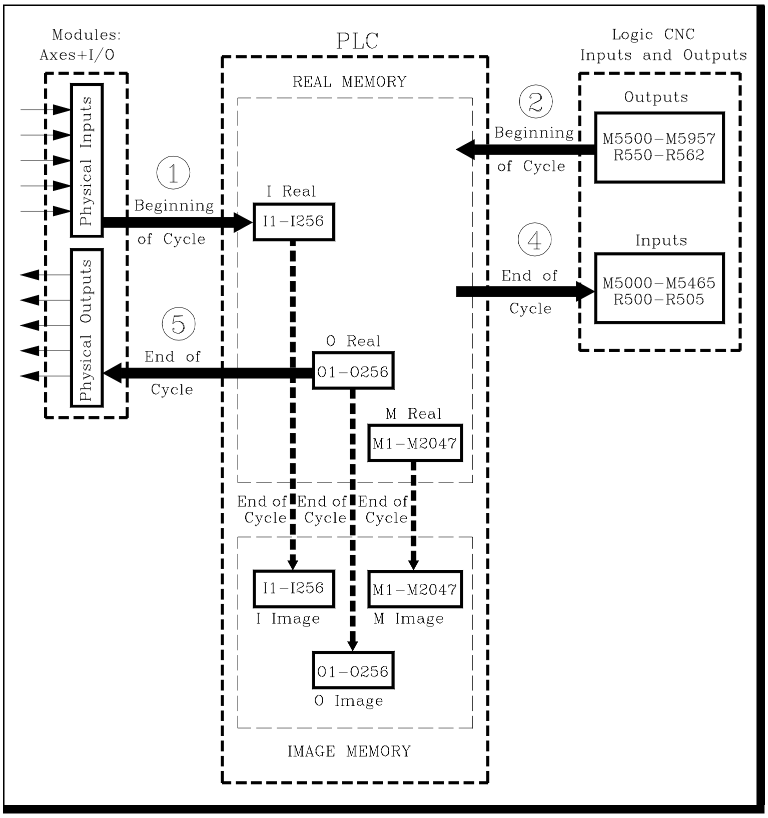
All the steps described so far work with Real memory.

It is the same to say: Saying "the value of such and such register" is the same as saying "the Real value of such and such register".

The Image Memory contains a copy of the values (status) that the resources had at the end of the previous cycle.

The PLC makes this copy at the end of the cycle.

The resources having an image value are: I1 thru I512, O1 thru O512 and M1 thru M2047



The next example shows how the PLC acts when operating with real and image values. :

PLC Program

PLC Program	() = M1	Assigns the value of “1” to mark M1
	M1 = M2	Assigns the value of M1 to M2
	M2 = M3	Assigns the value of M2 to M3
	M3 = O5	Assigns the value of M3 to output O5

	Real values				Image values			
	M1	M2	M3	O5	M1	M2	M3	O5
Beginning	0	0	0	0	0	0	0	0
End 1st scan	1	1	1	1	1	0	0	0
End 2nd scan	1	1	1	1	1	1	0	0
End 3rd scan	1	1	1	1	1	1	1	0
End 4th scan	1	1	1	1	1	1	1	1

Operating with real values

In the first scan, when execution the instruction M1 = M2, M1 has a real value of “1” set by the previous instruction.

The same is true for instructions M2=M3 and M3=O5

That is why real values are used, output O1 takes the value of “1” in the first scan.

Operating with image values

The first cycle (scan) sets the real value of M1=1; but its image value will not be set to "1" until the end of the cycle.

In the 2nd cycle (scan), the image value of M1 is "1" and the real value of M2 is set to "1". But the image value of M2 will not be set to "1" until the end of the cycle.

In the 3rd cycle (scan), the image value of M2 is "1" and the real value of M3 is set to "1". But the image value of M3 will not be set to "1" until the end of the cycle.

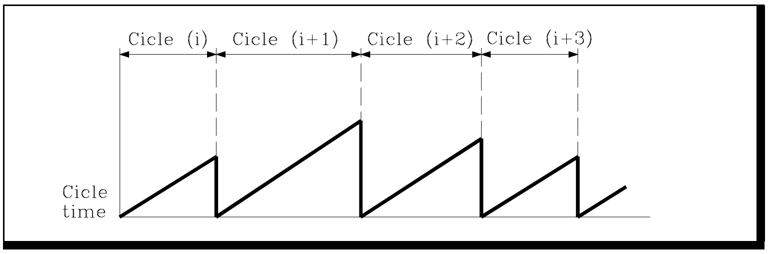
In the 4th cycle (scan), the image value of M3 is "1" and the real value of O5 is set to "1".

As can be observed, the system is faster when operating with real resource values.

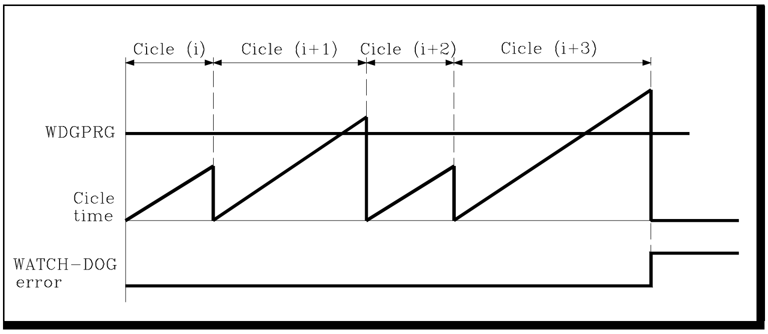
Operating with image values permits analyzing the same resource along the whole program with the same value regardless of its current (instantaneous) real value.

5.3 Cycle time

The time the PLC requires to execute the program is called cycle time and can vary in the successive cycles of a same program, as the conditions under which they are executed are not the same.



plc.m.p WDGPRG (P0) sets the maximum cycle execution time This is called WATCH-DOG time and if a cycle is executed which lasts longer than 1.5 times this time, or two cycles are executed, one after the other, taking longer than this time period, the CNC will display the WATCH-DOG error of the Main Module.



This way, the execution of cycles that, due to their duration, disturb the operation of the machine can be prevented and the PLC can be prevented from executing a cycle which has no end due to a programming error.

5.4 Modular structure of the program

The program to be executed by the PLC consists of a series of MODULES which are appropriately defined by means of DIRECTING INSTRUCTIONS.

The modules which can make up the program are:

Main module (PRG)
Periodic Execution module (PE)
First Cycle module (CY1)

Each module must begin with the directing instruction which defines it (PRG, PE, CY1) and end with the directing instruction END.

Should the main program contain the MAIN MODULE only it is not necessary to place the instructions PRG and END.

5.4.1 First Cycle module (CY1)

This module is optional and will only be executed when the PLC is turned on. It is used to initialize the different resources and variables with their initial values, before proceeding to execute the rest of the program.

This module operates by default with the real values of resources I, O, M.

It is not necessary for this to be at the beginning of the program, but must always be preceded by the instruction CY1.

5.4.2 Main module (PRG)

This module contains the user program. It will be executed cyclically and will be given the task of analyzing and modifying CNC inputs and outputs. Its execution time will be limited by the value of plc.m.p. WDGPRG (P0)

This module operates by default with the image values of resources I, O, M.

There can only be one main program and this must be preceded by the instruction PRG, it is not necessary to define it if it starts on the first line.

5.4.3 Periodic execution module (PE t)

This module is optional and will be executed every period of time t indicated in the directing instruction defining the module.

This module may be used to process certain critical inputs and outputs which cannot be checked or updated properly in the body of the main program due to its extended execution time.

Another application for this module is for those cases where specific tasks need not be evaluated at every PLC program cycle. Those tasks would be programmed in the periodic module and they would be executed with the frequency established by the execution time assigned to this module (for example: if t= 30,000; every 30 seconds).

A “t” value between 1 and 65535 milliseconds may be programmed.

The execution time of this module will be limited by the value of plc.m.p. WDGPER (P1)

This module operates by default with the real values of resources I, O, M.

Example:

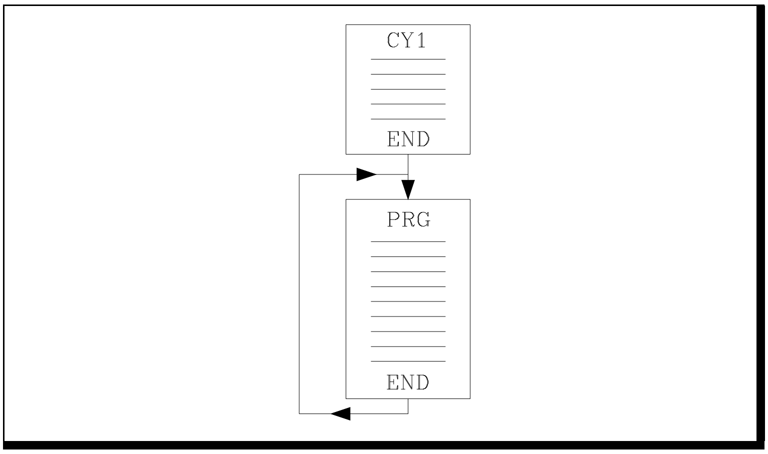
PE 10 Defines the beginning of the Periodic Module PE which will be executed every 10 milliseconds.

If this module is being executed with real values and acts on a physical output, this is updated at the end of the execution of the periodic module.

5.4.4 Priority of execution of the PLC modules

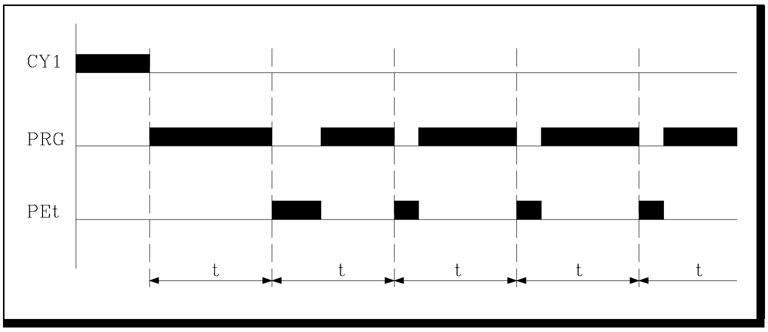
Every time the PLC program is started (command RUN) the first module to be executed is the first cycle Module (CY1). Once execution has been completed, it will continue with the main Module (PRG).

The main Module will be executed cyclically until the execution of the PLC has stopped (command STOP).



The Periodic Module will be executed every time the time indicated in the directing instruction “PE t” elapses. This count starts when the execution of the Main Module (the first time) begins.

Every time this module is executed, the execution of the Main Module is interrupted, and its execution resumes when the execution of the Periodic Module finishes.



User notes:

8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

PLC RESOURCES

6

6. PLC RESOURCES

6.1 Inputs

These are elements which supply information to the PLC from signals received from the outside world. They are represented by the letter I followed by the input number which is desired to reference, for example I1, I25, I102, etc.

The PLC may control 512 inputs although when communicating with the outside world it can only access the physical ones.

Local physical inputs are the ones corresponding to the Central unit.

Remote physical inputs are the ones corresponding to the remote modules

6.2 Outputs

These are elements which allow the PLC to activate or deactivate the different devices in the electrical cabinet. They are represented by the letter O followed by the output number which is desired to reference, for example O1, O25, O102, etc.

The PLC may control 512 outputs although when communicating with the outside world it can only access the physical ones.

Local physical outputs are the ones corresponding to the Central unit.

Remote physical outputs are the ones corresponding to the remote modules

Output O1 coincides with the emergency output of the CNC (connector); thus, it must be kept high (logic level 1).

6.3 Marks

These are elements capable of memorizing in one bit (as if they were an internal relay) information defined by the user, their value being inalterable even when the power supply to the system is turned off.

This will be programmed by the letter M followed by the number of the mark which it is wished to reference, for example, M1, M25, M102, etc.

The PLC controls the following marks:

User marks	M1 - M2000
Arithmetic flag marks	M2003
Clock marks	M2009 - M2024
Fixed status marks	M2046 and M2047
Marks associated with messages	M4000 - M4127
Marks associated with errors	M4500 - M4563
Screen marks	M4700 - M4955
CNC communication marks	M5000 - M5957

Marks M1 thru M2047 have image values unlike the remainder of the marks, and so the PLC will always work with their real values.

The arithmetic flag mark available at the PLC is:

M2003 Is the Zero flag and is set to 1 (high logic level) when the result of an AND, OR, XOR operation is 0.

The clock marks M2009 to M2024, make up internal clocks of different periods which can be used by the user.

The following table shows the available marks and the average period of each one.

M2009	100 ms.	M2017	1 sec.
M2010	200 ms.	M2018	2 sec.
M2011	400 ms.	M2019	4 sec.
M2012	800 ms.	M2020	8 sec.
M2013	1.6 sec.	M2021	16 sec.
M2014	3.2 sec.	M2022	32 sec.
M2015	6.4 sec.	M2023	64 sec.
M2016	12.8 sec.	M2024	128 sec.

The fixed status marks available at the PLC are:

M2046 Always has a value of 0.

M2047 Always has a value of 1.

The PLC allows, by means of the activation of a series of message marks, the PLC message corresponding to the PLC message table to be displayed on the CNC screen. They can be named by means of the mark M4000 - M4127 or by means of their associated ms MSG1 - MSG128:

M4000	M4001	M4002	--------	M4126	M4127
MSG1	MSG2	MSG3	--------	MSG127	MSG128

Likewise, 64 error marks are available which allow the error corresponding to the PLC error table to be displayed on the CNC screen as well as to interrupt the execution of the CNC program, stopping axis feed and spindle rotation. The activation of any of these marks does not activate the external CNC Emergency output.

They can be named by means of mark M4500-M4563 or by means of their associated mnemonic ERR1 - ERR64:

M4500	M4501	M4502	--------	M4562	M4563
ERR1	ERR2	ERR3	--------	ERR63	ERR64

Because the PLC program is not interrupted by these marks, it is advised to make it possible to change their status via accessible external inputs; otherwise, the CNC will keep receiving the same error at every PLC scan (cycle) thus preventing access to any PLC mode.

By activating one of the marks M4700-M4955 user pages 0-255 can be activated in the CNC. They can be named by means of mark M4700-M4955 or by means of their associated mnemonic PIC0 - PIC255:

M4700	M4701	M4702	--------	M4954	M4955
PIC0	PIC1	PIC2	--------	PIC254	PIC255

The PLC has marks M5000 through M5957 to exchange information with the CNC, all of which have associated mnemonics. See chapter ***9. Logic CNC inputs and outputs***

6.4 Registers

These are elements which store a numerical value in 32 bits, their value remaining unalterable even when the power supply to the system is cut off.

They do not have image values and are represented by the letter R, followed by the register number it is desired to reference, for example R1, R25, R102, etc.

The PLC has the following registers:

User registers	R1 - R499
Registers for communication with the CNC	R500 - R559

The PLC will consider each value stored in each register as an integer with a sign, and can be within ±2147483647.

It is also possible to make reference to a BIT of the REGISTER by putting the letter B and the bit number (0/31) in front of the selected register. For example:

B7R155 Refers to Bit 7 of Register 155.

The PLC considers bit 0 as being the one with least significance and bit 31 as being the one with most significance.

The value stored in a Register can be treated as being decimal, hexadecimal (preceded by “$”), binary (preceded by “B”) or in BCD. Example:

Decimal	156
Hexadecimal	$9C
Binary	B0000 0000 0000 0000 0000 0000 1001 1100

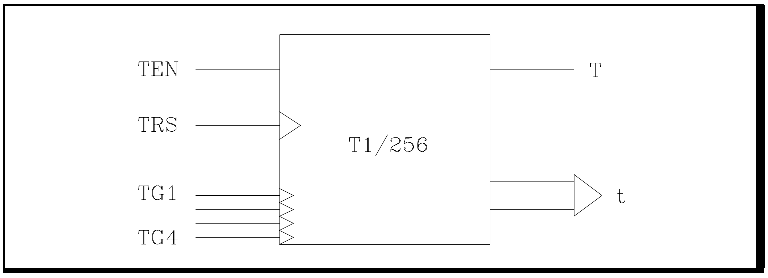
6.5 Timers

These are elements capable of maintaining their output at a determined logic level during a preset time (time constant), after which the output changes status.

They do not have image values and are represented by the letter T, followed by the number of the timer it is required to reference, for example, T1, T25, T102, etc.

The time constant is stored in a 32-bit variable, and so its value can be between 0 and 4294967295 milliseconds, which is equivalent to 1193 hours (almost 50 days).

The PLC has 256 timers, each of which has T status output and TEN, TRS, TG1, TG2, TG3 and TG4 inputs. It is also possible to consult at any moment the time which has elapsed from the moment it was activated.

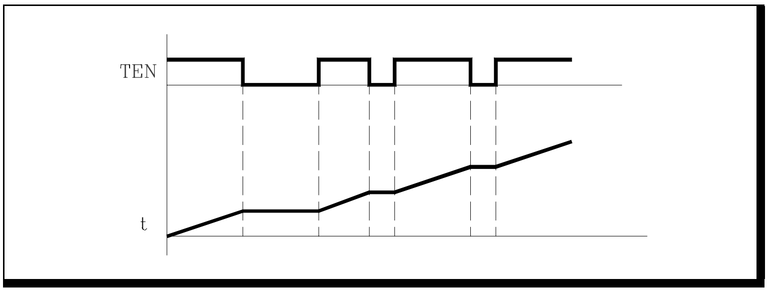


Enable input (TEN)

This input allows the timing of the timer to be stopped. It is referred to by the letter TEN followed by the number of the timer which is wished to reference, for example TEN 1, TEN 25, TEN 102, etc.

So that the time elapses within the timer this input must be at level “1”. By default and every time a timer is activated the PLC will assign this input a logic level of “1”.

If, once the timer is activated, TEN = 0 is selected, the PLC stops timing, and it is necessary to assign TEN = 1 for this timing to continue.



Example:

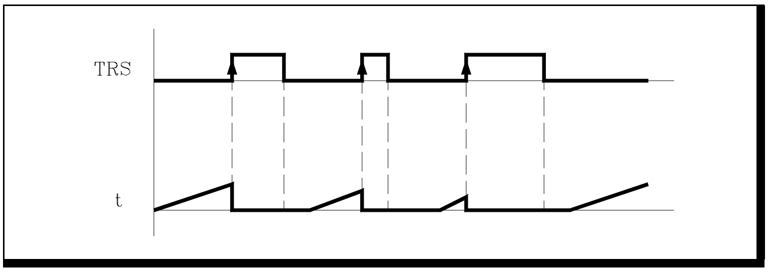
I2 = TEN 10 Input I2 controls the Enable input of timer T10.

Reset input (TRS)

This input allows the timer to be initialized, by assigning the value 0 to its T status and by canceling its count (it initializes this to 0). It is referred to by the letters TRS followed by the timer number it is wished to reference, for example TRS 1, TRS 25, TRS 102, etc.

This initialization of the timer will be made when a transition of logic level from “0” to “1” (leading edge) is produced. By default and every time a timer is activated the PLC will assign this input a logic level of “0”.

If, once the timer is activated, a leading edge is produced at the TRS input, the PLC initializes the timer, assigning value 0 to its T status and cancelling the count (it initializes this to 0). Additionally, the timer is deactivated it being necessary to activate its trigger input to activate it again.



Example:

I3 = TRS 10	Input I3 controls the Reset input of timer T10.

Trigger input (TG1, TG2, TG3, TG4)

These inputs allow the timer to be activated, and it begins to time. They are referred to by the letters TG1, TG2, TG3, TG4 followed by the number of the timer it is required to reference and the value which is required to start the count with (Time Constant).

For example TG1 1 100, TG2 25 224, TG3 102 0, TG4 200 500, etc.

The Time Constant value is defined in thousandths of a second, and it is possible to indicate this by means of a numerical value or by assigning it the internal value of an R register.

TG1 20 100	Activates timer T20 by means of trigger input TG1 and with a time constant of 100 milliseconds.
TG2 22 R200	Activates timer T22 by means of trigger input TG2 and with a time constant which will be defined (in thousandths of a second) by the value of Register R200 when the instruction is executed.

Inputs TG1, TG2, TG3 and TG4 are used to activate the timer in four different operating modes:

TG1 input in MONOSTABLE mode

TG2 input in DELAYED CONNECTION mode

TG3 input in DELAYED DISCONNECTION mode

TG4 input in SIGNAL LIMITING mode

This activation of the timer is made when a logic level transition of any of these inputs is produced, either from “0” to “1” or from “1” to “0” (leading or trailing edge) depending on the chosen input. By default and every time the timer is initialized by means of the RESET input (TRS), the PLC will assign logic level “0” to these inputs.

The operating mode of each of these trigger inputs is explained individually.

Status output (T)

This output indicates the logic status of the timer. It is referred to by the letter T followed by the number of the timer which it is required to reference, for example T1, T25, T102, etc.

The logic status of the timer depends on the operating mode selected by means of the trigger inputs TG1, TG2, TG3 and TG4, and so the activation or deactivation of this signal is explained in each of the PLC operating modes.

Elapsed time (T)

This output indicates the time elapsed in the timer since the moment it was activated. It is referred to by the letter T followed by the number of the timer which it is required to reference, for example T1, T25, T102, etc.

Although when written as T123 it coincides with the Status Output, both are different and they are also used in different types of instruction.

In binary type instructions, function T123 makes reference to the logic status of the timer.

T123 = M100 ; Assigns mark to M100 the status (0/1) of Timer 123

In arithmetic and comparison functions T123 makes reference to the time elapsed in the timer from the moment it was activated.

I2 = MOV T123 R200

Transfers the time of T123 to register R200

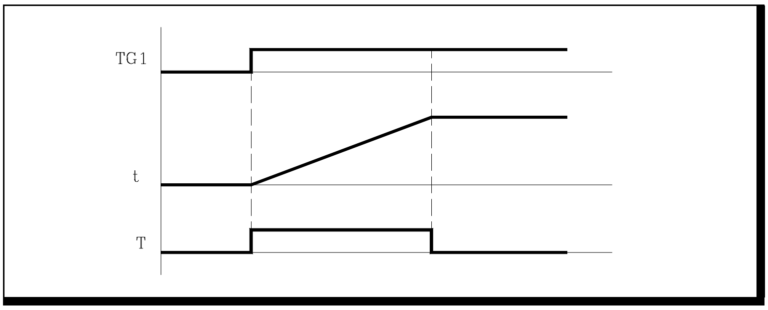
CPS T123 GT 1000 = M100

Compares whether the time of T123 is greater than 1000, in which case it activates mark M100.

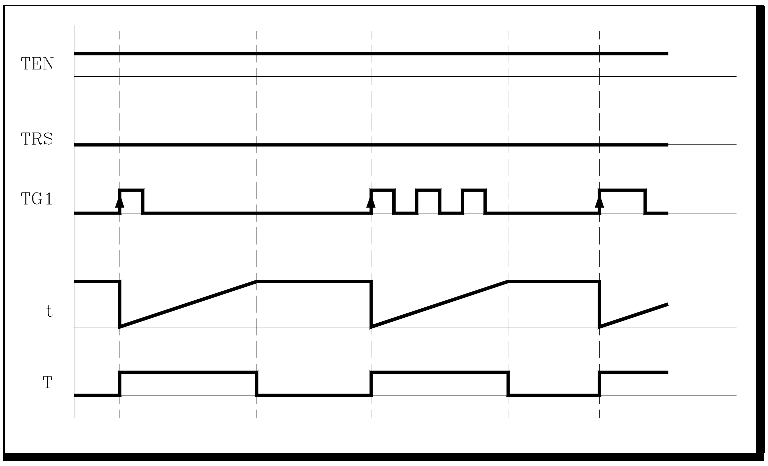
The PLC has a 32-bit variable to store the time of each timer.

6.5.1 Monostable mode. TG1 input

In this operational mode the timer status is kept at the high logic level (T=1) from the moment the TG1 input is activated until the time indicated by the time constant elapses.



If the timer is initialized with values TEN=1 and TRS=0, the timer will be activated when a leading edge is produced at input TG1. At that moment, the timer status output (T) changes status (T=1) and timing t starts from a value of 0.



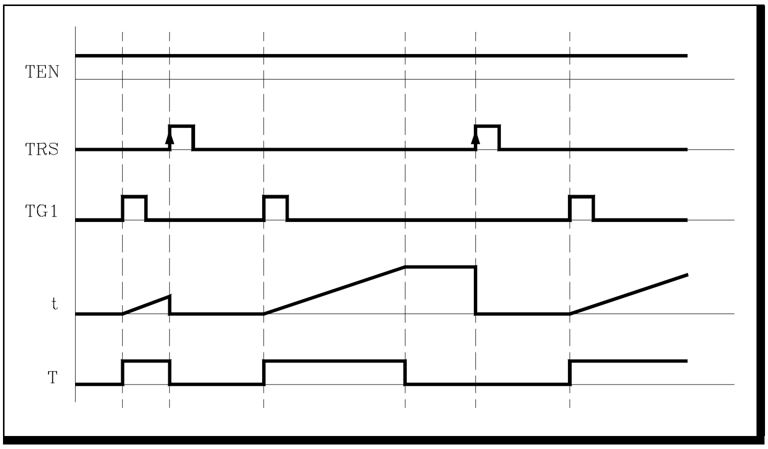
Once the time specified by the time constant has elapsed, timing will be considered as having finished. The timer status output (T) changes status (T=0) and the elapsed time will be maintained with the time value of the timer (T).

Any alteration which may be produced in input TG1 (leading or trailing edge) during the timing operation will have no effect whatsoever.

If, once the timing is complete it is required to activate the timer again, another leading edge must be produced at the TG1 input.

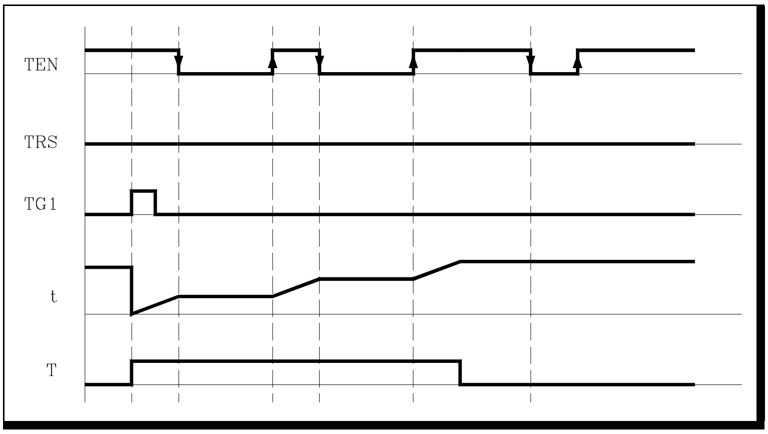
Operation of the TRS input in this mode

If a leading edge is produced at the TRS input at any moment during timing or afterwards, the PLC initializes the timer, assigning the value 0 to its T status and cancelling its count (it initializes this to 0). Due to the fact that the timer is initialized, it will be necessary to activate its trigger input to activate it again.



Operation of the TEN input in this mode

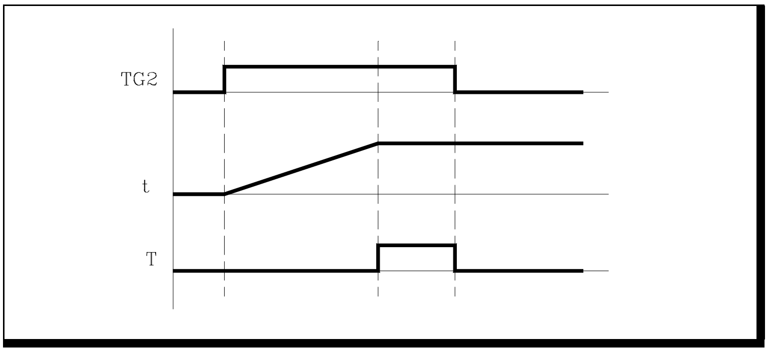
If, once the timer is activated, TEN = 0 is selected, the PLC stops timing, and it is necessary to assign TEN = 1 for this timing to continue.



6.5.2 Delayed connection mode. TG2 input

This operating mode allows a delay to be made between the activation of the trigger input TG2 and the activation of the T status of the timer.

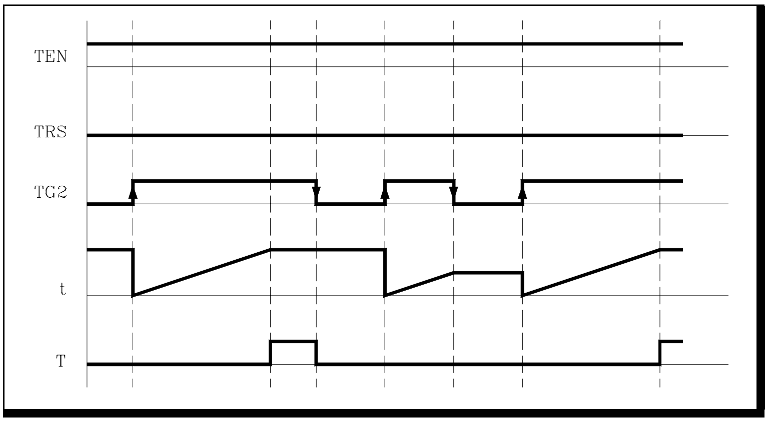
The duration of the delay is determined by the time constant.



If the timer is initialized with values TEN=1 and TRS=0, the timer will be activated when a leading edge is produced at TG2 input. At that moment, timing t will start from a value of 0.

Once the time specified by the time constant has elapsed the timing operation will be considered as having completed and the timer status output (T=1) will be activated and will remain in this status until the trailing edge is produced in the trigger input TG2.

The elapsed time will remain as a timer time value (T) once timing has been completed.

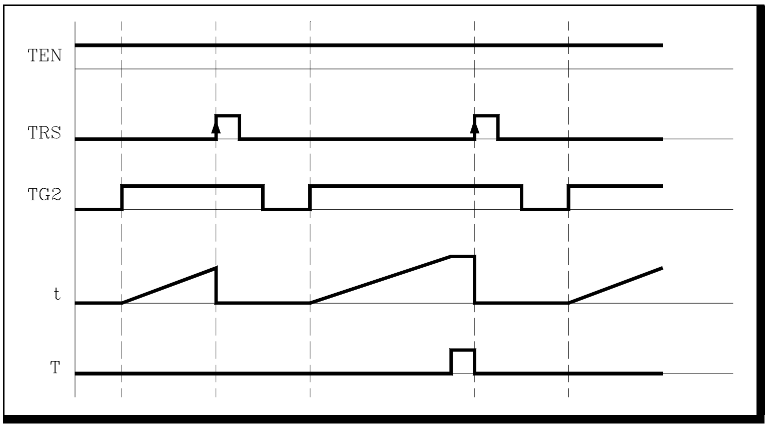


If, once the timing has finished, it is required to activate the timer again, another leading edge must be produced in the TG2 input.

If the trailing edge of the trigger input TG2 is produced before the time specified by the time constant has elapsed, the PLC will consider that the timing operation has concluded, maintaining the time count it had at that moment as the timer time (T).

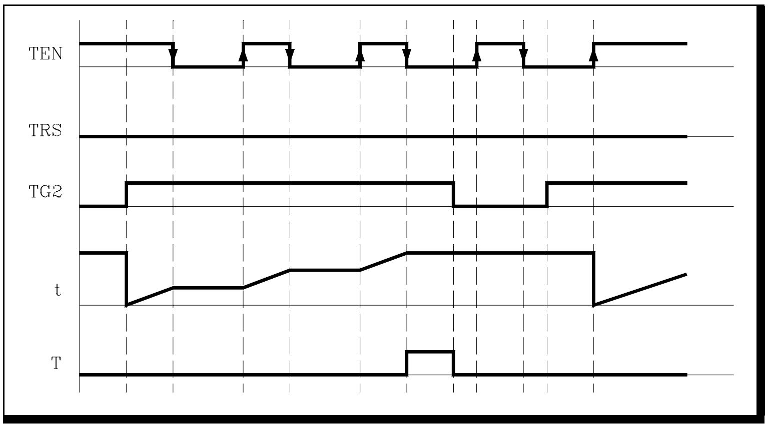
Operation of the TRS input in this mode

If a leading edge is produced at the TRS input at any moment during timing or afterwards, the PLC initializes the timer, assigning the value 0 to its T status and cancelling its count (it initializes this to 0). Due to the fact that the timer is initialized, it will be necessary to activate its trigger input to activate it again.



Operation of the TEN input in this mode

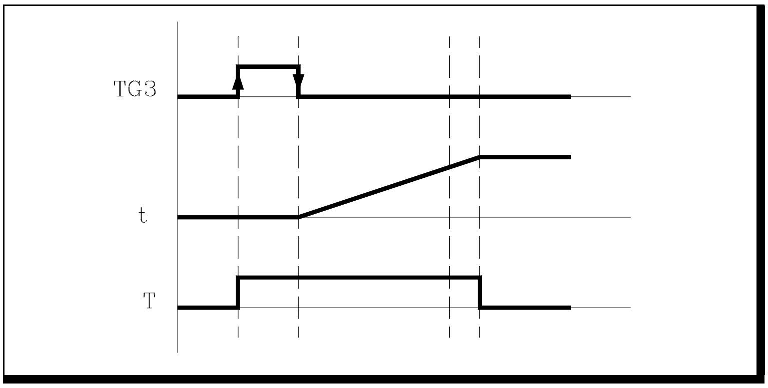
If, once the timer is activated, TEN = 0 is selected, the PLC stops timing, and it is necessary to assign TEN = 1 for this timing to continue.



6.5.3 Delayed disconnection mode. TG3 input

This operating mode allows a delay to be made between the deactivation of the trigger input TG3 and the activation of the T status of the timer.

The duration of the delay is determined by the time constant.

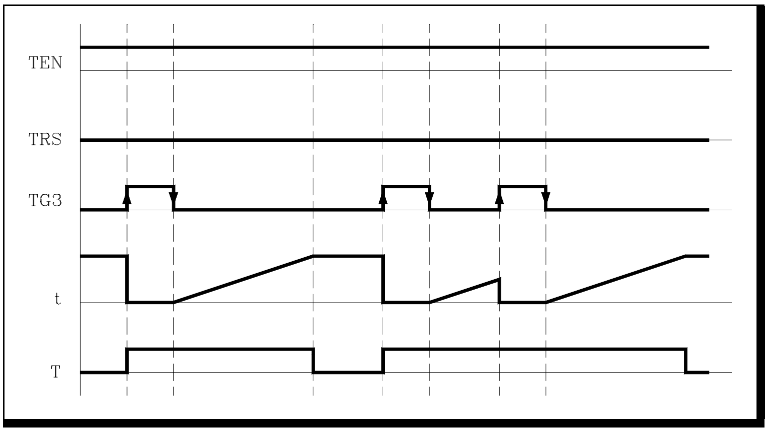


If the timer is initialized with values TEN=1 and TRS=0, the timer will be activated when a leading edge is produced at the TG3 input. At that moment, the timer status output will have a value of T=1.

The timer will wait a trailing edge of the TG3 input to start timing t from a value of 0.

Once the time specified by the time constant has elapsed the timing operation will be considered as having completed and the timer status output will be deactivated (T=0).

The elapsed time will remain as a timer time value (T) once timing has been completed.

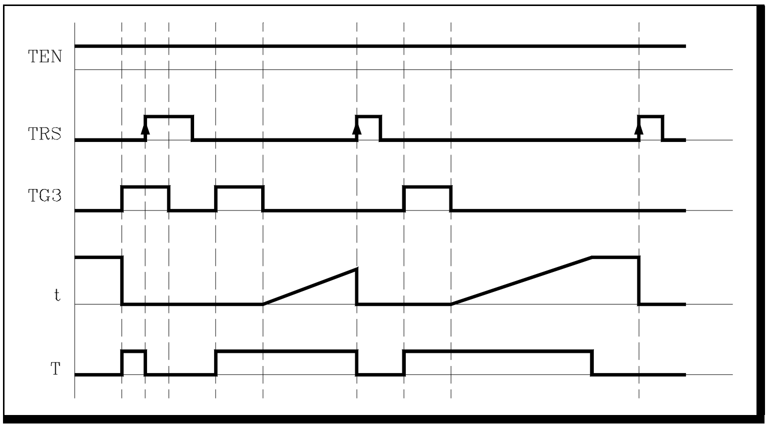


If, once the timing has finished, it is required to activate the timer again, another leading edge must be produced at the TG3 input.

If another leading edge of the trigger input TG3 is produced before the time specified by the time constant has elapsed, the PLC will consider that the timer has been activated again, maintaining its status (T=1) and initializing timing at 0.

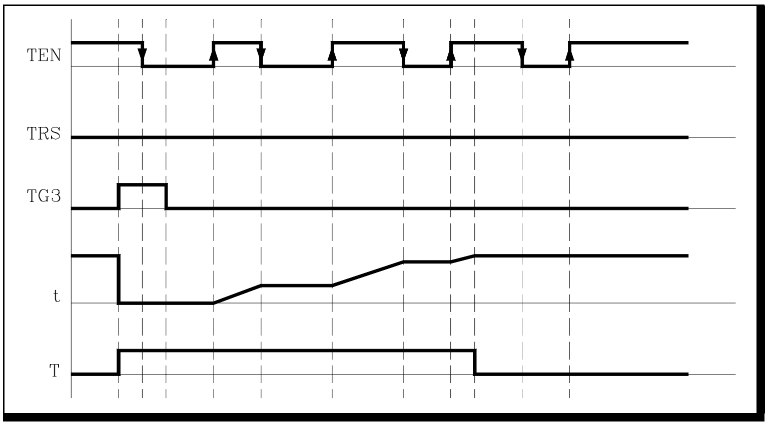
Operation of the TRS input in this mode

If a leading edge is produced at the TRS input at any moment during timing or afterwards, the PLC initializes the timer, assigning the value 0 to its T status and cancelling its count (it initializes this to 0). Due to the fact that the timer is initialized, it will be necessary to activate its trigger input to activate it again.



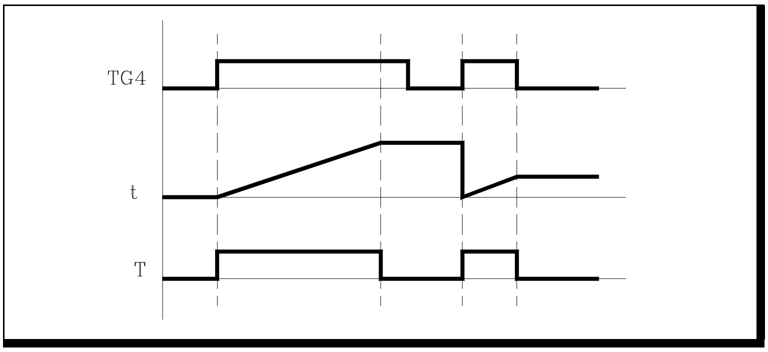
Operation of the TEN input in this mode

If, once the timer is activated, TEN = 0 is selected, the PLC stops timing, and it is necessary to assign TEN = 1 for this timing to continue.

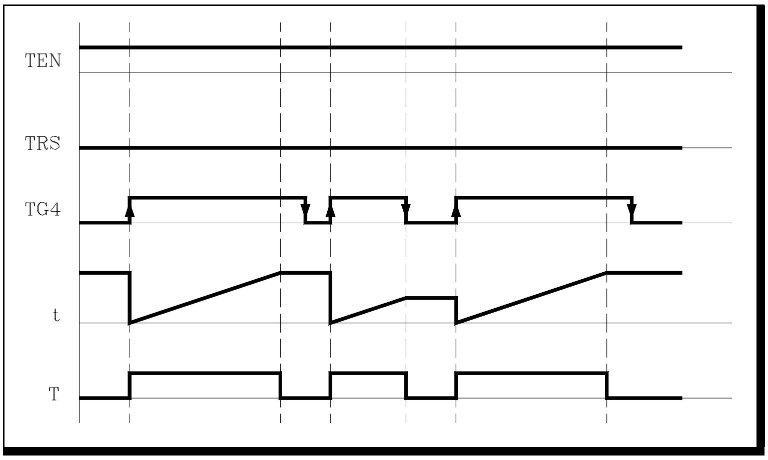


6.5.4 Signal limiting mode. TG4 input

In this operating mode the timer status is kept at a high logic level (T=1) from the moment when the TG4 input is activated until the time indicated by the time constant has elapsed, or until a down flank is produced at the TG4 input.



If the timer is initialized with values TEN=1 and TRS=0, the timer will be activated when a leading edge is produced at the TG4 input. At that moment, the timer status output (T) changes status (T=1) and timing t starts from a value of 0.



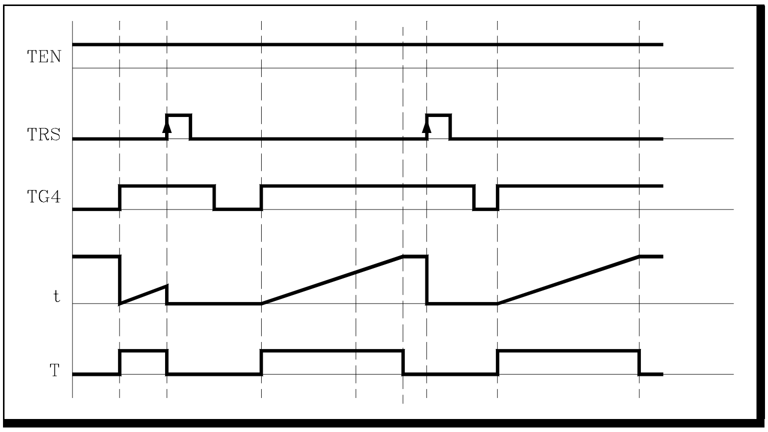
Once the time specified by the time constant has elapsed, timing will be considered as having finished. The time status output (T) changes status (T=0) and the elapsed time will be kept as a timer time value (T).

If, before the time specified by the time constant has elapsed, a trailing edge is produced in the trigger input TG4, the PLC will consider that the timing operation has concluded it will deactivate the status output (T=0) and maintain the value it has at that moment as the timer time value (T).

If, once the timing has concluded, it is required to activate the timer again, another leading edge must be produced at the TG4 input.

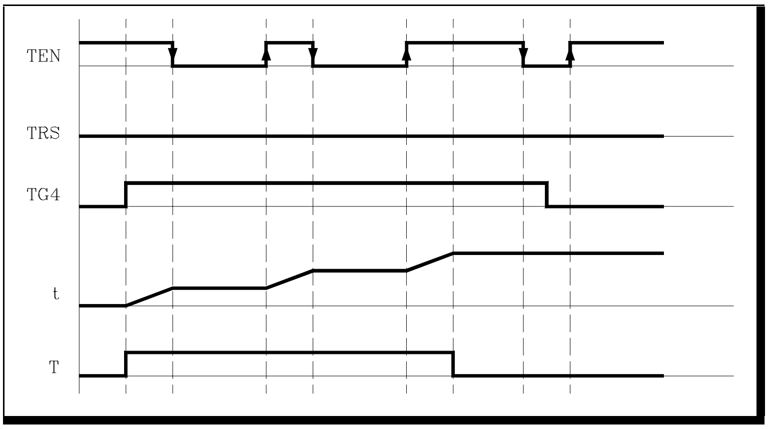
Operation of the TRS input in this mode

If a leading edge is produced at the TRS input at any moment during timing or afterwards, the PLC initializes the timer, assigning the value 0 to its T status and cancelling its count (it initializes this to 0). Due to the fact that the timer is initialized, it will be necessary to activate its trigger input to activate it again.



Operation of the TEN input in this mode

If, once the timer is activated, TEN = 0 is selected, the PLC stops timing, and it is necessary to assign TEN = 1 for this timing to continue.

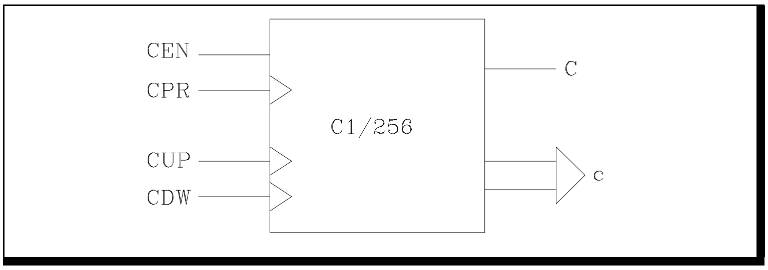


6.6 Counters

These are elements capable of counting up or down a specific amount of events. They do not have image values and are represented by the letter C, followed by the counter number which it is required to reference, for example C1, C25, C102, etc.

The count of a counter is stored in a 32-bit variable, thus having a possible value of up to +2147483647.

The PLC has 256 counter, each of which has the C status output and CUP, CDW, CEN and CPR inputs. It is also possible to consult the count value at any time.



Count-up input (CUP)

This input allows the counter count to be increased in a unit every time a leading edge is produced in it. It is referred to by the letters CUP followed by the counter number which is required to reference, for example CUP 1, CUP 25, CUP 102, etc.

Example:

I2 = CUP 10	Every time a leading edge is produced at input I2 the counter count C10 will be increased.

Count-down input (CDW)

This input allows the counter count to be decreased in a unit every time a leading edge is produced in it. It is referred to by the letters CDW followed by the counter number which is required to reference, for example CDW 1, CDW 25, CDW 102, etc.

Example:

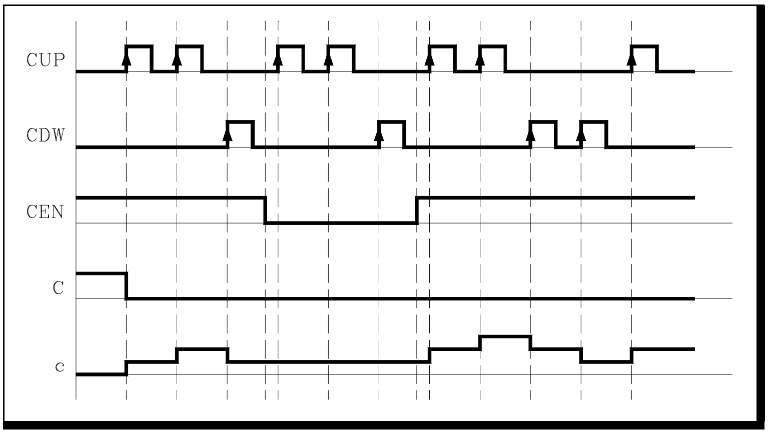
I3 = CDW 20	Every time a leading edge is produced at input I3 the counter count C20 will be decreased.

ENABLE INPUT (CEN)

This input allows the internal counter count to be stopped. It is referred to by the letters CPR followed by the number of the counter which is required to reference for example CEN 1, CEN 25, CEN 102, etc.

In order to be able to modify the internal count by means of the inputs CUP and CDW this input must be at logic level “1”. By default and every time a counter is activated the PLC will assign this input a logic level of “1”.

If CEN = 0 is selected the PLC stops the counter count, ignoring the inputs CUP and CDW until this input allows it (CEN = 1).



Example:

I10 = CEN 12	Input I10 controls the Enable input of counter C12.

PRESET INPUT (CPR)

This input allows the counter to be preset with the desired value. It is referred to by the letters CPR followed by the number of the counter which is required to reference and the value to be assigned to the counter count.

For example CPR 1 100, CPR 25 224, CPR 102 0, CPR 200 500, etc.

The value of the count can be indicated by means of a numerical value or by assigning to it the internal value of an R register.

CPR 20 100	Presets the C20 counter to a value of 100.
CPR 22 R200	Presets the C22 counter with the value of the Register 200 when the instruction is executed.

The counter is preset with the value when a leading edge is produced at the CPR input.

STATUS OUTPUT (C)

This output indicates the logic status of the counter. It is referred to by the letter C, followed by the counter number which is required to reference, for example C1, C25, C102, etc.

The logic status of the counter will be C=1 when the value of the count is zero and C=0 in the remainder of cases.

COUNT VALUE (C)

This output indicates the value of the internal counter count. It is referred to by the letter C, followed by the counter number which is required to reference, for example C1, C25, C102, etc.

Although when written C123 it coincides with the Status Output, both are different and, are used in different types of instructions.

In binary type instructions function C123 makes reference to the counter's logic status.

C123 = M100	Assigns mark to M100 the (0/1) status of counter 123

In arithmetic and function comparison instructions C123 makes reference to the internal counter count.

I2 = MOV C123 R200

Transfers the count of C123 to register R200

CPS C123 GT 1000 = M100

Compares whether the count of C123 is greater than 1000, in which case it activates mark M100.

The PLC has a 32-bit variable to store the count of each counter.

6.6.1 Operating mode of a counter

If the CEN counter input is initialized (CEN=1), the counter allows its count to be increased and decreased by means of the CUP and CDW inputs.

Operation of CUP and CDW inputs

Every time a leading edge is produced at the CUP input the counter increases its count by one count.

Every time a leading edge is produced at the CDW input the counter decreases its count by one count.

Operation of the CPR input

If a leading edge is produced at the CPR input the internal count value will take the new value assigned.

Operation of the CEN input

If CEN = 0 is selected the counter ignores both up-count (CUP) and down-count (CDW) inputs, it being necessary to assign CEN = 1 for the counter to take notice of these inputs.

User notes:

8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

PLC PROGRAMMING

7

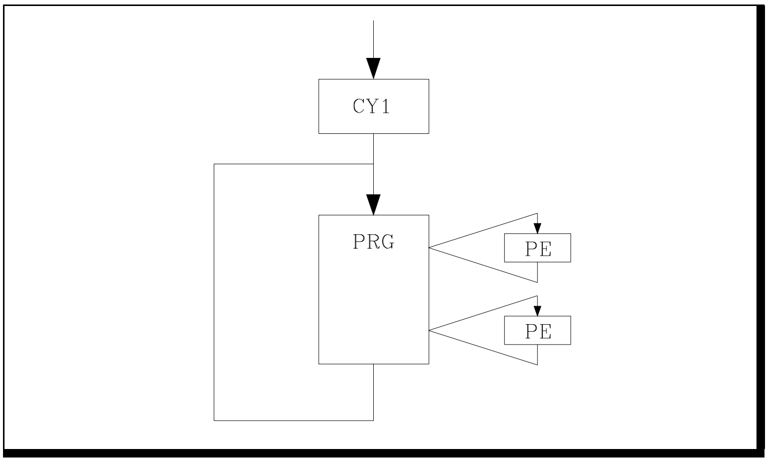
7. PLC PROGRAMMING

The PLC program is structured by modules and it could consist of:

Main module (PRG)
Periodic Execution module (PE)
First Cycle module (CY1)

Every time the PLC program starts running, the CNC will execute first, if it has been defined, the First Cycle module (CY1). Then it will execute the Main Program module (PRG) continuously until the PLC program is stopped.

The periodic execution modules (PE) will be executed every so often with the frequency established for each of them. This time period starts counting from the time the CY1 cycle is ended. The execution of a periodic module temporarily interrupts the execution of the main module.



When defining the PLC program, both the processing of the main module (PRG) and the periodic modules (PE) must be taken into consideration.

The main module (PRG) will be processed cyclically and it done as described in section*5.2 PLC program execution*

The periodic module is optional and it is executed every so often as indicated by the directing instruction defining the module.

It is used to process certain critical inputs and outputs which cannot be properly evaluated within the main module because the cycle scan time for the main module would be too long for these resources to be checked and reacted upon.

It does not modify the status of the PLC resources. Therefore, the main module will resume execution as if the Periodic Module had not been executed at all.

The periodic module (PE) is processed as follows:

1. The PLC takes into account the current values, as just before executing the PE module, of the local physical inputs (connectors of the central unit).
2. It runs the Periodic Module.
3. It assigns the current values of the PLC's "O" resources to the local physical outputs (connectors of the central unit)
4. It ends the execution of the Periodic Module and resumes the execution of the Main Module.

To work with remote physical inputs and outputs, use the IREMRD and OREMWR instructions.

7.1 Module structure

The modules which make up the PLC program (main module "PRG", periodic modules "PE" and first cycle module "CY1") consist of a series of Instructions which, depending on their functionality, can be divided into:

Directing instructions.
Executable instructions.

Directing Instructions provide the PLC with information on the type of module (PRG, CY1,...) and on how it must execute it (REA, IMA,...).

Executable Instructions allow inquiries to be made on and/or alterations to the status of PLC resources and consist of:

Logic expressions (Boolean 0/1)	I28 AND I30
Action instructions.	= O25

Logic expressions consist of:

Consulting instruction	I28, O25
Operators.	AND

All comments must begin with a “;” : ";". The lines that begin with the “;” character are considered as a comment and are not executed.

Programming example:

```
PRG            ; Directing instruction
; Example       Comment
I100 = M102    ; Executable Instruction
I28 AND I30    ; Logic expression
= O25          ; Action instruction
I32 \          ; Consulting instruction (1st part of the expression)
AND I36        ; Consulting instruction (2nd part of the expression)
= M300         ; Action instruction
END            ; Directing instruction
```

See appendix ***Summary of PLC programming commands***

Warning:

Empty lines are not allowed, they must contain at least one comment.

7.2 Directing instructions

These provide the PLC with information on the type of module and the way it must be executed.

The directing instructions available at the PLC are:

PRG, PEt, CY1 Define the module type.

PRG	Main module.
CY1	First cycle module.
PE	Periodic module. It is executed every t milliseconds. For example: PE 100 it will be executed every 100 ms.

END Indicates the end of the module. If this is not defined, the PLC understands that this module ends in the last block of the program.

Example of programming using the directing instruction END:

CY1	Beginning of module CY1
——-	
END	End of module CY1
PRG	Beginning of module PRG
——-	
END	End of module PRG
PE 100	Beginning of module PE
——-	
END	End of module PE

Example of programming without using the directing instruction END:

CY1	Beginning of module CY1
——-	
PRG	Beginning of module PRG
——-	
PE 100	Beginning of module PE
——-	
——	End of modules CY1, PRG and PE

L Label. Used to identify a program line, and is only used when references or program jumps are made.

It will be represented with the letter L followed by three figures (1-256), it not being necessary to follow any order and numbers out of sequence are permitted.

If there are 2 or more labels with the same number in a single program, the PLC will show the corresponding error when compiling it.

DEF Symbol definition. Allows a symbol to be associated with any PLC variable, it being possible to reference this variable throughout the program by means of the variable name or by means of the associated symbol.

Example:

DEF EMERG I1

Assigns the EMERG symbol to input 11, so any reference throughout the program to EMERG will be interpreted by the PLC as a reference to I1.

It is also possible to associate a symbol to any number which can be given in decimal, with or without a sign, or hexadecimal format preceded with the "$" sign.

This option, among other applications, makes programming and later understanding of the PLC program much easier when trying to control the CNC by simulating its keyboard from the PLC program.

Example:

DEF HELP $FFF2

Assigns the "HELP" symbol to the code for the HELP key.

() = MOV HELP R101

Assigns the code corresponding to the "HELP" key to register R101.

CNCWR (R101, KEY, M101)

Indicates to the CNC that the key whose code is stored in register R101 and corresponds to the HELP key.

The PLC allows up to 200 symbol definitions which must always be programmed at the beginning of the program, before any other instruction, be this directing or executing.

A symbol will be made up with up to 8 characters, and must not coincide with any of the words reserved for instructions, nor be formed by the characters space" ", equal "=", open and close parentheses "( )", comma and semicolon ", ;".

Duplicate symbols are not allowed; but, one resource may have more than one symbol.

Example:

DEF EMRGOUT O1
DEF SALEMRG O1

The symbols associated to specialized marks and register (M> 2047 y R > =500) are pre-defined in the PLC and, therefore, it is not necessary to define them, nevertheless and if required, the PLC allows a different symbol to be assigned to them.

REA, IMA Indicate to the PLC that the consultations defined below will be made on the real (REA) or image (IMA) values of I, O, M resources.

Counters. Timers and Registers do not have image values, so their real values will always be evaluated.

Action instructions (=O32) will always update the real values of PLC resources.

Example:

```
IMA           Consultations will evaluate Image values.
I1 AND I2 = 01
---------
REA           Consultations will evaluate Real values.
IMA I3 AND REA M4 = 02
              Evaluates the Image of I3 and the Real of M4
IMA I5 REA = O3
              Evaluates the Image of I5 and the next ones in Real
```

IRD
IREMRD Update the real values of the local inputs (IRD) and the remote ones (IREMRD) after reading the relevant physical inputs.

Care must be taken when using these instructions since the current real values of the inputs will be lost.

OWR
OREMWR Update the local physical outputs (OWR) and the remote ones (OREMWR) with the current real values of the corresponding O resources.

MRD Updates the values of resources M5000/5957 and R500/559 with the values of the logic outputs of the CNC.

Care must be taken when using this instruction since the current values of those resources will be lost. After executing this instruction, the new values will match those of the logic outputs of the CNC (internal variables).

MWR Updates the logic inputs of the CNC (internal variables) with the current real values of resources M5000/5957 and R500/559.

TRACE This instruction is used when working with the Logic Analyzer in order to capture data during the execution of the PLC cycle.

It must be born in mind that the logic analyzer performs a data capture at the beginning of each cycle (PRG and PE) after reading the physical inputs and updating the marks corresponding to the CNC logic outputs and just before starting the program execution.

Use this instruction to carry out another data capture while executing the PLC cycle.

Example of how to use the "TRACE" instruction:

```
PRG
          -----------
          TRACE      Data capture
          -----------
          TRACE      Data capture
          -----------
          TRACE      Data capture
          -----------
END
PE 5
          -----------
          TRACE      Data capture
          -----------
END
```

The data capture in the execution of the trace in this program takes place:

- At the beginning of each PRG cycle
- Every time the periodic cycle (PE) is executed (every 5 milliseconds)
- 3 times while executing the PRG module.
- Once while executing the PE module.

This way, by means of the "TRACE" instruction the data capture can be done any time, especially at those program points considered more critical.

This instruction must only be used when debugging the PLC program and it should be avoided once the PLC program is fully debugged.

7.3 Consulting instructions

They are used to evaluate the PLC resources as well as the marks and registers of the CNC-PLC communication. They are divided into:

Simple Consulting Instructions

Flank Detection Consulting Instructions

Comparative Consulting Instructions

All the consulting instructions allow the previous operand NOT, which reverses the result of the preceding consultation.

Example:

NOT I1 This Consultation will return a "0" if input I1 is at 1; and a "1" when input I1 is at 0.

Simple

They test the status of the resources and return their logic state.

I	1/512	Inputs
O	1/512	Outputs
M	1/5957	Marks
T	1/256	Timers
C	1/256	Counters
B	0/31 R 1/499	Register Bits

Example:

I12 It will return a 1 if input 12 is active and a 0 if otherwise.

For flank detection

They check if the status of the resource has changed since the last time this consultation was made.

This consultation may be made on Real or Image values.

There are two types of instructions:

DFU Detects whether an Up Flank (leading edge), a change of status from 0 to 1 has been produced in the specified variable. It will return a "1" if that is the case.

DFD Detects whether a Down Flank (trailing edge), a change of status from 1 to 0 has been produced in the specified variable. It will return a "1" if that is the case.

The programming format of the different combinations is:

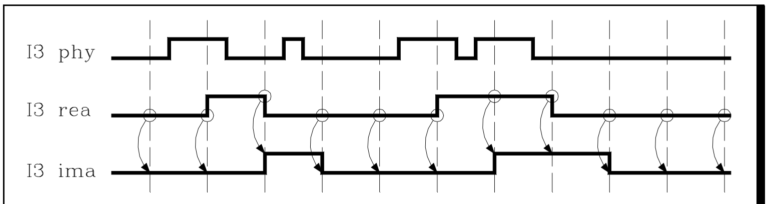
DFU (Up flank detection.) DFD (Down flank detection)	I 1/512 O 1/512 M 1/5957

The consulting instructions to detect the flanks of marks M4000 thru M4127, M4500 thru M4563, M4700 thru M4955 and M5000 thru M5957 will be executed with their real values even when working with image values since these marks have no image values.

Considering that these instructions can evaluate real and image values, the following points must be taken into account:

- The PLC updates the real values of the inputs at the beginning of the cycle, taking the values of the physical inputs.

- The image values of the inputs, outputs and marks are updated after executing the program cycle.



Examples:

DFU I23 DFU B3R120 DFU AUXEND

For comparison

CPS Used to compare two operands, checking whether the first one is greater than (GT), greater than or equal to (GE), equal to (EQ), not equal to (NE), smaller than or equal to (LE) or less than (LT) the second one.

The following may be used as operands: Timers (internal count), Counters (internal count), Registers, CNC-PLC communication registers and numbers (#) within ±2147483647 or between 0 and $FFFFFFFF

The programming format of the different combinations is:

CPS	T 1/256	GT	T 1/256
	C 1/256	GE	C 1/256
	R 1/559	EQ	R 1/559
	#	NE	#
		LE	
		LT	

If the required condition is met, the consulting instruction will return the logic value “1”; otherwise, value “0” is returned.

Programming examples:

CPS C12 GT R14 = M100

If the internal count of counter C12 is GREATER than the value of register R14, the PLC will make M100 = 1 and M100=0 in the opposite case.

CPS T2 EQ 100 = TG1 5 2000

When the time elapsed on the counter T2 is EQUAL to the value of 100, timer T5 will be activated working as a monostable and with a time constant of 2 seconds.

7.4 Operators and symbols

Used to group and operate with different consulting instructions.

The available operators are: NOT AND OR XOR

The available symbols are: ( )

Operators are associated from left to right and the priorities, ordered from highest to lowest, are the following:

NOT AND XOR OR

The "(" and ")" signs are used to clarify and select the order (sequence) used to evaluate the logic expression.

Example: (I2 OR I3) AND (I4 OR (NOT I5 AND I6)) = O7

NOT Reverses the result of the consultation.

NOT I2 = O3

Output O3 will be active when input I2 is not.

AND Logic function "AND".

I4 AND I5 = O6

Output O6 will be active when both inputs (I4, I5) are active.

OR Logic function "OR".

I7 OR I8 = O9

Output O9 will be active when one of the inputs or both are active.

XOR Logic function "Exclusive OR".

I10 XOR I11 = O12

Output O12 will be active when inputs I10 and I11 have different logic levels.

( ) Open and close parenthesis

They are used to clarify and select the order (sequence) used to evaluate the logic expression.

Example: (I2 OR I3) AND (I4 OR (NOT I5 AND I6)) = O7

A consulting instruction formed exclusively by the operators "(" and ")" always has a value of "1", i.e.:

( ) = O2

Output O2 will always show the logic value "1".

7.5 Action instructions

The action instructions, depending on the result obtained in the logic expression may be used to alter the status of the PLC resources and CNC-PLC communication marks.

Logic expression = Action instruction

There may be several action instructions associated with a single logic expression. All the action instructions must be preceded by the "=" sign.

All Action Instructions allow a previous NOT, which reverses the result of the expression for that action.

Example:

I2 = O3 = NOT M100 = NOT TG1 2 100 = CPR 1 100

- Output O3 will show the status of input I2.
- Mark M100 will show the negated status of input I2.
- A trailing edge (down flank) at input I2 will activate the trigger input TG1 of timer T2.
- A leading edge (up flank) at input I2 will preset counter C1 with value 100.

Action instructions are divided into:

- Assignment Binary Action Instructions
- Conditioned Binary Actions Instructions
- Sequence Breaking Action Instructions
- Arithmetic Action Instructions
- Logic Action Instructions
- Specific Action Instructions

Action instructions can alter the status of all the PLC resources except that of the physical inputs being used.

When seeing the "I 1/1024" field, it means that only the status of the unused inputs may be modified.

For example, when using physical inputs I1 through I32, only inputs I33 through I1024 may be changed.

7.5.1 Binary assignment instructions

They assign the value obtained from evaluating the logic expression (0/1) to the indicated resource.

= I	1/512	Inputs
= O	1/512	Outputs
= M	1/5957	Marks
= TEN	1/256	Timer enable
= TRS	1/256	Timer reset
= TGn	1/256 n/R	Timer trigger input
= CUP	1/256	Counter count up
= CDW	1/256	Counter count down
= CEN	1/256	Counter enable
= CPR	1/256 n/R	Counter preset
= B	0/31 R 1/499	Register Bits

I3 = TG1 4 100

It assigns the status of input I3 to the trigger input TG1 of timer T4, thereby the leading edge at input I3 will activate trigger input TG1 of timer T4.

(I2 OR I3) AND (I4 OR (NOT I5 AND I6)) = M111

It assigns to Mark M111 the value obtained in the evaluation of the Logic Expression (I2 OR I3) AND (I4 OR (NOT I5 AND I6)) .

7.5.2 Conditioned binary actions instructions

There are 3 instructions SET, RES and CPL that may be used to change the status of the indicated resource.

Their programming format is:

= SET	I 1/512
= RES	O 1/512
= CPL	M 1/5957
	B 0/31 R 1/559

= SET If expression “1”, it assigns a “1” to the resource

If the result of evaluating the logic expression is a “1”, it assigns a “1” to the indicated resource. If the result is a “0”, it does not change the resource.

Example: CPS T2 EQ 100 = SET B0R100

When the time elapsed in the timer T2 equals 100, bit 0 of register R100 will be set to “1”.

= RES If expression “1”, it assigns a “0” to the resource

If the result of evaluating the logic expression is a “1”, it assigns a “0” to the indicated resource. If the result is a “0”, it does not change the resource.

Example: I12 OR NOT I22 = RES M55 = NOT RES M65

When the result of the logic expression is a “1”, the PLC sets “M55=0” and does not change M65.

When the result of the logic expression is a “0”, the PLC sets “M65=0” and does not change M55.

= CPL If expression = 1, it complements the resource

If the result of evaluating the logic expression is a “1”, it complements the status of the indicated resource. If the result is a “0”, it does not change the resource.

Example: DFU I8 OR DFD M22 = CPL B12R35

Every time an Up Flank (leading edge) is detected at input I8 or a Down Flank (trailing edge) in mark M22 the PLC will complement the status of bit 12 of Register R35.

7.5.3 Sequence breaking action instructions

These actions interrupt the sequence of a program and it continues somewhere else in the program.

That area must be identified with a label (L 1/256).

A subroutine is any part of the program that starts with a label (L1/256) and end with the directing instruction END.

= JMP Unconditional Jump.

If the result of evaluating the logic expression is a “1”, it causes jump to the indicated label. If the result is a “0”, it goes on to the next line in the program.

Example:

```
I8 = JMP L12                  If I8 = 1 it goes on to L12
M14 AND B7R120 = O8           If I8=1 it is not executed
CPS T2 EQ 2000 = O12          If I8=1 it is not executed
L12
(I12 AND I23) OR M54 = O6
```

= CAL Call to a Subroutine.

If the result obtained in the evaluation of the logic Expression is a “1” this action will execute the indicated subroutine.

Once this action instruction has concluded, the PLC will continue with the execution of the next action instruction or executing instruction programmed after the CAL L1/256 command.

If the result obtained in the evaluation of the logic Expression is a “0” this action will be ignored by the PLC without executing the subroutine.

Example: I2 = CAL L5 = O2

If I2=1, it will execute subroutine L5 and when done, the PLC sets output O2 to the value of input I2 (1).

If I2=0, it does not execute the subroutine and the PLC sets output O2 to the value of input I2 (0).

= RET Return or End of Subroutine.

If the result obtained in the evaluation of the logic Expression is a “1” this action will be treated by the PLC as if it involved the directing instruction END. If the result is a “0”, it will be ignored by the PLC.

If while executing a subroutine, the PLC detects a validated RET, it will end the subroutine.

If END is not programmed as end of subroutine, the PLC will continue executing until the end of the module (END) or the end of the program and it will finish the execution of the subroutine at that point.

It is advisable to place the subroutines after the END of the main program since if these are placed at the beginning, the PLC will start to execute them and will interpret the END of the subroutine as the END of the module, and it will consider that this has finished because no call was made to the subroutine.

7.5.4 Arithmetic action instructions

= MOV Used to move information from one PLC resource to another.

The programming format is:

	Code	Code	Source code	Destination code	Number of bits to transmit
MOV	I 1/512	I 1/512	(Bin)	0 (Bin)	32
	O 1/512	O 1/512	1(BCD)	1(BCD)	28
	M 1/5957	M 1/5957			24
	T 1/256	R 1/559			20
	C 1/256				16
	R 1/559				12
	#				8
					4

The source and target codes indicate the original and destination format (binary or BCD) of the data. 4, 8, 12, 16, 20, 24, 28 or 32 bits may be transmitted.

When defining neither the codes nor the number of bits to be moved, it will assume a 32-bit (0032) binary-to-binary format.

MOV	I12	M100	0032	from Binary to Binary in 32 bits
MOV	O21	R100	0012	from Binary to Binary in 12 bits
MOV	C22	O23	0108	from Binary to BCD in 8 bits
MOV	T10	M112	1020	from BCD to Binary in 20 bits

If the number to be converted from binary to BCD is greater than the maximum allowed in BCD, the value will be truncated ignoring the most significant bits.

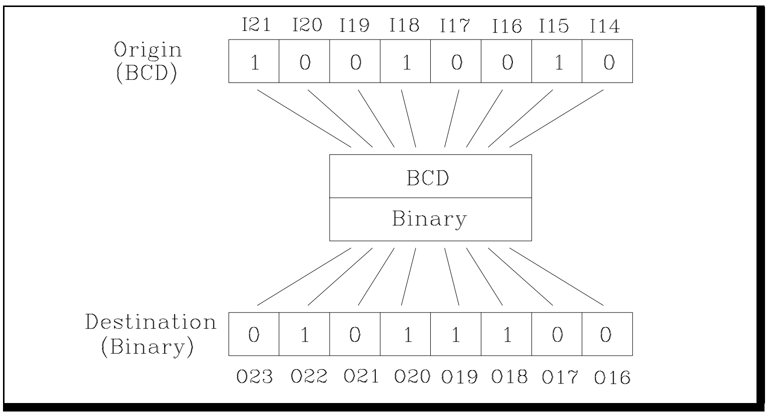
The maximum value that may be converted into BCD is:

9	with 4 bits	9999	with 16 bits	9999999	with 28 bits
99	with 8 bits	99999	with 20 bits	99999999	with 32 bits
999	with 12 bits	999999	with 24 bits		

In this cases, it is recommended to do the transfer by expanding the number of bits using, if necessary, registers or marks in intermediate steps.

Example: I11 = MOV I14 O16 108

If input I11 has a value of "1" the PLC transfers the logic states of input I14 and the next 7 inputs in BCD code, to the 8 outputs starting from O16, in binary code.



= NGU

Complements the bits of a Register.

It complements all 32 bits of the register (changes the state of each bit).

Example: I15 = NGU R152

If the input I15 has a value of "1" the PLC negates the 32 bits of register R152.

R152 before	0001	0001	0001	0001	0001	0001	0001	0001
R152 after	1110	1110	1110	1110	1110	1110	1110	1110

= NGS

Register sign change.

Example: I16 = NGS R89

If the input I16 has a value of "1" the PLC changes the sign of the contents of register R89.

R89 before	0001	0001	0001	0001	0001	0001	0001	0001
R89 after	1110	1110	1110	1110	1110	1110	1110	1111

= ADS
= SBS
= MLS
= DVS
= MDS

They may be used to carry out arithmetic operations such as addition (ADS), subtraction (SBS), multiplication (MLS), division (DVS) and module or remainder of a division (MDS).

Its programming format is:

ADS	R1/559	R1/559	R1/559
SBS	#	#	
MLS			
DVS			
MDS			

The following may be used as operands: Registers, CNC-PLC communication registers and numbers (#) within ±2147483647 or between 0 and $FFFFFFFF

The result of the operation may be stored in a register or in CNC-PLC communication register

Examples with R100=1234 and R101=100

() = ADS	R100	R101	R102	R102 = 1234 + 100 = 1334
() = SBS	R100	R101	R103	R103 = 1234 - 100 = 1134
() = MLS	R100	R101	R104	R104 = 1234 x 100 = 123400
() = DVS	R100	R101	R105	R105 = 1234 : 100 = 12
() = MDS	R100	R101	R106	R106 = 1234 MOD 100 = 34
() = ADS	1563	R101	R112	R112 = 1563 + 100 = 1663
() = SBS	R100	1010	R113	R113 = 1234 - 1010 = 224
() = MLS	1563	100	R114	R114 = 1563 x 100 = 156300
() = DVS	R100	1000	R115	R115 = 1234 : 1000 = 1
() = MDS	8765	1000	R116	R116 = 8765 MOD 1000= 765

Warning:

If a division by "0" is performed in the DVS operation, the CNC stops the execution of the PLC program and it displays the corresponding error message.

7.5.5 Logic action instructions

= AND
= OR
= XOR

They may be used to carry out logic operations AND, OR and XOR between the contents of registers or between a number an a register content. The result will always be placed in a register.

Its programming format is:

AND	R1/559	R1/559	R1/559
OR	#	#	
XOR			

Registers (R1/559) or numbers expressed in decimal, hexadecimal or binary format can be defined as first or second operand.

The destination register indicates where the result of the operation will be deposited and will be defined by means of a register (R1/559).

The mark M2003 is called Zero flag and indicates whether the result of an AND, OR, XOR, operation equals zero, in which case it follows that M2003=1.

Examples with R200 = B1001 0010
R201 = B0100 0101

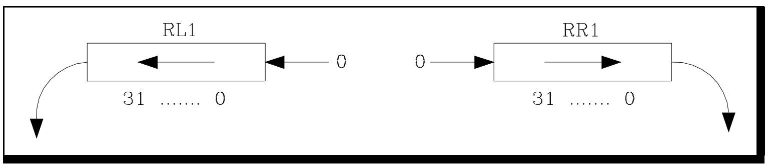
()=AND	R200	R201	R202	R202=B0	M2003=1
()=OR	R200	R201	R203	R203=B11010111	M2003=0
()=XOR	R200	R201	R204	R204=B11010111	M2003=0
()=AND	B1111	R201	R205	R205=B00000101	M2003=0
()=OR	R200	B1111	R206	R206=B10011111	M2003=0
()=XOR	B1010	B1110	R207	R207=B00000100	M2003=0

= RR
= RL

Used to rotate registers clockwise (RR) or counterclockwise (RL). There are two types of rotations: type 1 (RR1 or RL1) and type 2 (RR2 or RL2).

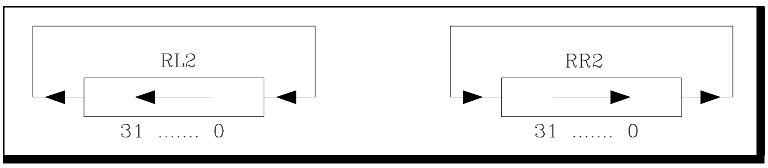
Type 1 rotation (RL1 or RR1):

It enters a 0 in the least significant bit (RL1) or in the most significant bit (RR1), by shifting the remaining bits in the register. The value of the last bit disappears.



Type 2 rotation (RL2 or RR2):

Rollover rotation of the register in the indicated direction.



Its programming format is:

	Code	Nr of repetitions	Code
RR1	R1/559	R1/559	R1/559
RR2		0/31	
RL1			
RL2			

The source and target registers must always be defined even when they are the same. The number of repetitions indicates the number of times the register will be rotated.

Examples:

RR1 R100 1 R200	1 type 1 rotation to the right of the contents of R100 leaving the result in R200
RL2 R102 4 R101	4 type 2 rotations to the left of the contents of R102 leaving the result in R101

() = RL2 R17 4 R20

R17 =	0011	0000	1100	1100	0100	0110	1101	0100
R20 =	0000	1100	1100	0100	0110	1101	0100	0011

7.5.6 Specific action instructions

= ERA Used to delete a group of resources. Indicate the first and last resource to be deleted.

Its programming format is:

ERA	I	1/512	1/512
	O	1/512	1/512
	M	1/5957	1/5957
	T	1/256	1/256
	C	1/256	1/256
	R	1/559	1/559

The Marks can be M1/2047, M4000/4127, M4500/4563, M4700/4955 or M5000/5957 and Registers R1/559

When deleting a group of I, O, M or R, the PLC sets them to "0".

If a group of timers is erased this is the equivalent of Resetting them and if a group of counters is erased this is similar to making a Preset with a value 0 for them.

This action is especially appropriate for execution in the first cycle module (CY1) with the aim of setting the required resources in the initial working conditions.

Examples:

I12 = ERA O5 12

If input I12 has a value of "1" the PLC will set to 0 outputs O5 thru O12.

I23 = ERA C15 18

If input I23 has a value of "1" the PLC will preset counters C15 thru C18 to 0.

= CNCRD
= CNCWR

Access to the internal CNC variables

They may be used to read (CNCRD) and write (CNCWR) internal CNC variables and their programming format is:

CNCRD (Variable, Register, Mark)
CNCWR (Register, Variable, Mark)

The CNCRD action loads the contents of the variable into the register and the CNCWR action reads the contents of the register into the variable.

The internal CNC variables are described in the chapter on "CNC-PLC communication".

The mark is set to "1" when the operation begins and it keeps this value until the end of the operation.

When requesting the information of a nonexistent variable (for example the coordinate of an axis that does not exist) it will show an error message.

Examples:

CNCRD (FEED, R150, M200)

Loads into register R150 the feedrate value selected at the CNC by means of function G94.

CNCWR (R92, TIMER, M200)

Presets the timer enabled by the PLC with the value contained in Register R92.

= PAR It checks the parity type of a register.

Its programming format is:

PAR R1/559 M1/5957

If the register being checked has an EVEN parity, this instruction will set the indicated mark to “1” and if its parity is ODD, it will set it to “0”.

Example:

I15 = PAR R123 M222

If I15 = 1 the PLC checks the parity of register R123 and sets M222 = 1 if it is EVEN or M222 = 0 if it is ODD.

User notes:

User notes:

8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

CNC-PLC COMMUNICATION

8

8. CNC-PLC COMMUNICATION

The exchange of information between the CNC and the PLC allows:

- The control of logic inputs and outputs from the CNC by means of an exchange of information between both systems, which is done periodically and by means of specific PLC Marks and Registers.
- The transfer from the CNC to the PLC of M, S and T auxiliary functions.
- Display screens which have been defined previously by the user, as well as generating messages and errors in the CNC, by means of specific PLC Marks.
- Reading and writing internal CNC variables from the PLC.
- Access to all PLC variables from any part program.
- Monitoring on the CNC screen of PLC variables.
- Access to all PLC variables from a computer, via DNC through RS 232 C and RS 422 serial lines.

8.1 Auxiliary M, S, T functions

MBCD1 (R550)
MBCD2 (R551)
MBCD3 (R552)
MBCD4 (R553)
MBCD5 (R554)
MBCD6 (R555)
MBCD7 (R556)
MBCDP1 (R565)
MBCD2 (R566)
MBCD3 (R567)
MBCD4 (R568)
MBCDP5 (R569)
MBCD6 (R570)
MBCD7 (R571)

MBCD* resgisters correspond to the main channel whereas MBCDP registers are for the PLC channel.

The CNC tells the PLC by means of these 32 bit registers, the miscellaneous M functions programmed in the block being executed.

If there are less than 7 miscellaneous M functions in each block, the CNC will send the information in the lower-numbered registers, assigning the value $FFFFFFFF to those which are left free.

This way, if a block contains functions M100, M120 and M135, the CNC will transfer the following information:

MBCD1 (R550)	= $100
MBCD2 (R551)	= $120
MBCD3 (R552)	= $135
MBCD4 (R553)	= $FFFFFFFF
MBCD5 (R554)	= $FFFFFFFF
MBCD6 (R555)	= $FFFFFFFF
MBCD7 (R556)	= $FFFFFFFF

Use one of the following methods to determine whether or not a specific “M” function has been programmed in a block which is being executed:

1. Check all MBCD registers one by one until the specific “M” function is found or until one of them contains the $FFFFFFFF value.
2. Use the “MBCD*” format which permits checking all MBCD registers at the same time.

   Example: CPS MBCD* EQ $30 = ...

   It returns a “1” if it detects an M30, and a “0” if otherwise.

The miscellaneous M functions can be executed at the beginning or end of the block, according to how these are set in the miscellaneous M function table.

Besides, this table will indicate whether the CNC must wait, or not, for the general logic input AUXEND to consider the execution of the corresponding M as having been completed

SBCD (R557)

This register will be used when using a spindle operating with BCD coded S signal. s.m.p. SPDLTYPE (P0).

The auxiliary S function will always be executed at the beginning of the block and the CNC will wait for the general logic input AUXEND to be activated to consider the execution completed.

If S output in 2-digit BCD is used the CNC will tell the PLC, by means of this register the selected spindle speed according to the following conversion table:

Programmed S	S BCD	Programmed S	S BCD	Programmed S	S BCD
0	00	50-55	54	800-899	78
1	20	56-62	55	900-999	79
2	26	63-70	56	1000-1119	80
3	29	71-79	57	1120-1249	81
4	32	80-89	58	1250-1399	82
5	34	90-99	59	1400-1599	83
6	35	100-111	60	1600-1799	84
7	36	112-124	61	1800-1999	85
8	38	125-139	62	2000-2239	86
9	39	140-159	63	2240-2499	87
10-11	40	160-179	64	2500-2799	88
12	41	180-199	65	2800-3149	89
13	42	200-223	66	3150-3549	90
14-15	43	224-249	67	3550-3999	91
16-17	44	250-279	68	4000-4499	92
18-19	45	280-314	69	4500-4999	93
20-22	46	315-354	70	5000-5599	94
23-24	47	355-399	71	5600-6299	95
25-27	48	400-449	72	6300-7099	96
28-31	49	450-499	73	7100-7999	97
32-35	50	500-559	74	8000-8999	98
36-39	51	560-629	75	9000-9999	99
40-44	52	630-709	76		
45-49	53	710-799	77		

If a value over 9999 is programmed the CNC will tell the PLC the spindle speed corresponding to value 9999.

If S output in 8-digit BCD is used the CNC will indicate the programmed spindle speed to the PLC by means of this register.

This value will be coded in BCD format (8 digits) in thousandths of a revolution per minute.

S 12345.678 = 0001 0010 0011 0100 0101 0110 0111 1000

If no S has been programmed in the block, the CNC will assign a value of $FFFFFFFF to this register.

TBCD (R558) The CNC tells the PLC by means of this 32-bit register, the pocket number in the magazine where the selected tool is.

If the g.m.p. RANDOMTC (P25) has been set so it is not a random magazine, the magazine pocket position coincides with the tool number.

This will be coded in BCD format (8 digits).

T 123 = 0000 0000 0000 0000 0000 0001 0010 0011

If no T has been programmed in the block, the CNC will assign a value of $FFFFFFFF to this register.

The T function will always be executed at the beginning of the block and the CNC will wait for the general logic input AUXEND to be activated to consider the execution completed.

T2BCD (R559) This register is used when a special tool change has been made (family code >=200) or with machining centers with a non-random tool magazine (general machine parameter RANDOMTC (P25).

The CNC tells the PLC by means of the 32 bit register, the position of the magazine (empty pocket) in which the tool which was on the spindle must be deposited.

This will be coded in BCD code (8 digits). If a second T function is not required the CNC will assign a value $FFFFFFFF to the register.

The second T function will be sent together with M06 and the CNC will wait for the general logic input AUXEND to the activated to consider the execution completed.

8.2 Auxiliary M, S, T function transfer

Every time a block is executed in the CNC, information is passed to the PLC about the M, S, and T functions which are active.

M function: The CNC analyzes the M functions programmed in the block and in accordance with how these are defined, will send these to the PLC before and/or after the movement.

To do this, it uses variables “MBCD1” to “MBCD7” (R550 to R556) and activates the general logic output “MSTROBE” to indicate to the PLC that it must execute them.

Depending on how these functions are defined on the table, the CNC must wait, or not, for the general input “AUXEND” to be activated to consider the execution completed.

S function: If an S function has been programmed and the spindle has BCD input, the CNC will send this value to the variable “SBCD” (R557) and will activate the general logic output “SSTROBE” to indicate to the PLC that it must be executed.

This transmission is made at the beginning of the block execution and the CNC will wait for the general input “AUXEND” to be activated to consider the execution completed.

T function: The CNC will indicate via the variable “TBCD” (R558) the T function which has been programmed in the block and activates the general logic output “TSTROBE” to tell the PLC that it must execute it.

This transmission is made at the beginning of the block execution and the CNC will wait for the general input “AUXEND” to be activated to consider the execution completed.

Second T function: If this involves changing a special tool or a machining center with non-random tool magazine, the CNC will indicate, on executing the M06 function, the position of the magazine (empty pocket) in which the tool which was on the spindle must be deposited.

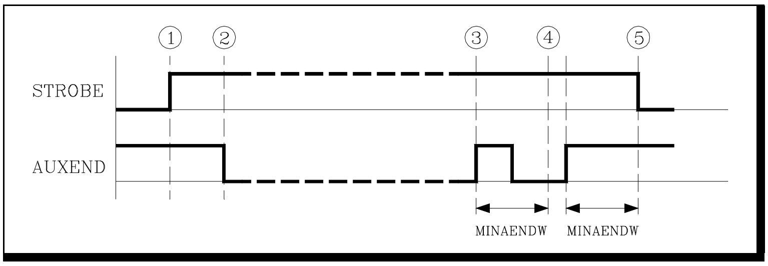
This indication will be made by means of the variable “T2BCD” (R559) and by activating the general logic output “T2STROBE” to tell the PLC that it must execute it. The CNC will wait for the general input AUXEND to be activated to consider the execution completed.

Warning:

It must be borne in mind that at the beginning of the execution of the block, the CNC can tell the PLC the execution of the M, S, T and T2 functions by activating their STROBE signals together and waiting for a single “AUXEND” signal for all of them.

8.2.1 Transferring M, S, T using the AUXEND signal

1. Once the block has been analyzed and after sending the corresponding values in the variables “MBCD1-7”, “SBCD”, “TBCD” and “T2BCD”, the CNC will tell the PLC by means of the general logic outputs “MSTROBE”, “SSTROBE”, “TSTROBE” and “T2STROBE” that the required auxiliary functions must be executed.



2. When the PLC detects that one of the STROBE signals is active, it must deactivate the general logic input “AUXEND” to tell the CNC that the execution of the corresponding function or functions is starting.
3. The PLC will execute all the auxiliary functions required, it being necessary to analyze the “MSTROBE”, “SSTROBE”, “TSTROBE”, “T2STROBE” general logic outputs and the “MBCD1-7”, “SBCD”, “TBCD” and “T2BCD” variables in order to do this.

   Once this has been executed the PLC must activate the general logic input “AUXEND” to indicate to the CNC that the processing of the required functions was completed.
4. Once the general “AUXEND” input is activated, the CNC will require that this signal be kept active for a time period greater than the value given to the g.m.p. “MINAENDW” (P30).

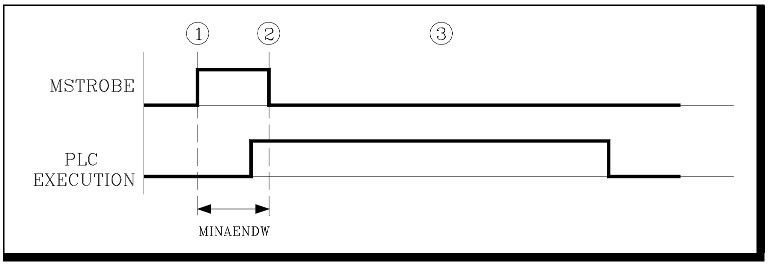
   This way, erroneous interpretations of this signal by the CNC due to an improper PLC program logic are avoided .
5. Once the period of time MINAENDW has elapsed with the general input “AUXEND” at a high logic level, the CNC will deactivate the general logic outputs “MSTROBE”, “SSTROBE”, “TSTROBE”, “T2STROBE” to tell the PLC that the execution of the required auxiliary function or functions has been completed.

Warning:

When the block being executed has several auxiliary functions (M, S, T), the CNC waits a time period set by g.m.p. MINAENDW (P30) between two consecutive transfers.

8.2.2 Transferring the auxiliary (miscellaneous) M functions without the AUXEND signal

1. Once the block has been analyzed and after passing the corresponding values in variables “MBCD1-7”, the CNC will tell the PLC through the general logic output “MSTROBE” that the required auxiliary function or functions must be executed.



2. The CNC will keep the general logic output “MSTROBE” active during the time indicated by means of g.m.p. MINAENDW (P30).

   Once this period of time has elapsed the CNC will continue to execute the program.

   It is advisable for the “MINAENDW” value to be equal to or greater than the duration of a PLC cycle, in order to ensure the detection of this signal by the PLC.

3. When the PLC detects the activation of the general logic signal “MSTROBE” it will execute the required miscellaneous “M” functions in the “MBCD1-7” variables.

8.3 Displaying messages, errors and screens

The PLC has a series of marks that allow messages and errors to be displayed in the CNC, as well as displaying screens which have been defined previously by the user.

Displaying messages

The PLC has 128 marks, with their corresponding mnemonic for displaying messages in the CNC.

M4000	MSG001	M4100	MSG101	M4125	MSG126
M4001	MSG002	M4101	MSG102	M4126	MSG127
M4002	MSG003	M4102	MSG103	M4127	MSG128
-----	-----	-----	-----	-----	-----
-----	-----	-----	-----	-----	-----

If one of these marks is activated (high logic level), the CNC will display the selected message number and its associated text on the PLC message display window (upper right-hand part).

The CNC allows a text to be associated to each PLC message (PLC message editing mode).

If the PLC activates 2 or more messages, the CNC will always display the message with the highest priority, this being understood as being the message with the lowest number. In this way, MSG1 will have the highest priority and MSG128 the lowest priority.

In this same message display window, the CNC can show the character + (plus sign), which indicates that there are more messages activated by the PLC, and these can be displayed if the active message page option is accessed in the PLC operating mode.

A message can be erased by deactivating it from the PLC program (low logic level) or from the CNC keyboard, after selecting it on the active messages page.

Nevertheless and depending on the program, the PLC may reactivate this message in the following cycle.

Example: DFU I10 = MSG1
I10 = MSG2

1. Input I10 changes from 0 to 1
   Messages MSG1 and MSG2 are activated.
2. The user deletes the messages using the keyboard
3. In the next PLC cycle, since I10 is kept at “1”, MSG2 is activated again

Displaying errors

The PLC has 64 marks, with their corresponding mnemonic, for displaying errors at the CNC.

M4500	ERR001	M4530	ERR031	M4561	ERR062
M4501	ERR002	M4531	ERR032	M4562	ERR063
M4502	ERR003	M4532	ERR033	M4563	ERR064
-----	------	-----	------	-----	------
-----	------	-----	------	-----	------

When one of these marks is activated (they are set high), they interrupt CNC part-program execution. It also displays the selected error message and its associated text in the middle of the screen.

The CNC allows a text to be associated to each PLC error (PLC error editing mode).

It is recommended to change the state of these marks by means of accessible external inputs since the PLC will not stop and the CNC will receive the error message in each new PLC cycle scan; thus preventing access to any of the PLC modes.

Displaying screens (pages)

The PLC has 256 marks with their corresponding mnemonic, for displaying screens (pages) at the CNC.

M4700	PIC000	M4900	PIC200	M4953	PIC253
M4701	PIC001	M4901	PIC201	M4954	PIC254
M4702	PIC002	M4902	PIC202	M4955	PIC255
-----	------	-----	------	-----	------
-----	------	-----	------	-----	------

If one of these marks is activated (high logic level), the CNC will display the character * (asterisk) on the PLC message display window (upper right-hand part) indicating that at least one of the 256 screens (pages) defined by the user in the graphic editor mode is activated.

The selected screens (pages) will be displayed, one by one, if the active page (screen) option is accessed in the PLC operating mode.

A page can be deactivated from the PLC program (by placing the corresponding mark at the low logic level) or, from the CNC keyboard, after selecting it in the active page mode.

8.4 Access to the PLC from the CNC

The CNC is provided with an operating mode in which it can:

Monitor the user PLC program.
Monitor PLC resources.
Modify PLC resources.
Execute PLC commands (compile, execute, etc.).
Etc.

Likewise, the CNC allows access to all PLC variables of any part program and is provided with several high level language instructions for this purpose, which allow Inputs, Outputs, Marks, Registers, Timers and Counters to be read or modified.

8.5 Access to the PLC from a PC, via DNC.

The CNC allows the PLC to communicate with a computer via DNC through the RS232C and RS422 serial lines.

In this way a computer can access the PLC carrying out:
Transfer and reception of the user PLC program.
Monitoring of the user PLC program.
Monitoring of PLC resources.
Consultation or modification of PLC resources.
Execution of PLC commands (compile, execute, etc.).
Etc.

The DNC manual can be applied for from the Commercial Department of FAGOR AUTOMATION S. COOP.

User notes:

User notes:

8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3X)
(SOFT T: 6.3X)

CNC LOGIC INPUTS AND OUTPUTS

9

9. LOGIC CNC INPUTS AND OUTPUTS

Physical inputs and outputs are the names given to the set of inputs and outputs of the CNC system which, being controlled by the PLC, communicate with the outside through CNC connectors.

The CNC also has a series of logic inputs and outputs for the internal exchange of information with PLC Marks and Registers. This type of marks do not have images on the PLC.

Each of these logic CNC inputs and outputs can be referred to by means of the corresponding PLC resource or by means of their associated Mnemonic. For example:

M5000	/EMERGEN
M5016	AUXEND
M5104	MIRROR1
M5507	/ALARM

Mnemonics which begin with "/" indicate that the signal is active low (0 V.).

All the mnemonics refer to their associated variable, it being necessary to use the NOT operator to refer to its negation, for example:

NOT M5000	NOT /EMERGEN
NOT M5016	NOT AUXEND

CNC logic inputs and outputs can be grouped in:

- General logic inputs.
- Axis logic inputs.
- Spindle logic inputs.
- Logic inputs of the auxiliary spindle
- Key inhibiting logic inputs.
- Logic inputs of the PLC channel

- General logic outputs.
- Axis logic outputs.
- Spindle logic outputs.
- Logic outputs of the auxiliary spindle
- Logic outputs of key status
- Logic inputs of the PLC channel

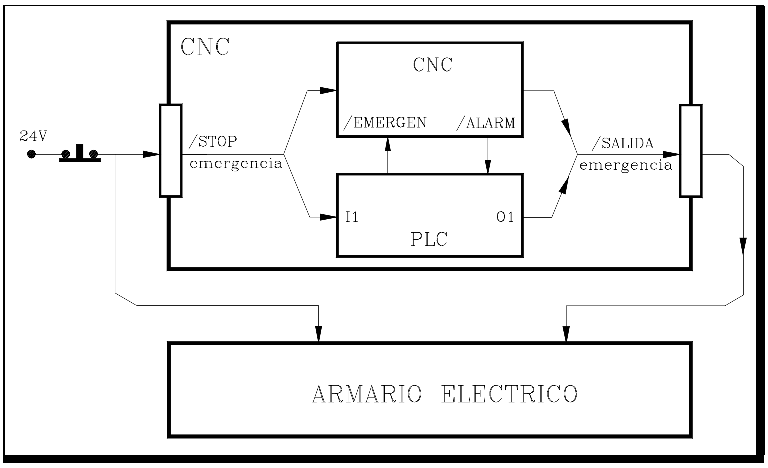
9.1 General logic inputs

Warning:

These inputs must always be defined in the PLC program.

/EMERGEN	*(M5000)*	*/STOP*	*(M5001)*
/FEEDHOL	*(M5002)*	*/XFERINH*	*(M5003)*
CYSTART	*(M5007)*		

/EMERGEN (M5000) There are to ways to cause an emergency at the CNC, by activating the physical input /EMERGENCY STOP (pin 10 of connector X2) or the general logic input "/EMERGEN" from the PLC.



When the PLC sets the "/EMERGEN" input low (0V), the CNC stops the axes and the spindle and it displays the corresponding error message.

Also, the CNC activates the "/EMERGENCY OUTPUT" and "/ALARM" signals to let the outside world and the PLC know that an emergency has occurred at the CNC.

The CNC does not allow executing programs and it aborts any attempt to move the axes or the spindle while the "/EMERGEN" input is low (0V).

When the PLC brings the "/EMERGEN" input back high (24V), the CNC deactivates the "/EMERGENCY OUTPUT" and "/ALARM" signals to let the outside world and the PLC know that there is no longer an emergency at the CNC.

Example: I-EMERG AND (rest of conditions) = /EMERGEN

If the external emergency input is activated or any other emergency occurs, the general logic input /EMERGEN of the CNC. When there is no emergency, this signal must remain high.

/STOP (M5001) When the PLC sets this signal low, the CNC stops the part program, and maintains spindle rotation.

In order to continue executing the program, as well as setting this signal at a high logic level, the general logic input CYSTART must be activated.

The treatment which this /STOP signal receives is similar to that given to the STOP key on the CNC Front Panel keeping all the keys enabled even when the /STOP signal is at low logic level (0) .

Example: ( ) = /STOP

There is always permission to execute the part program.

/FEEDHOL (M5002) When the PLC sets this signal low, the CNC stops the axes (maintaining spindle rotation). When the signal returns to the high logic level, the movement of the axes continues.

If the /FEEDHOL signal is activated (0V) in a block without motion, the CNC will continue the execution of the program until detecting a block with motion.

Example: ( ) = /FEEDHOL

There is always permission to move the axes.

/XFERINH (M5003) If the PLC sets this signal low, the CNC prevents the following block from starting, but finishes the one it is executing. When the signal returns to high logic level, the CNC continues to execute the program.

Example: ( ) = /XFERINH

There is always permission to execute the next block.

CYSTART (M5007) If the START key is pressed on the Front Panel of the CNC, this is indicated to the PLC by means of the general logic output START.

If the PLC program considers that there is nothing to prevent the part program form being executed, the CYSTART signal must be set at a high logic level, thus beginning the execution of the program.

The CNC will indicate by means of the general logic output INCYCLE that the program is being executed. As of that moment the CYSTART can return to low logic level.

Example: START AND (rest of conditions) = CYSTART

When the cycle START key is pressed, the CNC activates the general logic output START.

The PLC must check that the rest of the conditions (hydraulic, safety devices, etc.) are met before setting the general input CYSTART high in order to start executing the program

SBLOCK (M5008) When the PLC sets this signal high, the CNC changes to the Single Block execution mode.

The treatment this signal receives is similar to that given to the Single Block softkey.

MANRAPID (M5009) If the PLC sets this signal at a high logic level, the CNC selects rapid feed for all the movements executed in JOG Mode.

When the signal returns to a low logic level, the movements executed in JOG mode are made at the previously-selected feedrate.

The treatment which this signal receives is similar to that given to the Rapid Feedrate key on the Control Panel.

The ***EXRAPID (M5057)*** signal is similar, but for movements in execution.

OVRCAN (M5010) If the PLC sets this signal at a high logic level, the CNC selects 100% feedrate OVERRIDE, irrespective of whether this is selected by the PLC, DNC, program or by the Front Panel switch.

While the OVERCAN signal is activated (logic 1), the CNC will apply in each mode 100 % of the feedrate corresponding to that mode.

LATCHM (M5011) This allows the type of JOG key operation to be selected in JOG Mode.

If the PLC sets this signal low, the axes will only move while the corresponding JOG key is pressed.

If the PLC sets this signal at a high logic level, the axes will move from the moment the corresponding JOG key is pressed until the STOP key or other JOG key is pressed. In this case, the movement will be transferred to that indicated by the new key.

ACTGAIN2 (M5013) The axes and the spindle can have 2 ranges of gains and accelerations.

By default, the first range is always assumed. The one indicated by the a.m.p. and s.m.p.: ACCTIME (P18), PROGAIN (P23), DERGAIN (P24) and FFGAIN (P25)

g.m.p. ACTGAIN2 (P108) indicates with which functions or in which mode the second range is applied, the one set by a.m.p. ACCTIME2 (P59), PROGAIN2 (P60), DERGAIN2 (P61) and FFGAIN2 (P62) or s.m.p. ACCTIME2 (P47), PROGAIN2 (P48), DERGAIN2 (P49) and FFGAIN2 (P50).

The gains and accelerations can also be changed from the PLC regardless of the active operating mode or function. To do this, use general input ACTGAIN2 (M5013).

If ACTGAIN2 (M5013) = 0

The CNC assumes the first range.

If ACTGAIN2 (M5013) = 1

The CNC assumes the 2nd range.

Warning:

The change of gains and accelerations is always made at the beginning of the block.

When working in round corner (G5), the change does not take place until G07 is programmed.

RESETIN (M5015) This signal will be treated by the CNC when the JOG mode is selected and there is no movement of the axes or when a program to be executed is selected and it is not running.

When there is a rising edge (leading edge) of this signal (change from low to high) the CNC assumes the initial machining conditions selected by the machine parameter.

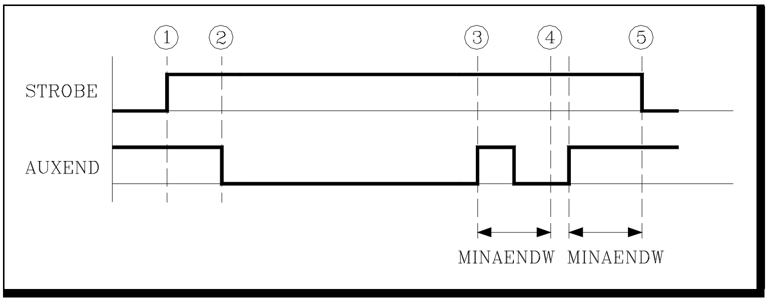
The CNC will indicate by means of the general logic output RESETOUT that this function has been selected.

The treatment received by this signal is similar to that given to the RESET key on the Front Panel.

AUXEND (M5016) This signal is used in the execution of auxiliary functions M, S and T, to tell the CNC that the PLC is executing them.

It operates in the following way:

1. Once the block has been analyzed and after sending the corresponding values in the variables "MBCD1-7", "SBCD", "TBCD" and "T2BCD", the CNC will tell the PLC by means of the general logic outputs "MSTROBE", "SSTROBE", "TSTROBE" and "T2STROBE" that the required auxiliary functions must be executed.



2. When the PLC detects that one of the STROBE signals is active, it must deactivate the general logic input "AUXEND" to tell the CNC that the execution of the corresponding function or functions is starting.
3. The PLC will execute all the auxiliary functions required, it being necessary to analyze the "MSTROBE", "SSTROBE", "TSTROBE", "T2STROBE" general logic outputs and the "MBCD1-7", "SBCD", "TBCD" and "T2BCD" variables in order to do this.

   Once this has been executed the PLC must activate the general logic input "AUXEND" to indicate to the CNC that the processing of the required functions was completed.
4. Once the general "AUXEND" input is activated, the CNC will require that this signal be kept active for a time period greater than the value given to the g.m.p. "MINAENDW" (P30).

   This way, erroneous interpretations of this signal by the CNC due to an improper PLC program logic are avoided .
5. Once the period of time MINAENDW has elapsed with the general input "AUXEND" at a high logic level, the CNC will deactivate the general logic outputs "MSTROBE", "SSTROBE", "TSTROBE", "T2STROBE" to tell the PLC that the execution of the required auxiliary function or functions has been completed.

TIMERON (M5017) The CNC is provided with a timer which can be enabled and disabled. By means of this logic CNC input, it will be enabled (timing) when the PLC sets the signal TIMERON at a high logic level.

This general purpose timer can be accessed by means of the internal variable TIMER. An application of this timer is to monitor tool life.

TREJECT (M5018) The PLC sets this signal at a high logic level in order to tell the CNC to reject the tool in use, even though it may not have come to the end of its service life. An important application is to replace the tool when the PLC detects that it is broken.

PANELOFF (M5019) The PLC sets this signal high in order to tell the CNC that the front panel keyboard (MONITOR/KEYBOARD) and the keyboard of the CONTROL PANEL of the CNC are deactivated.

It is recommended to change the state of this mark by means of an accessible external input since the PLC will not stop and the CNC will receive the error message in each new PLC cycle scan; thus preventing access to any of the PLC modes.

PLCABORT (M5022) The PLC sets this signal high to indicate to the CNC that it must stop the PLC axes. It also cancels the rest of the movement and the possible blocks that might have been sent from the PLC.

Once this process is ended, the CNC automatically deactivates this signals.

The following example shows how the axes controlled by the PLC may be moved by means of external push-buttons.

The PLC will order to move the "C" axis by 1 meter every time the "C+" button is pressed, but stopping it when this key is released.

```
DEF CPLUS I2  ; Push-button "C+"
DFU CPLUS =CNCEX (G91 G1 C1000 F3000, M1)
              ; Move 1000mm when button is pressed
DFD CPLUS = SET PLCABORT
              ; Stop when button is released
```

On power-up, the CNC sets this mark to "0".

PLCREADY (M5023) This mark indicates the PLC status.

PLCREADY = 0	PLC stopped
PLCREADY = 1	PLC in execution

If this mark is set to 0. The PLC program will stop.

This mark MUST be set to 1 so the CNC allows the spindle and/or the axes to be moved. Otherwise, it will issue the corresponding error message.

INT1 (M5024)
INT2 (M5025)
INT3 (M5026)
INT4 (M5027)

The PLC sets one of these signals to logic state "1" to "tell" the CNC to interrupt the execution of the currently running program and jump to execute the interruption subroutine whose number is indicated in the general machine parameter "INT1SUB" (P35), "INT2SUB" (P36), "INT3SUB" (P37) or "INT4SUB" (P38) respectively.

All these inputs have the same priority and are active by level (not by flank or edge). Only the first one being detected high ("1") will be attended to.

The status of these signals "INT1", "INT2", "INT3", "INT4" are not stored; therefore, it is recommended to activate these marks at the PLC by means of an instruction of the "=SET" type. These marks will be deactivated automatically when starting the execution of the corresponding subroutine.

An interruption subroutine cannot, in turn, be interrupted.

BLKSKIP1 (M5028) The PLC sets this signal at a high logic level to tell the CNC that the block skip condition "/ or /1" is met, therefore, the blocks which have this block skip condition will not be executed.

BLKSKIP2 (M5029) The PLC sets this signal at a high logic level to tell the CNC that the block skip condition "/ or /2" is met, therefore, the blocks which have this block skip condition will not be executed.

BLKSKIP3 (M5030) The PLC sets this signal at a high logic level to tell the CNC that the block skip condition "/ or /3" is met, therefore, the blocks which have this block skip condition will not be executed.

M01STOP (M5031) The PLC sets this signal at a high logic level to tell the CNC to stop the execution of the part program when the auxiliary (miscellaneous) M01 function is executed.

TOOLINSP (M5050) The CNC takes into consideration this input in MC, MCO, TC and TCO modes.

It indicates whether the "T" key must be pressed or not after executing the operation or the part in order to inspect the tool.

TOOLINSP = 0 The tool inspection mode is available when interrupting the execution.

TOOLINSP = 1 To access the tool inspection mode, first interrupt the execution and then press the "T" key

RETRACE (M5051) The CNC takes into account this input when Retracing is available, g.m.p. RETRACAC (P133) = 1.

If while executing a part-program the PLC sets this signal high, retracing is activated. The CNC interrupts program execution and starts executing backwards what has been done so far.

When the PLC sets this signal back low, retracing is canceled. The CNC starts executing forward what was done backwards and it will go on to execute the part of the program that was not machined.

Retracing executes backwards the current block plus up to 50 blocks that were already executed.

The retracing function ends in the following cases:

- When the previous 50 blocks are retraced.
- When retraced all the way to the beginning of the program
- When finding a block that contains no M, S or T function.
- When finding a high-level language block.

In all cases, the CNC activates the RETRAEND (M5522) signal to let the PLC know that all possible blocks have been executed.

While the Retracing function is active, neither tool inspection nor MDI operations are possible.

Retracing cannot be activated while a canned cycle is active or when working in “look ahead”.

ACTLIM2 (M5052) The PLC sets this signal high to "tell" to the CNC to activate the second travel limits set by means of variables LIMPL(X-C) and LIMMI(X-C)

The second travel limit of each axis will be taken into account if the first one has been set using a.m.p. LIMIT+ (P5) and LIMIT- (P6).

HNLINARC (M5053) This signal is used when selecting the “Path handwheel” work mode has been selected using general input "MASTRHND (M5054)". It allows selecting the type of movement.

M5053 = 0	Along a linear path.
M5053 = 1	Along an arc.

For a linear path, the path angle must be indicated by the MASLAN variable and for an arc, the center coordinates must be indicated by the MASCFI and MASCSE variables

The MASLAN, MASCFI and MASCSE variables can read or written from the CNC, DNC and the PLC .

MASTRHND (M5054) The PLC sets this signal high to "tell" the CNC to activate the "Path Handwheel" mode

M5054 = 0	Normal handwheels.
M5054 = 1	Path handwheel mode ON.

EXRAPID (M5057) The CNC takes it into account when g.m.p. ***RAPIDEN (P130)***=1

If the PLC sets this signal high, the programmed movements are executed in rapid traverse.

When the signal is set back low, the movements are executed at the programmed feedrate.

The treatment which this signal receives is similar to that given to the Rapid Feedrate key on the Control Panel.

The ***MANRAPID (M5009)*** signal is similar, but for movements in jog mode.

9.2 Axis logic inputs.

There are several groups of logic inputs (LIMIT, DECEL, etc.) which refer to the possible axes of the machine by means of digits 1 through 7 (LIMIT+2, DECEL1,etc.)

These numbers have nothing to do with the values assigned to the g.m.p. "AXIS1" through "AXIS8".

These variables are numbered according to the logic order of the axes.

For example, if the CNC controls the X, Y, Z, B, C and U axis, the order will be: X, Y, Z, U, B, C and, therefore:

LIMIT+1, LIMIT-1, DECEL1, etc.	for the X axis:
LIMIT+2, LIMIT-2, DECEL2, etc.	for the Y axis:
LIMIT+3, LIMIT-3, DECEL3, etc.	for the Z axis:
LIMIT+4, LIMIT-4, DECEL4, etc.	for the U axis:
LIMIT+5, LIMIT-5, DECEL5, etc.	for the B axis
LIMIT+6, LIMIT-6, DECEL6, etc.	for the C axis:

LIMIT+1 (M5100) and LIMIT-1 (M5101)
LIMIT+2 (M5150) and LIMIT-2 (M5151)
LIMIT+3 (M5200) and LIMIT-3 (M5201)
LIMIT+4 (M5250) and LIMIT-4 (M5251)
LIMIT+5 (M5300) and LIMIT-5 (M5301)
LIMIT+6 (M5350) and LIMIT-6 (M5351)
LIMIT+7 (M5400) and LIMIT-7 (M5401)

The PLC sets these signals at a high logic level in order to tell the CNC that the corresponding axis has overrun the end of its range of movement in the positive (+) or negative (-) direction indicated by the limit switch.

In this case, the CNC stops axis feed and spindle rotation and displays the corresponding error on screen.

In Manual (JOG) Operating Mode the axis which has overrun its range of travel can be moved in the correct direction in order to place it within the correct range of travel.

DECEL1 (M5102)
DECEL2 (M5152)
DECEL3 (M5202)
DECEL4 (M5252)
DECEL5 (M5302)
DECEL6 (M5352)
DECEL7 (M5402)

These signals are used by the CNC when machine reference search is made.

If the PLC sets one of these signals high, this indicates to the CNC that the machine reference search switch of the corresponding axis has been pressed.

When this signal is activated in the machine reference search mode, the CNC decelerates the axis, changing the rapid approach feedrate indicated by the a.m.p. “REFEED1”, with the slow feedrate indicated by the a.m.p. “REFEED2”. After decelerating it accepts the following reference signal from the corresponding axis feedback system as being valid.

INHIBIT1 (M5103)
INHIBIT2 (M5153)
INHIBIT3 (M5203)
INHIBIT4 (M5253)
INHIBIT5 (M5303)
INHIBIT6 (M5353)
INHIBIT7 (M5403)

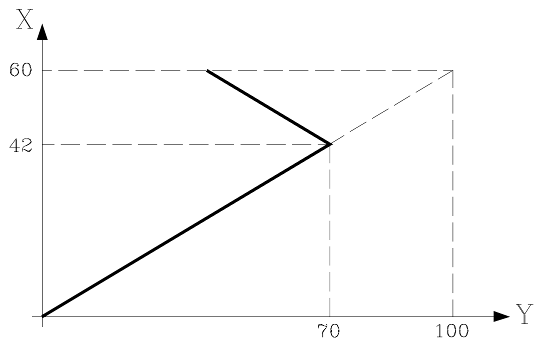
The PLC sets one of these signals at a high logic level in order to tell the CNC to prevent any movement of the corresponding axis. This movement will continue when the PLC sets this signal at the low logic level once more.

If the inhibited axis is moving together with other axes, all these stop moving until the signal returns to the low logic level.

MIRROR1 (M5104)
MIRROR2 (M5154)
MIRROR3 (M5204)
MIRROR4 (M5254)
MIRROR5 (M5304)
MIRROR6 (M5354)
MIRROR7 (M5404)

If the PLC sets one of these signals at a high logic level, the CNC applies mirror image to the movement of the corresponding axis.

It must be borne in mind that if this signal is activated during a programmed movement, the CNC will only apply mirror image to the movement, not to the final coordinate.



N00 G01 X0 Y0 F1000
N10 G01 X70 Y42
N20 G01 X100 Y60
N30 M30

If, when executing the programmed movement in block N20 the signal corresponding to the X axis “MIRROR1” is active, the CNC will apply mirror image to the remaining movement in X.

This way, the new end of travel point will be X40 Y60.

By means of the activation of these signals, symmetrical parts can be executed by using a single program, for example, soles of shoes.

In order to obtain the same effect as functions G11, G12, G13 and G14, it is necessary for the corresponding axis or axes to be positioned at part zero when these signals are activated.

SWITCH1 (M5105)
SWITCH2 (M5155)
SWITCH3 (M5205)
SWITCH4 (M5255)
SWITCH5 (M5305)
SWITCH6 (M5355)
SWITCH7 (M5405)

When having 2 axes controlled by the same servo drive, this mark may be used to toggle the velocity commands between the two axes.

See section ***4.11 Axes (2) controlled by a single drive***

DRO1 M5106
DRO2 M5156
DRO3 M5206
DRO4 M5256
DRO5 M5306
DRO6 M5356
DRO7 M5406

These inputs, together with the corresponding "SERVOON" inputs make it possible to operate with the axes as DRO.

In order for the axis to work in DRO mode, its "DRO" input must be high and its corresponding "SERVOON" input must be low.

When an axis works as a DRO, the positioning loop is open and its following error is ignored while in motion

If the DRO signal is brought back low, the axis will no longer behave as a DRO and the CNC will take as position value its current position assigning a 0 value to the following error.

SERVO1ON (M5107)
SERVO2ON (M5157)
SERVO3ON (M5207)
SERVO4ON (M5257)
SERVO5ON (M5307)
SERVO6ON (M5357)
SERVO7ON (M5407)

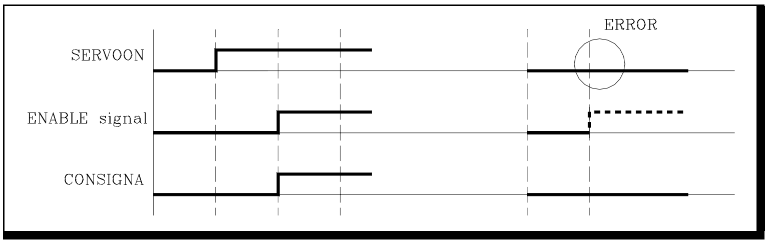
When one of these signals is set high, the CNC closes the positioning loop of the corresponding axis.

If set low, the CNC does not close the position loop of the axis. Any position deviation is stored as following error, thus when the signal gets back high, the axis moves to return to position

These signals are controlled by the PLC and when the positioning loop is to be closed, they will be processed by the CNC according to the value given to machine parameter “DWELL” (P17) for the axes.

DWELL = 0

When a.m.p. DWELL (P17) for the axis to be moved is set to 0, the CNC will check the status of the SERVOON signal at the time when the ENABLE must be output.



If the SERVOON signal is high, the CNC allows the movement of this axis by activating the ENABLE signal and outputting the required analog voltage.

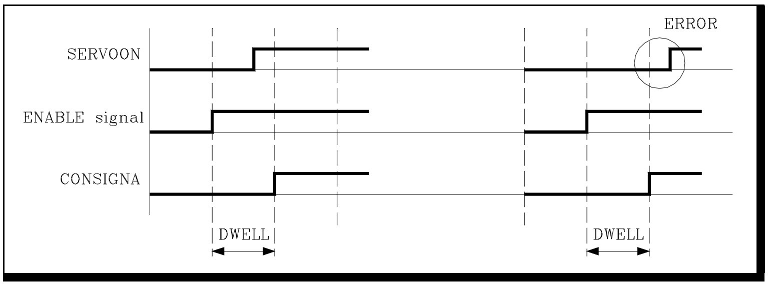
On the other hand, if the SERVOON signal is low or if it changes during the movement of the axes, the CNC stops the axes feed and the spindle rotation displaying the corresponding error message.

DWELL<>0

When a.m.p. DWELL (P17) for the axis to be moved is set to other than “0”, the CNC will check the status of the SERVOON signal at the time when the ENABLE must be output.

When this signal (SERVOON) is high, the CNC allows the movement of the axis by activating the ENABLE signal and providing the required analog output voltage.

On the other hand, if the SERVOON signal is low, the CNC activates the ENABLE signal and after “waiting” for a time period indicated in DWELL, it checks again the status of the SERVOON signal. If it is high, the required spindle analog voltage will be output. If low, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.



Also, if the SERVOON signal changes states during the movement of the axis, the CNC stops the axes feed and the spindle rotation displaying the corresponding error message.

AXIS+1 (M5108) and AXIS-1 (M5109)
AXIS+2 (M5158) and AXIS-2 (M5159)
AXIS+3 (M5208) and AXIS-3 (M5209)
AXIS+4 (M5258) and AXIS-4 (M5259)
AXIS+5 (M5308) and AXIS-5 (M5309)
AXIS+6 (M5358) and AXIS-6 (M5359)
AXIS+7 (M5408) and AXIS-7 (M5409)

The CNC uses these signals when working in the Manual (JOG) Operating Mode.

If the PLC sets one of these signals high, the CNC will move the corresponding axis in the direction indicated, positive (+) or negative (-). This movement will be performed at the feedrate override % currently selected.

The treatment which these signals receive is similar to that given to the JOG keys of the Control Panel.

SPENA1 (M5110) and DRENA1 (M5111)
SPENA2 (M5160) and DRENA2 (M5161)
SPENA3 (M5210) and DRENA3 (M5211)
SPENA4 (M5260) and DRENA4 (M5261)
SPENA5 (M5310) and DRENA5 (M5311)
SPENA6 (M5360) and DRENA6 (M5361)
SPENA7 (M5410) and DRENA7 (M5411)

The CNC uses these signals when communicating with the drive via Sercos.

Every time the PLC sets one of these signals high, the CNC lets the corresponding drive know about it via Sercos.

These signals correspond to the "Speed enable" and "Drive enable" signals of the drive.

The operation of these signals is described in the drive manual. Nevertheless, remember that:

- Both signals must be initialized low when powering up the PLC.
- For normal drive operation, both signals must be set high.
- A down flank (trailing edge) of the DRENA signal (Drive enable) turns off the power circuit of the drive and the motor loses its torque. In this situation, the motor is no longer governed and it will stop when its kinetic energy runs out. (Stop by friction).

- A trailing edge of the SPENA signal (Speed enable) switches the "Internal Velocity Reference" to "0" rpm and brakes the motor while maintaining its torque. Once the motor has stopped, it turns off the power circuit of the drive and the motor loses its torque.

SYNCHRO1 (M5112)
SYNCHRO2 (M5162)
SYNCHRO3 (M5212)
SYNCHRO4 (M5262)
SYNCHRO5 (M5312)
SYNCHRO6 (M5362)
SYNCHRO7 (M5412)

The PLC sets one of these signals high to synchronize the corresponding axis to the axis defined by the a.m.p. “SYNCHRO”.

ELIMINA1 (M5113)
ELIMINA2 (M5163)
ELIMINA3 (M5213)
ELIMINA4 (M5263)
ELIMINA5 (M5313)
ELIMINA6 (M5363)
ELIMINA7 (M5413)

If the PLC sets one these signals high, the CNC does not display the corresponding axis but keeps controlling it. Same as when setting a.m.p. DFORMAT (P1) =3

The "ELIMINA" mark can be activated and deactivated at any time and it also cancels the feedback alarms which the machine parameter does not do.

When the axis is controlled via Sercos and the PLC sets the relevant ELIMINA signal high, the commands to park that axis are generated automatically via Sercos. The drive of that axis will not issue, for example, when removing the feedback, the CNC will not issue a feedback error.

SMOTOF1 (M5114)
SMOTOF2 (M5154)
SMOTOF3 (M5214)
SMOTOF4 (M5254)
SMOTOF2 (M5154)
SMOTOF6 (M5354)
SMOTOF7 (M5414)

The SMOTIME (P58) filter set for each axis with parameter P58 can be canceled from the PLC.

This SMOTIME filter will be activated or deactivated at the beginning of the block. If one of these logic inputs is activated or deactivated while the CNC is overlapping blocks being executed in round corner, it will be ignored until that operation is finished.

LIM1OFF (M5115)
LIM2OFF (M5165)
LIM3OFF (M5215)
LIM4OFF (M5265)
LIM5OFF (M5315)
LIM6OFF (M5365)
LIM7OFF (M5415)

The PLC sets one of these signals high so that the CNC ignores the software limits of the corresponding axis.

9.3 Spindle logic inputs.

This CNC can handle 2 spindles: a main spindle and a second spindle. They both can be operative simultaneously, but only one can be controlled at a time. This selection can be made via part-program by means of functions G28 and G29.

LIMIT+S (M5450) and LIMIT-S (M5451) main spindle
LIMIT+S2 (M5475) and LIMIT-S2 (M5476) Second spindle

The CNC uses this signal while searching home when the spindle changes to working in closed loop (M19). The CNC only considers the signals for the currently selected spindle.

The PLC sets one of the signals high to tell the CNC that the spindle has overrun its range of travel in the positive (+) or negative (-) direction.

In this case, the CNC stops axis feed and spindle rotation and displays the corresponding error on screen.

DECELS (M5452) Main spindle
DECELS2 (M5477) Second spindle

The CNC uses this signal while searching home when the spindle changes to working in closed loop (M19). The CNC only considers the signals for the currently selected spindle.

The PLC sets this signal high to indicate to the CNC that the reference search switch is pressed.

When this signal is activated in the reference search mode the CNC decelerates the spindle, changing the rapid approach speed indicated by the s.m.p. REFEED1 (P34), with the slow feedrate indicated by the s.m.p. REFEED2 (P35). After decelerating, it accepts the following reference signal from the spindle feedback systems as being valid.

SPDLEINH (M5453) Main spindle
SPDLEIN2 (M5478) Second spindle

The CNC considers these 2 signals at all times so both spindles can be controlled by the PLC.

When the PLC sets this signal high, the CNC outputs a zero analog for the spindle.

SPDLEREV (M5454) Main spindle
SPDLERE2 (M5479) Second spindle

The CNC considers these 2 signals at all times so both spindles can be controlled by the PLC.

When the PLC sets this signal high, the CNC reverses the programmed spindle turning direction.

If while being this signal high, a block containing an M3 or M4 is executed, the spindle will start turning in the opposite direction.

SMOTOFS (M5455) Main spindle
SMOTOFS2 (M5480) Second spindle

The SMOTIME (P46) filter set for the main and second spindles with parameter P46 can be canceled from the PLC.

This SMOTIME filter will be activated or deactivated at the beginning of the block. If one of these logic inputs is activated or deactivated while the CNC is overlapping blocks being executed in round corner, it will be ignored until that operation is finished.

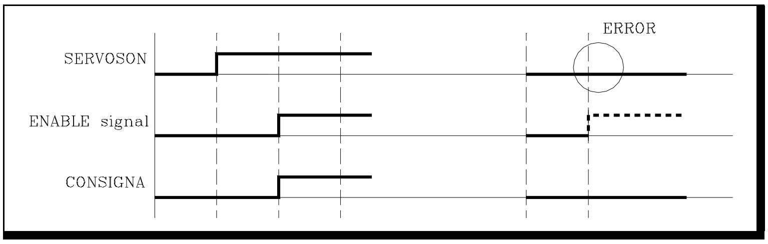
SERVOSON (M5457) Main spindle
SERVOSO2 (M5482) Second spindle

These signals are controlled by the PLC and the CNC will process them only when the spindle is working in closed loop (M19). Its treatment depends on the value assigned to s.m.p. DWELL (P17).

DWELL = 0

If s.m.p. DWELL (P17) has been set to "0", the CNC will check the status of the SERVOSON signal at the time when the ENABLE signal is to be output.

If the SERVOSON signal is high, the CNC will allow the spindle to rotate by activating the ENABLE signal and providing the required analog output voltage.



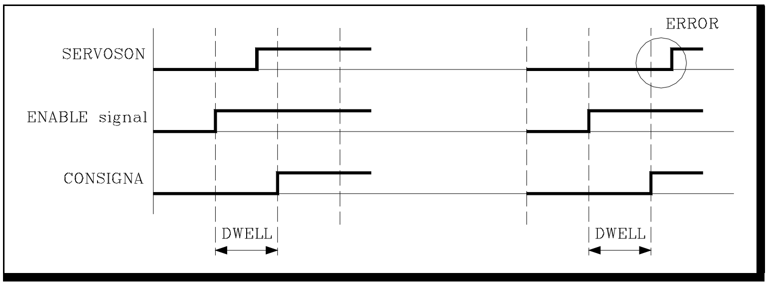
On the other hand, if the SERVOSON signal is low or if it changes to low during the rotation of the spindle, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.

DWELL<>0

If s.m.p. DWELL (P17) has been set to other than "0", the CNC will check the status of the SERVOSON signal at the time when the ENABLE signal is to be output.

If the SERVOSON signal is high, the CNC will allow the spindle to rotate by activating the ENABLE signal and providing the required analog output voltage.

On the other hand, if the SERVOSON signal is low, the CNC will activate the ENABLE signal and, after waiting for a time period indicated by the value given to "DWELL", the CNC checks the SERVOSON signal again. If it is high, the required spindle analog voltage will be output. If low, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.



Also, if it changes to low during the rotation of the spindle, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.

GEAR1 (M5458), GEAR2 (M5459), GEAR3 (M5460), GEAR4 (M5461) Main spindle GEAR12 (M5483), GEAR22 (M5484), GEAR32 (M5485), GEAR42 (M5486) Second spindle

The PLC uses these signals to indicate to the CNC which spindle speed range is currently selected (high logic level). The CNC only considers the signals for the currently selected spindle.

When any of the miscellaneous functions M41, M42, M43 or M44 is programmed, the CNC will "tell" the PLC so it selects the desired gear range even if it is already selected.

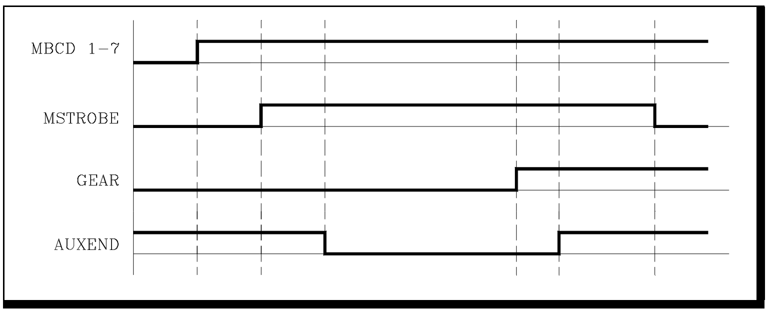
When working with automatic tool changer, the CNC will check the currently selected gear (GEAR1... GEAR4) and if it does not match the selected speed, the CNC will let the PLC know using the relevant auxiliary function (M41, M42, M43 or M44) to select it.

Once the PLC selects the proper gear, it indicates it to the CNC by means of the logic input corresponding to the spindle (GEAR1 ...). GEAR4).

The spindle gear change depends on the setting of functions M41 through M44 in the M function table:

The M41, M42, M43 or M44 function uses the "AUXEND" signal:

- The CNC indicates to the PLC the selected gear range M41, M42, M43 or M44 in one of the registers "MBCD1" through "MBCD7" and it activates the "MSTROBE" signal to let the PLC "know" that it must execute it.
- When the PLC detects the activation of the "MSTROBE" signal it must deactivate the general logic input "AUXEND" to "tell" the CNC that the execution of the function has started.
- Once executed this function, the PLC will inform the CNC that the new gear has been selected by means of the logic input corresponding to the spindle ("GEAR1".... "GEAR4").
- The PLC, then, activates the logic input "AUXEND" to "tell" the CNC that the execution of the gear change has been completed.



- Once the “AUXEND” input is activated, the CNC will require that this signal be kept active for a time period greater than the value given to the g.m.p. “MINAENDW” (P30).
- This way, erroneous interpretations of this signal by the CNC due to an improper PLC program logic are avoided .
- Once the “MINAENDW” time has elapsed with the “AUXEND” general input kept high, the CNC will check whether the new gear range has been selected by verifying that the corresponding input (GEAR1... GEAR4) is set high.
- If it is, it will cancel the general logic output “MSTROBE” to “tell” the PLC that the gear change has finished and if the corresponding input (GEAR1... GEAR4) is not selected, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.

If the M41, M42, M43 or M44 function does not use the “AUXEND” signal:

- The CNC indicates to the PLC the selected gear range M41, M42, M43 or M44 in one of the registers “MBCD1” through “MBCD7” and it activates the “MSTROBE” signal to let the PLC “know” that it must execute it.
- The CNC will keep the output “MSTROBE” active during the time indicated by means of g.m.p. MINAENDW (P30).
- After this time, the CNC will check whether the new gear range has been physically selected by verifying that the corresponding GEAR input (GEAR1... GEAR4) is set high.
- If it is not selected, the CNC will stop the axes feed and the spindle rotation displaying the corresponding error message.

SPENAS (M5462) and DRENAS (M5463) Main spindle
SPENAS2 (M5487) and DRENAS2 (M5488) Second spindle

The CNC uses these signals when communicating with the drive via Sercos.

Every time the PLC sets one of these signals high, the CNC lets the corresponding drive know about it via Sercos.

These signals correspond to the "Speed enable" and "Drive enable" signals of the drive.

The operation of these signals is described in the drive manual. Nevertheless, remember that:

- Both signals must be initialized low when powering up the PLC.
- For normal drive operation, both signals must be set high.
- A down flank (trailing edge) of the DRENA signal (Drive enable) turns off the power circuit of the drive and the motor loses its torque. In this situation, the motor is no longer governed and it will stop when its kinetic energy runs out. (Stop by friction).
- A trailing edge of the SPENA signal (Speed enable) switches the "Internal Velocity Reference" to "0" rpm and brakes the motor while maintaining its torque. Once the motor has stopped, it turns off the power circuit of the drive and the motor loses its torque.

PLCFM19 (M5464) and M19FEED (R505) Main spindle
PLCFM192 (M5489) and M19FEED2 (R507) Second spindle

The CNC only considers the signals for the currently selected spindle.

The PLC uses the "PLCM19" signal to indicate to the CNC the positioning and rapid synchronized speed value to assume when operating in closed loop (M19).

When this input is low, the CNC assumes the value set by s.m.p. "REFEED1" (P34)

When this input is high, the CNC assumes the value set by the spindle input register "M19FEED" (R505).

The "M19FEED" value is given in 0.0001º/min.

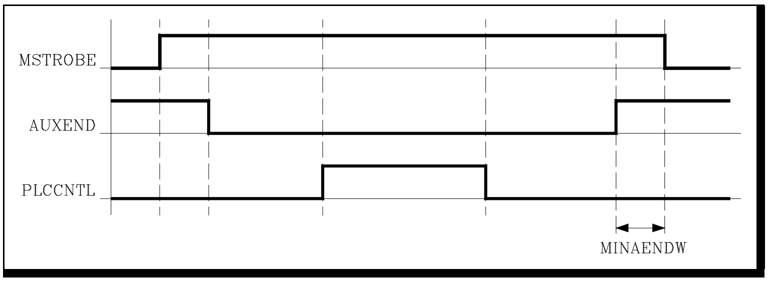
PLCCNTL (M5465) Main spindle
PLCCNTL2 (M5490) Second spindle

The CNC considers these 2 signals at all times so both spindles can be controlled by the PLC.

This is used to tell the CNC that the spindle is controlled directly by the PLC (high logic level).

It is used, for example, for oscillating the spindle during gear changes or for changing tools.

The following example shows how a new spindle speed is selected involving a range change.



After analyzing the block and detecting the speed change the CNC indicates this to the PLC in one of the “MBCD1-7” Registers (M41 to M44) and will activate the general logic output “MSTROBE” to tell the PLC that it must execute it.

The PLC will deactivate the logic input AUXEND to tell the CNC that the treatment of the auxiliary function is starting.

After calculating the value corresponding to the residual output S for the range change, the PLC will indicate this to the CNC by means of the Register “SANALOG”, afterwards setting the signal “PLCCNTL” at a high logic level.

At this time the CNC will send out the output indicated in the Register SANALOG.

Once the requested speed change has been made, the new active speed will be indicated to the CNC (spindle logic inputs GEAR1 to GEAR4).

In order to give the control of the spindle back to the CNC, the signal “PLCCNTL” must be set low.

Finally, the PLC will activate the logic input AUXEND once more to tell the CNC that the execution of the auxiliary function has been completed.

SANALOG (R504) Main spindle
SANALOG2 (R506) Second spindle

The CNC considers these 2 signals at all times so both spindles can be controlled by the PLC.

The PLC will indicate by means of this 32 bit register the spindle analog output which the CNC must send out when it is controlled by the PLC.

SANALOG=32767 corresponds to an analog output of 10 V.

(10/32767) 0.305185 millivolts of analog output correspond to SANALOG=1.

This way, for 4V of analog voltage, the following must be programmed:

SANALOG = (4 x 32767)/10 = 13107

For -4V of analog voltage, the following must be programmed:

SANALOG = (-4 x 32767)/10 = -13107

9.4 Logic inputs of the auxiliary spindle

SPENAAS (M5449) and DRENAAS and (M5448)

The CNC uses these signals when communicating with the drive via Sercos.

Every time the PLC sets one of these signals high, the CNC lets the corresponding drive know about it via Sercos.

These signals correspond to the "Speed enable" and "Drive enable" signals of the drive.

The operation of these signals is described in the drive manual. Nevertheless, remember that:

- Both signals must be initialized low when powering up the PLC.
- For normal drive operation, both signals must be set high.
- A down flank (trailing edge) of the DRENA signal (Drive enable) turns off the power circuit of the drive and the motor loses its torque. In this situation, the motor is no longer governed and it will stop when its kinetic energy runs out. (Stop by friction).
- A trailing edge of the SPENA signal (Speed enable) switches the "Internal Velocity Reference" to "0" rpm and brakes the motor while maintaining its torque. Once the motor has stopped, it turns off the power circuit of the drive and the motor loses its torque.

PLCCNTAS (M5056) It is used to "tell" the CNC that the auxiliary spindle is controlled directly by the PLC (active high).

SANALOAS (R509) With this 32-bit register, the PLC will "tell" the CNC what spindle analog voltage to output when the auxiliary spindle is controlled by the PLC or via Sercos.

A 10 V of analog voltage corresponds to SANALOAS=32767.

SANALOAS=1 corresponds (10/32767) 0.305185 millivolts of analog voltage.

This way, for 4V of analog voltage, the following must be programmed:

SANALOAS = (4x32767)/10 = 13107

For -4V of analog voltage, the following must be programmed:

SANALOAS = (-4x32767)/10 = -13107

9.5 Key inhibiting logic inputs.

KEYDIS1 (R500)
KEYDIS2 (R501)
KEYDIS3 (R502)
KEYDIS4 (R503)

The PLC can individually inhibit the operation of the panel keys, setting the corresponding bit of one of these four 32-bit registers high.

Register	Bit	Key inhibited	Register	Bit	Key inhibited
KEYDIS1	0	F	KEYDIS2	0	B
KEYDIS1	1	L	KEYDIS2	1	H
KEYDIS1	2	Q	KEYDIS2	2	N
KEYDIS1	3	W	KEYDIS2	3	S
KEYDIS1	4	SHIFT	KEYDIS2	4	Y
KEYDIS1	5	9	KEYDIS2	5	RESET
KEYDIS1	6	6	KEYDIS2	6	ESC
KEYDIS1	7	3	KEYDIS2	7	MAIN MENU
KEYDIS1	8	E	KEYDIS2	8	A
KEYDIS1	9	K :	KEYDIS2	9	G
KEYDIS1	10	P	KEYDIS2	10	M
KEYDIS1	11	V	KEYDIS2	11	R
KEYDIS1	12	CAPS	KEYDIS2	12	X
KEYDIS1	13	8	KEYDIS2	13	ENTER
KEYDIS1	14	5	KEYDIS2	14	HELP
KEYDIS1	15	2	KEYDIS2	15	
KEYDIS1	16	D:	KEYDIS2	16	.
KEYDIS1	17	J	KEYDIS2	17	0
KEYDIS1	18	O	KEYDIS2	18	-
KEYDIS1	19	U	KEYDIS2	19	+
KEYDIS1	20	SP	KEYDIS2	20	
KEYDIS1	21	7	KEYDIS2	21	
KEYDIS1	22	4	KEYDIS2	22	
KEYDIS1	23	1	KEYDIS2	23	
KEYDIS1	24	C	KEYDIS2	24	Next page
KEYDIS1	25	I	KEYDIS2	25	Previous page
KEYDIS1	26	Ñ	KEYDIS2	26	Up arrow
KEYDIS1	27	T	KEYDIS2	27	Down arrow
KEYDIS1	28	Z	KEYDIS2	28	Right arrow
KEYDIS1	29	=	KEYDIS2	29	Left arrow
KEYDIS1	30	/	KEYDIS2	30	CL
KEYDIS1	31	*	KEYDIS2	31	INS

The keys inhibited by register KEYDIS3 (R502) depends on the CNC model available (M or T model).

<table>
<tr><th rowspan="2">Register</th><th rowspan="2">Bit</th><th colspan="2">Key inhibited</th><th rowspan="2">Register</th><th rowspan="2">Bit</th><th rowspan="2">Key inhibited</th></tr>
<tr><th>M</th><th>T</th></tr>
<tr><td>KEYDIS3</td><td>0</td><td colspan="2">F1</td><td>KEYDIS4</td><td>0</td><td>Handwheel x100</td></tr>
<tr><td>KEYDIS3</td><td>1</td><td colspan="2">F2</td><td>KEYDIS4</td><td>1</td><td>Handwheel x10</td></tr>
<tr><td>KEYDIS3</td><td>2</td><td colspan="2">F3</td><td>KEYDIS4</td><td>2</td><td>Handwheel x1</td></tr>
<tr><td>KEYDIS3</td><td>3</td><td colspan="2">F4</td><td>KEYDIS4</td><td>3</td><td>JOG 10000</td></tr>
<tr><td>KEYDIS3</td><td>4</td><td colspan="2">F5</td><td>KEYDIS4</td><td>4</td><td>JOG 1000</td></tr>
<tr><td>KEYDIS3</td><td>5</td><td colspan="2">F6</td><td>KEYDIS4</td><td>5</td><td>JOG 100</td></tr>
<tr><td>KEYDIS3</td><td>6</td><td colspan="2">F7</td><td>KEYDIS4</td><td>6</td><td>JOG 10</td></tr>
<tr><td>KEYDIS3</td><td>7</td><td colspan="2"></td><td>KEYDIS4</td><td>7</td><td>JOG 1</td></tr>
<tr><td>KEYDIS3</td><td>8</td><td>X +</td><td>3rd axis +</td><td>KEYDIS4</td><td>8</td><td>Feed Override 0%</td></tr>
<tr><td>KEYDIS3</td><td>9</td><td>Y +</td><td rowspan="3">X +</td><td>KEYDIS4</td><td>9</td><td>Feed Override 2%</td></tr>
<tr><td>KEYDIS3</td><td>10</td><td>Z +</td><td>KEYDIS4</td><td>10</td><td>Feed Override 4%</td></tr>
<tr><td>KEYDIS3</td><td>11</td><td>4 +</td><td>KEYDIS4</td><td>11</td><td>Feed Override 10%</td></tr>
<tr><td>KEYDIS3</td><td>12</td><td>5 +</td><td>4th axis +</td><td>KEYDIS4</td><td>12</td><td>Feed Override 20%</td></tr>
<tr><td>KEYDIS3</td><td>13</td><td colspan="2">Speed Override +</td><td>KEYDIS4</td><td>13</td><td>Feed Override 30%</td></tr>
<tr><td>KEYDIS3</td><td>14</td><td colspan="2">Spindle clockwise</td><td>KEYDIS4</td><td>14</td><td>Feed Override 40%</td></tr>
<tr><td>KEYDIS3</td><td>15</td><td colspan="2">Start</td><td>KEYDIS4</td><td>15</td><td>Feed Override 50%</td></tr>
<tr><td>KEYDIS3</td><td>16</td><td colspan="2"></td><td>KEYDIS4</td><td>16</td><td>Feed Override 60%</td></tr>
<tr><td>KEYDIS3</td><td>17</td><td></td><td>Z -</td><td>KEYDIS4</td><td>17</td><td>Feed Override 70%</td></tr>
<tr><td>KEYDIS3</td><td>18</td><td colspan="2">Rapid</td><td>KEYDIS4</td><td>18</td><td>Feed Override 80%</td></tr>
<tr><td>KEYDIS3</td><td>19</td><td></td><td>Z +</td><td>KEYDIS4</td><td>19</td><td>Feed Override 90%</td></tr>
<tr><td>KEYDIS3</td><td>20</td><td colspan="2"></td><td>KEYDIS4</td><td>20</td><td>Feed Override 100%</td></tr>
<tr><td>KEYDIS3</td><td>21</td><td colspan="2" rowspan="3">Spindle stop</td><td>KEYDIS4</td><td>21</td><td>Feed Override 110%</td></tr>
<tr><td>KEYDIS3</td><td>22</td><td>KEYDIS4</td><td>22</td><td>Feed Override 120%</td></tr>
<tr><td>KEYDIS3</td><td>23</td><td>KEYDIS4</td><td>23</td><td></td></tr>
<tr><td>KEYDIS3</td><td>24</td><td>X -</td><td>3rd axis -</td><td>KEYDIS4</td><td>24</td><td></td></tr>
<tr><td>KEYDIS3</td><td>25</td><td>Y -</td><td></td><td>KEYDIS4</td><td>25</td><td></td></tr>
<tr><td>KEYDIS3</td><td>26</td><td>Z -</td><td>X -</td><td>KEYDIS4</td><td>26</td><td></td></tr>
<tr><td>KEYDIS3</td><td>27</td><td>4 -</td><td></td><td>KEYDIS4</td><td>27</td><td></td></tr>
<tr><td>KEYDIS3</td><td>28</td><td>5 -</td><td>4th axis -</td><td>KEYDIS4</td><td>28</td><td></td></tr>
<tr><td>KEYDIS3</td><td>29</td><td colspan="2">Speed Override -</td><td>KEYDIS4</td><td>29</td><td></td></tr>
<tr><td>KEYDIS3</td><td>30</td><td colspan="2">Spindle counterclockwise</td><td>KEYDIS4</td><td>30</td><td></td></tr>
<tr><td>KEYDIS3</td><td>31</td><td colspan="2">Stop</td><td>KEYDIS4</td><td>31</td><td></td></tr>
</table>

Should one of the inhibited positions of the Feedrate Override switch be selected, the CNC will take the value corresponding to the nearest uninhibited position below it. If all of them are inhibited, the lowest will be taken (0%).

For example, if only positions 110% and 120% of the switch are allowed and position 50% is selected, the CNC will take a value of 0%.

9.6 Logic inputs of the PLC channel

To govern the axes managed by PLC.

/FEEDHOP (M5004) It is similar to general logic input /FEEDHOL (M5002), but for the PLC channel.

When the PLC sets this signal low, the CNC stops the axes (maintaining spindle rotation). When the signal returns to the high logic level, the movement of the PLC axes continues.

This input must always be defined in the PLC program.

/XFERINP (M5005) It is similar to general logic input /XFERINH (M5003), but for the PLC channel.

If the PLC sets this signal low, the CNC prevents the following block from starting, but finishes the one it is executing. When the signal returns to high logic level, the CNC continues to execute the program.

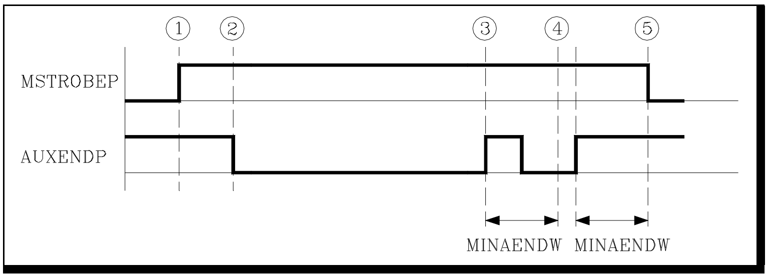
This input must always be defined in the PLC program.

AUXENDP (M5006) It is similar to general logic input AUXEND (M5016), but for the PLC channel.

This signal is used in the execution of auxiliary M functions to tell the CNC that the PLC is executing them.

It operates in the following way:

1. Once the block has been analyzed and after passing the corresponding values in variables “MBCD1-7”, the CNC will tell the PLC through the general logic output “MSTROBEP” that the required auxiliary function or functions must be executed.



2. When the PLC detects the activation of the “MSTROBEP” signal it must deactivate the general logic input “AUXENDP” to “tell” the CNC that the execution of the function has started.

3. The PLC will execute all the required auxiliary functions analyzing general logic output “MSTROBEP” and variables “MBCDP1” through “MBCDP7” (R565 through R571).

Once this has been executed the PLC must activate the general logic input “AUXENDP” to indicate to the CNC that the processing of the required functions was completed.

4. Once the general “AUXENDP” input is activated, the CNC will require that this signal be kept active for a time period greater than the value given to the g.m.p. “MINAENDW” (P30).

This way, erroneous interpretations of this signal by the CNC due to an improper PLC program logic are avoided .

5. Once the period of time MINAENDW has elapsed with the general input “AUXENDP” at a high logic level, the CNC will deactivate the general logic output "MSTROBEP" to tell the PLC that the execution of the required auxiliary function or functions has been completed.

9.7 General logic outputs

CNCREADY (M5500) The CNC activates and maintains this signal high if the autotest which the CNC makes when it is powered up has not detected any problem.

Should any hardware error be detected (RAM, over-temperature, etc.) this signal is set low.

Example: CNCREADY AND (other conditions) = O1

The emergency output O1 of the PLC must be normally high. Should any problem come up on CNC power-up (CNCREADY), emergency output O1 must be set low (0V).

START (M5501) The CNC sets this signal high in order to tell the PLC that the START key on the Front Panel has been pressed.

If the PLC program considers that there is nothing to prevent the part program from starting, it must set the general logic input CYSTART at a high logic level, thereby starting the execution of the program.

When the CNC detects an up flank (logic level change from low to high) at the CYSTART signal, it reset the START signal to low.

Example: START AND (rest of conditions) = CYSTART

When the cycle START key is pressed, the CNC activates the general logic output START.
The PLC must check that the rest of the conditions (hydraulic, safety devices, etc.) are met before setting the general input CYSTART high in order to start executing the program

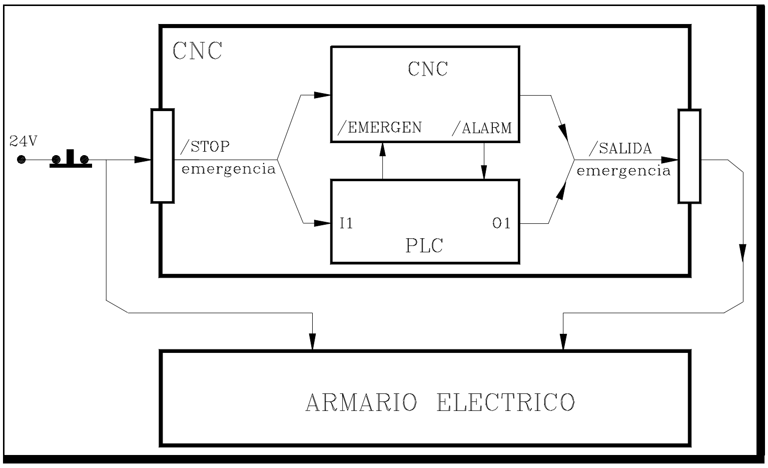
FHOUT (M5502) The CNC sets this signal high in order to tell the PLC that the execution of the program is stopped due to one of the following causes:

- Because the CONTROL PANEL STOP key has been pressed.
- Because the general logic input /STOP has been set low, even though later it has returned high.
- Because the general logic input /FEEDHOL is low.

RESETOUT (M5503) The CNC sets this signal high for 100 milliseconds, in order to tell the PLC that it is under initial conditions because the Reset key on the Front Panel has been pressed or because the general logic input RESETIN has been activated.

LOPEN (M5506) The CNC sets this signal high in order to tell the PLC that the positioning loop of the axes is open since an error has occurred.

/ALARM (M5507) The CNC sets this signal low in order to tell the PLC that an alarm or emergency condition has been detected. This signal will be set high once again, once the message from the CNC has been eliminated and the cause of the alarm has disappeared.



Likewise, while this signal is low, the CNC keeps the Emergency output (pin 2 of connector X2) active (low).

Example: /ALARM AND (other conditions) = O1

The emergency output O1 of the PLC must be normally high.

If an alarm or an emergency is detected at the CNC, the emergency output O1 must be set low (0V).

MANUAL (M5508) The CNC sets this signal high to tell the PLC that the JOG (Manual) Operating Mode is selected.

AUTOMAT (M5509) The CNC sets this signal high to tell the PLC that the Automatic Operating Mode is selected.

MDI (M5510) The CNC sets this signal high to tell the PLC that the MDI Mode (manual data input) is selected in one of the operating modes (JOG, Automatic, etc).

SBOUT (M5511) The CNC sets this signal high to tell the PLC that the Single Block Execution Mode is selected.

CUSTOM (M5512) It indicates to the CNC which work mode is currently selected:

CUSTOM = 0 M or T work mode

CUSTOM = 1 MC, MCO, TC or TCO work mode

When having 2 keyboards, this variable may be used in the PLC to:

- govern the keyboard switching board.
- know the source of the keys and inhibit the desired keys.

INCYCLE (M5515) The CNC sets this signal high while executing a block or moving an axis.

Once the execution of the program has been requested by the PLC to the CNC by means of the logic input CYSTART, the latter will indicate that it is being executed by setting the INCYCLE signal high.

This signal is maintained high until the CNC finishes the part program or when this is stopped by means of the STOP key on the CONTROL PANEL or the general logic input /STOP.

If the CNC is in the Single Block Execution Mode, the INCYCLE signal is set low as soon as the block execution is concluded.

If the CNC is in the JOG Mode, the INCYCLE signal is set low as soon as the position indicated has been reached.

If the CNC is in JOG mode and the axes are being jogged, the "INCYCLE" signal goes high while any of the jog keys are pressed.

RAPID (M5516) The CNC sets this signal high to tell the PLC that a rapid positioning (G00) is being executed.

TAPPING (M5517) The CNC sets this signal high to tell the PLC that a tapping canned cycle is being executed (G84).

THREAD (M5518) The CNC sets this signal high to tell the PLC that a threading block is being executed (G33).

PROBE (M5519) The CNC sets this signal high to tell the PLC that a probing movement is being executed (G75/G76).

ZERO (M5520) The CNC sets this signal high to tell the PLC that a machine reference search is being executed (G74).

RIGID (M5521) This output is only available on the Mill model. The CNC set this signal high to indicate to the PLC that a RIGID TAPPING operation (G84) is being performed. Mill model.

RETRAEND (M5522) The CNC sets this signal high to indicate to the “PLC” that while retracing is active all the possible blocks have been retraced.

For further information, see general input “Retrace (M5051)”.

CSS (M5523) The CNC sets this signal high to tell the PLC that the constant cutting speed function is selected (G96). Lathe Model.

SELECT0 (M5524)
SELECT1 (M5525)
SELECT2 (M5526)
SELECT3 (M5527)
SELECT4 (M5528)
SELECTOR (R564)

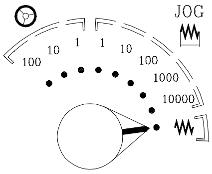
The CNC uses these signals to indicate to the PLC the position selected at each keyboard switch.

SELECTOR indicates the position currently selected.

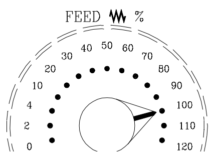
SELECT indicates the value applied by the CNC.

Usually, the two values coincide, except when a position has been selected which has been disabled with the KEYDIS4 input (R503).

If while being the 60% and 120% inhibited, the 100% position is selected, SELECTOR will show the selected position (100%) and SELECT will show the value being applied (50%)



Selected position Value applied	SELECTOR(3) SELECT3	SELECTOR(2) SELECT2	SELECTOR(1) SELECT1	SELECTOR(0) SELECT0
Handwheel x100	0	0	0	0
Handwheel x10	0	0	0	1
Handwheel x1	0	0	1	0
JOG 10000	0	0	1	1
JOG 1000	0	1	0	0
JOG 100	0	1	0	1
JOG 10	0	1	1	0
JOG 1	0	1	1	1
Continuous JOG	1	0	0	0



Selected position Value applied	SELECTOR(7) SELECT7	SELECTOR(6) SELECT6	SELECTOR(5) SELECT5	SELECTOR(4) SELECT4
Feed Override 0%	0	0	0	0
Feed Override 2%	0	0	0	1
Feed Override 4%	0	0	1	0
Feed Override 10%	0	0	1	1
Feed Override 20%	0	1	0	0
Feed Override 30%	0	1	0	1
Feed Override 40%	0	1	1	0
Feed Override 50%	0	1	1	1
Feed Override 60%	1	0	0	0
Feed Override 70%	1	0	0	1
Feed Override 80%	1	0	1	0
Feed Override 90%	1	0	1	1
Feed Override 100%	1	1	0	0
Feed Override 110%	1	1	0	1
Feed Override 120%	1	1	1	0

MSTROBE (M5532)

The CNC sets this signal high to tell the PLC that it must execute the auxiliary M function or functions which are indicated in Registers “MBCD1” to “MBCD7” (R550 to R556).

SSTROBE (M5533)

This signal will be used when operating a spindle with BCD coded S signal. s.m.p. SPDLTYPE (P0).

The CNC sets this signal high to tell the PLC that it must execute the auxiliary S function which is indicated in the Register “SBCD” (R557).

TSTROBE (M5534) The CNC sets this signal high to tell the PLC that it must execute the auxiliary S function which is indicated in the Register "TBCD" (R558).

In this register the CNC will tell the PLC the position of the magazine where the selected tool is.

If the g.m.p. RANDOMTC (P25) has been set so it is not a random magazine, the magazine pocket position coincides with the tool number.

T2STROBE (M5535) This register is used when a special tool change has been made, family code or 200 or with machining centers with a non-random tool magazine (g.m.p. RANDOMTC (P25).

The CNC sets this signal high to tell the PLC that it must execute a second auxiliary T function indicated in the Register "T2BCD" (R559).

In this register the CNC indicates to the PLC the position of the magazine in which the tool which was on the spindle must be placed.

S2MAIN (M5536) It indicates which spindle is controlled by the CNC. This selection can be made via part-program by means of functions G28 and G29.

If the CNC controls the main spindle S2MAIN is low

If the CNC controls the second spindle S2MAIN is high

ADVINPOS (M5537) It is used on punch presses having an eccentric cam as a punching system.

The CNC sets this signal high a specific time period before the axes reach position. This time is set by g.m.p. ANTIME (P69).

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

INTEREND (M5538)
INPOS (M5539) The CNC uses these two signals to let the PLC "know" that the theoretical interpolation between axes has been completed (INTEREND) and that all the axes involved in the interpolation are in position (INPOS).

The CNC sets the "INTEREND" signal high when the interpolation is ended being low while in execution.

When the CNC verifies that all the axes have been within the dead band (in position zone INPOSW P19) for a time period indicated in the a.m.p INPOTIME (P20), it will consider that all of them are in position and it will inform the PLC by setting the logic output "INPOS" high.

The logic output "INTEREND" can be used when it is required to activate mechanisms before the axes reach their position.

DM00 (M5547) The CNC sets this signal high to tell the PLC that the auxiliary function M00 (program stop) is programmed in the block being executed.

DM01 (M5546) The CNC sets this signal high to tell the PLC that the auxiliary function M01 (conditional stop) is programmed in the block being executed.

DM02 (M5545) The CNC sets this signal high to tell the PLC that the auxiliary function M02 (program end) is programmed in the block being executed.

DM03 (M5544) The CNC sets this signal high to tell the PLC that the spindle is turning clockwise or that the auxiliary function M03 is programmed in the block being executed.

DM04 (M5543) The CNC sets this signal high to tell the PLC that the spindle is turning counter-clockwise or that the auxiliary function M04 is programmed in the block being executed.

DM05 (M5542) The CNC sets this signal high to tell the PLC that the spindle is stopped or that the auxiliary function M05 is programmed in the block being executed.

DM06 (M5541) The CNC sets this signal high to tell the PLC that the spindle is stopped or that the auxiliary function M06 is programmed in the block being executed (tool change).

DM08 (M5540) The CNC sets this signal high to tell the PLC that the coolant output is activated or that the auxiliary function M08 is programmed in the block being executed.

DM09 (M5555) The CNC sets this signal high to tell the PLC that the coolant output is deactivated or that the auxiliary function M09 is programmed in the block being executed.

DM19 (M5554) The CNC sets this signal high to tell the PLC that it is working with spindle orientation or that the auxiliary function M19 is programmed in the block being executed.

DM30 (M5553) The CNC sets this signal high to tell the PLC that the auxiliary function M30 (program end) is programmed in the block being executed.

DM41 (M5552) The CNC sets this signal high to tell the PLC that the first spindle speed range is selected or that the auxiliary function M41 is programmed in the block being executed.

DM42 (M5551) The CNC sets this signal high to tell the PLC that the second spindle speed range is selected or that the auxiliary function M42 is programmed in the block being executed.

DM43 (M5550) The CNC sets this signal high to tell the PLC that the third spindle speed range is selected or that the auxiliary function M43 is programmed in the block being executed.

DM44 (M5549) The CNC sets this signal high to tell the PLC that the fourth spindle speed range is selected or that the auxiliary function M44 is programmed in the block being executed.

DM45 (M5548) The CNC sets this signal high to tell the PLC that the miscellaneous function M45 has been programmed and, therefore, the "auxiliary spindle or live tool" is active.

TANGACT (M5558) It indicates that the tangential control function, G45, is active.

SYNCPOSI (M5559) Indicates that the spindles are synchronized in position (set high). In other words, that the second spindle follows behind the main spindle at an angular distance set by G30.

It goes low when the following error between them exceeds the maximum allowed by s.m.p. "SYNPOSOF (P53)".

SYNSPEED (M5560) Indicates that the spindles are synchronized in speed (set high). In other words, that the second spindle turns at the same speed as the main spindle.

It goes low when the following error between them exceeds the maximum allowed by s.m.p. SYNSPEOF (P54).

SYNCHRON (M5561) Indicates that the G77S function is currently selected (spindle synchronization).

SERPLCAC (M5562) It is used in the data exchange, via Sercos between the CNC and the drives.

The CNC sets this signal high to "tell" the PLC that the requested change of parameter sets and gear ratios is in progress.

While this mark is on, no other change may be requested because the command would be lost.

9.8 Logic outputs of the axes

There are several groups of logic outputs (ENABLE, DIR, etc.) which refer to the possible axes of the machine by means of digits 1 through 7 (ENABLE2, DIR1,etc.)

These numbers have nothing to do with the values assigned to the g.m.p. "AXIS1" through "AXIS8".

These variables are numbered according to the logic order of the axes.

For example, if the CNC controls the X, Y, Z, B, C and U axis, the order will be: X, Y, Z, U, B, C and, therefore:

ENABLE1, DIR1, REFPOIN1, INPOS1	for the X axis:
ENABLE2, DIR2, REFPOIN2, INPOS2	for the Y axis:
ENABLE3, DIR3, REFPOIN3, INPOS3	for the Z axis:
ENABLE4, DIR4, REFPOIN4, INPOS4	for the U axis:
ENABLE1, DIR1, REFPOIN1, INPOS1	for the B axis
ENABLE2, DIR2, REFPOIN2, INPOS2	for the C axis:

ENABLE1 (M5600)
ENABLE2 (M5650)
ENABLE3 (M5700)
ENABLE4 (M5750)
ENABLE5 (M5800)
ENABLE6 (M5850)
ENABLE7 (M5900)

The CNC sets these signals at a high logic level to tell the PLC to allow the corresponding axis to move.

DIR1 (M5601)
DIR2 (M5651)
DIR3 (M5701)
DIR4 (M5751)
DIR5 (M5801)
DIR6 (M5851)
DIR7 (M5901)

The CNC uses these signals to tell the PLC in which direction the axes move.

If the signal is high this indicates that the corresponding axis moves in a negative direction.

If the signal is low this indicates that the corresponding axis moves in a positive direction.

REFPOIN1 (M5602)
REFPOIN2 (M5652)
REFPOIN3 (M5702)
REFPOIN4 (M5752)
REFPOIN5 (M5802)
REFPOIN6 (M5852)
REFPOIN7 (M5902)

The CNC sets these signals high to tell the PLC that the machine reference search has been made already.

It is set low

- after turning on the CNC.
- after doing a Shift Reset
- if a feedback alarm or a measurement loss occurs
- if feedback reading is lost via Sercos due to a power outage
- when modifying certain machine parameters, for example: number of axes.

In all these cases, a home search must be carried out so the signal is set back high.

DRSTAF1 (M5603) and DRSTAS1 (M5604)
DRSTAF2 (M5653) and DRSTAS2 (M5654)
DRSTAF3 (M5703) and DRSTAS3 (M5704)
DRSTAF4 (M5753) and DRSTAS4 (M5754)
DRSTAF5 (M5803) and DRSTAS5 (M5804
DRSTAF6 (M5853) and DRSTAS6 (M5854)
DRSTAF7 (M5903) and DRSTAS7 (M5904)

The CNC uses these signals when communicating with the drive via Sercos and they indicate the drive status.

	DRSTAF*	DRSTAS*
After actuating the main switch of the electrical cabinet, 24 Vdc are applied to the drive	0	0
The drive runs an internal test. If OK, it activates the “System OK” output From this moment on, the power supply must be applied power.	0	1
When there is power at the drive Bus, it is ready to output torque. To do that, activate the “Drive enable” and “Speed enable” inputs.	1	0
Once the “Drive enable” and “Speed enable” are activated, the drive is running properly.	1	1

When an internal error comes up in the drive, the DRSTAF* and DRSTAS* signals are set low.

ANT1 (M5606)
ANT2 (M5656)
ANT3 (M5706)
ANT4 (M5756)
ANT5 (M5806)
ANT6 (M5856)
ANT7 (M5906)

These signals are related to a.m.p. MINMOVE (P54).

If the axis move is smaller than the value indicated by this a.m.p. MINMOVE (P54), the corresponding axis logic output "ANT1 thru "ANT7" goes high.

INPOS1 (M5607)
INPOS2 (M5657)
INPOS3 (M5707)
INPOS4 (M5757)
INPOS5 (M5807)
INPOS6 (M5857)
INPOS7 (M5907)

The CNC sets these signals high to tell the PLC that the corresponding axis is in position.

There is also the general logic output INPOS in which the CNC indicates to the PLC that all the axes have reached their position.

9.9 Spindle logic outputs.

This CNC can handle 2 spindles: a main spindle and a second spindle. They both can be operative simultaneously, but only one can be controlled at a time. This selection can be made via part-program by means of functions G28 and G29.

ENABLES (M5950) Main spindle
ENABLES2 (M5975) Second spindle

This signal is used when working with the spindle in closed loop (M19). The CNC only considers the signals for the currently selected spindle.

The CNC sets this signal high to tell the PLC to allow the spindle to move.

DIRS (M5951) Main spindle
DIRS2 (M5976) Second spindle

This signal is used when working with the spindle in closed loop (M19). The CNC only considers the signals for the currently selected spindle.

The CNC uses this signal to tell the PLC in which direction the spindle is moving.

If the signal is at a high logic level, this indicates that the spindle moves in a negative direction.

If the signal is low, this indicates that the spindle moves in a positive direction.

REFPOINS (M5952) Main spindle
REFPOIS2 (M5977) Second spindle

This signal is used when working with the spindle in closed loop (M19). The CNC only considers the signals for the currently selected spindle.

The CNC sets this signal high to tell the PLC that the spindle reference point search has already been made.

This is set low when the CNC is powered up, after executing the Shift Reset sequence or a feedback alarm occurs due to loss of count, and every time a change is made from closed loop (M19) to open loop.

DRSTAFS (M5953) and DRSTASS (M5954) Main spindle
DRSTAFS2 (M5978) and DRSTASS2 (M579) Second spindle

The CNC uses these signals when communicating with the drive via Sercos and they indicate the drive status.

	DRSTAF*	DRSTAS*
After actuating the main switch of the electrical cabinet, 24 Vdc are applied to the drive	0	0
The drive runs an internal test. If OK, it activates the "System OK" output From this moment on, the power supply must be applied power.	0	1
When there is power at the drive Bus, it is ready to output torque. To do that, activate the "Drive enable" and "Speed enable" inputs.	1	0
Once the "Drive enable" and "Speed enable" are activated, the drive is running properly.	1	1

When an internal error comes up in the drive, the DRSTAF* and DRSTAS* signals are set low.

REVOK (M5956) Main spindle
REVOK2 (M5981) Second spindle

The CNC only considers the signals for the currently selected spindle.

When working with M03 and M04 the CNC sets this signal high to tell the PLC that the real spindle revolutions correspond to those programmed.

The CNC will activate this signal every time the real revolutions are within the range defined by s.m.p. "LOSPDLIM" and "UPSPDLIM".

When working with the spindle in closed loop (M19), the CNC sets this signal high if the spindle is stopped.

INPOSS (M5957) Main spindle
INPOSS2 (M5982) Second spindle

This signal is used when working with the spindle in closed loop (M19). The CNC only considers the signals for the currently selected spindle.

The CNC sets this signal high to tell the PLC that the spindle is in position.

9.10 Logic outputs of the auxiliary spindle

DRSTAFAS (M5557) and DRSTASAS (M5556)

The CNC uses these signals when communicating with the drive via Sercos and they indicate the drive status.

	DRSTAF*	DRSTAS*
After actuating the main switch of the electrical cabinet, 24 Vdc are applied to the drive	0	0
The drive runs an internal test. If OK, it activates the “System OK” output From this moment on, the power supply must be applied power.	0	1
When there is power at the drive Bus, it is ready to output torque. To do that, activate the “Drive enable” and “Speed enable” inputs.	1	0
Once the “Drive enable” and “Speed enable” are activated, the drive is running properly.	1	1

When an internal error comes up in the drive, the DRSTAF* and DRSTAS* signals are set low.

9.11 Logic outputs of key status

KEYDIS1 (R560)
KEYDIS2 (R561)
KEYBD3 (R562)
KEYBD4 (R563)

These registers indicate whether or not one of the keys on the keyboard or on the operator panel is pressed.

When one of these keys is pressed, the corresponding bit will be set high and it will return low when the key is released.

Register	Bit	Key inhibited	Register	Bit	Key inhibited
KEYBD1	0	F	KEYBD2	0	B
KEYBD1	1	L	KEYBD2	1	H
KEYBD1	2	Q	KEYBD2	2	N
KEYBD1	3	W	KEYBD2	3	S
KEYBD1	4	SHIFT	KEYBD2	4	Y
KEYBD1	5	9	KEYBD2	5	RESET
KEYBD1	6	6	KEYBD2	6	ESC
KEYBD1	7	3	KEYBD2	7	MAIN MENU
KEYBD1	8	E	KEYBD2	8	A
KEYBD1	9	K :	KEYBD2	9	G
KEYBD1	10	P	KEYBD2	10	M
KEYBD1	11	V	KEYBD2	11	R
KEYBD1	12	CAPS	KEYBD2	12	X
KEYBD1	13	8	KEYBD2	13	ENTER
KEYBD1	14	5	KEYBD2	14	HELP
KEYBD1	15	2	KEYBD2	15	
KEYBD1	16	D:	KEYBD2	16	.
KEYBD1	17	J	KEYBD2	17	0
KEYBD1	18	O	KEYBD2	18	-
KEYBD1	19	U	KEYBD2	19	+
KEYBD1	20	SP	KEYBD2	20	
KEYBD1	21	7	KEYBD2	21	
KEYBD1	22	4	KEYBD2	22	
KEYBD1	23	1	KEYBD2	23	
KEYBD1	24	C	KEYBD2	24	Next page
KEYBD1	25	I	KEYBD2	25	Previous page
KEYBD1	26	Ñ	KEYBD2	26	Up arrow
KEYBD1	27	T	KEYBD2	27	Down arrow
KEYBD1	28	Z	KEYBD2	28	Right arrow
KEYBD1	29	=	KEYBD2	29	Left arrow
KEYBD1	30	/	KEYBD2	30	CL
KEYBD1	31	*	KEYBD2	31	INS

The values returned by register KEYBD (R562) depend on the CNC model available (M or T model).

Register	Bit	Key inhibited M	Key inhibited T	Register	Bit	Key inhibited
KEYBD3	0	F1		KEYBD4	0	Handwheel x100
KEYBD3	1	F2		KEYBD4	1	Handwheel x10
KEYBD3	2	F3		KEYBD4	2	Handwheel x1
KEYBD3	3	F4		KEYBD4	3	JOG 10000
KEYBD3	4	F5		KEYBD4	4	JOG 1000
KEYBD3	5	F6		KEYBD4	5	JOG 100
KEYBD3	6	F7		KEYBD4	6	JOG 10
KEYBD3	7			KEYBD4	7	JOG 1
KEYBD3	8	X +	3rd axis +	KEYBD4	8	Feed Override 0%
KEYBD3	9	Y +		KEYBD4	9	Feed Override 2%
KEYBD3	10	Z +	X +	KEYBD4	10	Feed Override 4%
KEYBD3	11	4 +		KEYBD4	11	Feed Override 10%
KEYBD3	12	5 +	4th axis +	KEYBD4	12	Feed Override 20%
KEYBD3	13	Speed Override +		KEYBD4	13	Feed Override 30%
KEYBD3	14	Spindle clockwise		KEYBD4	14	Feed Override 40%
KEYBD3	15	Start		KEYBD4	15	Feed Override 50%
KEYBD3	16			KEYBD4	16	Feed Override 60%
KEYBD3	17		Z -	KEYBD4	17	Feed Override 70%
KEYBD3	18	Rapid		KEYBD4	18	Feed Override 80%
KEYBD3	19		Z +	KEYBD4	19	Feed Override 90%
KEYBD3	20			KEYBD4	20	Feed Override 100%
KEYBD3	21			KEYBD4	21	Feed Override 110%
KEYBD3	22	Spindle stop		KEYBD4	22	Feed Override 120%
KEYBD3	23			KEYBD4	23	
KEYBD3	24	X -	3rd axis -	KEYBD4	24	
KEYBD3	25	Y -		KEYBD4	25	
KEYBD3	26	Z -	X -	KEYBD4	26	
KEYBD3	27	4 -		KEYBD4	27	
KEYBD3	28	5 -	4th axis -	KEYBD4	28	
KEYBD3	29	Speed Override -		KEYBD4	29	
KEYBD3	30	Spindle counterclockwise		KEYBD4	30	
KEYBD3	31	Stop		KEYBD4	31	

User notes:

User notes:

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

ACCESS TO THE INTERNAL CNC VARIABLES

10

10. ACCESS TO THE INTERNAL CNC VARIABLES

The PLC provides two instructions (actions) which permit to read or modify the various internal variables of the CNC.

CNCRD: Allows reading access to the CNC internal variables. It is programmed as follows:

CNCRD (Variable, Register, Mark)

This instruction loads the selected PLC register with the contents of the indicated CNC variable.

If this instruction has been executed properly, the PLC will assign a value of “0” to the indicated “error detection” Mark and “1” if otherwise.

Example: CNCRD (FEED, R150, M200)

It loads the value of the feedrate selected at the CNC when working in G94 into the PLC register R150.

When requesting information about a nonexisting variable (i.e. the position value of a nonexisting axis), this instruction will not alter the contents of the register and it will set the selected error mark indicating that the variable does not exist.

CNCWR: Allows writing access to internal CNC variables. It is programmed as follows:

CNCWR (Register, Variable, Mark)

This instruction loads the contents of the indicated PLC register into the selected CNC variable.

If this instruction has been executed properly, the PLC will assign a value of “0” to the indicated “error detection” Mark and “1” if otherwise.

Example: CNCWR (R92, TIMER, M200)

Presets the timer enabled by the PLC with the value contained in Register R92.

When trying to modify the contents of a nonexisting variable or assign an improper value to it, the selected “error mark” will be set to “1” which will indicate that this instruction is incorrect.

When performing an improper reading or writing request, the PLC will continue the execution of the program unless interrupted by the programmer after having analyzed the “error” mark defined in the instruction.

The internal CNC variables which can be accessed by the PLC can be read-only or read-and-write variables.

Every internal CNC table (tools, tool offset, Zero offsets, etc.) has a mnemonic to identify the fields. Use one of the following formats to access a specific variable:

- The corresponding mnemonic followed by the element number of that table. Example (TOR3):

  CNCRD(TOR3,R100,M102);

  Loads register R100 with the R value of tool offset 3.

- The corresponding mnemonic and a register containing the element number of that table. Example (TOR R222):

  CNCRD (TOR R222, R100, M102)

  It assigns the R value of the offset indicated by Register R222 to register R100

The variables available at the CNC can be classified as follows:

- Variables associated with tools.
- Variables associated with zero offsets.
- Variables associated with machine parameters
- Variables associated with work zones
- Variables associated with feedrates
- Variables associated with coordinates
- Variables associated with the spindle
- Variables associated with local and global parameters
- Other variables

10.1 Variables associated with tools.

These variables are associated with the tool offset table, tool table and tool magazine table, so the values which are assigned to or read from these fields will comply with the formats established for these tables.

Tool offset table of the MILL model CNC:

R, L, I, K In the units set by g.m.p. "INCHES" (P8):
- If INCHES = 0, in 0.0001 mm. (±999999999)
- If INCHES = 1, in 0.00001 inch. (±393700787)
- If rotary axis, in 0.0001º (±999999999).

Tool offset table of the LATHE model CNC:

X, Z, R, I, K In the units set by g.m.p. "INCHES" (P8):
- If INCHES = 0, in 0.0001 mm. (±999999999)
- If INCHES = 1, in 0.00001 inch. (±393700787)
- If rotary axis, in 0.0001º (±999999999).

F Integer value between 0 and 9.

Tool table for Mill model CNC:

Tool offset number 0 ...NTOFFSET (maximum 255)

Family code
- If normal tool, 0 =< n < 200
- If special tool, 200 =< n < 255

Nominal life 0...65535 minutes or operations.

Real life 0...9999999 hundredths of a minute or 99999 operations.

Tool table for lathe model CNC:

Tool offset number 0 ...NTOFFSET (maximum 255)

Family code
- If normal tool, 0 =< n < 200
- If special tool, 200 =< n < 255

Nominal life 0...65535 minutes or operations.

Real life 0...9999999 hundredths of a minute or 99999 operations.

Cutter angle
- In 0.00010 units up to 359.9999º.

Cutter width

In the units set by g.m.p. "INCHES" (P8):
- If INCHES = 0, in 0.0001 mm. (±999999999)
- If INCHES = 1, in 0.00001 inch. (±393700787)

Cutting angle
- In 0.00010 units up to 359.9999º.

Tool magazine table:

Contents of each magazine position
Tool number 1 ...NTOOL (maximum 255)

0	Empty
-1	Cancelled

Tool position in magazine
Position number 1 ..NPOCKET (maximum 255)

0	On spindle
-1	Not found
-2	In change position

Read-only variables

TOOL Returns the active tool number

CNCRD(TOOL,R100,M100)
Loads register R100 with the number of the active tool

TOD Returns the active tool offset number

NXTOOL Returns the next tool number, which is selected but is awaiting the execution of M06 to be active.

NXTOD Returns the number of the tool offset corresponding to the next tool, which is selected but is awaiting the execution of M06 to be active.

TMZPn Returns the position occupied in the tool magazine by the indicated tool (n).

Read-and-write variables

TLFDn This variable allows the tool offset number of the indicated tool (n) to be read or modified in the tool table.

CNCRD(TFLD3,R100,M102)
Loads register R100 with the tool offset number of tool 3.

CNCWR(R101,TFLD3,M101)
Assigns the tool offset number indicated in register R101 to tool number 3.

TLFFn This variable allows the family code of the indicated tool (n) to be read or modified in the tool table.

TLFNn This variable allows the value assigned as the nominal life of the indicated tool (n) to be read or modified in the tool table.

TLFRn This variable allows the value corresponding to the real life of the indicated tool (n) to be read or modified in the tool table.

TMZTn This variable allows the contents of the indicated position (n) to be read or modified in the tool magazine table.

Read-and-write variables of the MILL model

TORn This variable allows the value assigned to the Radius of the indicated tool offset (n) in the tool offset table to be read or modified.

CNCRD(TOR3,R100,M102);
Loads register R100 with the R value of tool offset 3.

CNCWR(R101,TOR3,M101)
Assigns the value indicated in R101 to the R of tool offset 3.

TOLn This variable allows the value assigned to the Length of the indicated tool offset (n) to be read or modified in the tool offset table.

TOIn This variable allows the value assigned to the wear in radius (I) of the indicated tool offset (n) to be read or modified in the tool offset table.

TOKn This variable allows the value assigned to the wear in length (K) of the indicated tool offset (n) to be read or modified in the tool offset table.

Read-and-write variables of the LATHE model

TOXn This variable allows reading or modifying the length value along the X axis assigned to the indicated tool offset (n).

CNCRD (TOX3, R100, M102)
Loads R100 with the length value along X of the tool offset 3.

CNCWR (R101, TOX3, M101)
Assigns the value indicated in R101 to the length along X of the tool offset 3.

TOZn This variable allows reading or modifying the length value along the Z axis assigned to the indicated tool offset (n).

TOFn This variable allows reading or modifying the location code (F) of the indicated tool offset (n).

TORn This variable allows reading or modifying the radius R value of the indicated tool offset (n).

TOIn This variable allows reading or modifying the length wear value (I) along the X axis of the indicated tool offset (n).

TOKn This variable allows reading or modifying the length wear value (K) along the Z axis of the indicated tool offset (n).

NOSEAn: This variable allows reading or modifying the cutter angle assigned to the indicated tool (n) in the tool table.

NOSEWn This variable allows reading or modifying the cutter width assigned to the indicated tool (n) in the tool table.

CUTAn This variable allows reading or modifying the cutting angle assigned to the indicated tool (n) in the tool table.

10.2 Variables associated with zero offsets.

These variables are associated with the zero offset table, due to which the values that will be assigned to or read from these fields will comply with the formats established for this table.

The zero offsets, in addition to the additive offset indicated by the PLC, are G54, G55, G56, G57, G58 and G59.

The values of each axis are given in the units set by machine g.m.p. "INCHES".

If "INCHES" = 0, in 0.0001 mm. (±999999999)
If "INCHES" = 1, in 0.00001 inch. (±393700787)
If rotary axis, in 0.0001º (±999999999).

Although there are variables which refer to each axis, the CNC only allows those referring to the selected axes in the CNC. Thus, if the CNC controls axes X, Y, Z, U and B, it only allows the variables ORGX, ORGY, ORGZ,. ORGU and ORGB in the case of ORG(X-C).

Read-only variables

ORG(X-C) Returns the value of the active zero offset in the selected axis. The value of the additive offset indicated by the PLC is not included in this value.

Read-and-write variables

ORG(X-C)n This variable allows the value of the selected axis to be read or modified in the table corresponding to the indicated zero offset (n).

CNCRD(ORGX 55,R100,M102)
Loads register R100 with the X value of G55 in the zero offset table.

CNCWR(R101,ORGY 54,M101)
Assigns the value indicated in R101 to the Y value of G54 in the zero offset table.

PLCOF(X-C) This variable allows the value of the selected axis to be read or modified in the additive zero offset table indicated by the PLC.

10.3 Variables associated with machine parameters

Variables associated with machine parameters are read-only variables.

In order to become familiar with the values returned it is advisable to consult the chapter ***3. Machine parameters***, taking into account the following indications:

Values 1/0 correspond to the parameters which are defined with YES/NO, +/- and ON/OFF.

Values regarding position and feedrate values will be given in the units set by g.m.p. INCHES (P8):

If “INCHES” = 0, in 0.0001 mm. (±999999999)

If “INCHES” = 1, in 0.00001 inch. (±393700787)

Values regarding the spindle (when working in M19) and rotary axes will be given in 0.0001 degree units. Maximum. ±999999999

Read-only variables

MPGn Returns the value assigned to general machine parameter (n).

CNCRD (MPG 8,R100,M102)

Loads register R100 with the value of g.m.p INCHES (P8), If mm, R100 = 0; and if inch, R100 =1.

MP(X-C)n Returns the value assigned to the machine parameter (n) of the indicated axis (X-C).

CNCRD (MPY 1,R100,M102)

Loads register R100 with the value of machine parameter P1 (DFORMAT) for the Y axis which indicates the display format used for this axis.

MPSn Returns the value assigned to the indicated machine parameter (n) of the main spindle.

MPSSn Returns the value assigned to the indicated machine parameter (n) of the second spindle.

MPASn Returns the value of the indicated machine parameter (n) of the auxiliary spindle.

MPLCn Returns the value assigned to the indicated machine parameter (n) of the PLC.

10.4 Variables associated with work zones

The values of the limits are given in the units set by g.m.p. INCHES (P8):

If "INCHES" = 0, in 0.0001 mm. (±999999999)
If "INCHES" = 1, in 0.00001 inch. (±393700787)
If rotary axis, in 0.0001º (±999999999).

The status of the work zones are defined according to the following code:

0	Disabled
1	Enabled as no-entry zone.
2	Enabled as no-exit zone.

Read-and-write variables

FZONE
FZLO(X-C)
FZUP(X-C)

Variables associated with work zone 1

Status (FZONE) and upper (FZUP) and lower limits (FZLO) on the indicated axis (X-C).

SZONE
SZLO(X-C)
SZUP(X-C)

Variables associated with work zone 2

Status (SZONE) and upper (SZUP) and lower limits (SZLO) on the indicated axis (X-C).

TZONE
TZLO(X-C)
TZUP(X-C)

Variables associated with work zone 3

Status (TZONE) and upper (TZUP) and lower limits (TZLO) on the indicated axis (X-C).

FOZONE
FOZLO(X-C)
FOZUP(X-C)

Variables associated with work zone 4

Status (FOZONE) and upper (FOZUP) and lower limits (FOZLO) on the indicated axis (X-C).

FIZONE
FIZLO(X-C):
FIZUP(X-C)

Variables associated with work zone 5

Status (FIZONE) and upper (FIZUP) and lower limits (FIZLO) on the indicated axis (X-C).

The following example shows how it is possible to define as forbidden zone for the X axis the area between 0 and 100mm (1000000 tenths of microns).

```
condition   = MOV 0 R1         = CNCWR (R1,FZLOX,M1)
            = MOV 1000000 R1   = CNCWR (R1,FZUPX,M1)
            = MOV 1 R1         = CNCWR (R1,FZONE,M1)
```

10.5 Variables associated with feedrates

Variables associated with real feedrate

FREAL Real CNC feedrate It takes into account the Feedrate Override and the acc/dec of the machine. Read-only variable.

In 0.0001 mm/min. or 0.00001 inch/min units.

On Laser cutting machines, it is recommended to use this variable to make the power of the Laser proportional to the actual feedrate at all times.

Variables associated with function G94

FEED Feedrate active in G94 (ignoring the Feedrate Override). Read-only variable.

It may be set via program, PLC or DNC. The one indicated via DNC has the highest priority and the one indicated by program the lowest.

In 0.0001 mm/min. or 0.00001 inch/min units.

DNCF
PLCF
PRGF

Feedrate currently selected via DNC, PLC or program (PRGF). If they have a value of 0 it means that they are not selected.

PLCF is a read-write variable, the rest are read-only.

Variables associated with function G95

FPREV Feedrate active in G95 (ignoring the Feedrate Override). Read-only variable.

It may be set via program, PLC or DNC. The one indicated via DNC has the highest priority and the one indicated by program the lowest.

In 0.0001 mm/rev. or 0.00001 inch/rev units.

DNCFPR
PLCFPR
PRGFPR

Feedrate currently selected via DNC, PLC or program (PRGFPR). If they have a value of 0 it means that they are not selected.

PLCFPR is a read-write variable, the rest are read-only.

Variables associated with function G32

PRGFIN Feedrate selected by program, in 1/min. Read-only variable.

The FEED variable, associated with G94, indicates the resulting feedrate in mm/min or inches/min.

Variables associated with Feedrate Override

FRO Feedrate Override or % of feedrate. Read-only variable.

It may be set via program, PLC, DNC or from the front panel. Order of priority (from highest to lowest): by program, by DNC, by PLC and from the front panel switch.

It is given in integer values between 0 and "MAXFOVR" (maximum 255).

PRGFRO
DNCFRO
PLCFRO
CNCFRO

Feedrate Override (%) currently selected via program (PRGFRO), DNC, PLC or by front panel switch (CNCFRO) If they have a value of 0 it means that they are not selected.

PLCFRO is a read-write variable, the rest are read-only.

PLCCFR Feedrate Override (%) for the PLC channel. It can only be set via PLC using an integer between 0 and 255. Read-write variable.

10.6 Variables associated with coordinates

The values of each axis are given in the units set by machine g.m.p. INCHES (P8).

If “INCHES” = 0, in 0.0001 mm. (±999999999)
If “INCHES” = 1, in 0.00001 inch. (±393700787)
If rotary axis, in 0.0001º (±999999999).

Read-only variables

POS(X-C) Real position value of the selected axis referred to machine reference zero (home).

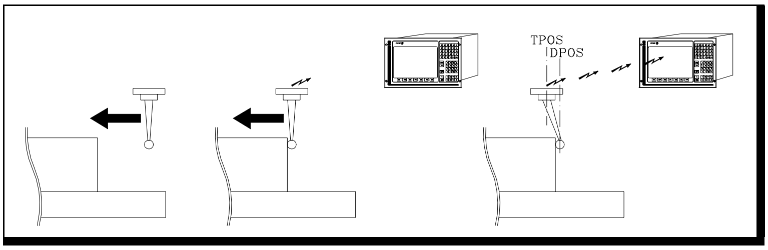
On the Lathe model, the coordinates of each axis are shown in either radius or diameter depending on the setting of a.m.p. DFORMAT (P1).

TPOS(X-C) Theoretical position value (real + following error) of the selected axis referred to machine reference zero (home).

On the Lathe model, the coordinates of each axis are shown in either radius or diameter depending on the setting of a.m.p. DFORMAT (P1).

DPOS(X-C) The CNC updates this variable whenever probing operations "G75, G76" and probing cycles "Probe" are carried out.

When the digital probe communicates with the CNC via infrared beams, there could be some delay (milliseconds) from the time the probe touches the part to the instant the CNC receives the probe signal.



Although the probe keeps moving until the CNC receives the probing signal, the CNC takes into account the value assigned to g.m.p. PRODEL (P106) and provides the following information (variables associated with coordinates):

TPOS Actual position of the probe when the CNC receives the probe signal.

DPOS Theoretical position of the probe when the probe touched the part.

FLWE(X-C) Following error of the indicated axis.

DEFLEX
DEFLEY
DEFLEZ These variables can only be used on the Mill model. They return the amount of deflection obtained at the time by the Renishaw probe SP2 on each axis X, Y, Z.

Read-and-write variables

DIST(X-C) Distance travelled by the indicated axis.

This value is accumulative and is very useful when it is required to perform an operation which depends on the distance travelled by the axes, for example:

The CNC will set this value to 0 when changing the software version or when a checksum error occurs.

LIMPL(X-C)
LIMMI(X-C) With these variables, it is possible to set a second travel limit for each axis: LIMPL for the upper limit and LIMMI for the lower one.

The PLC activates and deactivates these second limits through general logic input ACTLIM2 (M5052)

The second travel limit will be taken into account if the first one has been set using a.m.p. LIMIT+ (P5) and LIMIT- (P6).

10.7 Variables associated with electronic handwheels

Read-only variables

HANPF
HANPS
HANPT
HANPFO

They return the pulses of the first (HANPF), second (HANPS), third or fourth (HANPT) handwheel received since the CNC was turned on. It doesn't matter if the handwheel is connected to the connector or to the PLC inputs.

Read-and-write variables

HANFCT to set a different multiplying factor for each handwheel.

It must be used when using several electronic handwheels or when using a single handwheel but different multiplying factors (x1, x10, x100) are to be applied to each axis.

		C			B			A			W			V			U			Z			Y			X		
		c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a

Once the switch has been turned to one of the handwheel positions, the CNC checks this variable and, depending on the values assigned to each axis bit (c, b, a) it applies the multiplying factor selected for each one of them.

c	b	a	
0	0	0	The value indicated at the front panel or keyboard switch.
0	0	1	x1 factor
0	1	0	x10 factor
1	0	0	x100 factor

If there are more than one bit set to "1" for an axis, the least significant bit will be considered. Thus:

c	b	a	
1	1	1	x1 factor
1	1	0	x10 factor

Note: The screen always shows the value selected at the switch.

HBEVAR It must be used when having a Fagor HBE handwheel.

It indicates whether the HBE handwheel is enabled or not, the axis to be jogged and the multiplying factor to be applied (x1, x10, x100).

					C			B			A			W			V			U			Z			Y			X		
	*	^			c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a	c	b	a

(*) Indicates whether the HBE handwheel pulses will be taken into account or not in jog mode.

0 They are ignored.

1 They are taken into account.

(^) When the machine has a general handwheel and the individual handwheels (associated with an axis), it indicates which handwheel has priority when both are turned at the same time.

0 The individual handwheel has priority. The relevant axis ignores the pulses from the general handwheel, the rest of the axes don't.

1 The general handwheel has priority. It ignores the pulses from the individual handwheel.

The values assigned to the "c" "b" "a" bits indicate the axis to be moved and the multiplying factor currently selected.

c	b	a	
0	0	0	Not to be moved
0	0	1	x1 factor
0	1	0	x10 factor
1	0	0	x100 factor

If several axes are selected, the following order of priority is applied: X, Y, Z, U, V, W, A, B, C.

If there are more than one bit set to "1" for an axis, the least significant bit will be considered. Thus:

c	b	a	
1	1	1	x1 factor
1	1	0	x10 factor

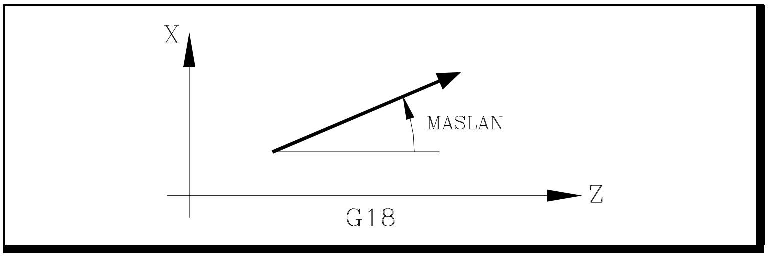
The HBE handwheel has priority. That is, regardless of the mode selected at the CNC switch (continuous or incremental JOG, handwheel), HBEVAR is set to other than "0", the CNC goes into handwheel mode.

It shows the selected axis in reverse video and the multiplying factor selected by the PLC. When the HBEVAR variable is set to "0", it shows the mode selected by the switch again.

See section *4.12 Fagor handwheels: HBA, HBE and LGB*

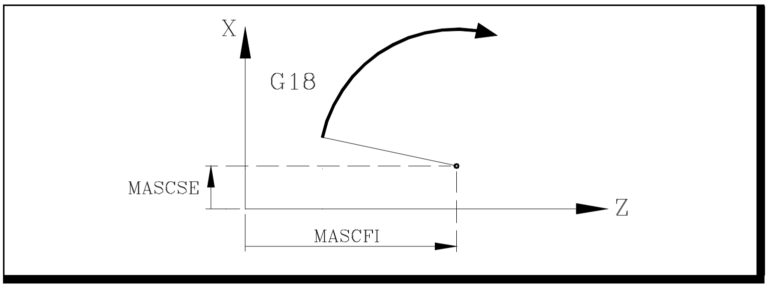
MASLAN Must be used when the Path Handwheel mode has been selected.

Indicates the angle of the linear path.



MASCFI
MASCSE Must be used when the Path Handwheel mode has been selected.

On circular paths (arcs), they indicate the center coordinates.



10.8 Variables associated with the main spindle

Variables associated with the real speed

SREAL Real spindle speed

This will be in 0.0001 rev./min. units.

Variables associated with spindle speed

SPEED Spindle turning speed which is selected at the CNC.

It may be set via program, PLC or DNC. The one indicated via DNC has the highest priority and the one indicated by program the lowest.

This will be in 0.0001 rev./min. units.

DNCS
PLCS
PRGS
Spindle speed selected via DNC, PLC or program (PRGS). If they have a value of 0 it means that they are not selected.

PLCS is a read-write variable, the rest are read-only.

Variables associated with constant cutting speed (Lathe model)

CSS Constant cutting speed selected at the CNC.

It may be set via program, PLC or DNC. The one indicated via DNC has the highest priority and the one indicated by program the lowest.

In the units set by g.m.p. "INCHES" (P8): If "INCHES"=0 in m/min and ifi "INCHES"=1 if feet/minute

DNCCSS
PLCCSS
PRGCSS
Constant cutting speed selected via DNC, PLC or program (PRGCSS). If they have a value of 0 it means that they are not selected.

PLCCSS is a read-write variable, the rest are read-only.

Variables associated with Spindle Override

SSO Spindle speed override (%) Read-only variable.

It may be set via program, PLC, DNC or from the front panel. Order of priority (from highest to lowest): by program, by DNC, by PLC and from the front panel.

It is given in integer values between 0 and “MAXFOVR” (maximum 255).

PRGSSO
DNCSSO
PLCSSO
CNCSSO
Spindle speed overrid (%) currently selected via program (PRGSSO), DNC , PLC or at the front panel switch (CNCSSO). If they have a value of 0 it means that they are not selected.

PLCSSO is a read-write variable, the rest are read-only.

Variables associated with spindle speed limits.

SLIMIT Spindle speed limit currently set.

It may be set via program, PLC or DNC. The one indicated via DNC has the highest priority and the one indicated by program the lowest.

This will be in 0.0001 rev./min. units. If it has a value of 0 it means that it is not defined.

DNCSL
PLCSL
PRGSL
Spindle speed limit selected via DNC, PLC or program (PRGSL). If they have a value of 0 it means that they are not selected.

PLCSL is a read-write variable, the rest are read-only.

Read-only variables associated with actual (real) position

POSS Real Spindle position.

Its value in 0.0001 degree units between ±999999999.

RPOSS Real spindle position in 360º module.

Its value in 0.0001 degree units between 0 and 360º.

Read-only variables associated with theoretical position

TPOSS Theoretical Spindle position (real + lag).

Its value in 0.0001 degree units between ±999999999.

RTPOSS Theoretical Spindle position (real + lag) in 360º.

Its value in 0.0001 degree units between 0 and 360º.

Read variables associated with the following error (axis lag)

FLWES Following error in 0.0001 of a degree.

SYNCER Amount of lag of the second spindle following the main spindle when they are both synchronized.

Its value in 0.0001 degree units between ±999999999.

If the error is smaller than the maximum allowed by s.m.p. SYNPOSOF (P53) for the main spindle and the general output SYNCPOSI (M5559) is set to "1".

10.9 Variables associated with the second spindle

Variables associated with the real speed

SSREAL Real spindle speed

This will be in 0.0001 rev./min. units.

Variables associated with spindle speed

SSPEED Spindle turning speed which is selected at the CNC.

It may be set via program, PLC or DNC. The one indicated via DNC has the highest priority and the one indicated by program the lowest.

This will be in 0.0001 rev./min. units.

SDNCS
SPLCS
SPRGS

Spindle speed selected via DNC, PLC or program (SPRGS). If they have a value of 0 it means that they are not selected.

SPLCS is a read-write variable, the rest are read-only.

Variables associated with constant cutting speed (Lathe model)

SCSS Constant cutting speed selected at the CNC.

It may be set via program, PLC or DNC. The one indicated via DNC has the highest priority and the one indicated by program the lowest.

In the units set by g.m.p. "INCHES" (P8): If "INCHES"=0 in m/min and ifi "INCHES"=1 if feet/minute

SDNCCS
SPLCCS
SPRGCS

Constant cutting speed selected via DNC, PLC or program (SPRGCS). If they have a value of 0 it means that they are not selected.

SPLCCS is a read-write variable, the rest are read-only.

Variables associated with Spindle Override

SSSO Spindle speed override (%) Read-only variable.

It may be set via program, PLC, DNC or from the front panel. Order of priority (from highest to lowest): by program, by DNC, by PLC and from the front panel.

It is given in integer values between 0 and “MAXFOVR” (maximum 255).

SPRGSO
SDNCSO
SPLCSO
SCNCSO

Spindle speed override (%) currently selected via program (SPRGSO), DNC , PLC or at the front panel switch (SCNCSO). If they have a value of 0 it means that they are not selected.

SPLCSO is a read-write variable, the rest are read-only.

Variables associated with spindle speed limits.

SSLIMI Spindle speed limit currently set.

It may be set via program, PLC or DNC. The one indicated via DNC has the highest priority and the one indicated by program the lowest.

This will be in 0.0001 rev./min. units. If it has a value of 0 it means that it is not defined.

SDNCSL
SPLCSL
SPRGSL

Spindle speed limit selected via DNC, PLC or program (SPRGSL). If they have a value of 0 it means that they are not selected.

SPLCSL is a read-write variable, the rest are read-only.

Read-only variables associated with actual (real) position

SPOSS Real Spindle position.

Its value in 0.0001 degree units between ±999999999.

SRPOSS Real spindle position in 360º module.

Its value in 0.0001 degree units between 0 and 360º.

Read-only variables associated with theoretical position

STPOSS Theoretical Spindle position (real + lag).

Its value in 0.0001 degree units between ±999999999.

SRTPOS Theoretical Spindle position (real + lag) in 360º.

Its value in 0.0001 degree units between 0 and 360º.

Read variables associated with the following error (axis lag)

SFLWES Following error in 0.0001 of a degree.

10.10 Variables associated with the live tool

Read-only variables

ASPROG It must be used inside the subroutine associated with function M45.

It returns the rpm programmed by M45 S

When programming M45 alone, the variable takes the value of "0".

The ASPROG variable is updated just before executing the M45 function so it is already updated when executing its associated subroutine.

LIVRPM must be used when operating in TC mode.

It returns the rpm selected by the user for the live tool when in TC mode.

10.11 Variables associated with local and global parameters

The CNC offers two types of general purpose variables, local parameters P0 through P25 and global parameters P100 through P299.

It is possible to assign local parameters to more than one subroutine. Up to 6 nesting levels of the local parameters are possible within the 15 nesting levels for the subroutines.

Therefore, each time a local parameter must be referred to, it is necessary to indicate its current nesting level.

Local and global parameters may be assigned a value within +2147483647.

When reading the value of one of these parameters by means of functions GUP and LUP the obtained value will always be an integer (dropping the decimals if it has them). If the parameter value is greater than +2147483647 the variable will return this maximum value.

Read-and-write variables

GUP n This variable allows reading or modifying the indicated global parameter (n) (P100-P299).

CNCRD (GUP 155, R100, M102)

Loads register R100 with the value of global parameter P155.

CNCWR (R101, GUP 155, M101)

Assigns the value in register R101 to global parameter P155.

LUP a b: This variable allows reading or modifying the indicated local parameter (b) (P0-P25) corresponding to a nesting level (a).

CNCRD (LUP 3 15, R100, M102)

Loads register R100 with the value of local parameter P15 corresponding to nesting level 3.

CNCWR (R101, LUP 2 15, M101)

Assigns the value of register R101 to local parameter P15 of level 2.

10.12 Sercos variables

They are used in the data exchange, via Sercos, between the CNC and the drives.

Write variables

SETGEX, SETGY, SETGZ	**for the axes**
SETGES	**for the main spindle**
SSETGS	**for the second spindle**

The drive may have up to 8 gear ratios (0 through 7). Sercos identifier 218, GearRatioPreselection.

It may also have up to 8 parameter sets (0 through 7). Sercos identifier 217, ParameterSetPreselection.

With these variables the work range or gear ratio and the parameter set of each drive may be modified.

The 4 least significant bits of these variables indicate the gear ratio and the other 4 the parameter set to be selected.

Since it takes the drive some time to change the parameter set and the gear ratios, mark SERPLCAC (M5562) will remain active from when the change is requested until the drive assumes the new values.

No other SETGE* change may be requested while this mark is active because the command would be lost.

10.13 Software & hardware configuration variables

Read-only variables

HARCON Indicates, with bits, the CNC hardware configuration.

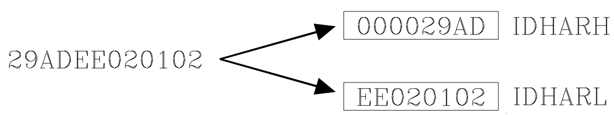
The bit will be set to "1" when its corresponding configuration is available.

bit		
0	Turbo board	
4,3,2,1	0100	8040 model
5	Sercos (digital model)	
6	Reserved	
9,8,7	000 001 010 011	Expansion board missing "Feedback + I/O" expansion board Feedback-only expansion board I/O-only expansion board
10	Axis board with 12-bit (=0) or 16-bit (=1) Digital/Analog converter.	
12, 11	Reserved	
14, 13	Reserved	
15	It has CAN (digital module)	
18,17,16	Keyboard type (technical service department)	
20,19	CPU type (technical service department)	
23,22,21	000 010 110 111	Memkey Card (4M) Memkey Card (24M) Memkey Card (512M) Memkey Card (2M)

IDHARH
IDHARL They return, in BCD code, the hardware identification number corresponding to the "Memkey Card". It is the number appearing on the software diagnosis screen.

Since the identification number has 12 digits, the IDHARL variable shows the 8 least significant bits and the IDHARH the 4 most significant bits.

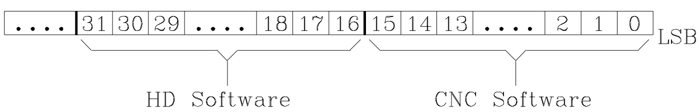
Example:



SOFCON They return the software version numbers for the CNC and the Hard Disk.

Bits 15-0 return the CNC software version (4 digits)

Bits 31-16 return the software version of the Hard Disk (HD) (4 digits)



For example, SOFCON 01010311 indicates

Hard Disk (HD) software version

CNC software version 0311

10.14 Variables associated with telediagnosis

Read-only variables

HARSWA
HARSWB

They return, in 4 bits, the central unit configuration, a "1" when present and a "0" when missing.

HARSWA								
bits	31 - 28	27 - 24	23 - 20	19 - 16	15 - 12	11 - 8	7 - 4	3 - 0
board	Large sercos	I/O 4	I/O 3	I/O 2	I/O 1	Axes:	Turbo	CPU

HARSWB								
bits	31 - 28	27 - 24	23 - 20	19 - 16	15 - 12	11 - 8	7 - 4	3 - 0
board						Small sercos		HD

The Sercos board may be large (occupying the whole module) or small installed in the CPU module ("1" If connected to COM1 or "2" if connected to COM2).

HARTST It returns the result of the hardware test. The data comes at the least significant bits with a "1" if it failed and with a "0" if OK or if the relevant board is missing.

<table>
<tr><td>bit 13</td><td>bit 12</td><td>bit 11</td><td>bit 10</td><td>bit 9</td><td>bit 8</td><td>bit7</td><td>bit 6</td><td>bit 5</td><td>bit 4</td><td>bit 3</td><td>bit 2</td><td>bit 1</td><td>bit 0</td></tr>
<tr><td rowspan="2">Inside temperature</td><td colspan="5">Board voltage</td><td colspan="8">Supply voltages</td></tr>
<tr><td>I/O 3</td><td>I/O 2</td><td>I/O 1</td><td></td><td>Axes :</td><td>+3.3 V</td><td>GND</td><td>GNDA</td><td>-15 V</td><td>+15 V</td><td>Batt ery</td><td>-5 V</td><td>+5 V</td></tr>
</table>

MEMTST It returns the result of the memory test. Each data uses 4 bits that will be at "1" if the test is OK and will have a value other than "1" if there is an error.

bits	30	....	19 - 16	15 - 12	11 - 8	7 - 4	3 - 0
Test	Test status	....	Caché	Sdram	HD	Flash	Ram

Bit 30 stays at "1" during the test.

NODE It returns the number of the node assigned to the CNC in the sercos ring.

VCHECK It returns the checksum of the code for the software version installed. It is the value appearing on the code test.

10.15 Operating-mode related variables

Read-only variables related to the standard mode

OPMODE It returns the code corresponding to the selected operating mode.

0	Main menu.
10	Automatic execution.
11	Single block execution.
12	MDI in EXECUTION
13	Tool inspection
20	Theoretical path movement simulation
21	G functions simulation
22	G, M, S and T functions simulation
23	Simulation with movement in the main plane
24	Simulation with rapid movement
30	Normal editing
31	User editing
32	TEACH-IN editing
33	Interactive editor
34	Profile editor
40	Movement in continuous JOG
41	Movement in incremental JOG
42	Movement with electronic handwheel
43	HOME search in JOG
44	Position preset in JOG
45	Tool calibration
46	MDI in JOG
47	JOG user operation
50	Zero offset table
51	Tool Offset table
52	Tool table
53	Tool magazine table
54	Global parameter table
55	Local parameter table
60	Utilities
70	DNC status
71	CNC status
80	Editing PLC files
81	Compiling PLC program
82	PLC monitoring
83	Active PLC messages
84	Active PLC pages (screens)
85	Save PLC program
86	Restore PLC program
87	PLC usage maps
88	PLC statistics
90	Set-up
100	General machine parameter table
101	Axis machine parameter tables
102	Spindle machine parameter tables
103	Serial port machine parameter tables
104	PLC machine parameter table
105	M function table
106	Leadscrew and cross compensation table
107	Machine parameter table for Ethernet
110	Diagnosis: configuration

111	Diagnosis: hardware test
112	Diagnosis: RAM memory test
113	Diagnosis: Flash memory test.
114	User diagnosis
115	Hard disk diagnosis (HD)
116	Circle geometry test

Read-only variables related to the Conversational mode (MC, TC, MCO, TCO) and Configurable mode M(SHIFT-ESC)-T(SHIFT-ESC).

In these work modes, it is recommended to use variables OPMODA, OPMODB and OPMODC. The OPMODE variable is generic and contains different values to those of the standard mode.

OPMODE It returns the code corresponding to the selected operating mode.

0	CNC starting up
10	In Execution mode. Either executing or waiting for the CYCLE START key (Start key icon at the top)
21	In Graphic simulation mode
30	Cycle editing
40	Jog mode (Standard screen).
45	Tool calibration mode
60	Part management. PPROG mode

OPMODA Indicates the operating mode currently selected when working with the main channel.

Use the OPMODE variable to know at any time the selected operating mode (main channel, user channel, PLC channel).

This information is given at the least significant bits with a "1" when active and with a "0" when not active or when it is not available in the current version.

bit 0	Program in execution.
bit 1	Program in simulation.
bit 2	Block in execution via MDI, JOG
bit 3	Repositioning in progress.
bit 4	Program interrupted, by CYCLE STOP
bit 5	MDI, JOG Block interrupted
bit 6	Repositioning interrupted
bit 7	In tool inspection
bit 8	Block in execution via CNCEX1
bit 9	Block via CNCEX1 interrupted
bit 10	CNC ready to accept JOG movements: jog, handwheel, teach-in, inspection.
bit 11	CNC ready to receive the CYCLE START command: execution, simulation and MDI modes.
bit 12	The CNC is not ready to execute anything involving axis or spindle movement.

OPMODB Indicates the type of simulation currently selected. This information is given at the least significant bits indicating with a "1" the one currently selected.

bit 0	Theoretical path
bit 1	G functions
bit 2	G M S T functions
bit 3	Main plane
bit 5	Rapid (S=0)

OPMODC Indicates the axes selected by Handwheel. This information is given at the least significant bits indicating with a "1" the one currently selected.

bit 8	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
		Axis 7	Axis 6	Axis 5	Axis 4	Axis 3	Axis 2	Axis 1

The axis name corresponds to the number according to the programming order for them.

Example:

If the CNC controls axes X, Y, Z, U, B, C. Axis 1=X, Axis2=Y, Axis3=Z, Axis4=U, Axis5=B, Axis6=C.

10.16 Other variables

Read-only variables

NBTOOL Indicates the tool number being managed.

Example:

There is a manual tool changer. Tool T1 is currently selected and the operator requests tool T5.

The subroutine associated with the tools may contain the following instructions:

(P103 = NBTOOL)
(MSG "SELECT T?P103 AND PRESS CYCLE START")

Instruction (P103 = NBTOOL) assigns the number of the tool currently being managed to parameter P103. Therefore, P103=5

The message displayed by the CNC will be ""SELECT T5 AND PRESS CYCLE START".

PRGN Returns the program number being executed. Should none be selected, a value of -1 is returned.

BLKN Returns the label number of the block being executed or that of the last block executed. If none, it returns -1.

GGSA It returns the status of functions G00 through G24. The status of each one of these functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G24	G23	G22	G21	G20	....	G04	G03	G02	G01	G00

CNCRD (GGSA, R100, M102)

Loads register R100 with the status of functions G00 through G24.

GGSB It returns the status of functions G25 through G49. The status of each one of these functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G49	G48	G47	G46	G45	....	G29	G28	G27	G26	G25

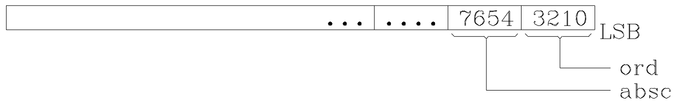
GGSC It returns the status of functions G50 through G74. The status of each one of these functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G74	G73	G72	G71	G70	....	G54	G53	G52	G51	G50

GGSD It returns the status of functions G75 through G99. The status of each one of these functions will be given in the 25 least significant bits and it will be indicated by a 1 when active and a 0 when not active or when not available in the current software version.

G99	G98	G97	G96	G95	....	G79	G78	G77	G76	G75

PLANE Returns data on the abscissa axis (bits 4 to 7) and the ordinate axis (bits 0 to 3) of the active plane in 32 bits and in binary.



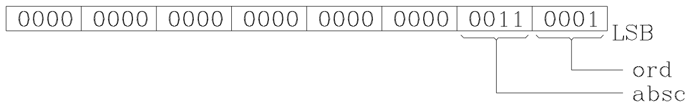
The axes are coded in 4 bits and indicate the axis number according to the programming order.

Example:

If the CNC controls the X,Y,Z,U,B,C axes and the ZX plane (G18) is selected.

CNCRD (PLANE, R100, M102);

Loads register R100 with Hexadecimal value $31.



Abscissa axis	= 3 (0011)	⇒	Z axis
Ordinate axis	= 1 (0001)	⇒	X axis

LONGAX This variable can only be used on the Mill model. It returns the number according to the programming order corresponding to the longitudinal axis. This will be the one selected with the G15 function and by default the axis perpendicular to the active plane, if this is XY, ZX or YZ.

Example:

If the CNC controls the X, Y, Z, U, B, C axes and the U axis is selected.

CNCRD (LONGAX, R100, M102);

Loads register R100 with a value of 4.

MIRROR Returns in the least significant bits of the 32-bit word, the status of the mirror image of each axis, 1 in the case of being active and 0 if not.

bit 8	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
		Axis 7	Axis 6	Axis 5	Axis 4	Axis 3	Axis 2	Axis 1

The axis name corresponds to the number according to the programming order for them.

Example:

If the CNC controls axes X, Y, Z, U, B, C. Axis 1=X, Axis2=Y, Axis3=Z, Axis4=U, Axis5=B, Axis6=C.

SCALE It returns the general scaling factor being applied. It will be multiplied by 10000.

SCALE(X-C) Returns the specific scaling factor of the indicated axis (X-C). They will be multiplied by 10000.

ORGROT This variable can only be used on the Mill model. It returns the rotation angle of the coordinate system currently selected with G73. Its value in 0.0001 degree units.

PRBST Returns probe status.

0 The probe is not touching the part.
1 The probe is touching the part.

CLOCK Returns the time in seconds indicated by the system clock. Possible values 0...4294967295

TIME Returns the time in hours-minutes-seconds format.

CNCRD (TIME, R100, M102)

Loads register R100 hh-mm-ss. For example if the time is: 34sec. R100 = 182234.

DATE Returns the date in year-month-day format.

CNCRD (DATE, R100, M102)

Loads register R100 with year-month-day. For example: if the date is April 25th 1992, R100 = 920425.

CYTIME Returns in hundredths of a second the time elapsed in making the part not counting the time period when the program execution might have been stopped. Possible values 0...4294967295

The CNC will consider the execution of the program finished after executing the last block of the program or after executing a block containing an M02 or M30 miscellaneous function.

FIRST Indicates whether it is the first time that a program has been run or not. It returns a value of 1 if it is the first time and 0 if not.

A first-time execution is considered as being one which is done:

- After turning on the CNC.
- After pressing the “Shift-Reset” keys.
- Every time a new program is selected.

ANAI(n) It returns the value of the indicated analog input (n) and one of the CNC’s analog inputs (1-8) may be selected. Its value is given in 0.0001V units within ±5 V.

CNCERR Returns the Error code active at the CNC.

If none, it returns “0”.

DNCERR Returns the Error code generated via DNC. If none, it returns “0”.

AXICOM Returns in the 3 least significant bits the axis pairs switched with function G28.

			Pair 3		Pair 2		Pair 2
			Axis 2	Axis 1	Axis 2	Axis 1	Axis 2	Axis 1

The axes are coded in 4 bits and indicate the axis number (1 to 7) according to their programming order.

If the CNC controls the X, Y, Z, B, C axes and G28BC has been programmed, the AXICOM variable will show:

			Pair 3		Pair 2		Pair 2	
			0000	0000	0000	0000	0101	0100

TANGAN This variable is associated with the tangential control function, G45. It indicates the programmed angular position.

Read-and-write variables

TIMER This variable allows reading or modifying the time, in seconds, indicated by the clock enabled by the PLC. Possible values 0...4294967295

The CNC will set this value to 0 when changing the software version or when a checksum error occurs.

PARTC The CNC has a part counter whose count increases every time M30 or M02 is executed and this variable allows its value to be read or modified. This value will be between 0 and 4294967295

The CNC will set this value to 0 when changing the software version or when a checksum error occurs.

KEY This variable allows reading the last accepted keystroke or simulating the CNC keyboard assigning the desired key code to it.

CNCRD (KEY, R100, M102)

Loads register R100 with the value of the last key accepted.

To simulate the CNC keyboard from the PLC, follow these steps:

R111=1 R110=0

CNCWR (R111, KEYSRC, M101)

Indicates to the CNC that only keystrokes coming from the PLC must be processed (CNC keyboard inhibited).

CNCWR (R101, KEY, M101)

Indicates to the CNC that the key corresponding to the code contained in R101 has been pressed.

CNCWR (R110, KEYSRC, M101)

Process only keystrokes coming from the CNC.

KEYSRC This variable allows reading or modifying the source of keystrokes, possible values being:

0	Keyboard
1	PLC
2	DNC

The CNC only allows modification of this variable if it is set to "0" or "1".

Once the keystroke simulation is finished, it is advisable to re-enable the CNC keyboard in order to be able to access the various operating modes of the CNC.

The CNC will assign a value of 0 to this variable on power-up and after pressing SHIFT, RESET.

ANAO(n) This variable allows the required analog output (n) to be read or modified. The value assigned will be expressed in 0.0001 volt units and within +/-10 Volts.

The analog outputs which are free among the eight (1..8) available at the CNC may be modified, the corresponding error being displayed if an attempt is made to write in one which is occupied.

User notes:

8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

AXES CONTROLLED FROM THE PLC

11

11. AXES CONTROLLED FROM THE PLC

The PLC offers actions CNCEX and CNCEX1 to send commands to the CNC.

CNCEX	sends commands to the CNC so it executes movements on one or several axes.
CNCEX1	sends commands to the CNC so it executes any kind of block.

The CNCEX action is executed through the execution channel of the PLC.

The CNCEX1 action is executed via main channel of the CNC and as long as the JOG keyboard is enabled. Its execution can be interrupted by pressing [CYCLE STOP] or even canceled by pressing [RESET].

If a CNCEX1 action is received when the JOG keyboard is disabled, the CNC ignores this command.

The programming format for these actions is:

CNCEX (ASCII block, Mark)
CNCEX1 (ASCII block, Mark)

By means of these actions, the PLC sends to the CNC the command indicated in the "ASCII Block" to be executed.

If the "ASCII Block" has been accepted by the CNC, the PLC will set the indicated mark to "0" or to "1" if otherwise. The CNC only indicates that the "ASCII Block" has been accepted. It is up to the operator to verify whether the command has actually been executed by the CNC or not.

Examples:

CNCEX (G1 U125 V300 F500, M200)

Sends to the CNC the command "G1 U125 V300 F500" so it executes a linear interpolation of the U and V axes at a feedrate of F500 being the end point: U125 V300.

CNCEX1 (T5, M200)

Selects the tool T5 in the tool changer.

Example of how to use action CNCEX1 when using a tool changer controlled by the PLC.

The T executed last at the CNC is T1. Therefore, it is the active T.
A new tool is selected, for example T5.

- If carried out by means of action CNCEX1, the change is made by the CNC and it assumes T5 as the new active tool.
- If not carried out by means of action CNCEX1, the change is made by the PLC and T1 remains as the active tool.

Then, an operation programmed with T1 is carried out.

- If the change was made with action CNCEX1, the CNC detects the tool change (from T5 to T1) and carries out the change.
- If the change was not made with action CNCEX1, the CNC does not detect the tool change (T1), it does not make the change and carries out the operation with the selected tool T5 with the problems this may cause.

11.1 PLC execution channel

The CNC offers a parallel execution channel to execute commands received from the PLC. This channel will have its own history and it permits the execution of blocks programmed from the PLC regardless of the operating mode being selected at the CNC at the time.

When the CNC receives a command from the PLC and it is executing another command received earlier, it will store the new one in an internal buffer. This new command will be executed after finishing the one being executed.

The internal buffer can store up to 3 commands received from the PLC besides the one currently in execution.

11.1.1 Considerations

Set-up

The a.m.p. "AXISTYPE" of each axis of the machine must be set properly indicating whether that axis is controlled by the CNC or from the PLC.

The axes of the PLC channel can only be governed from the PLC.

They may be edited and part-programs may be generated with axes of the PLC channel. This permits generating part-programs or subroutines associated with the PLC channel.

It issues an error message when trying to execute, from the CNC channel, a program block that includes a PLC axis.

When all the axes of the machine are set to be governed from the CNC, with the CNCEX action only blocks programmed in high level language may be executed through the PLC execution channel.

Axis control

To govern axes managed by PLC, use the following marks associated with “Feed-hold” and “Transfer Inhibit”:

/FEEDHOP	(M5004)	Similar to the /FEEDHOL signal
FHOUTP	(M5504)	Similar to the /FHOUT signal
/XFERINP	(M5005)	Similar to the /XFERINH signal

Auxiliary (miscellaneous) M functions

To control the M functions managed by the PLC, the following marks and registers are generated:

MBCDP1 through MBCDP7 (R565 through R571)
similar to signals MBCD1 through MBCD7

AUXENDP (M5006)
Similar to the AUXEND signal

MSTROBEP (M5505)
Similar to the MSTROBE signal

Data transfer

If when executing at the PLC the action "CNCEX (ASCII Block, Mark)", the CNC detects that the contents of the ASCII block being received is erroneous, it will set the indicated Mark to "1". The PLC program will keep executing while it is up to the programmer to check whether the function was executed correctly or not.

The CNC considers the contents of the ASCII block incorrect in the following instances:

- When the syntax is incorrect.
- When programming a not-permitted preparatory function (G code).
- When programming an auxiliary function M, S, T or tool offset D.
- When programming a high level language block.
- When the axis to be moved cannot be controlled from the PLC.
- When the internal buffer for PLC command storage is full.

Errors during execution

When the CNC detects an execution error in one of the two execution channels (for example, travel limit overrun), it will show the corresponding error code.

If it must also stop the movement of the axes and the spindle rotation, the CNC will stop the movement of all the axes regardless of whether they are controlled from the CNC or the PLC.

Also, if the detected error stops the program execution, the CNC will stop the execution of both channels and each one of them will act as follows:

CNC channel

Once the cause of the error has been removed, select again the execution or simulation mode and continue with the program execution.

PLC channel

The PLC program does not stop and continues running.

The commands sent by means of action "CNCEX" will not be executed until removing the cause of the error.

Once the cause of the error removed, the CNC will execute all the new commands sent by the PLC.

To know from the PLC program whether any CNC error is active, this information can be requested by accessing the internal CNC variable "CNCERR". This variable indicates the error number being active at the CNC and if none is active, it returns a 0 value.

11.1.2 Blocks which can be executed from the PLC

It is possible to execute blocks that contain G codes, axis position values, feedrates, M functions and high level language programming.

Auxiliary functions S, T and D cannot be programmed.

The ASCII block to be sent to the CNC by means of the action CNCEX to be executed in the PLC execution channel must be written in the CNC's own programming format.

Preparatory functions

The preparatory functions which can be used in the PLC execution channel are the following:

G00	Rapid travers
G01	Linear interpolation
G02	Clockwise circular (helical) interpolation
G03	Counterclockwise circular (helical) interpolation
G04	Interrupt block preparation of the PLC channel.
G04K	Dwell
G05	Round corner
G06	Arc center in absolute coordinates
G07	Square corner
G09	Arc defined by three points
G16	Main plane selection by two addresses
G32	Feedrate "F" as an inverted function of time.
G50	Controlled corner rounding
G52	Movement until making contact
G53	Programming with respect to machine reference zero (home)
G70	Inch programming
G71	Metric programming
G74	Home search
G75	Probing move until touching
G76	Probing move while touching
G90	Absolute programming
G91	Incremental programming
G92	Preset
G93	Polar origin preset
G94	Feedrate in millimeters (inches) per minute
G95	Feedrate in millimeters (inches) per revolution.

All these functions must be programmed as described in the programming manual.

Move the axes

Only those axes set by means of a.m.p. AXISTYPE (P0) for each axis as to be controlled by the PLC can be mentioned.

The position values of these axes, which can be either linear or rotary, can be programmed in either Cartesian or polar coordinates.

These coordinates can also be defined via parametric programming using any global arithmetic parameters (P100 thru P299)

When using parametric programming, it is recommended to previously assign a value to the corresponding global parameter by means of the instruction: CNCWR.

Example:

...... = MOV 150 R1
Assigns the value of 150 to register R1

...... = CNCWR (R1, GUP200, M100)
Assigns the value of R1 to parameter P200, (P200=150)

...... = CNCEX (G90 G1 U P200, M100)
Requests the CNC to execute the command: G90 G1 U150.
The U axis will go to position 150.

To govern axes managed by PLC, use the following marks associated with “Feed-hold” and “Transfer Inhibit”:

/FEEDHOP	(M5004)	Similar to the /FEEDHOL signal
FHOUTP	(M5504)	Similar to the /FHOUT signal
/XFERINP	(M5005)	Similar to the /XFERINH signal

Axis feedrate

The programming format for the axis feedrate (F5.5) depends on the function (G94 or G95) and on the work units selected for this execution channel.

- If G94, in mm/min. or inches/min.
- If G95, in mm/rev or inches/rev.
  It must be borne in mind that this feedrate depends on the actual spindle rpm which is in the main execution channel.

If the move corresponds to a rotary axis, the CNC will assume the feedrate to be programmed in degrees/min.

Modify the feedrate (override)

The PLCCFR variable sets, from the PLC, the % of feedrate selected by the execution channel of the PLC.

G.m.p. MAXFOVR (P18) limits the value of the percentage applied to both execution channels (main and PLC).

The OVRCAN (M5020) mark sets the feedrate override of the main channel to 100%. It does not affect the feedrate override of the PLC channel

Same as with the main channel, the following movements have a special treatment:

- When searching home, the value of PLCCFR is ignored.
- In G0, it considers the value of g.m.p. "RAPIDOVR (P17)"
  If "P17=NO" always 100%, except if PLCCFR=0. In that case, the movement stops.
  If "P17=YES" considers PLCCFR, but it limits its value to 100%.
- In G1, G2, G3 it is always applied except when operating at maximum feedrate (F0).
- In that case, it is limited to 100%. In G75, G76, it is only applied when g.m.p. FOVRG75 (P126) = YES.

Blocks programmed in high-level language

The high-level statements that can be used in the PLC execution channel are:

(IF condition <action1> ELSE <action2>)
(CALL (expression))

Examples:

CNCEX ((CALL 100), M1000)
Sends the (CALL 100) command to the CNC so it executes (calls) subroutine 100

CNCEX ((P100=P100+2), M1000)
Sends the (P100=P100+2) command to the CNC to increment the value of parameter P100 in 2 units.

Programming high-level blocks has the following restrictions:

- The programmed blocks can only work with global parameters.
- Up to 5 nesting levels of standard subroutines are allowed (neither parametric nor global).

Example in mm:

Move the W axis to the coordinate indicated by register R101.

When the PLC works with integers (32 bits), the value of register R2 is given in tenths of microns (0.0001 mm).

CNCWR (R101, GUP 155, M101)
Assigns the value indicated in R101 to global parameter P155.

CNCEX ((P155=P155/10000), M101)
Converts the value of P155 into mm

CNCEX (G1 WP155 F2000, M101)
Movement of the W axis

Interrupt block preparation

Same as in the CNC channel, blocks are also prepared in advance in the PLC channel.

CNCEX (G1 W100, M101)
    Movement of the W axis
CNCEX (IF P100=0 <action1>)
    P100 is analyzed during block preparation

The value of P100 may be different before, during and after the movement of the W axis. If it is to be analyzed after moving the axis, function G4 must be programmed.

CNCEX (G1 W100, M101)
    Movement of the W axis
CNCEX (G4, M102)
    Interrupts block preparation
CNCEX (IF P100=0 <action1>)
    P100 is analyzed after moving the axis.

Likewise, every time a PLC resource is accessed (I, O, M, R), block preparation is interrupted.

CNCEX (G1 W100, M101)
    Movement of the W axis
CNCEX (IF PLCI8=1 <action2>)
    I8 is checked after moving the axis.

Auxiliary (miscellaneous) M functions

The M functions programmed in the PLC channel may be defined in the M function table.

In the PLC channel, the following functions cannot be programmed: M0, M1, M2, M3, M4, M5, M6, M19, M30, M41, M42, M43, M44 and M45.

The following marks and registers are generated for managing the M functions, :

MBCDP1 through MBCDP7 (R565 through R571)
    similar to signals MBCD1 through MBCD7
AUXENDP (M5006)
    Similar to the AUXEND signal
MSTROBEP (M5505)
    Similar to the MSTROBE signal

11.1.3 Control of the PLC program from the CNC

The section of the PLC program regarding the "axes controlled from the PLC" can be controlled from the CNC itself.

To do this, the inputs, outputs, marks, registers, timers or counters of the PLC itself are used.

The CNC has the following PLC related variables to read or change the status of the selected resource.

PLCI	To read or modify up to 32 PLC inputs.
PLCO	To read or modify up to 32 PLC outputs.
PLCM	To read or modify up to 32 PLC marks (internal relays).
PLCR	To read or modify the status of a register.
PLCT	To read or modify the count of a timer.
PLCC	To read or modify the count of a counter.

With these variables, the desired values will be assigned, in the part-program of the CNC, to the PLC resources used in the communication. The setting of these values will be carried out whenever an axis or axes are to be controlled from the PLC.

In turn, the PLC program must check the status of such resources and when detecting that one of them is activated, it must execute the corresponding section of the PLC program.

It is also possible to transfer data from the CNC to the PLC via global and local arithmetic parameters. The PLC has the following variables related to those CNC parameters:

GUP	To read or modify a global parameter of the CNC.
LUP	To read or modify a local parameter of the CNC.

Example:

The "U" axis is controlled by the PLC and we want to command it from any part-program of the CNC in such way that we could select the type of move (G00 or G01), the positioning coordinate and the feedrate for that move.

In order to command it from any part-program, it is convenient to have in a subroutine the section of the CNC program allowing the data transfer with the PLC.

This example uses subroutine SUB1 and, for data exchange, it uses global CNC parameters.

P100	Type of move. If P100 = 0, then G00; If P100 = 1, then G01.
P101	"U" axis positioning coordinate.
P102	Feedrate. It only makes sense when moving in G01.

To indicate to the PLC that it must execute this move, it activates the following PLC resource:

M1000	Command to begin movement.

Any part-program of the CNC may contain a block of the type:

(PCALL 1, G1, U100, F1000)

This block calls subroutine SUB1 and it transfers the local parameters G, U and F with the following information:

G	Type of movement
U	"U" axis positioning coordinate
F	Feedrate for the movement

Subroutine SUB1 can be programmed as follows:

```
( SUB 1)
(P100 = G, P101 = U, P102 = F)
    Data transfer to global parameters
( PLCM1000 = PLCM1000 OR 1 )
    Execution command for the PLC.
(RET)
```

The PLC program, in turn, will have to contain the following instructions:

```
M1000 = CNCEX (G90 GP100 UP101 FP102, M111)
    ;When mark  M1000 is active, it sends the indicated block to
    the CNC
NOT M111 = RES M1000
    If the CNC accepts this block, it resets mark M1000.
```

11.2 Action CNCEX1

The CNCEX1 action is executed via main channel of the CNC and as long as the JOG keyboard is enabled. Its execution can be interrupted by pressing [CYCLE STOP] or even canceled by pressing [RESET].

If a CNCEX1 action is received when the JOG keyboard is disabled, the CNC ignores this command.

The block to be executed must be written in the programming format of the CNC itself.

Any type of block can be sent which is edited in ISO or high level language. It admits preparatory functions, auxiliary functions, calls to subroutines, etc.

User notes:

User notes:

8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

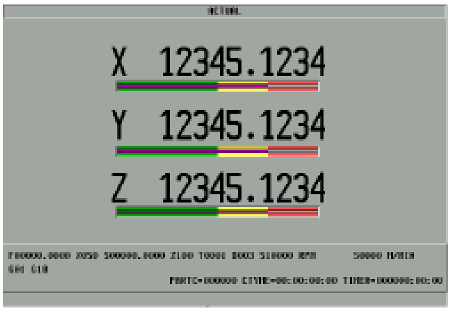
CUSTOMIZABLE SCREENS

12

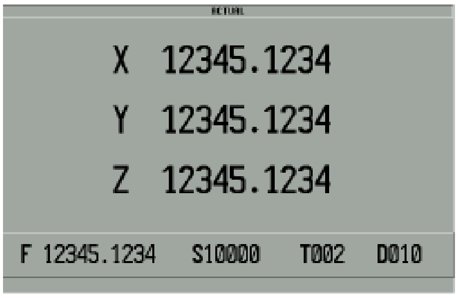
12. CUSTOMIZABLE SCREENS

The machine manufacturer may customize some of the CNC screens to:

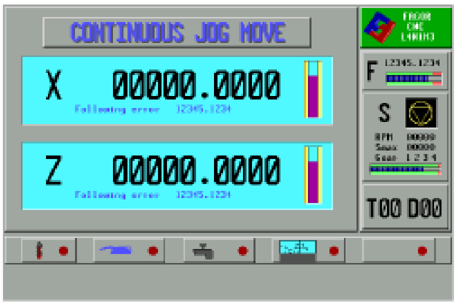
(1) display more information



(2) display the same information but in a different way



(3) display a completely different screen in contents and looks.



All of them use OEM screens that have been created on a PC using the Fagor Wgdraw application software and have been sent out to the CNC using the Fagor Windnc application software.

- In (1) the OEM screen (consumption led bar graph) is superimposed on the standard CNC screen.
- In (2) the upper area corresponds to the standard screen and the lower area shows the part of the machine manufacturer (OEM).
- In (3) the OEM screen replaces completely the standard CNC screen.

Use the configuration file at the CNC to define how the screens are to be laid out and which values must be displayed on the OEM screen.

12.1 Configuration file

It is a program which describes the operating characteristics of the graphic elements of the screen.

Set g.m.p. "CFGFILE (127)" with the number of the program for the configuration file:

The configuration file is a CNC program edited in high level language (configuration language) which is described later on. It may be edited both at the CNC and at a PC.

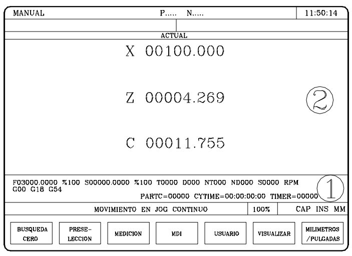
It may be stored in user RAM memory or in CARD A. If it is stored in both places, the one in user RAM memory will be used. It is recommended to store it only in CARD A once it has been fully debugged.

The configuration file must contain all the information regarding all the screens being customized.

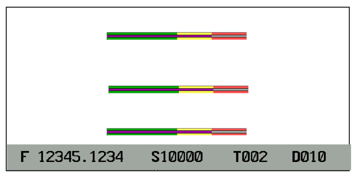
Next, all the screens that may be customized will be shown and the nomenclature to be used in the configuration file. Numbers 1, 2, 3, 4 and 5 indicate the areas each screen is divided into.

When editing a screen, the CNC overlaps the OEM screen over the standard CNC screen. The DISABLE instruction of the configuration file serves to indicate which of the standard screen areas are eliminated.

Example: Standard screen + OEM screen + Disable 1

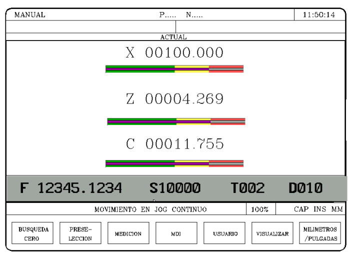


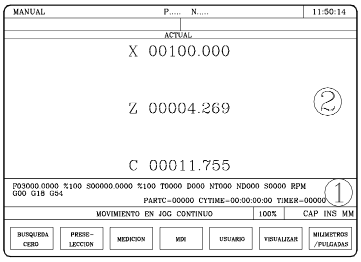
Standard screen



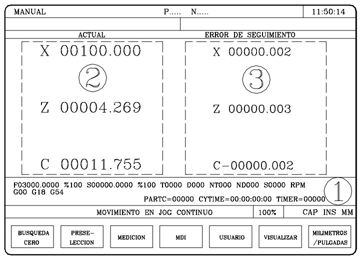
OEM screen

Both screens are overlapped; but "Disable 1" indicates that the area 1 of the standard screen is not displayed. Therefore:

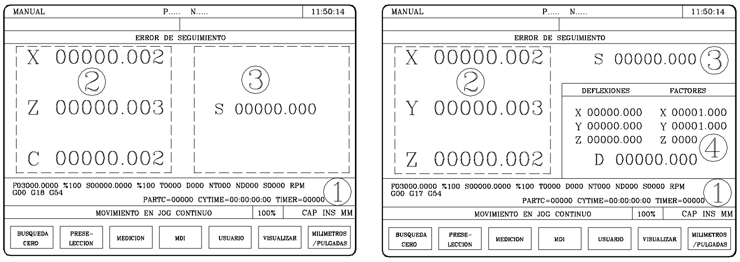




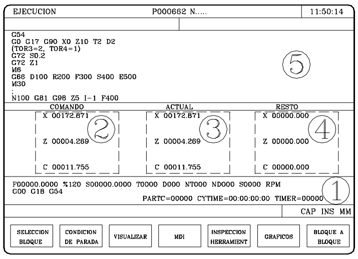
[JOG]
JOG mode - Actual



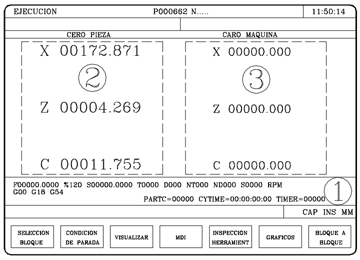
[JOGAFL] JOG mode - Actual and Following Error



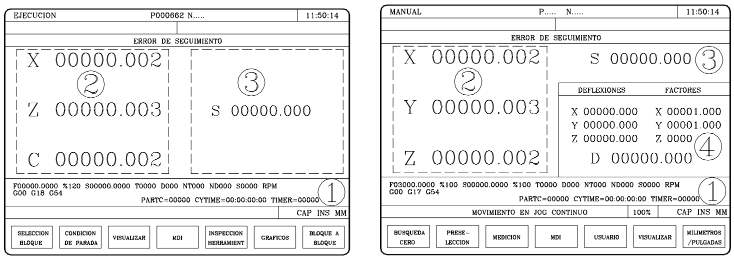
[JOGFLW] JOG mode - Following error



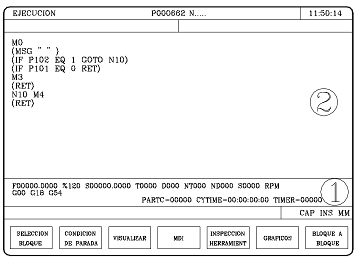
[STD]
Execution mode - Standard



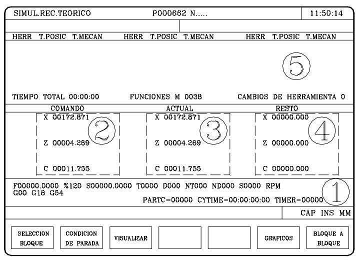
[POS]
Execution mode - Position



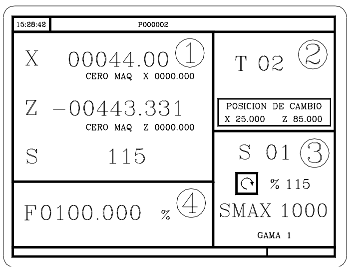
[FLW] Execution mode - Following error



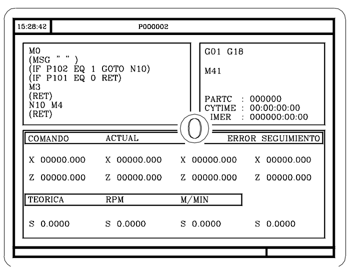
[PRG]
Execution mode - Program



[SUB]
Execution mode - Subroutines



[STDCONV]
Conversational mode - Standard



[AUXCONV] Conversational mode - Auxiliary for execution

12.2 Configuration language

The general characteristics of the configuration language are:

- All instructions are preceded by ";" and enclosed in parenthesis.
- The comments must be alone and preceded by ";;"
- The configuration file must begin with the line ;(PRGSCRIPT 1)
- It indicates that it is a configuration file corresponding to the version being used (in this case "1")
- The configuration file should end with the line ;(END)
- While debugging the program, the ;(DEBUG) instruction should be used.
- If an error occurs while checking the configuration file, the CNC will inform about it in program 999500.

The configuration language consists of:

- A series of key words or tokens.
- The names of the internal CNC variables
- Numbers that may be associated with the previous two items.
- Various punctuation signs.

Example of a configuration file:

```
;(PRGSCRIPT 1)
;;==================================
;;   SCREEN LAYOUT
;;     Screen in JOG mode - Actual
;;==================================
;[JOG],PLCM1125
;(DEBUG)
;(DISABLE 0)
;(WGDWIN 201)
;;-------------------- X axis, coordinate, error and consumption
;(AUTOREFRESH W1=POSX)
;(AUTOREFRESH W2=FLWEX)
;(FORMAT W3,LEDBARDEC)
;(AUTOREFRESH W3=ANAI1)
;;------------------- Z axis, position, error and consumption
;(AUTOREFRESH W4=POSZ)
;(AUTOREFRESH W5=FLWEZ)
;(FORMAT W6,LEDBARDEC)
;(AUTOREFRESH W6=ANAI2)
;;------------------- Feedrate, F, %
;(AUTOREFRESH W7=FEED)
;(AUTOREFRESH W8=FRO)
;;------------------- Spindle, S, Smax, %
;(AUTOREFRESH W9=SPEED)
;(AUTOREFRESH W10=SLIMIT)
;(AUTOREFRESH W11=SSO)
;;------------------- Tool and Offset (T, D)
;(AUTOREFRESH W12=TOOL)
;(AUTOREFRESH W13=TOD)
;(END)
```

12.3 Key words

;(PRGSCRIPT 1) Header of the configuration file and version used to edit it (in this case "1"). It must always be defined.

;[JOG],PLCM1125 Screen to be customized and condition

The screens that may be customized are:

[JOG]	JOG mode - Actual
[JOGFLW]	JOG mode - Following Error
[JOGAFL]	JOG mode - Actual and Following Error
[STD]	Execution mode - Standard
[FLW]	Execution mode - Following error
[POS]	Execution mode - Position
[PRG]	Execution mode - Program
[SUB]	Execution mode - Subroutines
[STDCONV]	Conversational mode - Standard
[AUXCONV]	Conversational mode - Auxiliary for execution

The screens may be active at all times or only when the set condition is met, Thus: Thus:

;[JOG]	Is always active
;[PRG],PLCM1000	Active with M1000=1. If M1000=0, standard screen

;(END) End of the screen definition.

The configuration file must contain all the screens to be customized. Each screen starts with the [xxx] instruction and ends with the (END) instruction .

;(DEBUG) It is optional. It indicates on which line program 999500 starts giving out information of the errors that have come up when debugging the configuration file.

The configuration file only debugs the portion of the selected screen. It starts with the [xxx] instruction and end with the (END) instruction.

It is recommended to program a (DEBUG) in the definition of each screen.

;(DISABLE x) Indicates the area of the standard screen to be eliminated.

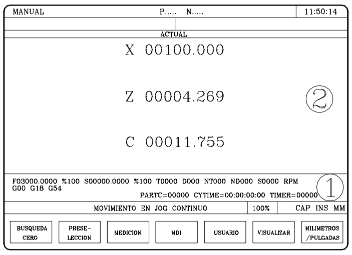
When editing a screen, the CNC overlaps the OEM screen over the standard CNC screen. The DISABLE instruction serves to indicate which areas of the standard display are to be eliminated (not displayed)

(DISABLE 1)	Eliminates area 1 of the standard display
(DISABLE 2)	Eliminates area 2 of the standard display

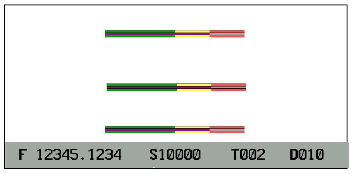
It is possible to define as many Disable instructions as screen areas are to be eliminated.

To eliminate the whole standard screen, program (DISABLE 0). In this case, only the OEM screen will be displayed.

Examples:



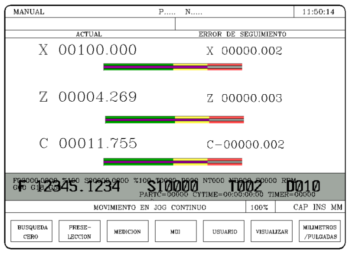
Standard screen



OEM screen

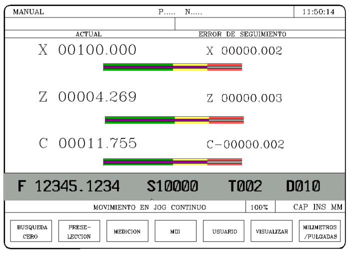
Without "Disable"

Both screens overlap.
There are areas with information overlapped.
In this case, area 1.



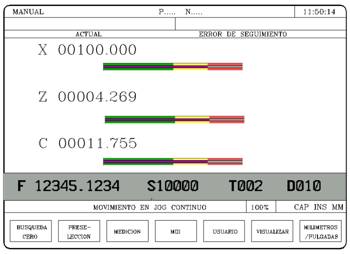
With (DISABLE 1)

Area 1 of the standard screen is not displayed.



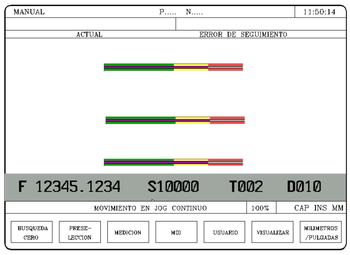
With (DISABLE 1) and (DISABLE 3)

Areas 1 and 3 of the standard display are not displayed.



With (DISABLE 0)

All areas of the standard screen are turned off
Only the OEM screen is displayed.



;(WGDWIN 201) It must ALWAYS be defined. It indicates the number of the OEM screen to be overlapped, edited with the Fagor WGDRAW application software.

;(W1=GUP100) Associates the value of a global parameter with the (W) data.

;(W2=PLCFRO) Associates the value of a variable with the (W) data.

;(W3=PLCR127) Associates the value of a PLC resource with the (W) data.

a register	;(W6=PLCR127)	
a mark	;(W6=PLCM1000,1)	first and how many
a group of inputs	;(W6=PLCI8,4)	first and how many
a group of outputs	;(W6=PLCO10,3)	first and how many

Associate only resources that are defined in the PLC program.

For marks, inputs and outputs, one must indicate how many of them, if none is indicated, 32 are assigned.

;(W6=PLCO11,4)	Assigns the value of O11, O12, O13, O14
;(W6=PLCO11)	Assigns the value of O11, O12 ... O41, O42

If a field (W) has a parameter, variable or resource associated with it, it acts as follows:

It assumes the value that its associate has when accessing the page.

- To continuously update the field value, use the (AUTOREFRESH) instruction as described later on.
- If its associate is a read-only type, the user will not be able to change the field value.
- It its associate is read/write type, the user may change the field value.
- When changing the value of the field (W), the value of its associate is also changed.
- On the other hand, when using the (AUTOREFRESH) instruction and the CNC or PLC changes the value of the associate, the value of the field is also changed.

;(AUTOREFRESH W2=FLWEX)

If (W2=FLWEX), it assigns the value of the X axis following error to the graphic element W2. This instruction updates that value periodically.

;(FORMAT W8,LEDBARDEC)

It must be used with Ledbar type (W) data that have a decimal variable associated with them. (For example: X axis following error).

The values assigned, at the PLC, to the end and intermediate values of a LEDBAR element must be integer values and must be related to the variable associated at the CNC.

When the associated variable has a decimal format the following instruction must be used:

;(FORMAT W8,LEDBARDEC)

This instruction is used to convert coordinate values (decimal) to integer values by multiplying them by 10000

Examples:

To represent the % of axis feedrate, the FRO variable is used.
The FRO values are integers (between 0 and 120) and, therefore, do not require LEDBARDEC

;(AUTOREFRESH W9=FRO)

On the other hand, to represent the amount of following error on the X axis, the FLWEX variable is used.
The values of FLWEX are not integers and, therefore, require LEDBARDEC (multiplying it by 10000) in order to make them integers.

;(FORMAT W11,LEDBARDEC)
;(AUTOREFRESH W11=FLWEX)

12.4 Example of a configuration file

;(PRGSCRIPT 1)

Header

;;==

;; Screen (201) in JOG mode - Actual

;;==

Comment

;[JOG],PLCM1125

To show the "JOG mode - Actual" screen when mark M1125=1

;(DEBUG)

Starting at this line, program 999500 keeps a log of the errors originated when debugging the configuration file.

;(DISABLE 0)

The OEM screen will replace the standard CNC screen.

;(WGDWIN 201)

The OEM screen is 201

;;--------------------- X axis, coordinate, error and consumption

;(AUTOREFRESH W1=POSX)

The graphic element W1 will always show the X axis position.

;(AUTOREFRESH W2=FLWEX)

The graphic element W2 will always show the X axis following error.

;(FORMAT W3,LEDBARDEC)

;(AUTOREFRESH W3=ANAI1)

The graphic element W3 (ledbar) will always show the X axis consumption (input ANAI1)

;;-------------------- Z axis, position, error and consumption

;(AUTOREFRESH W4=POSZ)

The graphic element W4 will always show the Z axis position.

;(AUTOREFRESH W5=FLWEZ)

The graphic element W5 will always show the Z axis following error.

;(FORMAT W6,LEDBARDEC)

;(AUTOREFRESH W6=ANAI2)

The graphic element W6 (ledbar) will always show the Z axis consumption (input ANAI2)

;;-------------------- Feedrate, F, %

;(AUTOREFRESH W7=FEED)

The graphic element W7 will always show the feedrate of the axes

;(AUTOREFRESH W8=FRO)

The graphic element W8 will always show the selected % of feedrate override for the axes.

;;-------------------- Spindle, S, Smax, %

;(AUTOREFRESH W9=SPEED)

The graphic element W9 will always show the spindle speed.

;(AUTOREFRESH W10=SLIMIT)

The graphic element W10 will always show the maximum spindle speed allowed.

;(AUTOREFRESH W11=SSO)

The graphic element W11 will always show the selected % of spindle speed override.

;;-------------------- Tool and Offset (T, D)

;(AUTOREFRESH W12=TOOL)

The graphic element W12 will always show the number of the selected tool.

;(AUTOREFRESH W13=TOD)

The graphic element W10 will always show the number of the selected tool offset.

;(END)

End of debug and end of the section corresponding to the screen

;;===

;; Screen (202) in MC/TC/CO mode

;;===

;[STDCONV],PLCM1125

We wish to customize the "Standard screen of the Conversational mode" when mark M1125=1

;(DEBUG)

Starting at this line, program 999500 keeps a log of the errors originated when debugging the configuration file.

;(DISABLE 0)

The OEM screen will replace the standard CNC screen.

;(WGDWIN 202)

The OEM screen is 202

;;-------------------- Coordinates of the Z and X axes

;(AUTOREFRESH W1=POSZ)

The graphic element W1 will always show the Z axis position (coordinates)

;(AUTOREFRESH W2=POSX)

The graphic element W2 will always show the X axis position (coordinates)

;;------------------- Machine cursors

;(AUTOREFRESH W3=POSZ)

The graphic element W3 (ledbar) will always show the Z axis position.

;(AUTOREFRESH W4=POSX)

The graphic element W4 (ledbar) will always show the X axis position.

;;-------------------- Axes feedrate (F)

;(AUTOREFRESH W5=FEED)

The graphic element W5 will always show the feedrate of the axes.

;(END)

End of debug, end of the portion of the configuration file corresponding to the screen and end of the configuration file.

12.5 Error log file (P999500)

Every time a customized screen is accessed, the CNC checks the section of that screen in the configuration file.

If it has errors, it displays the standard screen instead of the customized one.

If the error has been detected in a section defined after the DEBUG instruction, it generates, in program P999500 several lines indicating the detected error or errors.

The error log file (P999500) contains all the errors detected since the CNC was turned on. When the CNC is turned off, this error log file (P999500) is deleted.

Examples of detected errors:

```
Error due to a nonexistent variable. It must be FLWEX
;(AUTOREFRESH W2=FLWEXX)
    ;     Syntax error...
    ;          Unknown CNC variable name
    ;          Error on line: 12
    ;          Error on character: LF
```

```
Error caused for referring to a nonexistent graphic element
(W33).
;(AUTOREFRESH W33=PLCR124)
    ;     Warning...
    ;          Programmed Widget does not exist.
    ;          Warning in line: 15
```

User notes:

8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

CONFIGURABLE WORK MODE

13

13. CONFIGURABLE WORK MODE

The CNC offers a work mode that may be configured by the machine manufacturer. The basic screen provided by Fagor permits controlling the axes, the tool and the spindle.

In the configurable work mode, the manufacturer (OEM) can:

- Configure, partly or completely, the basic screen provided by Fagor.
- Create diagnosis screens
- Create screens to consult and/or modify internal variables of the CNC, PLC or drive.
- Create screens for the operator to set zero offsets, etc.

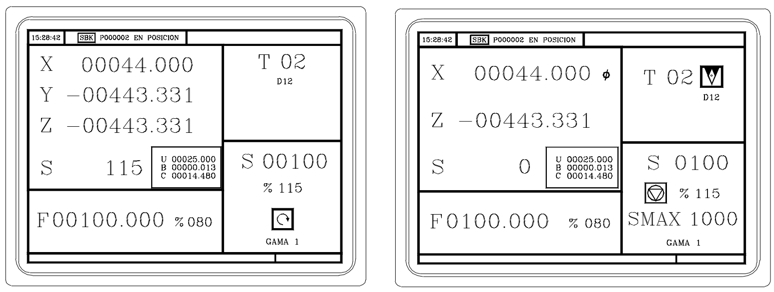
Since all the OEM screens have a subroutine associated, they can also create OEM cycles to:

- Consult inputs and outputs
- Adjust the machine axes
- Manage tool magazines
- Manage external devices
- etc.

The OEM machining canned cycles may even be used to machine parts. A machining operation may be repeated as often as you wish, but it cannot be stored in memory.

With the keystroke sequence [SHIFT] [ESC] , it is possible to switch from M/T mode to Configurable mode and viceversa.

Depending on the model, when accessing the configurable work mode, the CNC shows the following screen:

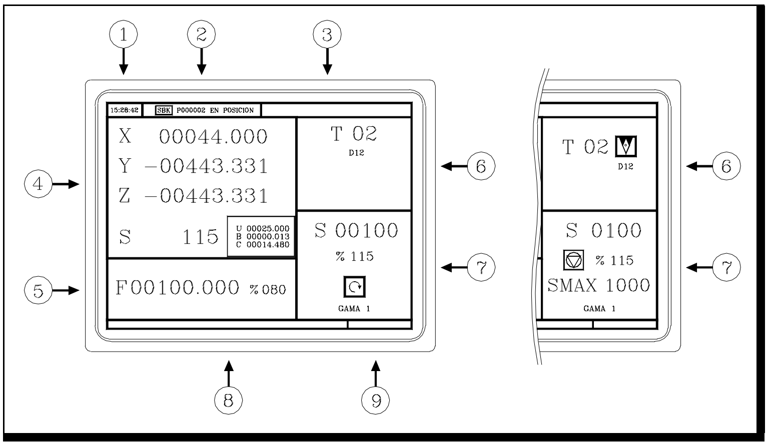


M model T model

The way to operate with either model is very similar. If any of the features described here is not common to both models, it will clearly indicate which model it corresponds to.

These screens may be customized entirely or by areas. See section *12.1 Configuration file*

The screen provided by Fagor contains the following information:



1. Clock
2. This window may show the following data:

SBK	when “Single Block” execution mode is selected.
DNC	when the DNC mode is active.
P.....	number of the program currently selected
Message	«In position» - «Execution» - «Interrupted» - «RESET»
PLC messages	

3. This window shows the CNC messages.
4. The window provided by Fagor shows the position of the axes (those of the auxiliary axes in a frame) and the real spindle rpm (S).
5. The window provided by Fagor shows the axis feedrate (F) currently selected and the % of F being applied.
6. The window provided by Fagor shows the tool number (T) and the tool offset (D). If the tool number and offset number are the same, the CNC will not show the “D” value.

   The T model also shows the graphic representation for the location code (shape) associated with the tool.
7. The window provided by Fagor shows the associated spindle speed (S), the % being applied, the selected turning direction and the active spindle range (gear).

   The T model also shows the maximum rpm and the “CSS” value when working at Constant Surface Speed.
8. This window shows the help texts associated with the OEM screens. See Wgdraw manual.
9. Reserved

13.1 Axis control

When accessing the customizable mode, the CNC assumes the work units “mm or inches”, “radius or diameters”, “mm/min. or mm/rev”, etc. selected by machine parameter.

Preset

It must be done on one axis at a time and proceeding as follows:

[X] or [Y] or [Z] Value [ENTER]

The CNC requests confirmation of the command.

Axis feedrate (F)

To set the axis feedrate, key in:

[F] Value [START]

Jog movement

Besides the continuous, incremental or handwheel jog, it is also possible to move the axes to a programmed position.

It is done on one axis at a time. With the feedrate “F” and % currently selected. To do that, press:

[X] or [Y] or [Z] Value [START]

13.2 Tool control

To select another tool, press:

[T] Value [START]

The CNC will manage the tool change and the T model updates its graphic representation.

Another offset may be assigned to the tool temporarily without modifying the one associated with it.

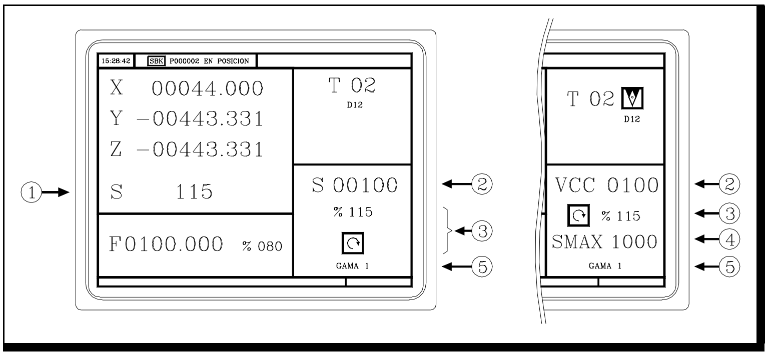
To access the “D” field, press:

[T] [↓] Tool offset number [START]

The CNC temporarily assumes the new offset for the current tool. The internal table is not modified, the tool’s associated offset is still the one assigned to it when it was calibrated.

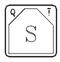
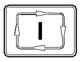
13.3 Spindle control

The CNC displays the following information:



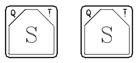
1. Real spindle speed in rpm.
2. Theoretical spindle speed in rpm or m/min (ft/min) when working at Constant Surface Speed (CSS).

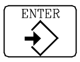
   To select another speed, press:

    Value 

   The CNC assumes this value and if the spindle is turning, it updates the real spindle speed (in rpm)
3. Spindle status and % of spindle speed being applied.
4. Maximum spindle rpm (T model)

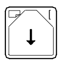
   To select another speed, press:

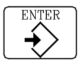
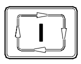
    The CNC frames the current value.

   Value  The CNC assumes this value and does not allow the spindle to exceed these rpm.
5. Spindle range (gear) currently selected.

   This value cannot be changed when using an automatic gear change.

   To change gears, press

    an d  until the new value is framed.

   Gear (range) number  or 

Note: When the machine does not use spindle gears, this message makes no sense. That is why the CNC does not show this message when text 28 or program 999997 has not been defined.

13.4 MDI

To use the MDI option in this work mode, send the key code “$F01E (61470)” from the PLC to the CNC.

In the next example, there is an external push button connected to input I13 and the MDI mode is activated every time this button is pressed or when pressing a free key of the front panel.

( ) = MOV 0 R100 = MOV 1 R101 = MOV $F01E R102

Initializations

DFU I13 OR DFU B?? R???

Every time one of the keys is pressed...

= CNCWR(R101, KEYSRC, M1)

... it indicates to the CNC that the keys come from the PLC

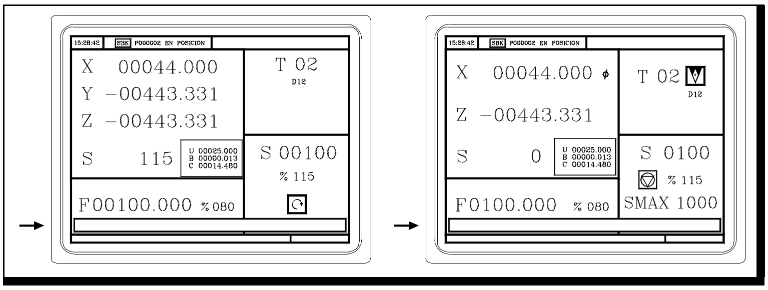
= CNCWR(R102, KEY, M1)

... a code is sent out to activate the MDI mode

= CNCWR(R100, KEYSRC, M1)

... and it indicates to the CNC that the keys come from the CNC

In MDI mode, the CNC shows a window at the bottom of the screen.



In this window, a block may be edited and later executed.

13.5 Screens, Subroutines and Cycles

User Fagor's Wgdraw (PC based) application to define the screens

Screens 201 through 255 must be used to customize the screens already stored at the CNC.

Screens 001 through 200 may be used to create new screens for diagnosis, variable consultation, setup, adjustments, device control, OEM canned cycles, etc.

Regardless of how they are used, screens 001 through 200 the have associated with them:

- The configuration file P999xxx (P999001 through P999200) that must be defined by the OEM.
- The subroutine that contains the executable 9xxx (9001 through 9200) that must be defined by the OEM.
- Program P999995 that contains the texts used by all the screens. See Wgdraw manual
- Program P999994 that contains the help texts used by all the screens. See Wgdraw manual

Thus, the configuration file P999004 and subroutine 9004 are associated with screen 4.

HELP

When pressing [HELP] , the CNC shows all the screens defined by the OEM with the WGDRAW application.

CLEAR

To delete any of them, position the cursor on it and press [CLEAR].

The CNC requests confirmation and the access code for customizing, if the OEM has previously defined it.

ENTER

To access one of the screens 001 through 2000, position the cursor on it and press [ENTER].

Screens 201 through 255 cannot be accessed from this mode. Select the CNC screen associated with it.

Warning

In the Wgdraw application, the text for to the screen title should be defined with the same number as the screen.

This way, when pressing, the key [HELP], CNC will show the number and title (text of program P999995 with the same number) of the available screens.

13.6 Associated keys

Screens 001 through 200 are grouped in 20 groups. Keys F1 through F7 allow direct access to the first 7 groups; the rest of the grops are accessed by sending a key code from the PLC.

Each group has 10 different levels or screens. Once a group has been selected, send the code “$F01C (61468)” for the [LEVEL CYCLE] key from the PLC to see the next level.

Now, let us see how the screens are grouped and how to access them.

Key or code	Screens								
F1 key	1	21	41	61	81	101	121	141	161
F2 key	2	22	42	62	82	102	122	142	162
F3 key	3	23	43	63	83	103	123	143	163
F4 key	4	24	44	64	84	104	124	144	164
F5 key	5	25	45	65	85	105	125	145	165
F6 key	6	26	46	66	86	106	126	146	166
F7 key	7	27	47	67	87	107	127	147	167
$F108 (61704)	8	28	48	68	88	108	128	148	168
$F109 (61705)	9	29	49	69	89	109	129	149	169
$F10A (61706)	10	30	50	70	90	110	130	150	170
$F10B (61707)	11	31	51	71	91	111	131	151	171
$F10C (61708)	12	32	52	72	92	112	132	152	172
$F10D (61709)	13	33	53	73	93	113	133	153	173
$F10E (61710)	14	34	54	74	94	114	134	154	174
$F10F (61711)	15	35	55	75	95	115	135	155	175
$F110 (61712)	16	36	56	76	96	116	136	156	176
$F111 (61713)	17	37	57	77	97	117	137	157	177
$F112 (61714)	18	38	58	78	98	118	138	158	178
$F113 (61715)	19	39	59	79	99	119	139	159	179
$F114 (61716)	20	40	60	80	100	120	140	160	180

When accessing a group, the CNC will show the last screen (level) used in that group.

To exit the screen:

- Press the key again or send the group code
  it shows the basic screen
- Press the key or send the code of another group
  shows the screen of the new group.
- Press [ESC] twice
  it shows the basic screen.

Example. It has screens 1, 21, 41, 2 and 22

Press F1
it shows screen 1
It sends the level key code
it shows screen 21
It sends the level key code
it shows screen 41
Press F2
it shows screen 2
It sends the level key code
it shows screen 22
Press F2

it exits the screen and shows the basic screen
Press F1
it shows screen 41 (last one used in the group)
It sends the level key code
it shows screen 1 (there are only 1, 21, 41)
Press F2
it shows screen 22 (last one used in the group)
It sends the level key code
it shows screen 2 (there are only 2, 22)
Press [ESC] twice
it shows the basic screen.

The next example shows an external push button connected to input I27 that selects and unselects the group of screens 13, 33, 53, etc.

( ) = MOV 0 R100 = MOV 1 R101 = MOV $F10D R102

Initializations

DFU I27
Every time the external key is pressed...

= CNCWR(R101, KEYSRC, M1)
.. it indicates to the CNC that the keys come from the PLC

= CNCWR(R102, KEY, M1)
... it sends the key code for the group 13, 33, 53, etc.

= CNCWR(R100, KEYSRC, M1)
... and it indicates to the CNC that the keys come from the CNC

13.7 OEM text in several languages.

By default, the texts defined by the OEM are in a single language and stored in several programs:

PLCMSG	Texts for PLC messages
PLCERR	Texts for PLC error messages
P999995	Texts and titles used by all the OEM screens.
P999994	Help texts of the OEM screens or cycles.

To have texts and messages in several languages, they must be grouped in a single program and set g.m.p. MSGFILE (P131) with that program number.

The MSGFILE program may be in user memory or in the Memkey Card. If it is in both places, it takes the one in user memory.

MSGFILE program structure:

On each line, a text is defined preceded by ";" the "text number", a blank space and the "$" sign. Examples:

```
;116    $Axis feedrate (F)
;117    $Tool (T)
```

The texts must be grouped by subjects and languages.

The labels identify a group with a mnemonic in brackets and preceded by ";"

;[PLCMSG]	Texts for PLC messages (up to 256).
;[PLCERR]	Texts for PLC error messages (up to 265).
;[CO999995]	Texts and titles used by all the OEM screens (up to 256).
;[CO999994]	Help texts of the OEM screens or cycles (up to 256).
;[OEMMSG]	Other texts used in OEM programs (up to 768).

The language number is indicated after the label and separated by a comma "," the same number used by g.m.p. LANGUAGE (P122):

(0) English	;[PLCMSG],0	(1) Spanish	;[PLCMSG],1
(2) French	;[PLCMSG],2	(3) Italian	;[PLCMSG],3
(4) German	;[PLCMSG],4	(5) Dutch	;[PLCMSG],5
(6) Portuguese	;[PLCMSG],6	(7) Czech	;[PLCMSG],7
(8) Polish	;[PLCMSG],8		

The text groups may be defined in the desired order, grouping them by subjects, languages, etc.

Programming example using MSGFILE (P131) = 12345:

P12345	
;[PLCMSG],0 ;1 $Text 1 ;2 $Text 2 --------------	Texts for PLC messages in English
;[PLCERR],0 ;1 $Text 1 ;2 $Text 2 --------------	Texts for PLC error messages in English
;[CO999994],0 ;1 $Text 1 ;2 $Text 2 --------------	Help texts of the OEM screens or cycles in English.
;[CO999995],0 ;1 $Text 1 ;2 $Text 2 --------------	Texts and titles used by all the OEM screens in English.
;[OEMMSG],0 ;1 $Text 1 ;2 $Text 2 --------------	Other texts used in OEM programs in English.

The texts of the [OEMMSG] are directed to:

- Texts of the user channel used with the Program control instructions MSG, ERR, IB, SK that depend on the selected work language.

MSG "KKK")	displays the text KKK on the screen.
(MSG 200)	displays the text 200 of the [OEMMSG] group in the currently selected language.
(MSG $C8)	similar to the previous one. It is the text 200 in hexadecimal format.
(MSG P100)	shows the text of the [OEMMSG] group whose number is the same as the value of parameter 100.

- Texts associated with screens and symbols generated with program Draw55.
  - Key in the text and press ENTER.
  - Assign to it the number of one of the internal CNC texts.
  - Assign to it the number of one of the OEM texts defined in the [OEMMSG] group. New option that is selected with the F2 softkey (OEM TEXT).

    Note: This type of texts can only be displayed properly at the CNC. The Draw55 application shows OEMtxtnn where “nn” means the number of the associated text; for example OEMtxt25.

The OEM screens or cycles are created with the Wgdraw application.

The texts used by the application are created in the Wgdraw.txt program and must be copied to the CNC as program P999995 or as part of the MSGFILE program within the [CO999995] group.

Use the Draw55 application to generate symbols or drawings used by those screens. If the texts of the symbols depend on the language, define them as part of the MSGFILE within the [OEMMSG] group.

Notes: On power-up or after a Shift-Reset, the CNC runs the following checkl to find the work texts of each group or text type:

- It takes the group of the MSGFILE program if it has been defined in the selected language.
- If it is not defined, it takes the texts of the first group defined (another language).
- If there is none, it takes the texts of program PLCMSG, PLCERR, P999995 or P999994

It then looks for texts in those groups.

13.8 Associated programs

Programs P900000 through P999999 are reserved for the CNC itself; in other words, the user cannot use them as part-programs.

Some have a special meaning and must be used by the machine manufacturer (OEM).

P999001 through P999200	Configuration files for OEM screens. P999001 corresponds to screen 1, P999002 to screen 2 and so on up to P999200 that corresponds to screen 200.
P999500	Program that logs (stores) the errors occurred when interpreting a configuration file.
P999994	Help texts of the OEM screens or cycles. See Wgdraw manual.
P999995	Texts and titles used by all the OEM screens; see Wgdraw manual.
P999999	Reserved for storing all the OEM subroutines It is empty.

The rest of the reserved programs are used internally by the CNC and cannot be erased.

13.9 Associated subroutines

Subroutines 0000 through 8999 are free to use and subroutines 9000 through 9999 are reserved for customizing the CNC.

It is recommended to define all the OEM subroutines in a program with a high number to prevent the user from modifying them. Program P999999 is free and may be used for this purpose.

When executing a cycle (screen), its associated subroutine is called upon indicating in local parameters A - Z (P0 - P25) the value used to define each field.

(PCALL 9001, A=11, B=22, C=33, D=44, E=0, F=1234, G=9999, H=1, I=1, J=12.34)

Parameter A (P0) indicates the value of the first field that may be edited, B (P1) that of the 2nd field and so on up to Z (P25) that indicates the value of the 26th field. The order is set when designing the screen with Wgdraw.

All the screens (cycles) have an associated subroutine, 9000 + "screen number" Subroutine 9001 corresponds to screen (cycle) 001, 9002 to 002 and so on up to 9200 that corresponds to screen (cycle) 200.

All these subroutines must be defined by the OEM and must contain all the necessary instructions to run the canned cycle.

```
Example  ( SUB 9005)  ; Definition of subroutine 9005
:                     ; Program blocks defined by the OEM.
         (RET)        ; End of subroutine
```

13.10 Configuration file

The configuration file is a CNC file written in high-level language (configuration language) that describes the operating characteristics of the various screen elements.

There is a configuration file for each screen. P999001 corresponds to screen 001, P999002 to 002 and so on up to P999200 that corresponds to screen 200.

The general characteristics of the configuration language are:

- All instructions are preceded by ";" and enclosed in parenthesis.
- The comments must be alone and preceded by ";;"
- The configuration file must begin with the line ;(PRGSCRIPT 1)
  It indicates that it is a configuration file corresponding to the version being used (in this case "1")
- The configuration file should end with the line ;(END)
- It is recommended to use the ;(DEBUG) instruction for the CNC to check the configuration file. If an error occurs, it will log it in program 999500.

The configuration language has the following key words:

;(PRGSCRIPT 1)

Header of the configuration file and version used to edit it (in this case "1"). It must always be defined.

;(DEBUG)

It is optional. It indicates on which line program 999500 starts giving out information of the errors that have come up when debugging the configuration file.

The debugging of the configuration file begins on the first line ;(PRGSCRIPT 1) and ends on the ;(END) line.

;(DISABLE 1)

It indicates that the blue frame at the top right must not be displayed; it shows the current status of the machine (coordinates and machining conditions)

;(DISABLE 20)

It indicates that the “CYCLE START” icon must not be displayed when pressing [ESC]

To exit the screen, press [ESC] twice. The first time, it displays the “CYCLE START” icon at the top right of the screen offering the chance to either execute or simulate the associated program.

Some screens are merely informative and no associated program must be executed.

In these cases, it is recommended to program ;(DISABLE20) to exit the screen when press [ESC].

;(DISABLE 21) It indicates that the spindle is not to be stopped at the end of the cycle.

By default, if ;(DISABLE21) is not programmed, the CNC adds the M5 instruction at the end of the associated program to stop the spindle at the end of the cycle.

;(HOTKEY W4,88) It permits associating a key to the (W) data.

In this case, pressing the X key (ASCII value 88) selects the W4 data.

;(TEACHIN W5=POSX)

Permits assigning the value of a CNC variable to the data.

In this case, While the W5 data is selected, when pressing the Recall key, W5 will show the variable of POSX (X axis position). If Enter is then pressed, the cycle assumes this value.

;(FORMAT W7,INCH) The value is shown in the work units (mm / inches) set by g.m.p. INCHES (P8).

;(PROFILE W12) Permits associating a profile with the (W) field. The field must be an unsigned 3-digit integer.

Select the (W) field, enter the number of the profile to be edited, press Recall and it will access the profile editor.

The edited profile is stored as program P994xxx.

P994001 corresponds to profile 001, P994002 to 002 and so on up to P994999 that corresponds to profile 999.

;(P100=W13) Permits assigning the value of the (W) data to a global parameter only when calling a subroutine.

When executing a cycle, the CNC calls the associated subroutine indicating with local parameters which values have been defined in each field. For example:

(PCALL 9001, A10, B12, C5, D8)

When using global parameters, the CNC uses another PCALL instruction to transfer the global parameters. For example:

(PCALL 9301, P100=22, P101=32, P102=48)
(PCALL 9001, A10, B12, C5,..... Y8, Z100)

The subroutine associated with the cycle is	9000 + cycle number
The auxiliary subroutine associated with the cycle is	9300 + cycle number

When using a (P100=W13) type instruction, the auxiliary subroutine must also be defined even if it only contains the SUB and RET instructions.

;(W1=GUP100) Associates the value of a global parameter with the (W) data.

;(W2=PLCFRO) Associates the value of a variable with the (W) data.

;(W3=PLCR127)

Associates the value of a PLC resource with the (W) data.

the value of a Register	;(W6=PLCR127)	
that of a Mark	;(W6=PLCM1000,1)	first and how many
that of a group of inputs	;(W6=PLCI8,4)	first and how many
that of a group of outputs	;(W6=PLCO10,3)	first and how many

Associate only resources that are defined in the PLC program.

For marks, inputs and outputs, one must indicate how many of them, if none is indicated, 32 are assigned.

;(W6=PLCO11,4)	Assigns the value of O11, O12, O13, O14
;(W6=PLCO11)	Assigns the value of O11, O12 ... O41, O42

If a field (W) has a parameter, variable or resource associated with it, it acts as follows:

- It assumes the value that its associate has when accessing the page.
  To continuously update the field value, use the (AUTOREFRESH) instruction as described later on.
- If its associate is a read-only type, the user will not be able to change the field value.
- It its associate is read/write type, the user may change the field value.
  When changing the value of the field (W), the value of its associate is also changed.
  On the other hand, when using the (AUTOREFRESH) instruction and the CNC or PLC changes the value of the associate, the value of the field is also changed.
- If an error occurs because the variable does not exist (PLC register) or the communication fails (drive variable), the field is momentarily disabled and it displays a grey window. The field is enabled again after 10 seconds.

;(AUTOREFRESH W6=FLWEX)

Refreshes (updates periodically) the value of graphic element W6 assigning it the X axis following error value.

;(SAVEINSUB)

It is programmed as a prefix of the instructions

;(W1=GUP100)
Associates the value of a global parameter, variable or resource of the PLC with the (W) data.

;(AUTOREFRESH W6=FLWEX)
Refreshes the value of graphic element W6.

The resulting new instructions are:

;(W1=GUP170)
;(SAVEINSUB W1=GUP170)

;(AUTOREFRESH W6=FLWEX)
;(SAVEINSUB AUTOREFRESH W6=FLWEX)

When recalling a cycle stored in memory [PPROG] , the CNC analyzes the type of instruction that each graphic element has associated with it and acts as follows:

- If it is of the ;(W1=GUP170) type, it assigns the current value of arithmetic parameter P170 to W1.
- If it is of the ;(SAVEINSUB W1=GUP170) , it restores the value that the arithmetic parameter had when the program was edited (when it was saved into memory) and it assigns that value to the graphic element W1 and to the arithmetic parameter P170.

Recommendations to use SAVEINSUB:

- Restrict its usage to when it is absolutely necessary because every time a stored cycle is saved or recalled, the value of the variable associated with the element changes.
- Use it only with global parameters and variables that may be read and written from the CNC See appendix ***Internal CNC variables***

;(FORMAT W8,LEDBARDEC)

It must be used with Ledbar type (W) data that have a decimal variable associated with them. (For example: X axis following error).

The values assigned, at the PLC, to the end and intermediate values of a LEDBAR element must be integer values and must be related to the variable associated at the CNC.

When the associated variable has a decimal format the following instruction must be used:

;(FORMAT W8,LEDBARDEC)

This instruction is used to convert coordinate values (decimal) to integer values by multiplying them by 10000

Examples:

To represent the % of axis feedrate, the FRO variable is used. The FRO values are integers (between 0 and 120) and, therefore, do not require LEDBARDEC

;(AUTOREFRESH W9=FRO)

On the other hand, to represent the amount of following error on the X axis, the FLWEX variable is used. The values of FLWEX are not integers and, therefore, require LEDBARDEC (multiplying it by 10000) in order to make them integers.

;(FORMAT W11,LEDBARDEC)
;(AUTOREFRESH W11=FLWEX)

;(MODALCYCLE) It indicates that cycle is modal. See programming manual.

The subroutine call is of the (MCALL 9001, A10, B12, C5,....) type

If after executing the cycle, several movements are carried out, the cycle will be executed again after each movement calling the (MCALL 9001, A10, B12, C5,...) subroutine again.

When using global parameters, the CNC only transfers the global parameters the first time.

First time:
(PCALL 9301, P100=22, P101=32, P102=48)
(MCALL 9001, A10, B12, C5,..... Y8, Z100)

The rest of the times:
(MCALL 9001, A10, B12, C5,..... Y8, Z100)

To cancel this mode, execute the (MDOFF) instruction.

;(END) It indicates that it has finished debugging the configuration file.

It ignores the instructions programmed next.

13.11 Error log file (P999500)

There is a configuration file for each screen. P999001 corresponds to screen 001, P999002 to 002 and so on up to P999200 that corresponds to screen 200.

The CNC checks these programs when accessing each screen for the first time. When detecting an error, it displays a message window.

In all of them, if the error has been detected in a section defined after the DEBUG instruction, the generate, in program P999500 several lines indicating the detected error or errors.

The error log file (P999500) contains all the errors detected since the CNC was turned on. When the CNC is turned off, this error log file (P999500) is deleted.

Examples of detected errors:

Error due to a nonexistent variable. It must be FLWEX

```
;(AUTOREFRESH W2=FLWEXX)
    ; Syntax error...
    ;       Unknown CNC variable name
    ;       Error on line: 12
    ;       Error on character: LF
```

Error caused for referring to a nonexistent graphic element (W33).

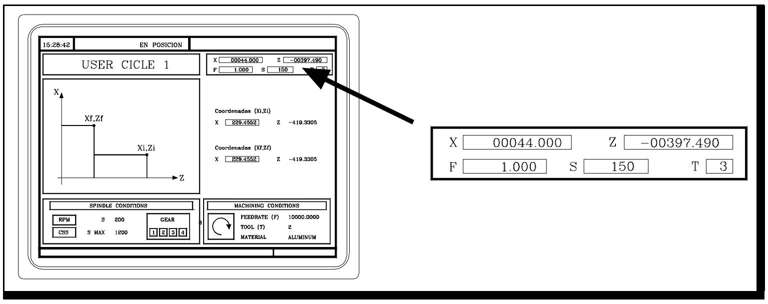
```
;(AUTOREFRESH W33=PLCR124)
    ; Warning...
    ;       Programmed Widget does not exist.
    ;       Warning in line: 15
```

Warning

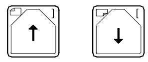
After modifying the configuration file, reset the CNC to debug it again when accessing its associated screen.

13.12 Cycle data entry

Once the cycle has been selected, the CNC shows the relevant screen. It may have a blue frame at the top right indicating the current status of the machine. Coordinates and machining conditions:



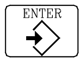
One of the data that define the cycle will be highlighted indicating that it is selected.

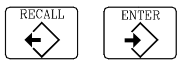
Use the keys to select another data 

Select a numeric data

The numeric data is used for coordinates, feedrate, spindle speed, tool number, etc. (value in Wgdraw).

X 229.4552

Key in the desired value and press 

If the Teachin instruction is associated with it, the value of an internal variable (coordinate, tool number, etc.) may be assigned to this field. In those cases, press 

If this [Recall] key is not available, send the key code $F006 (61446) from the PLC to the CNC.

The following example shows how to use an external push button connected to input I25 so as to act like the [Recall] key.

( ) = MOV 0 R100 = MOV 1 R101 = MOV $F006 R102

Initializations

DFU I27

Every time the external key is pressed...

= CNCWR(R101, KEYSRC, M1)

.. it indicates to the CNC that the keys come from the PLC

= CNCWR(R102, KEY, M1)

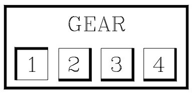
... the code for the Recall key is sent out.

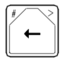
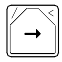
= CNCWR(R100, KEYSRC, M1)

... and it indicates to the CNC that the keys come from the CNC

Select an option among the ones available

It is used to select a button in a group of Wgdraw buttons.



Use the keys  

Select one of the representations in an icon

It is used to select an icon in a multiple representation of Wgdraw.

Press the key or send the key code $F01D (61469) from the PLC to the CNC until the desired icon or text appears.

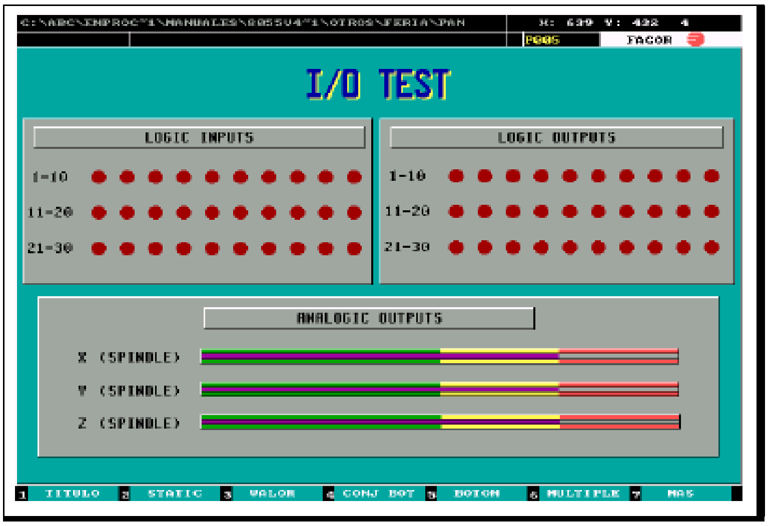
When pressing [ESC], the top of the screen shows a CYCLE START icon. It is then possible to:

- press [ESC] again to exit the cycle
- press the [CYCLE START] key to call the associated subroutine (9001) to execute the cycle.

13.13 Example. Consult inputs and outputs

Screen 005 is used. The data shown by this screen have the following identifier (W).

Inputs I1 through I30	W1 through W30
Outputs O1 through O30	W31 through W60
X, Y, Z axis (spindle) output	W61, W62, W63



This screen has the P999005 configuration file associated with it which is defined as follows:

;(PRGSCRIPT 1) Header and version

The debugging of the configuration file begins.

;(DEBUG) Starting at this line, program 999500 keeps a log of the errors originated when debugging the configuration file.

;(DISABLE 1) The blue frame at the top right of the screen is not to be displayed.

;(AUTOREFRESH W1=PLCI1,1)

Refreshes (updates periodically) the value of graphic element W1 assigning the value of input I1 to it. Repeat this instruction for all the inputs.

;(AUTOREFRESH W31=PLCO1,1)

Refreshes (updates periodically) the value of graphic element W31 assigning the value of output O1 to it. Repeat this instruction for all the outputs.

;(FORMAT W61,LEDBARDEC)
;(AUTOREFRESH W61=ANAO1)

Refreshes (updates periodically) the value of the W61 ledbar assigning the value of analog output 1 (X axis analog voltage) to it. Repeat this instruction for all three axes.

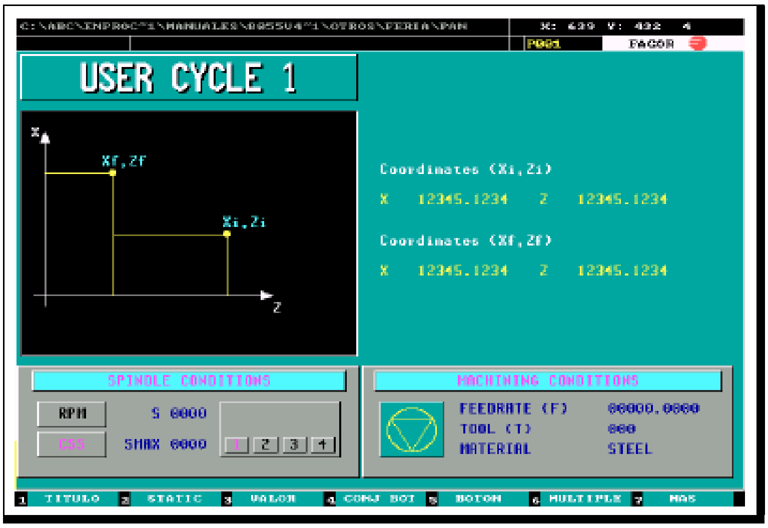
;(END) End of the debugging of the configuration file and end of program.

Since this screen does not belong to a cycle, its associated subroutine (9005) needs not be defined.

13.14 Example. Machining canned cycle

Screen 001 is used. The data shown by this screen have the following identifier (W).

The data that the user may edit in this cycle have the W identifier, the number associated with each one indicates the order in which they are selected (W1, W2, etc.) W11, W12).



This screen has the P999001 configuration file associated with it. Some of its fields are defined as follows:

;(PRGSCRIPT 1)

;(HOTKEY W1,88)
;(TEACHIN W1=POSX)

The W1 field has the X hotkey (88) and the X axis coordinate associated with it. In other words:

Pressing the X key selects this field.

While this field is selected, if the Recall key is pressed, it will show the X axis position. If Enter is then pressed, the cycle assumes that value.

Repeat these instructions for the fields: W2, W3 and W4

;(HOTKEY W6,83)
;(HOTKEY W7,83)

The W6 and W7 fields have the S (83) hotkey associated with them. In other words:

Pressing the S key selects the W6 field and pressing it again selects the W7 field.

Associate the F key with the W10 field and the T key with the W11 field.

;(END)

End of the debugging of the configuration file and end of program.

Pressing [ESC] displays a cycle-start icon at the top right of the screen and if the [CYCLE START] is pressed again, it calls the associated subroutine (9001) to execute the cycle.

The 9001 subroutine must be defined by the OEM and must contain all the necessary instructions to run the canned cycle.

User notes:

User notes:

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

PLC PROGRAMMING EXAMPLE

14

14. PLC PROGRAMMING EXAMPLE

It is a three-axes machine (X, Y, Z) having a spindle with two speed ranges.

The PLC, besides controlling the 3 axes and the spindle, is in charge of lubricating the axes as well as turning the coolant on and off.

CNC configuration

The PLC has 512 inputs and 512 outputs. Some of them, depending on the CNC configuration, communicate with external devices.

Warning

Input I1 is the emergency input of the CNC and must be supplied with 24V. Regardless of how it is treated by the PLC program, this signal is processed directly by the CNC at all times.

Output O1 is normally at 24V, high logic level, and it is set low, 0V, whenever an ALARM or an ERROR occurs at the PLC output O1.

14.1 Definition of symbols (mnemonics)

It is a possible to associate a symbol (name) to any PLC resource. It may have up to 8 characters so long as the name does not coincide with any of the reserved instructions. It may not contain the following characters: blank-space " ", equal sign "=", parenthesis "(" or ")", comma "," or semi-colon ";".

These symbols or names must always be defined at the beginning of the program. Duplicate symbols are not allowed; but, one resource may have more than one symbol.

For better clarification, the symbols used in this program are grouped by subjects.

Used in: Basic and necessary programming

DEF	I-EMERG	I1	External emergency input
DEF	I-CONDI	I70	Conditional mode. The CNC interrupts part-program execution when executing auxiliary function M01
DEF	SERVO-OK	I71	The servo drives are O.K.
DEF	O-EMERG	O1	Emergency output. It must be normally high.

Used in: Treatment of the axis overtravel limit switches

DEF	I-LIMTX1	I72	X axis positive overtravel limit switch
DEF	I-LIMTX2	I73	X axis negative overtravel limit switch
DEF	I-LIMTY1	I74	Y axis positive overtravel limit switch
DEF	I-LIMTY2	I75	Y axis negative overtravel limit switch
DEF	I-LIMTZ1	I76	Z axis positive overtravel limit switch
DEF	I-LIMTZ2	I77	Z axis negative overtravel limit switch

Used in: Treatment of the machine reference (home) switches

DEF	I-REF0X	I78	X axis home switch
DEF	I-REF0Y	I79	Y axis home switch
DEF	I-REF0Z	I80	Z axis home switch

Used in: Treatment of M, S, T functions

DEF	M-03	M1003	Auxiliary mark. Indicates that M03 must be executed
DEF	M-04	M1004	Auxiliary mark. Indicates that M04 must be executed
DEF	M-08	M1008	Auxiliary mark. Indicates that M08 must be executed
DEF	M -41	M1041	Auxiliary mark. Indicates that M41 must be executed
DEF	M -42	M1042	Auxiliary mark. Indicates that M42 must be executed

Used in: Machine way lubrication

DEF	I-LUBING	I81	Operator request to lubricate the ways of the machine
DEF	O-LUBING	=O2	Ways lubrication output

Used in: Coolant treatment

DEF	I-COOLMA	I82	The operator control the coolant. Manual mode.
DEF	I-COOLAU	I83	The CNC controls the coolant. Automatic mode.
DEF	O-COOL	O3	Coolant output

Used in: Spindle turning control

DEF	O-S-ENAB	O4	Spindle enable output

Used in: Treatment of the spindle speed range change

DEF	O-RANGE1	O5	Move gears to select range 1 (gear 1)
DEF	O-RANGE2	O6	Move gears to select range 2 (gear 1)
DEF	I-RANGE1	I84	Indicates that Range 1 is selected
DEF	I-RANGE2	I85	Indicates that Range 2 is selected

Used in: Keyboard simulation

DEF	I-SIMULA	I86	The operator requests the simulation of program P12
DEF	SENDKEY	M1100	Indicates that the code of a key is to be sent out to the CNC
DEF	KEYCODE	R55	Indicates the code of the key to be simulated
DEF	LASTKEY	R56	Indicates which is the last key accepted by the CNC
DEF	SENTOK	M1101	Indicates that the key code has been sent correctly
DEF	KEYBOARD	R57	Used to indicate to the CNC the source of the keys
DEF	CNCKEY	0	Used to indicate that the keys come from the CNC keyboard
DEF	PLCKEY	1	Used to indicate that the keys come from the PLC

DEF	MAINMENU	$FFF4	Code of the "MAIN MENU" key
DEF	SIMULATE	$FC01	Code of the "SIMULATE" key (F2)
DEF	KEY1	$31	Code of the "1" key
DEF	KEY2	$32	Code of the "2" key
DEF	ENTER	$0D	Code of the "ENTER" key
DEF	THEOPATH	$FC00	Code of the "THEORETICAL PATH" key (F1)
DEF	START	$FFF1	Code of the "START" key

14.2 First cycle module.

```
CY1
( ) = ERA O1 512 = ERA C1 256 = ERA T1 256 = ERA R1 256 = ERA M1 2000
( ) = ERA M4000 4127 = ERA M4500 4563 = ERA M4700 4955
```

Initializes all PLC resources to low logic level "0"

( ) = TG1 2 120000 Initializes the timer which controls the lubrication of the machine ways on power-up. This operation will be performed for 2 minutes.

( ) = TG2 4 3600000 Initializes the timer which controls the amount of time the axes are moving before they are lubricated. This lubrication lasts 5 minutes and it takes place after the axes have been moving for 1 hour.

END

14.3 Main module.

PRG
REA

---- Basic and necessary programming ----

( ) = /STOP	Permission to execute the part-program
( ) = /FEEDHOL	Permission to move the axes
( ) = /XFERINH	Permission to execute the next block

I-EMERG AND (rest of conditions) = /EMERGEN

If the external emergency input is activated or any other emergency occurs, the general logic input /EMERGEN of the CNC. When there is no emergency, this signal must remain high.

/ALARM AND CNCREADY = O-EMERG

The emergency output, O1, of the PLC (O-EMERG) must be normally high

If an alarm or emergency is detected at the CNC (/ALARM) or a problem was detected when powering the CNC up (CNCREADY), the emergency output O-EMERG must be brought low

I-CONDI = M01STOP When the operator selects the conditional mode (I-CONDI), the CNC general logic input M01STOP must be activated. It interrupts the program when executing M01

START AND (rest of conditions) = CYSTART

When the cycle START key is pressed, the CNC activates the general logic output START.

The PLC must check that the rest of the conditions (hydraulic, safety devices, etc.) are met before setting the general input CYSTART high in order to start executing the program

SERVO-OK AND NOT LOPEN = SERVO1ON = SERVO2ON = SERVO3ON

If the servo drives are OK and the CNC does not detect any errors in the positioning loop of the axes (LOPEN), the positioning loop must be closed on all axes. Axis logic inputs of the CNC: SERVO1ON, SERVO2ON, SERVO3ON.

----- Treatment of the axis overtravel limit switches -----

I-LIMTX1 = LIMIT+1
I-LIMTX2 = LIMIT-1
I-LIMTY1 = LIMIT+2
I-LIMTY2 = LIMIT-2
I-LIMTZ1 = LIMIT+3
I-LIMTZ2 = LIMIT-3

----- Treatment of the machine reference (home) switches -----

I-REF0X = DECEL1
I-REF0Y = DECEL2
I-REF0Z = DECEL3

----- Message treatment -----

The PLC allows displaying the corresponding PLC message at the CNC screen by activating marks MSG1 through MSG128,. This text must be previously edited at the PLC message table.

The following example shows how to generate a message to remind the operator to home the axes after powering the machine up.

(MANUAL OR MDI OR AUTOMAT) AND NOT (REFPOIN1 AND REFPOIN2 AND REFPOIN3) = MSG5

The message (MSG5) appears in the JOG, MDI or Automatic modes and only when the axes of the machine have not been referenced (homed). The CNC logic outputs "REFPOIN" indicate that the axes have been homed. ----- Error treatment -----

----- Error message treatment -----

The PLC permits displaying the corresponding error message on the CNC screen by activating marks ERR1 through ERR64 as well as interrupting the CNC program execution stopping the axes and the spindle. The activation of any of these marks does not activate the external CNC Emergency output.

Because the PLC program is not interrupted by these marks, it is advised to make it possible to change their status via accessible external inputs; otherwise, the CNC will keep receiving the same error at every PLC scan (cycle) thus preventing access to any PLC mode.

The text associated to the error message must be previously edited at the PLC error table.

The next example shows how to generate the X axis overtravel limit overrun error when one of the overtravel limit switches is pressed.

NOT I-LIMTX1 OR NOT I-LIMTX2 = ERR10

----- Treatment of M, S, T functions -----

The CNC activates the general logic output MSTROBE to "tell" the PLC to execute the M functions indicated at the variables MBCD1 through MBCD7.

It also activates: the SSTROBE output when the S function indicated at variable SBCD must be executed, the TSTROBE output when the T function indicated at variable TBCD must be executed and the T2STROBE output when the T function indicated at variable T2BCD must be executed.

Whenever the CNC activates one of these signals, it is convenient to deactivate the general CNC input AUXEND in order to interrupt the execution of the CNC. When the PLC concludes the processing of the required function, this AUXEND signal must be activated back so that the CNC resumes the execution of the interrupted program.

This example deactivates the AUXEND signal for 100 milliseconds using the timer T1.

MSTROBE OR SSTROBE OR TSTROBE OR T2STROBE = TG1 1 100

The activation of the STROBE signals activates timer T1 in the mono-stable mode for 100 milliseconds.

Whenever timer T1 is active, the PLC must set the AUXEND signal low as described in: "Treatment of the general CNC input AUXEND".

When the CNC activates the MSTROBE signal, the contents of variables MBCD1 through MBCD7 must be analyzed in order to know which auxiliary functions are to be executed. All MBCD variables may be analyzed at the same time by using "MBCD*".

This example SETs the auxiliary marks so they can be analyzed later. Once analyzed, they must be RESet so that the PLC does not analyze them again on the next cycle (scan)

DFU MSTROBE AND CPS MBCD* EQ $0 = RES M-08
DFU MSTROBE AND CPS MBCD* EQ $2 = RES M-08

Functions M00 and M02 cancel the coolant (M08)

DFU MSTROBE AND CPS MBCD* EQ $3 = SET M-03 = RES M-04
DFU MSTROBE AND CPS MBCD* EQ $4 = SET M-04 = RES M-03
DFU MSTROBE AND CPS MBCD* EQ $5 = RES M-03 = RES M-04

Functions M03 and M04 are incompatible with each other and M05 cancels both.

DFU MSTROBE AND CPS MBCD* EQ $8 = SET M-08
DFU MSTROBE AND CPS MBCD* EQ $9 = RES M-08
DFU MSTROBE AND CPS MBCD* EQ $30 = RES M-08

Functions M09 and M30 cancel the coolant (M08)

DFU MSTROBE AND CPS MBCD* EQ $41 = SET M-41 = RES M-42
DFU MSTROBE AND CPS MBCD* EQ $42 = SET M-41 = RES M-42

Functions M41 and M42 are incompatible with each other.

----- Spindle turning control -----

The spindle enable output O-S-ENAB will be activated when selecting function M03 or M04.

M-03 OR M-04 = O-S-ENAB

----- Treatment of spindle speed range change -----

The spindle in this example has two ranges (high and low). To perform a range change, the following steps must be taken:

- Deactivate the general CNC input AUXEND
- Remove the control of the spindle back to the CNC Controlled by PLC
- Output an oscillating analog signal to change gears.
- Move the gears
- Verify that the gear change has been completed
- Remove the oscillating analog signal
- Return the control of the spindle back to the CNC
- Activate the general CNC input AUXEND

Deactivate the general CNC input AUXEND

While changing gears (ranges), general CNC input AUXEND should be canceled in order to interrupt the execution of the CNC. "Treatment of the general CNC input AUXEND".

Remove the control of the spindle from the CNC Controlled by PLC

Output an oscillating analog signal to change gears.

DFU M-41 OR DFU M-42

When a range (gear) change is requested...

= MOV 2000 SANALOG

... A 0.610V analog signal for the spindle is prepared and...

= SET PLCCNTL ... the PLC grabs the control of the spindle loop

PLCCNTL AND M2011

While the PLC has the spindle control...

= SPDLEREV ... the spindle turning direction is changed every 400 milliseconds

Move the gears

The corresponding range output (O-RANGE) is kept active until the range selection is completed (I-RANGE).

M-41 AND NOT I-RANGE1 = O-RANGE1
M-42 AND NOT I-RANGE2 = O-RANGE2

Verify that the gear change has been completed

Remove the oscillating analog signal

Return the control of the spindle back to the CNC

(M-41 AND I-RANGE1) OR (M-42 AND I-RANGE2)

Once the gear change has concluded, the following must be done:

= RES M-41 = RES M-42

... remove the request for range change (M-41, M-42), ...

= MOV 0 SANALOG ... remove the spindle analog voltage,...

= RES PLCCNTL ... Return the control of the spindle to the CNC

I-RANGE1 = GEAR1
I-RANGE2 = GEAR2

The corresponding CNC logic input (GEAR1, GEAR2) must be activated to confirm the gear (range) change.

----- Lubrication of the machine ways -----

In this example, the machine axes are lubricated in the following instances:

- On machine power-up. For 2 minutes.
- When requesting a manual lubrication. For 5 minutes.
- After the axes have been moving for 1 hour. For 5 minutes.
- After an axis has travelled a specific distance since last lubricated. For 4 minutes.

Lubrication on machine power-up.

This operation will be performed for 2 minutes.

Whenever the machine is powered up, the PLC program starts running. Therefore, the first cycle module CY1 must activate timer T2 in the mono-stable mode for 2 minutes (120000 milliseconds).

( ) = TG1 2 120000

Manual lubrication

This operation will last 5 minutes and it will be performed at operator's request.

DFU I-LUBING = TG1 3 300000

Whenever the operator requests the lubricating (lubing) operation, T3 must be activated in the mono-stable mode for 5 minutes (300000 milliseconds).

Lubrication every hour of axis motion

This operation takes place when the axes of the machine have been moving for an accumulated time period of 1 hour. They will be lubricated for 5 minutes.

Timer T4 is used to keep track of the axis accumulated moving time and T5 to time the 5 minute lubrication period.

The first cycle module CY1 must activate timer T4 in the delayed activation mode with a time constant of 1 hour (3600 000 milliseconds).

( ) = TG2 4 3600000

ENABLE1 OR ENABLE2 OR ENABLE3 = TEN 4

T4 only times when any of the axis is moving.

T4 = TG1 5 300000

After having timed 1 hour, T5 must be activated in the mono-stable mode for 5 minutes. (300000 milliseconds)

T5 = TRS 4 = TG2 4 3600000

Resets the axis-motion timer T4 to zero.

Lubrication when an axis has traveled a specific distance since the last time it was lubricated

PLC machine parameters USER12 (P14), "USER13 (P15) and USER14 (P16) are used to indicate the distance each axis must travel before it gets lubricated.

( ) = CNCRD(MPLC12,R31,M302) = CNCRD(MPLC13,R32,M302) = CNCRD(MPLC14,R33,M302)

Assigns to registers R31, R32 and R33 the values of PLC machine parameters USER12 (P14), "USER13 (P15) and USER14 (P16)

( ) = CNCRD(DISTX,R41,M302) = CNCRD(DISTY,R42,M302) = CNCRD(DISTZ,R43,M302)

Assigns to registers R41, R42 and R43 the distance each axis has travelled.

CPS R41 GT R31 OR CPS R42 GT R32 OR CPS R43 GT R33

If the distance traveled by any axis exceeds the one set by machine parameter,......

= TG1 6 240000

....timer T6 must be activated in the mono-stable mode for 4 minutes (240000 milliseconds) and ......

= MOV 0 R39

= CNCWR(R39,DISTX,M302) = CNCWR(R39,DISTY,M302) = CNCWR(R39,DISTZ,M302)

.... reset to "0" the count of the distance travelled by each axis.

Activate the lubricating (lubing) operation

T2 OR T3 OR T5 OR T6 = O-LUBING

If any of these conditions is met, the lubing output will be activated.

DFD O-LUBING = TRS2 = TRS3 = TRS4 = TRS5 = TRS6

Once the lubricating operation has concluded, All timers must be reset to "0".

---- *Coolant treatment* ----

The CNC executes function M08 to turn the coolant on and function M09 to turn it off.

Also, in this case, the operator has a switch to select whether the coolant is activated manually by the operator or automatically by the CNC.

I-COOLMA	The operator control the coolant. Manual mode.
I-COOLAU	The CNC controls the coolant. Automatic mode.
O-COOL	Coolant on/off output.

I-COOLMA OR (I-COOLAU AND M-08) = O-COOL

Coolant ON.

RESETOUT = NOT O-COOL = RES M-08

The coolant will be turned off when the CNC is reset to initial conditions (RESETOUT) or when executing functions M00, M02, M09 and M30.

This instruction does not contemplate functions M00, M02, M09 and M30 since the treatment of M, S, T functions turns mark M-08 off when activating any of them.

----- *Treatment of the general CNC input AUXEND* -----

It is advisable to have one single instruction to control each one of the logic CNC inputs, thus preventing undesired functioning.

When having several instructions which can activate or deactivate an input, the PLC will always assign the result of analyzing the last one of those instructions.

This example shows how to group in a single instruction all the conditions that activate or deactivate one logic CNC input.

NOT T1 AND NOT M-41 AND NOT M-42 = AUXEND

Input AUXEND will remain low while:

- The "Treatment of the MSTROBE, TSTROBE, STROBE signals" is in progress (timer T1 active)
- A spindle range change is being performed (M-41, M-42)

----- *Keyboard simulation* -----

With this example it is possible to simulate the theoretical path of part-program P12 whenever the operator requests it.

To do this, follow these steps:

- Indicate to the CNC that from now on the keys will come from the PLC.
- Simulate all the necessary steps sending the code of each one of the keys.
- Indicate to the CNC that from now on the keys will be coming from the CNC keyboard, not from the PLC.

In order to make sending the keys easier, a subroutine is used which utilizes the following parameters:

SENDKEY	(Send Key) Calling parameter that must be activated whenever a key is to be sent.
KEYCODE	(Code of the key) Calling parameter that must contain the code corresponding to the key being simulated.
SENTOK	(Sent OK) Outgoing parameter indicating that the key code has been sent successfully.

DFU I-SIMULA = SET M120 = ERA M121 126

Whenever the operator requests the simulation (I-SIMULA), marks M120 through M126 must be activated....

= MOV PLCKEY KEYBOARD = CNCWR (KEYBOARD, KEYSRC, M100)

.. indicate to the CNC that, from now on, the keys will be coming from the PLC (PLCKEY)

= MOV MAINMENU KEYCODE = SET SENDKEY

... and send the code for the "MAIN MENU" key.

M120 AND SENTOK = RES M120 = RES SENTOK = SET M121

If the previous key was sent out successfully (SENTOK), flags M120 and SENTOK will be turned off, the flag for the next stage (M121) is activated ....

= MOV SIMULATE KEYCODE = SET SENDKEY

... and the code for the SIMULATE key (F2) is sent out.

M121 AND SENTOK = RES M121 = RES SENTOK = SET M122

If the previous key was sent out successfully (SENTOK), flags M121 and SENTOK will be turned off, the flag for the next stage (M122) is activated ....

= MOV KEY1 KEYCODE = SET SENDKEY

... ...and the code for the "1" key is sent out.

M122 AND SENTOK = RES M122 = RES SENTOK = SET M123

If the previous key was sent out successfully (SENTOK), flags M122 and SENTOK will be turned off, the flag for the next stage (M123) is activated ....

= MOV KEY2 KEYCODE = SET SENDKEY

... ...and the code for the "2" key is sent out.

M123 AND SENTOK = RES M123 = RES SENTOK = SET M124

If the previous key was sent out successfully (SENTOK), flags M123 and SENTOK will be turned off, the flag for the next stage (M124) is activated ....

= MOV ENTER KEYCODE = SET SENDKEY

... ...and the code for the "ENTER" key is sent out.

M124 AND SENTOK = RES M124 = RES SENTOK = SET M125

If the previous key was sent out successfully (SENTOK), flags M124 and SENTOK will be turned off, the flag for the next stage (M125) is activated ....

= MOV THEOPATH KEYCODE = SET SENDKEY

... and the code for the "THEORETICAL PATH" (F1) is sent out.

M125 AND SENTOK = RES M125 = RES SENTOK = SET M126

If the previous key was sent out successfully (SENTOK), flags M125 and SENTOK will be turned off, the flag for the next stage (M126) is activated ....

= MOV START KEYCODE = SET SENDKEY

... and the code for the START key is sent out.

M126 AND SENTOK = RES M126 = RES SENTOK

If the last key was sent out successfully (SENTOK), flags M126 and SENTOK will be turned off....

= MOV CNCKEY KEYBOARD = CNCWR (KEYBOARD, KEYSRC, M100)

.. and the CNC is "told" that from now on the keys will be coming from CNC keyboard (CNCKEY), not from the PLC.

--- Subroutine used to send a key ---

SENDKEY =SET M100 =SET M101 =SET M102 =RES SENDKEY

To send a key (SENDKEY), set to "1" internal marks M100 through M102 and reset the SENDKEY flag to "0".

M100 = CNCWR (KEYCODE, KEY, M100)

Sends to the CNC the code of the key to be simulated (KEYCODE). If this command is not executed correctly (M100=1), the PLC will try again on the next cycle scan.

M101 AND NOT M100 = CNCRD (KEY, LASTKEY, M101)

If the previous command was executed correctly, (M100=0), it reads the last key accepted by the CNC (LASTKEY).

M102 AND NOT M101 AND CPS LASTKEY EQ KEYCODE

If the previous command was executed correctly (M101=0) and the CNC accepted the key sent to it (LASTKEY = KEYCODE), ...

= RES M102 = SET SENTOK

.... the flag is turned off (M102=0) and the key is considered to be sent out successfully (SENTOK=1)...

= NOT M101 ... But if the CNC did not accept the key sent to it, it waits until it does (M101=1).

End of subroutine

END End of program

User notes:

User notes:

8040 CNC

FAGOR

REF. 0204

(SOFT M: 5.3x)
(SOFT T: 6.3x)

APPENDIX

APPENDIX

TECHNICAL SPECIFICATIONS OF THE CNC

General characteristics

4 feedback inputs for the axes.

4 analog outputs to control the axes (±10 V).

1 feedback input for the spindle encoder.

1 analog output to control the spindle (±10 V).

2 feedback inputs for the electronic handwheels.

2 inputs for digital probes (TTL or 24 Vdc)

0.0001mm or 0.00001 inch resolution

Multiplying factor up to x 25 with sinewave input.

Feedrate from 0.0001 to 99999.9999 mm/min. (0.00001 - 3937 inches/min.).

Maximum travel ±99999.9999 mm (±3937 inches) (±3937 inches).

1 RS232C communication line

56 optocoupled digital inputs

32 optocoupled digital outputs

Remote modules for digital I/O expansion.

32-bit processor

Math coprocessor

Graphics coprocessor.

1Mb CNC program memory.

Block processing time of 6.5 ms.

Configurable sample time: 2, 3, 4, 5 or 6 ms.

System software in 7 languages.

Approximate weight: 7.5 Kg.

Maximum consumption in normal operation : 60 W.

Color monitor

Technology: Color TFT LCD

Diagonal display area dimension: 10,4”

Resolution: VGA 3 x 640 x 480 pixels.

Number of Colors: 262144 Colors (6 bit for each subpixel RGB)

Backlit with 2 cold-cathode fluorescent lamps.

Monochrome monitor

Technology: LCD STN

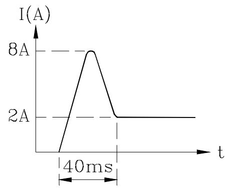
Diagonal display area dimension: 10,4”

Resolution: 640 x 480 pixels.

8 grey ranges

Backlit with 1 cold-cathode fluorescent lamp.

Power supply



Nominal voltage: 20V minimum and 30V maximum

Ripple: 4V

Nominal current: 2A

Current peak on power-up: 8A

The figure shows the shape of the supply current on power-up

PLC

Memory: 100 Kbytes

512 inputs.	512 outputs.
2047 user marks.	256 32-bit registers.
256 32-bit counters.	256 32-bit timers.

Programming in mnemonic.

1 millisecond time unit.

5V probe input.

Typical value 0.25 mA. @ Vin = 5V.

High threshold (logic level “1”) V_{IH}: from +2.4 Vdc up.

Low threshold (logic level “0”) V_{IL}: Below +0.9 Vdc

Maximum nominal voltage Vimax = +15 Vcc.

24V probe input.

Typical value 0.30 mA. @ Vin = 24V.

High threshold (logic level “1”) V_{IH}: from +12,5 Vdc up.

Low threshold (logic level “0”) V_{IL}: Below +4 Vdc

Maximum nominal voltage Vimax = +35 Vcc.

Digital inputs

Nominal voltage + 24 Vdc.

Maximum nominal voltage + 30 Vdc.

Minimum nominal voltage + 18 Vdc.

High threshold (logic level “1”) V_{IH}: from +18 Vdc up.

Low threshold (logic level “0”) V_{IL}: Under +5 Vdc or not connected.

Typical consumption of each input 5 mA.

Maximum consumption of each input 7 mA.

Protection by means of galvanic isolation by optocouplers.

Protection against reverse connection up to -30 Vdc.

Digital outputs

Nominal supply voltage + 24 Vdc.

Maximum nominal voltage + 30 Vdc.

Minimum nominal voltage + 18 Vdc.

Output voltage Vout = Supply voltage (Vdc) -3 V

Maximum output current 100 mA

Protection by means of galvanic isolation by optocouplers.

Protection against shortcircuits with external recovery diodes.

Feedback input for the axes

+5V power consumption. 1A (250 mA each axis)

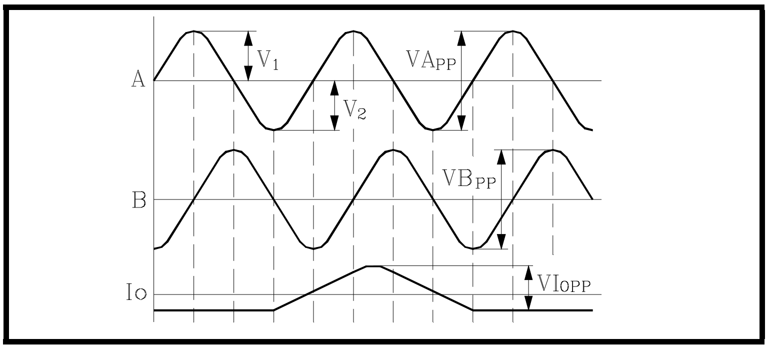
Work levels for differential square signal.

Maximum frequency: 400 KHz
Maximum separation between flanks: 460ns.
Phase shift 90° ±20°
Vmax. in common mode: ±7V.
Vmax. in differential mode: ±6V
Hysteresis: 0.2 V
Maximum differential input current: 3mA.

Work levels for non-differential square signal.

Maximum frequency: 400 KHz
Maximum separation between flanks: 460ns.
Phase shift 90° ±20°
High threshold (logic level 1) 1.25V < VIH < 7V.
Low threshold (logic level 0) -7V < VIH < 1V.
Vmax. ±7V
Hysteresis: 0.25 V
Maximum input current: 3mA.

Work levels for sinusoidal signal



Maximum frequency: 250 KHz

A and B signals

Amplitude: 0.6 ÷ 1.2Vpp
Centered: |V1-V2| / 2Vpp =< 6.5%
Relationship: VApp / VBpp = 0.8 ÷ 1.25
Phase shift: 90° ± 10°

Reference mark (I0)

Amplitude: 0.2 ÷ 0.85V
Width: T-90° =< I0 =< T+180°

Analog outputs for the axes

Command voltage within ±10 V, 16-bit solution

Minimum impedance of the connected connector 10 KΩ

Shielded cable should be used.

Spindle feedback input

+5V power consumption. 1A (250 mA each axis)

Work levels for differential square signal.

- Maximum frequency: 400 KHz
- Maximum separation between flanks: 460ns.
- Phase shift 90° ±20°
- Vmax. in common mode: ±7V.
- Vmax. in differential mode: ±6V
- Hysteresis: 0.2 V
- Maximum differential input current: 3mA.

Work levels for non-differential square signal.

- Maximum frequency: 400 KHz
- Maximum separation between flanks: 460ns.
- Phase shift 90° ±20°
- High threshold (logic level 1) 1.25V < VIH < 7V.
- Low threshold (logic level 0) -7V < VIH < 1V.
- Vmax. ±7V
- Hysteresis: 0.25 V
- Maximum input current: 3mA.

Analog outputs for the spindle

Command voltage within ±10 V, 16-bit solution

Minimum impedance of the connected connector 10 KΩ

Shielded cable should be used.

Feedback input for the handwheels

+5V power consumption. 1A (250 mA each axis)

Work levels for differential square signal.

- Maximum frequency: 200 KHz
- Maximum separation between flanks: 460ns.
- Phase shift 90° ±20°
- Vmax. in common mode: ±7V.
- Vmax. in differential mode: ±6V
- Hysteresis: 0.2 V
- Maximum differential input current: 3mA.

Work levels for non-differential square signal.

- Maximum frequency: 200 KHz
- Maximum separation between flanks: 460ns.
- Phase shift 90° ±20°
- High threshold (logic level 1) 1.25V < VIH < 7V.
- Low threshold (logic level 0) -7V < VIH < 1V.
- Vmax. ±7V
- Hysteresis: 0.25 V
- Maximum input current: 3mA.

Ambient conditions

Relative humidity: 30-95% non condensing

Operating temperature: between +5°C and +40°C with an average below +35°C.

Storage temperature: between -25° and +70°C

Maximum operating altitude : Meets the “IEC 1131-2” standard.

Packaging

Meets the “EN 60068-2-32” standard

Vibration

When running 10-50 Hz amplitude 0.2 mm. (1g)

While being shipped 10-50 Hz amplitude 1 mm (5g)

Free fall of packaged unit under Fagor ruling 1m.

Electromagnetic compatibility and Safety

See section ***0.6 Safety conditions***

Degree of protection

Central Unit : Front panel: IP54 and Rear panel: IP2X

Accessible parts inside the enclosure: IP1X

Operator panel: IP54

Battery

3.5 V lithium battery

Estimated life: 3 years

As from error indication (low battery) the information contained in the memory will be kept for 10 more days, with the CNC off. It must be replaced.

Precaution, due to the risk of explosion or combustion.

Warning:

To avoid excessive heating of internal circuits, the several ventilation slits must not be obstructed, it also being necessary to install a ventilation system which extracts hot air from the housing or desk which supports the Central Unit.

Due to the current state of the COLOR TFT LCD technology, all manufacturers accept the fact the LCD screens have a certain number of defective pixels.

The machine manufacturer must comply with the "EN 60204-1 (IEC-204-1)", standard regarding protection against electrical shock due to I/O contact failures with external power supply when not hooking up this connector before turning the power supply on.

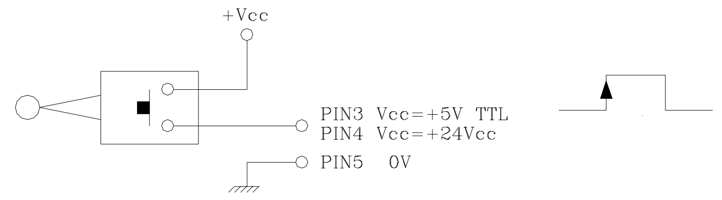
Access to the inside of the unit is absolutely forbidden to non authorized personnel.

Do not try to recharge the battery or expose it to temperatures over 100º C (212ºF). Do not short-circuit the terminals.

The CNC has two probe inputs, of 5Vdc and 24Vdc at connector X3

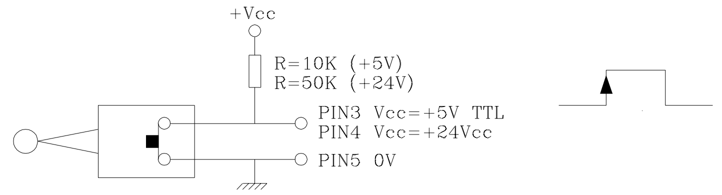
Depending on the type of connection applied the g.m.p. "PRBPULSE" (P39) must be set, indicating whether it operates with the leading edge or trailing edge of the signal which the probe provides.

Probe with "normally open contact" output



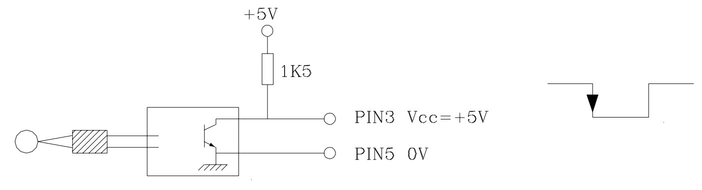
Active high (with an up flank)

Probe with "normally closed contact" output



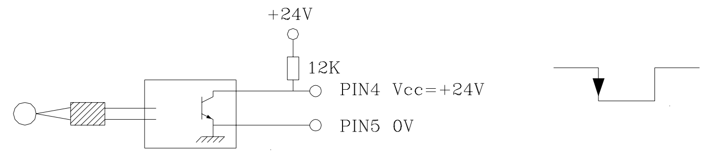
Active high (with an up flank)

Interface with output in open collector Connection to +5 V.



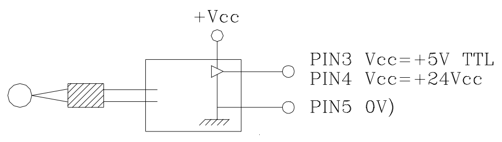
Active low (with a down flank)

Interface with output in open collector Connection to +24 V.



Active low (with a down flank)

Interface with output in PUSH-PULL



The active flank depends on the interface

SUMMARY OF PLC PROGRAMMING COMMANDS

Resources

Chapter 6.

Inputs:	I 1/512
Outputs:	O 1/512
User marks:	M 1/2000
Arithmetic flag marks:	M 2003
Clock marks:	M 2009/2024
Fixed status marks:	M 2046/2047
Marks associated with messages:	M 4000/4127
Marks associated with errors:	M 4500/4563
Screen marks:	M 4700/4955
CNC communication marks:	M 5000/5957
Timers:	T 1/256
Counters:	C 1/256
User registers	R 1/499
Registers for communication with the CNC	R 500/559

The value stored in each register will be considered by the PLC as a signed integer which could be referred to in the following formats:

Decimal

integer within ±2147483647.

Hexadecimal

Preceded by the $ sign and between 0 and FFFFFFFF

Binary

Preceded by the letter B and made up of up to 32 bits (1 or 0).

Directing instructions

Section 7.2

PRG	Main module.
CY1	First cycle module.
PE t	Periodic module. It will be executed every t time (in milliseconds).
END	End of module.
L 1/256	Label.
DEF	Symbol definition.
REA	All consultations will be performed on real values.
IMA	All consultations will be performed on image values.
IRD	Updates the "I" resources with the values of the physical inputs.
MRD	Updates resources M5000/5957 and R500/559 with the values of the logic CNC outputs.
OWR	Updates the physical outputs with the real values of the "O" resources.
MWR	Updates the logic CNC inputs (internal variables) with the values of resources M5000/5957 and R500/599
TRACE	Captures data for the Logic Analyzer while executing the PLC cycle.

Simple Consulting Instructions

Section 7.3

I	1/512	Inputs
O	1/512	Outputs
M	1/5957	Marks
T	1/256	Timers
C	1/256	Counters
B	0/31 R 1/499	Register Bits

Flank Detection Consulting Instructions

Section 7.3

DFU (Up flank detection) DFD (Down flank detection)	I 1/512 O 1/512 M 1/5957

Comparison Consulting Instructions

Section 7.3

CPS	T 1/256 C 1/256 R 1/559 #	GT GE EQ NE LE LT	T 1/256 C 1/256 R 1/559 #

Operators

Section 7.4

NOT	Inverts the result of the consulting instruction it precedes.
AND	Performs the logic function "AND" between consulting instructions.
OR	Performs the logic function "OR" between consulting instructions.
XOR	Performs the logic function "EXCLUSIVE OR" between consulting instructions.

Assignment Binary Action Instructions

Section 7.5.1

= I	1/512	Inputs
= O	1/512	Outputs
= M	1/5957	Marks
= TEN	1/256	Timer enable
= TRS	1/256	Timer reset
= TGn	1/256 n/R	Timer trigger input
= CUP	1/256	Counter count up
= CDW	1/256	Counter count down
= CEN	1/256	Counter enable
= CPR	1/256 n/R	Counter preset
= B	0/31 R 1/499	Register Bits

Conditioned binary actions instructions

Section 7.5.2

= SET If the logic expression is "1", this action assigns a "1" to the resource.

= RES If the logic expression is "1", this action assigns a "0" to the resource.

= CPL If the logic expression is "1", this action complements the logic state of the resource.

= SET = RES = CPL	I 1/512 O 1/512 M 1/5957 B 0/31 R 1/559

Sequence breaking action instructions

Section 7.5.3

= JMP L 1/256	Unconditional Jump.
= RET	Return or End of Subroutine.
= CAL L 1/256	Call to a Subroutine.

Arithmetic action instructions

Section 7.5.4

= MOV Transfers the logic states of the indicated source to the indicated destination.

= NGU Complements all register bits.

= NGS Changes the sign of the Register contents.

= ADS Adds the contents of a two registers or a number and a register content.

= SBS Subtracts between the contents of two registers or between a number and a register content.

= MLS Multiplies the contents of two registers or a number and a register content.

= DVS Divides the contents of two registers or a number and a register content.

= MDS Module between registers contents or between a number and a register content.

	Code	Code	Source code	Destination code	Number of bits to transmit
MOV	I 1/512 O 1/512 M 1/5957 T 1/256 C 1/256 R 1/559 #	I 1/512 O 1/512 M 1/5957 R 1/559	0 (Bin) 1 (BCD)	0 (Bin) 1 (BCD)	32 28 24 20 16 12 8 4

ADS SBS MLS DVS MDS	R1/559 #	R1/559 #	R1/559

Logic action instructions

Section 7.5.5

= AND Logic AND operation between register contents or between a number and a register content.

= OR Logic OR operation between register contents or between a number and a register content.

= XOR Logic XOR operation between register contents or between a number and a register content.

= RR 1/2 Right-hand register rotation.

= RR 1/2 Left-hand register rotation.

AND OR XOR	R1/559 #	R1/559 #	R1/559

	Code	Nr of repetitions	Code
RR1 RR2 RL1 RL2	R1/559	R1/559 0/31	R1/559

Specific action instructions

Section 7.5.6

= ERA Group erase

= CNCRD CNCRD (Variable, R1/559, M1/4955)
Read internal CNC variables.

= CNCWR CNCWR (R1/559, Variable, M1/5957)
Write internal CNC variables.

= PAR PAR R1/559 M1/5957
Parity of register

ERA	I O M T C R	1/512 1/512 1/5957 1/256 1/256 1/559	1/512 1/512 1/5957 1/256 1/256 1/559

INTERNAL CNC VARIABLES

Symbols R indicates that the variable can be read.
W indicates that the variable can be modified.

Variables associated with tools.

Variable	CNC	PLC	DNC	*Section 10.1*
TOOL	R	R	R	Number of active tool.
TOD	R	R	R	Number of active tool offset.
NXTOOL	R	R	R	Number of the next requested tool waiting for M06.
NXTOD	R	R	R	Number of the next tool's offset.
TMZPn	R	R	-	(n) tool's position in the tool magazine.
TLFDn	R/W	R/W	-	(n) tool's offset number.
TLFFn	R/W	R/W	-	(n) tool's family code.
TLFNn	R/W	R/W	-	Nominal life assigned to tool (n).
TLFRn	R/W	R/W	-	Real life value of tool (n).
TMZTn	R/W	R/W	-	Contents of tool magazine position (n).

Specific of Mill model

Variable	CNC	PLC	DNC	*Section 10.1*
TORn	R/W	R/W	-	Tool radius value of offset (n)
TOLn	R/W	R/W	-	Tool length value of offset (n)
TOIn	R/W	R/W	-	Tool radius wear (I) of offset (n)
TOKn	R/W	R/W	-	Tool length wear (K) of offset (n)

Specific of Lathe model

Variable	CNC	PLC	DNC	*Section 10.1*
TOXn	R/W	R/W	-	Tool length offset (n) along X axis.
TOZn	R/W	R/W	-	Tool length offset (n) along Z axis.
TOFn	R/W	R/W	-	Location code (F) of offset (n).
TORn	R/W	R/W	-	Tool radius (R) value of offset (n).
TOIn	R/W	R/W	-	Tool length wear (I) of offset (n) along X axis.
TOKn	R/W	R/W	-	Tool length wear (K) of offset (n) along Z axis.
NOSEAn:	R/W	R/W	-	Cutter angle of indicated tool.
NOSEWn	R/W	R/W	-	Cutter width of indicated tool.
CUTAn	R/W	R/W	-	Cutting angle of indicated tool.

Variables associated with zero offsets.

Variable	CNC	PLC	DNC	*Section 10.2*
ORG(X-C)	R	R	-	Active zero offset on the selected axis. The value of the additive offset indicated by the PLC is not included in this value.
PORGF	R	-	R	Abscissa coordinate value of polar origin.
PORGS	R	-	R	Ordinate coordinate value of polar origin.
ORG(X-C)n	R/W	R/W	R	Zero offset (n) value of the selected axis.
PLCOF(X-C)	R/W	R/W	R	Value of the additive Zero Offset activated via PLC.

Variables associated with machine parameters

Variable	CNC	PLC	DNC	Section 10.3
MPGn	R	R	-	Value assigned to general machine parameter (n).
MP(X-C)n	R	R	-	Value assigned to axis machine parameter (n) (X-C)
MPSn	R	R	-	Value assigned to machine parameter (n) of the main spindle.
MPSSn	R	R	-	Value assigned to machine parameter (n) of the second spindle.
MPASn	R	R	-	Value assigned to machine parameter (n) of the auxiliary spindle.
MPLCn	R	R	-	Value assigned to machine parameter (n) of the PLC.

Variables associated with work zones

Variable	CNC	PLC	DNC	Section 10.4
FZONE	R	R/W	R	Status of work zone 1.
FZLO(X-C)	R	R/W	R	Lower limit along the selected axis (X/C).
FZUP(X-C)	R	R/W	R	Upper limit along the selected axis (X/C).
SZONE	R	R/W	R	Status of work zone 2.
SZLO(X-C)	R	R/W	R	Lower limit along the selected axis (X/C).
SZUP(X-C)	R	R/W	R	Upper limit along the selected axis (X/C).
TZONE	R	R/W	R	Status of work zone 3.
TZLO(X-C)	R	R/W	R	Lower limit along the selected axis (X/C).
TZUP(X-C)	R	R/W	R	Upper limit along the selected axis (X/C).
FOZONE	R	R/W	R	Status of work zone 4.
FOZLO(X-C)	R	R/W	R	Lower limit along the selected axis (X/C).
FOZUP(X-C)	R	R/W	R	Upper limit along the selected axis (X/C).
FIZONE	R	R/W	R	Status of work zone 5.
FIZLO(X-C):	R	R/W	R	Lower limit along the selected axis (X/C).
FIZUP(X-C)	R	R/W	R	Upper limit along the selected axis (X/C).

Variables associated with feedrates

Variable	CNC	PLC	DNC	Section 10.5
FREAL	R	R	R	Real feedrate of the CNC in mm/min or inch/min.

Variables associated with function G94

Variable	CNC	PLC	DNC	Section 10.5
FEED	R	R	R	Active feedrate at the CNC in mm/min or inch/min.
DNCF	R	R	R/W	Feedrate selected via DNC.
PLCF	R	R/W	R	Feedrate selected via PLC.
PRGF	R	R	R	Feedrate selected by program.

Variables associated with function G95

Variable	CNC	PLC	DNC	Section 10.5
FPREV	R	R	R	Active feedrate at CNC, in m/rev or inch/rev.
DNCFPR	R	R	R/W	Feedrate selected via DNC.
PLCFPR	R	R/W	R	Feedrate selected via PLC.
PRGFPR	R	R	R	Feedrate selected by program.

Variables associated with function G32

Variable	CNC	PLC	DNC	Section 10.5
PRGFIN	R	R	R	Feedrate selected by program, in 1/min.

Variables associated with Feedrate Override (%)

Variable	CNC	PLC	DNC	Section 10.5
FRO	R	R	R	Feedrate Override (%) active at the CNC.
PRGFRO	R/W	R	R	Override (%) selected by program.
DNCFRO	R	R	R/W	Override (%) selected via DNC.
PLCFRO	R	R/W	R	Override (%) selected via PLC.
CNCFRO	R	R	R	Override (%) selected from the front panel knob.
PLCCFR	R	R/W	R	Override (%) of the PLC execution channel.

Variables associated with coordinates

Variable	CNC	PLC	DNC	Section 10.6
PPOS(X-C)	R	-	-	Programmed theoretical position value (coordinate).
POS(X-C)	R	R	R	Real position value of the indicated axis.
TPOS(X-C)	R	R	R	Theoretical position value (real + lag) of the indicated axis.
DPOS(X-C)	R	R	R	Theoretical position of the probe when the probe touched the part.
FLWE(X-C)	R	R	R	Following error of the indicated axis.
DEFLEX	R	R	R	Probe deflection along X axis. Mill model.
DEFLEY	R	R	R	Probe deflection along Y axis. Mill model.
DEFLEZ	R	R	R	Probe deflection along Z axis. Mill model.
DIST(X-C)	R/W	R/W	R	Distance travelled by the indicated axis.
LIMPL(X-C)	R/W	R/W	R	Second upper travel limit
LIMMI(X-C)	R/W	R/W	R	Second lower travel limit.

Variables associated with electronic handwheels

Variable	CNC	PLC	DNC	Section 10.7
HANPF	R	R	-	Pulses received from 1st handwheel since the CNC was turned on
HANPS	R	R	-	Pulses received from 2nd handwheel since the CNC was turned on
HANPT	R	R	-	Pulses received from 3rd handwheel since the CNC was turned on
HANPFO	R	R	-	Pulses received from 4th handwheel since the CNC was turned on
HANFCT	R	R/W	R	Multiplying factor different for each handwheel. When there are several
HBEVAR	R	R/W	R	HBE handwheel: reading enabled, axis being jogged and (x1, x10, x100) factor
MASLAN	R/W	R/W	R/W	Linear path angle for "Path handwheel" mode.
MASCFI	R/W	R/W	R/W	Arc center coordinates for "Path handwheel mode".
MASCSE	R/W	R/W	R/W	Arc center coordinates for "Path handwheel mode".

Variables associated with the main spindle

Variable	CNC	PLC	DNC	Section 10.8
SREAL	R	R	R	Real spindle speed in rpm.
SPEED	R	R	R	Active spindle speed at the CNC.
DNCS	R	R	R/W	Spindle speed selected via DNC.
PLCS	R	R/W	R	Spindle speed selected via PLC.
PRGS	R	R	R	Spindle speed selected by program.
SSO	R	R	R	Spindle Speed Override (%) active at the CNC.
PRGSSO	R/W	R	R	Override (%) selected by program.
DNCSSO	R	R	R/W	Override (%) selected via DNC.
PLCSSO	R	R/W	R	Override (%) selected via PLC.
CNCSSO	R	R	R	Spindle Speed Override (%) selected from front panel.
SLIMIT	R	R	R	Spindle speed limit, in rpm, active at the CNC.
DNCSL	R	R	R/W	Spindle speed limit selected via DNC.
PLCSL	R	R/W	R	Spindle speed limit selected via PLC.
PRGSL	R	R	R	Spindle speed limit selected by program.
POSS	R	R	R	Real Spindle position. Between ±999999999 ten-thousandths of a degree.
RPOSS	R	R	R	Real Spindle position. Between 0 and 360º (in ten-thousandths of a degree.
TPOSS	R	R	R	Theoretical Spindle position (real + lag). Between ±999999999 ten-thousandths of a degree.
RTPOSS	R	R	R	Theoretical Spindle position (real + lag). Between 0 and 360º (in ten-thousandths of a degree.
FLWES	R	R	R	Spindle following error in degrees.
SYNCER	R	R	R	Error of second spindle (synchronized) following the main spindle.

Lathe model related variables

Variable	CNC	PLC	DNC	Section 10.8
CSS	R	R	R	Constant surface speed active at the CNC in meters/min or ft/min.
DNCCSS	R	R	R/W	Constant surface speed selected via DNC.
PLCCSS	R	R/W	R	Constant surface speed selected via PLC.
PRGCSS	R	R	R	Constant surface speed selected by program.

Variables associated with the second spindle

Variable	CNC	PLC	DNC	*Section 10.9*
SSREAL	R	R	R	Real spindle speed in rpm.
SSPEED	R	R	R	Active spindle speed at the CNC.
SDNCS	R	R	R/W	Spindle speed selected via DNC.
SPLCS	R	R/W	R	Spindle speed selected via PLC.
SPRGS	R	R	R	Spindle speed selected by program.
SSSO	R	R	R	Spindle Speed Override (%) active at the CNC.
SPRGSO	R/W	R	R	Override (%) selected by program.
SDNCSO	R	R	R/W	Override (%) selected via DNC.
SPLCSO	R	R/W	R	Override (%) selected via PLC.
SCNCSO	R	R	R	Spindle Speed Override (%) selected from front panel.
SSLIMI	R	R	R	Spindle speed limit, in rpm, active at the CNC.
SDNCSL	R	R	R/W	Spindle speed limit selected via DNC.
SPLCSL	R	R/W	R	Spindle speed limit selected via PLC.
SPRGSL	R	R	R	Spindle speed limit selected by program.
SPOSS	R	R	R	Real Spindle position. Between ±999999999 ten-thousandths of a degree.
SRPOSS	R	R	R	Real Spindle position. Between 0 and 360º (in ten-thousandths of a degree.
STPOSS	R	R	R	Theoretical Spindle position (real + lag). Between ±999999999 ten-thousandths of a degree.
SRTPOS	R	R	R	Theoretical Spindle position (real + lag). Between 0 and 360º (in ten-thousandths of a degree.
SFLWES	R	R	R	Spindle following error in degrees.

Lathe model related variables

Variable	CNC	PLC	DNC	*Section 10.9*
SCSS	R	R	R	Constant surface speed active at the CNC in meters/min or ft/min.
SDNCCS	R	R	R/W	Constant surface speed selected via DNC.
SPLCCS	R	R/W	R	Constant surface speed selected via PLC.
SPRGCS	R	R	R	Constant surface speed selected by program.

Variables associated with the live tool

Variable	CNC	PLC	DNC	*Section 10.10*
ASPROG	R	R	-	Rpm programmed in M45 S (within the associated subroutine).
LIVRPM	R	R	-	Rpm of the live tool in TC mode.

Variables associated with the PLC.

Variable	CNC	PLC	DNC	*Programming Manual*
PLCMSG	R	-	R	Number of the active PLC message with the highest priority.
PLCIn	R/W	-	-	32 PLC inputs starting from (n).
PLCOn	R/W	-	-	32 PLC outputs starting from (n).
PLCMn	R/W	-	-	32 PLC marks starting from (n).
PLCRn	R/W	-	-	(n) Register.
PLCTn	R/W	-	-	Indicated (n) Timer's count.
PLCCn	R/W	-	-	Indicated (n) Counter's count.

Variables associated with local and global parameters

Variable	CNC	PLC	DNC	*Section 10.11*
GUP n	-	R/W	-	Global parameter (P100-P299) (n).
LUP (a,b)	-	R/W	-	Indicated local (P0-P25) parameter (b) of the nesting level (a).
CALLP	R	-	-	Indicates which local parameters have been defined by means of a PCALL or MCALL instruction (calling a subroutine).

Sercos variables

Variable	CNC	PLC	DNC	*Section 10.12*
SETGE(X-C)	W	W	-	Gear ratio and parameter set of the (X-C) axis drive
SETGES	W	W	-	Gear ratio and parameter set of the main spindle
SSETGS	W	W	-	Gear ratio and parameter set of the second spindle
SVAR(X-C) id	R/W	-	-	Sercos variable sercos for the (X-C) axis "id"
SVARS id	R/W	-	-	Sercos variable sercos for the main spindle "id"
SSVAR id	R/W	-	-	Sercos variable sercos for the second spindle "id"
TSVAR(X-C) id	R	-	-	Third attribute of the sercos variable for the (X-C) axis "id"
TSVARS id	R	-	-	Third attribute of the sercos variable for the main spindle "id"
TSSVAR id	R	-	-	Third attribute of the sercos variable for the second spindle "id"

Software & hardware configuration variables

Variable	CNC	PLC	DNC	*Section 10.13*
HARCON	R	R	R	Indicates, with bits, the CNC hardware configuration.
IDHARH	R	R	R	Hardware identifier (8 least significant bits)
IDHARL	R	R	R	Hardware identifier (4 most significant bits)
SOFCON	R	R	R	Software version of the CNC (bits 15-0) and HD (31-16)

Variables associated with telediagnosis

Variable	CNC	PLC	DNC	*Section 10.14*
HARSWA	R	R	R	Hardware configuration
HARSWB	R	R	R	Hardware configuration
HARTST	R	R	R	Hardware test
MEMTST	R	R	R	Memory test
NODE	R	R	R	Node number in the Sercos ring
VCHECK	R	R	R	Software version checksum

Operating-mode related variables

Variable	CNC	PLC	DNC	*Section 10.15*
OPMODE	R	R	R	Operating mode.
OPMODA	R	R	R	Operating mode when working in the main channel.
OPMODB	R	R	R	Type of simulation.
OPMODC	R	R	R	Axes selected by handwheel.

Other variables

Variable	CNC	PLC	DNC	*Section 10.16*
NBTOOL	R	R	R	Number of the tool being managed.
PRGN	R	R	R	Number of the program in execution.
BLKN	R	R	R	Label number of the last executed block.
GSn	R	-	-	Status of the indicated G function (n).
GGSA	-	R	R	Status of functions G00 thru G24.
GGSB	-	R	R	Status of functions G25 thru G49.
GGSC	-	R	R	Status of functions G50 thru G74.
GGSD	-	R	R	Status of functions G75 thru G99.
MSn	R	-	-	Status of the indicated M function (n)
GMS	-	-	R	Status of M functions: M (0..6, 8, 9, 19, 30, 41..44)
PLANE	R	R	R	Abscissa and ordinate axes of the active plane.
LONGAX	R	R	R	Axis affected by the tool length compensation (G15). Mill model.
MIRROR	R	R	R	Active mirror images.
SCALE	R	R	R	General scaling factor applied.
SCALE(X-C)	R	R	R	Scaling Factor applied only to the indicated axis.
ORGROT	R	R	R	Rotation angle (G73) of the coordinate system in degrees. Mill model.
ROTPF	R	-	-	Abscissa of rotation center. Mill model.
ROTPS	R	-	-	Ordinate of rotation center. Mill model.
PRBST	R	R	R	Returns probe status.
CLOCK	R	R	R	System clock in seconds.
TIME	R	R	R/W	Time in Hours, minutes and seconds.
DATE	R	R	R/W	Date in Year-Month-Day format
TIMER	R/W	R/W	R/W	Clock activated by PLC, in seconds.
CYTIME	R	R	R	Time to execute a part in hundredths of a second.
PARTC	R/W	R/W	R/W	Parts counter of the CNC.
FIRST	R	R	R	First time a program is executed.
KEY	R/W	R/W	R/W	keystroke code.
KEYSRC	R/W	R/W	R/W	Keystroke source, 0=keyboard, 1=PLC, 2=DNC
ANAIn	R	R	R	Voltage (in volts) of the indicated analog input (n).
ANAOn	R/W	R/W	R/W	Voltage (in volts) to apply to the indicated output (n).
CNCERR	-	R	R	Active CNC error number.
PLCERR	-	-	R	Active PLC error number.
DNCERR	-	R	-	Number of the error generated during DNC communications.
AXICOM	R	R	R	Pairs of axes switched with function G28.
TANGAN	R	R	R	Angular position, in degrees, with respect to the path.

Warning:

The "KEY" variable can be "written" at the CNC only via the user channel.

LOGIC CNC INPUTS AND OUTPUTS

General logic inputs

		Section 9.1
/EMERGEN	M5000	Stops the axes and the spindle. Displays the error.
/STOP	M5001	Stops execution of the part program, maintaining spindle rotation.
/FEEDHOL	M5002	Stops axis feed momentarily, maintaining spindle rotation.
/XFERINH	M5003	Prevents the next block from being executed, but finishes the one being executed.
CYSTART	M5007	Starts program execution.
SBLOCK	M5008	The CNC changes to Single Block execution mode.
MANRAPID	M5009	Selects rapid travel for all the movements that are executed in JOG Mode.
OVRCAN	M5010	Selects feedrate OVERRIDE at 100%.
LATCHM	M5011	The axes will move from the moment the corresponding JOG key is pressed until the STOP key is pressed.
ACTGAIN2	M5013	Indicates that the CNC assumes the 2nd range of gains.
RESETIN	M5015	Initial machining conditions selected by machine parameter.
AUXEND	M5016	Indicates that the execution of the M, S and T functions has completed.
TIMERON	M5017	Enables the timer:
TREJECT	M5018	Rejection of tool in use.
PANELOFF	M5019	Deactivation of keyboard.
PLCABORT	M5022	Possibility to abort the PLC channel
PLCREADY	M5023	PLC without errors.
INT1	M5024	Executes the interruption subroutine indicated in g.m.p. P35
INT2	M5025	Executes the interruption subroutine indicated in g.m.p. P36
INT3	M5026	Executes the interruption subroutine indicated in g.m.p. P37
INT4	M5027	Executes the interruption subroutine indicated in g.m.p. P38
BLKSKIP1	M5028	The “/ and /1” block skip condition is met.
BLKSKIP2	M5029	The “/2” block skip condition is met.
BLKSKIP3	M5030	The “/3” block skip condition is met.
M01STOP	M5031	Stops execution of the part program when the auxiliary M01 function is executed.
TOOLINSP	M5050	Tool inspection available in MC, MCO, TC, TCO modes
RETRACE	M5051	It activates function retracing
ACTLM2	M5052	Activates the second travel limits.
HNLINARC	M5053	Type of path with "Path Handwheel".
MASTRHND	M5054	Activates the "Path Handwheel" mode.
EXRAPID	M5057	Selects rapid travel for all the movements that are executed in Execution Mode.

Axis logic inputs.

	Axis 1 / Axis 2	Axis 3 / Axis 4	Axis 5 / Axis 6	Axis 7	Section 9.2
LIMIT+	M5100 M5150	M5200 M5250	M5300 M5350	M5400	Travel limit overrun. Stops the axes and the spindle. Displays the error.
LIMIT-*	M5101 M5151	M5201 M5251	M5301 M5351	M5401	Travel limit overrun. Stops the axes and the spindle. Displays the error.
DECEL*	M5102 M5152	M5202 M5252	M5302 M5352	M5402	Home switch pressed.
INHIBIT*	M5103 M5153	M5203 M5253	M5303 M5353	M5403	Inhibits axis movement.
MIRROR*	M5104 M5154	M5204 M5254	M5304 M5354	M5404	Applies mirror image.
SWITCH*	M5105 M5155	M5205 M5255	M5305 M5355	M5405	Swaps commands (axes with 1 drive)
DRO*	M5106 M5156	M5206 M5256	M5306 M5356	M5406	DRO axis. (DRO*=1 and SERVOON*=0).
SERVO*ON	M5107 M5157	M5207 M5257	M5307 M5357	M5407	Servo signal. (=1) closes the position loop
AXIS+*	M5108 M5158	M5208 M5258	M5308 M5358	M5408	Moves the axis in JOG mode. Similar to JOG keys.
AXIS-*	M5109 M5159	M5209 M5259	M5309 M5359	M5409	Moves the axis in JOG mode. Similar to JOG keys.
SPENA*	M5110 M5160	M5210 M5260	M5310 M5360	M5410	With Sercos. Speed Enable signal of the drive.
DRENA*	M5111 M5161	M5211 M5261	M5311 M5361	M5411	With Sercos. Drive Enable signal of the drive.
SYNCHRO*	M5112 M5162	M5212 M5262	M5312 M5362	M5412	Couples the axis with the one indicated by SYNCHRO*.
ELIMINA*	M5113 M5163	M5213 M5263	M5313 M5363	M5413	It does not display the axis and cancels the feedback alarms.
SMOTOF*	M5114 M5164	M5214 M5264	M5314 M5364	M5414	Cancels the SMOTIME filter, a.m.p. SMOTIME (P58).
LIM*OFF	M5115 M5165	M5215 M5265	M5315 M5365	M5415	It ignores software limits.

Spindle logic inputs.

Main		Second		Section 9.3
LIMIT+S	M5450	LIMIT+S2	M5475	Travel limit overrun. Stops the axes and the spindle. Displays the error.
LIMIT -S	M5451	LIMIT -S2	M5476	Travel limit overrun. Stops the axes and the spindle. Displays the error.
DECELS	M5452	DECELS2	M5477	Home switch pressed.
SPDLEINH	M5453	SPDLEIN2	M5478	Outputs a zero command for the spindle.
SPDLEREV	M5454	SPDLERE2	M5479	Reverses the spindle turning direction.
SMOTOFS	M5455	SMOTOFS2	M5480	Cancels the SMOTIME filter, s.m.p. SMOTIME (P46).
SERVOSON	M5457	SERVOSO2	M5482	Servo signal. (=1) to move the spindle in closed loop (M19).
GEAR1	M5458	GEAR12	M5483	Spindle gear 1 selected.
GEAR2	M5459	GEAR22	M5484	Spindle gear 2 selected.
GEAR3	M5460	GEAR32	M5485	Spindle gear 3 selected.
GEAR4	M5461	GEAR42	M5486	Spindle gear 4 selected.
SPENAS	M5462	SPENAS2	M5487	With Sercos. Speed Enable signal of the drive.
DRENAS	M5463	DRENAS2	M5488	With Sercos. Drive Enable signal of the drive.
PLCFM19	M5464	PLCFM192	M5489	Rapid synchronization feedrate, in M19.
M19FEED	R505	M19FEED2	R507	Rapid synchronization feedrate, in M19.

Main		Second		Section 9.3
PLCCNTL	M5465	PLCCNTL2	M5490	Spindle controlled directly by the PLC.
SANALOG	R504	SANALOG2	R506	Spindle analog voltage. Only for spindle controlled by PLC.

Logic inputs of the auxiliary spindle

		Section 9.4
SPENAAS	M5449	With Sercos. Speed Enable signal of the drive.
DRENAAS	M5448	With Sercos. Drive Enable signal of the drive.
PLCCNTAS	M5056	The auxiliary spindle is controlled directly by the PLC.
SANALOAS	R509	Analog voltage of the auxiliary spindle. Only for spindle controlled by PLC.

Key inhibiting logic inputs.

		Section 9.5
KEYDIS1	R500	Inhibit the operation of the panel keys.
KEYDIS2	R501	
KEYDIS3	R502	
KEYDIS4	R503	

Logic inputs of the PLC channel

		Section 9.6
/FEEDHOP	M5004	Stops the PLC axes momentarily, maintaining spindle rotation.
/XFERINP	M5005	Prevents the next block from being executed in the PLC channel, but finishes the one being executed.
AUXENDP	M5006	Indicates that the execution of the M, S and T functions has completed.

General logic outputs

		Section 9.7
CNCREADY	M5500	CNC without errors.
START	M5501	The CYCLE START key of the Front Panel has been pressed.
FHOUT	M5502	Indicates that program execution has been interrupted.
RESETOUT	M5503	Indicates that the CNC is set to initial conditions.
LOPEN	M5506	Indicates that the positioning loop for the axes is open.
/ALARM	M5507	An alarm or emergency condition was detected.
MANUAL	M5508	The Manual Operation (JOG) Mode has been selected.
AUTOMAT	M5509	The Automatic Operation Mode has been selected.
MDI	M5510	The MDI mode has been selected.
SBOUT	M5511	The Single Block Execution Mode has been selected.
CUSTOM	M5512	Selected work mode (=0) M or T; (=1) MC, MCO, TC or TCO
INCYCLE	M5515	The part program is being executed.
RAPID	M5516	A rapid traverse is being executed (G00).
TAPPING	M5517	A tapping cycle is being executed (G84).
THREAD	M5518	A threading block is being executed (G33).
PROBE	M5519	A probing movement is being executed (G75/G76).
ZERO	M5520	A machine reference search is being executed (G74).
RIGID	M5521	A rigid tapping block in execution. Mill model.
RETRAEND	M5522	Retrace function. All possible blocks have been retraced.
CSS	M5523	The G96 function is selected.
SELECT0	M5524	Position selected at the Front Panel switch
SELECT1	M5525	Position selected at the Front Panel switch
SELECT2	M5526	Position selected at the Front Panel switch
SELECT3	M5527	Position selected at the Front Panel switch
SELECT4	M5528	Position selected at the Front Panel switch
SELECTOR	R564	Position selected at the Front Panel switch
MSTROBE	M5532	Indicates that the auxiliary M functions which are indicated in registers R550 to R556 must be executed.

		Section 9.7
SSTROBE	M5533	Indicates that the auxiliary S function of register R557 must be executed.
TSTROBE	M5534	Indicates that the auxiliary T function of register R558 must be executed.
T2STROBE	M5535	Indicates that the auxiliary T function of register R559 must be executed.
S2MAIN	M5536	Indicates which spindle is controlled by the CNC.
ADVINPOS	M5537	For punch presses, it indicates that punching may begin.
INTEREND	M5538	It indicates that the interpolation is finished.
INPOS	M5539	The axes are in position.
DM00	M5547	Program interrupted after executing the auxiliary function M00.
DM01	M5546	Program interrupted after executing the auxiliary function M01.
DM02	M5545	The execution of the program has stopped after executing the auxiliary M02 function.
DM03	M5544	The spindle is turning clockwise (M03).
DM04	M5543	The spindle is turning counterclockwise (M04).
DM05	M5542	The spindle is stopped (M05).
DM06	M5541	The auxiliary M06 function has been executed.
DM08	M5540	The coolant output has been activated (M08).
DM09	M5555	The coolant output has been canceled (M09).
DM19	M5554	A block with spindle stop has been executed (M19).
DM30	M5553	The program concluded after executing the auxiliary M30 function.
DM41	M5552	First spindle speed gear (range) selected (M41).
DM42	M5551	Second spindle speed gear (range) selected (M42).
DM43	M5550	Third spindle speed gear (range) selected (M43).
DM44	M5549	Fourth spindle speed gear (range) selected (M44).
DM45	M5548	Auxiliary spindle or live tool selected (M45).
TANGACT	M5558	G45 active
SYNCPOSI	M5559	Spindles synchronized in position.
SYNSPEED	M5560	Spindles synchronized in speed.
SYNCHRON	M5561	G77S function selected (spindle synchronization).
SERPLCAC	M5562	Requested change of parameter set and gear ratio in progress.

Logic outputs of the axes

	Axis 1 Axis 2	Axis 3 Axis 4	Axis 5 Axis 6	Axis 7	Section 9.8
ENABLE*	M5600 M5650	M5700 M5750	M5800 M5850	M5900	Enables axis movement.
DIR*	M5601 M5651	M5701 M5751	M5801 M5851	M5901	Indicate axis moving direction.
REFPOIN*	M5602 M5652	M5702 M5752	M5802 M5852	M5902	Home search done.
DRSTAF*	M5603 M5653	M5703 M5753	M5803 M5853	M5903	With Sercos. They indicate servo drive status.
DRSTAS*	M5604 M5654	M5704 M5754	M5804 M5854	M5904	With Sercos. They indicate servo drive status.
ANT*	M5606 M5656	M5706 M5756	M5806 M5856	M5906	If distance < MINMOVE (P54), ANT*=1
INPOS*	M5607 M5657	M5707 M5757	M5807 M5857	M5907	Axis in position.

Spindle logic outputs.

Main		Second		Section 9.9
ENABLES	M5950	ENABLES2	M5975	Enables spindle movement.
DIRS	M5951	DIRS2	M5976	Spindle turning direction
REFPOINS	M5952	REFPOIS2	M5977	The spindle has been already referenced (homed).
DRSTAFS	M5953	DRSTAFS2	M5978	With Sercos. They indicate servo drive status.

Main		Second		Section 9.9
DRSTASS	M5954	DRSTASS2	M5979	With Sercos. They indicate servo drive status.
CAXIS	M5955	CAXIS2	M5980	"C" axis active.
REVOK	M5956	REVOK2	M5981	Spindle rpm correspond to programmed speed.
INPOSS	M5957	INPOSS2	M5982	Spindle in position.

Logic outputs of the auxiliary spindle

		Section 9.10
DRSTAFAS	M5557	With Sercos. They indicate servo drive status.
DRSTASAS	M5556	

Auxiliary M, S, T function transfer

		Section 8.1
MBCD1	R550	Auxiliary M function to be executed in the main channel.
MBCD2	R551	Auxiliary M function to be executed in the main channel.
MBCD3	R552	Auxiliary M function to be executed in the main channel.
MBCD4	R553	Auxiliary M function to be executed in the main channel.
MBCD5	R554	Auxiliary M function to be executed in the main channel.
MBCD6	R555	Auxiliary M function to be executed in the main channel.
MBCD7	R556	Auxiliary M function to be executed in the main channel.
MBCDP1	R565	Auxiliary M function to be executed in the PLC channel.
MBCDP2	R566	Auxiliary M function to be executed in the PLC channel.
MBCDP3	R567	Auxiliary M function to be executed in the PLC channel.
MBCDP4	R568	Auxiliary M function to be executed in the PLC channel.
MBCDP5	R569	Auxiliary M function to be executed in the PLC channel.
MBCDP6	R570	Auxiliary M function to be executed in the PLC channel.
MBCDP7	R571	Auxiliary M function to be executed in the PLC channel.
SBCD	R557	Spindle speed in BCD (2 or 8 digits).
TBCD	R558	Indicates the magazine position of the tool to be placed in the spindle.
T2BCD	R559	Magazine position (pocket) for the tool.

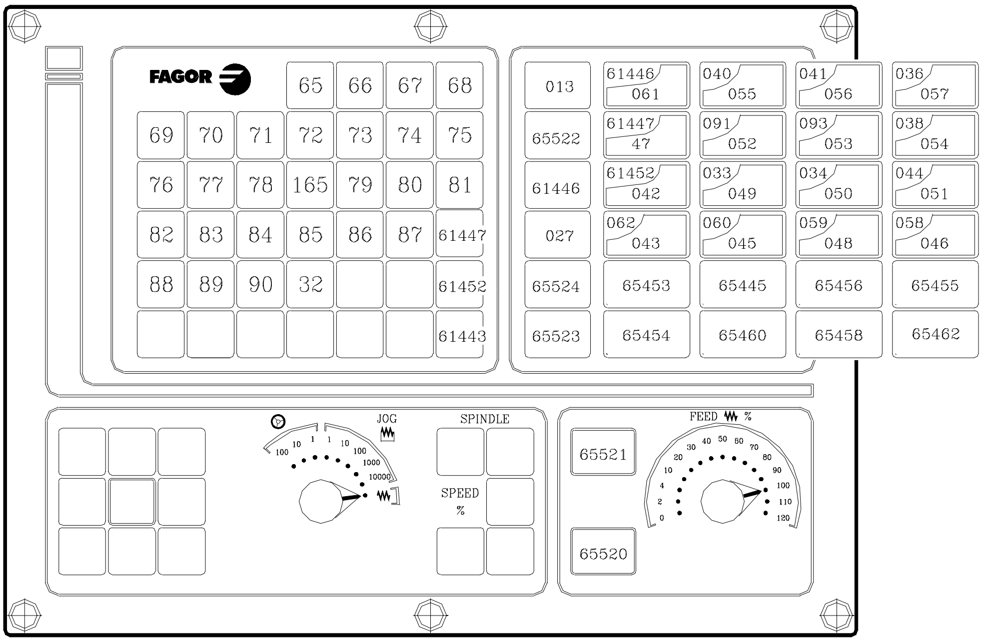
Logic outputs of key status

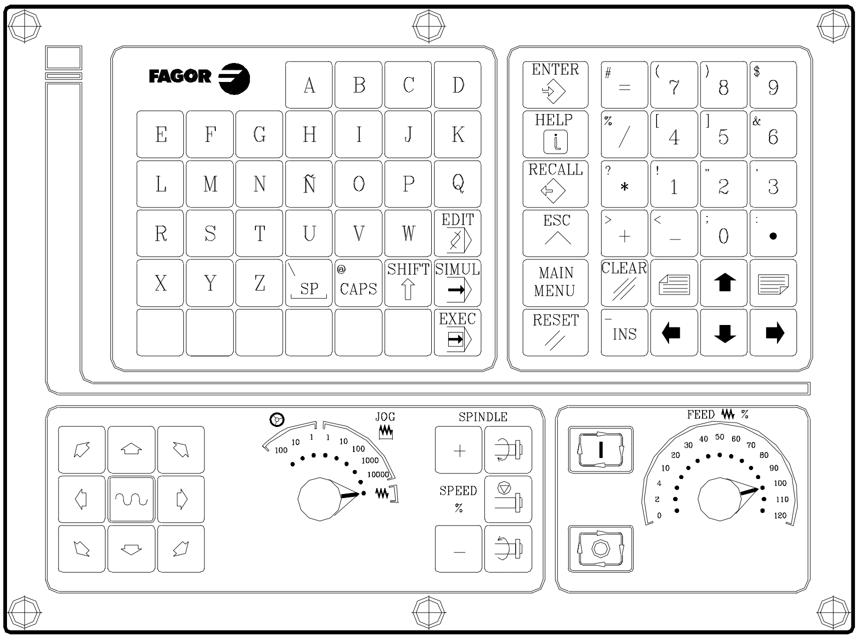
		Section 9.11
KEYBD1	R560	Indicate whether a key of the operator panel is pressed.
KEYBD2	R561	
KEYBD3	R562	
KEYBD4	R563	

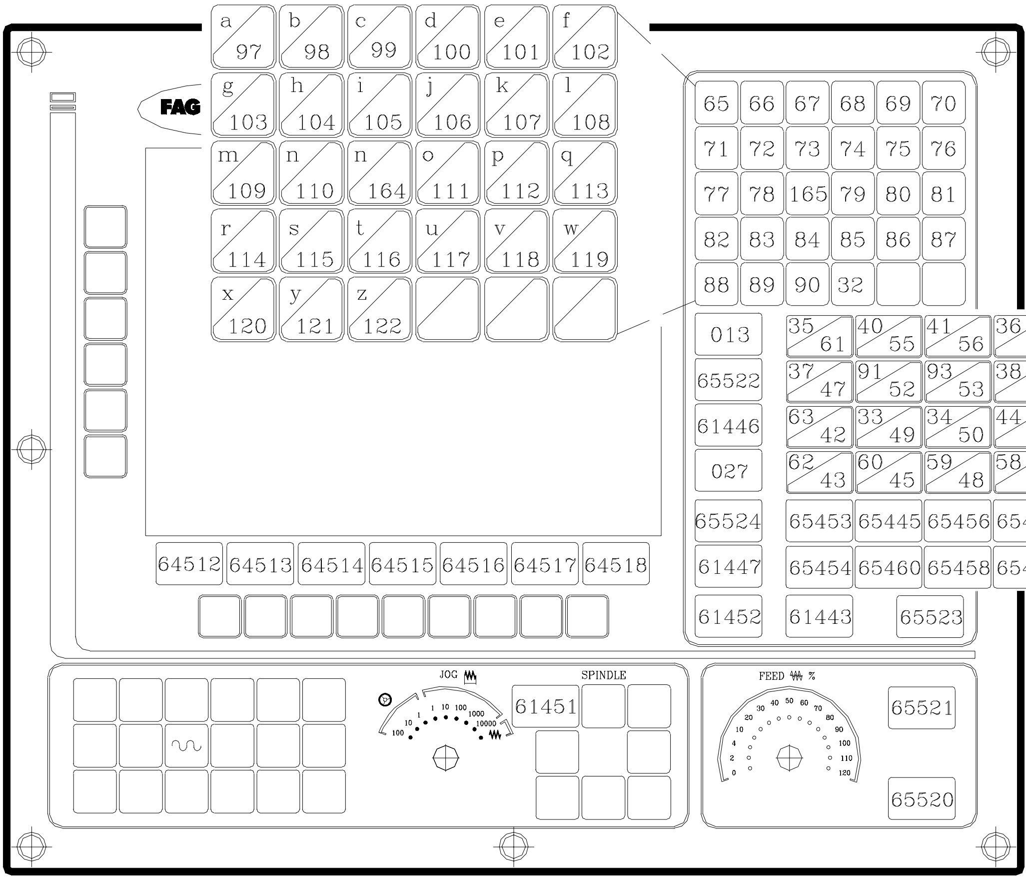
2-DIGIT BCD CODE OUTPUT CONVERSION TABLE

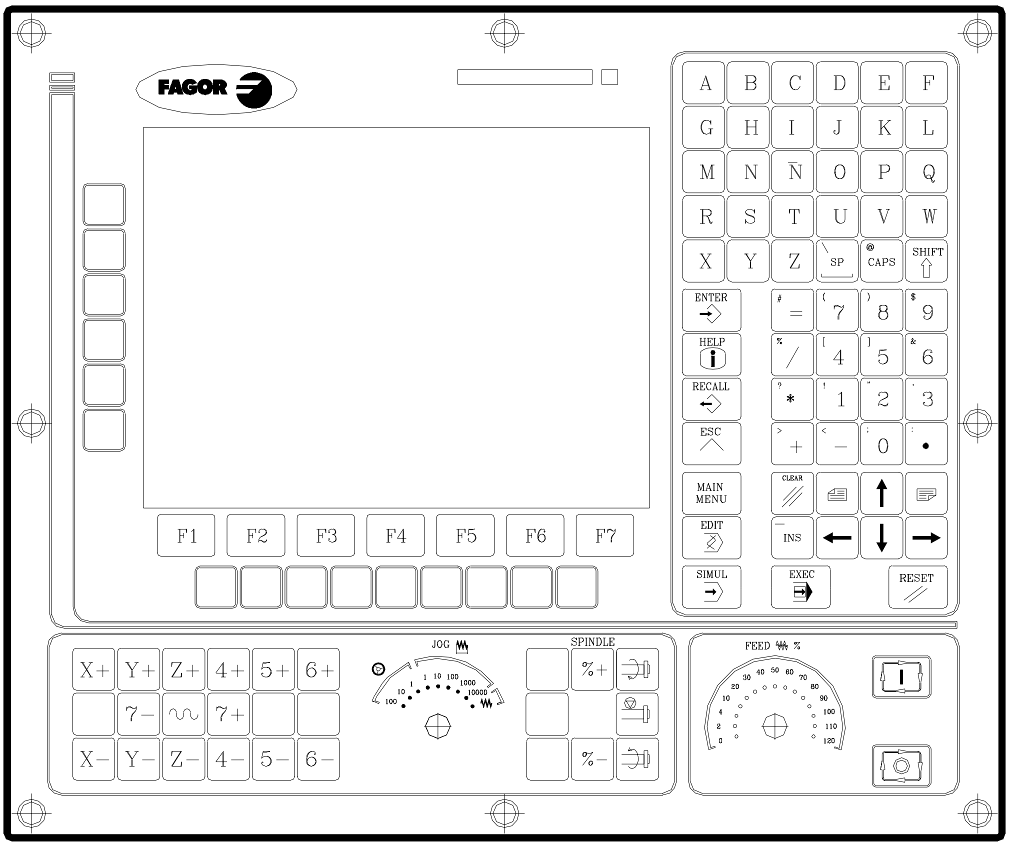
Programmed S	S BCD	Programmed S	S BCD	Programmed S	S BCD
0	00	50-55	54	800-899	78
1	20	56-62	55	900-999	79
2	26	63-70	56	1000-1119	80
3	29	71-79	57	1120-1249	81
4	32	80-89	58	1250-1399	82
5	34	90-99	59	1400-1599	83
6	35	100-111	60	1600-1799	84
7	36	112-124	61	1800-1999	85
8	38	125-139	62	2000-2239	86
9	39	140-159	63	2240-2499	87
10-11	40	160-179	64	2500-2799	88
12	41	180-199	65	2800-3149	89
13	42	200-223	66	3150-3549	90
14-15	43	224-249	67	3550-3999	91
16-17	44	250-279	68	4000-4499	92
18-19	45	280-314	69	4500-4999	93
20-22	46	315-354	70	5000-5599	94
23-24	47	355-399	71	5600-6299	95
25-27	48	400-449	72	6300-7099	96
28-31	49	450-499	73	7100-7999	97
32-35	50	500-559	74	8000-8999	98
36-39	51	560-629	75	9000-9999	99
40-44	52	630-709	76		
45-49	53	710-799	77		

Alphanumeric operator panel (M-T models)

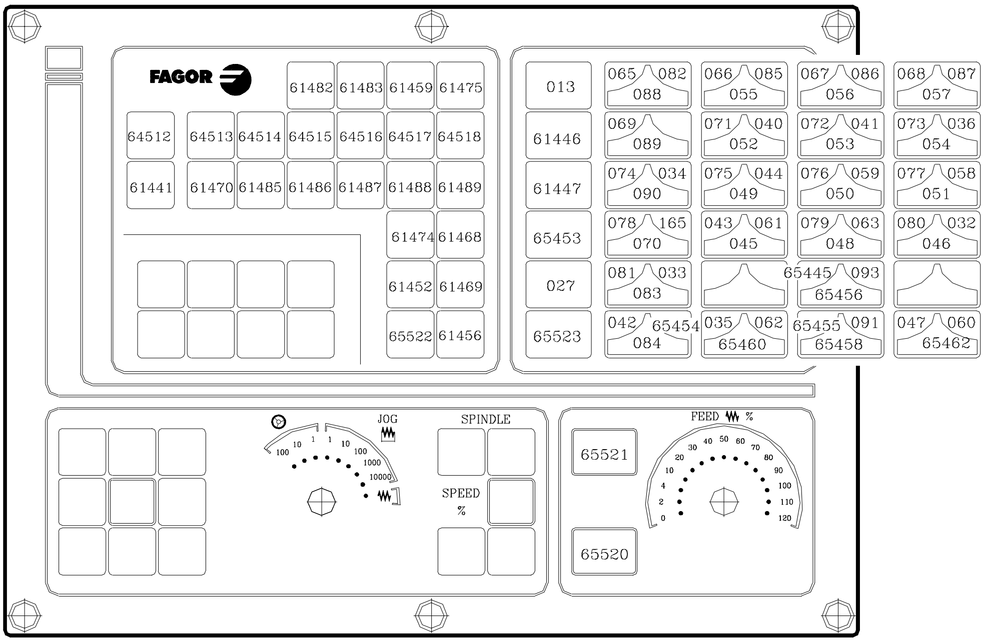


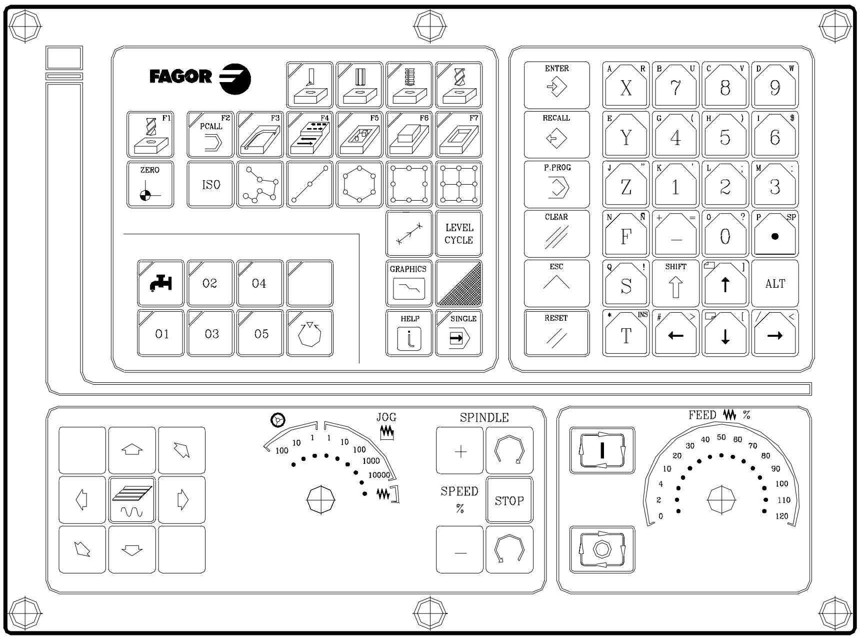


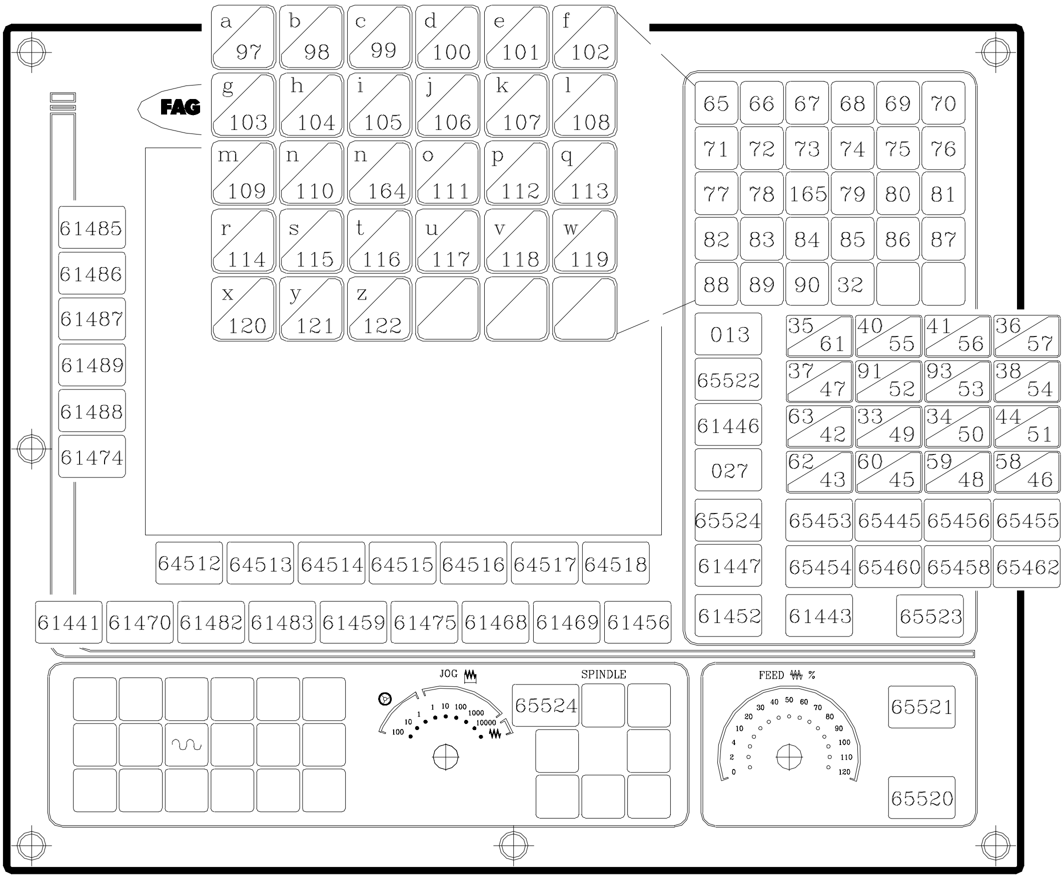

FAG
a 97 b 98 c 99 d 100 e 101 f 102
g 103 h 104 i 105 j 106 k 107 l 108
m 109 n 110 ñ 164 o 111 p 112 q 113
r 114 s 115 t 116 u 117 v 118 w 119
x 120 y 121 z 122
65 66 67 68 69 70
71 72 73 74 75 76
77 78 165 79 80 81
82 83 84 85 86 87
88 89 90 32
013 35 61 40 55 41 56 36 57
65522 37 47 91 52 93 53 38 54
61446 63 42 33 49 34 50 44 51
027 62 43 60 45 59 48 58 46
65524 65453 65445 65456 65455
61447 65454 65460 65458 65462
61452 61443 65523
64512 64513 64514 64515 64516 64517 64518
JOG
SPINDLE
61451
FEED %
65521
65520

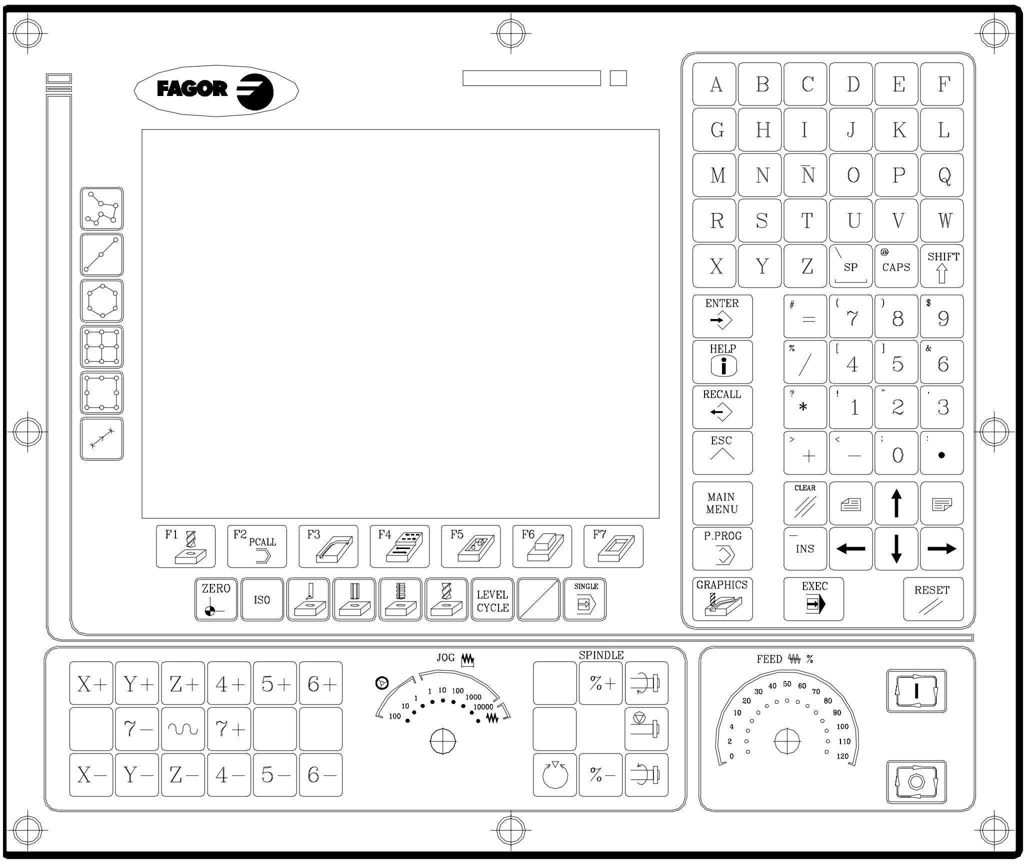

FAGOR
A B C D E F
G H I J K L
M N Ñ O P Q
R S T U V W
X Y Z SP CAPS SHIFT
ENTER
HELP
RECALL
ESC
MAIN MENU
EDIT
SIMUL
EXEC
RESET
CLEAR
INS
F1 F2 F3 F4 F5 F6 F7
X+ Y+ Z+ 4+ 5+ 6+
7− 7+
X− Y− Z− 4− 5− 6−
JOG
SPINDLE
%+ %−
FEED %

MC operator panel









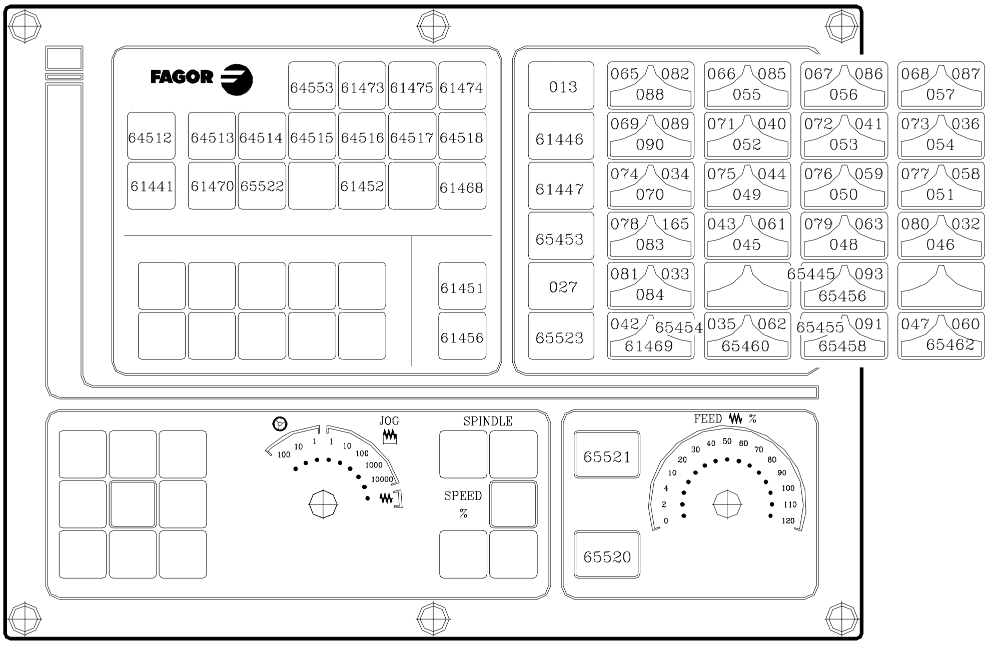
FAGOR

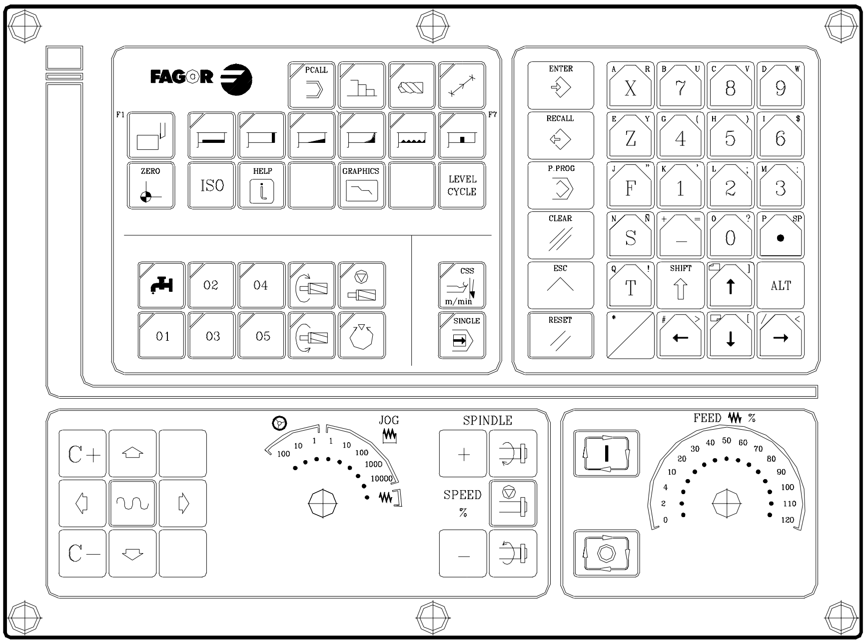
Installation manual
(Soft M: 5.3x)
(Soft T: 6.3x)

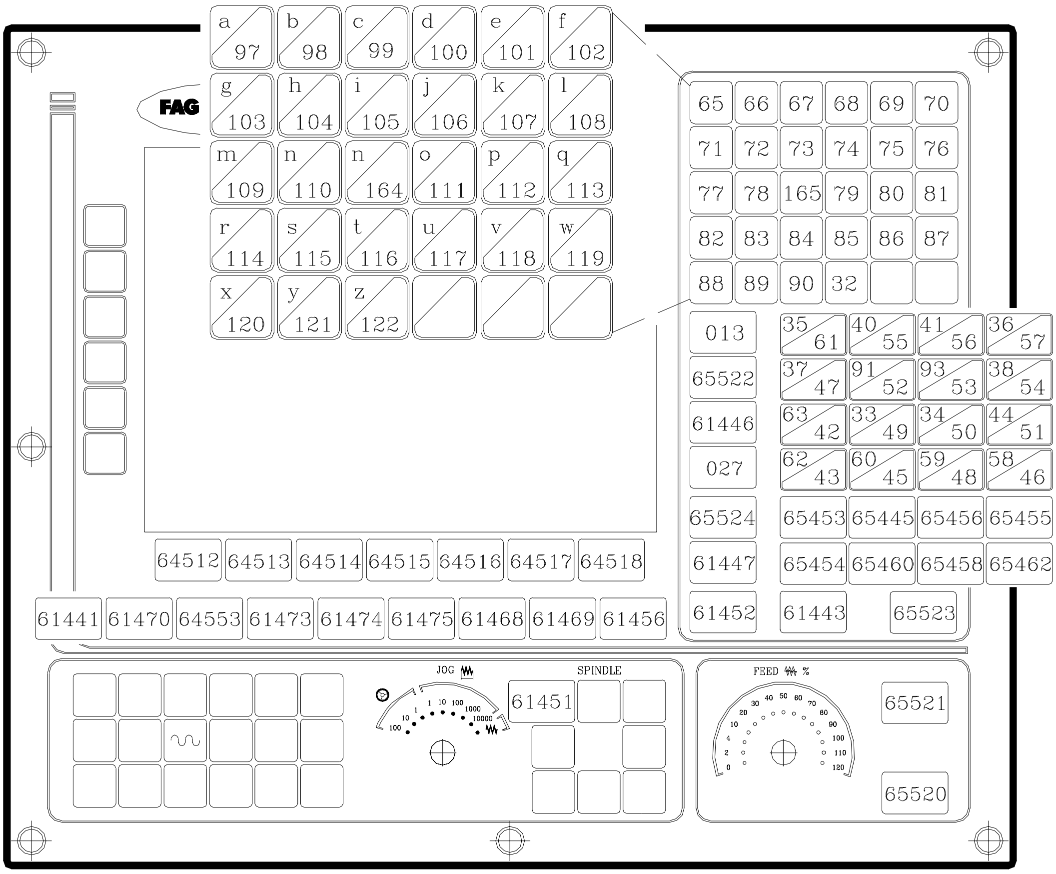
Appendix

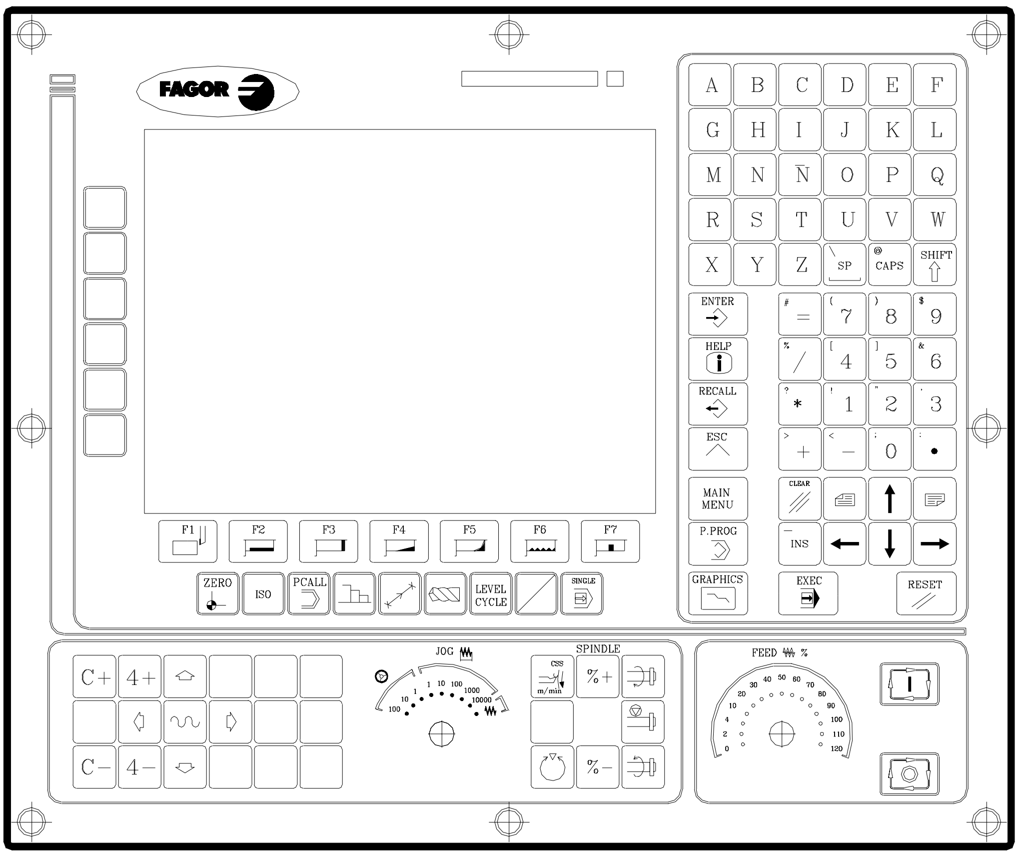
Key codes

TC operator panel

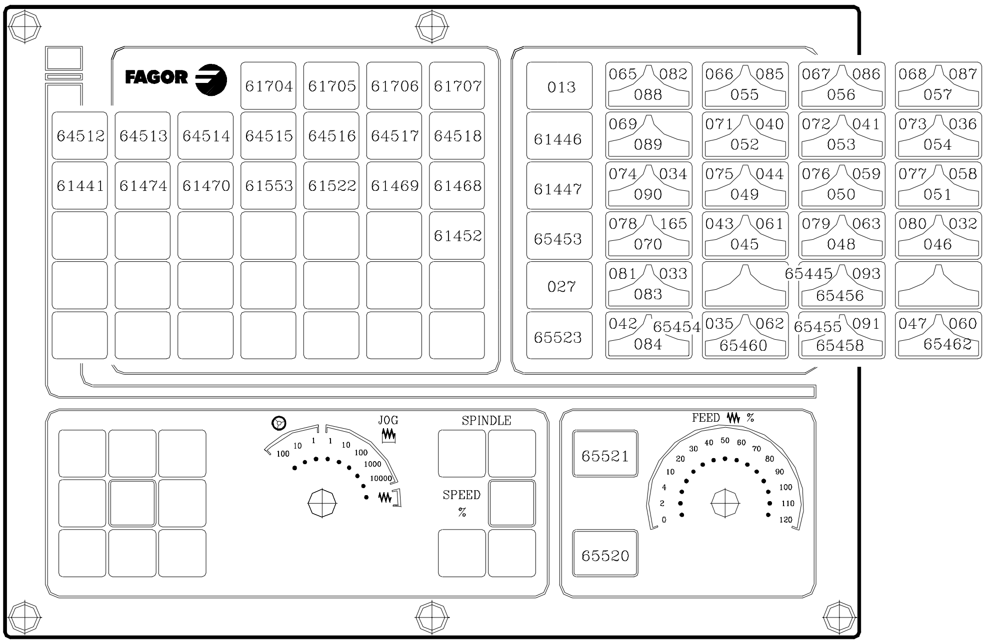


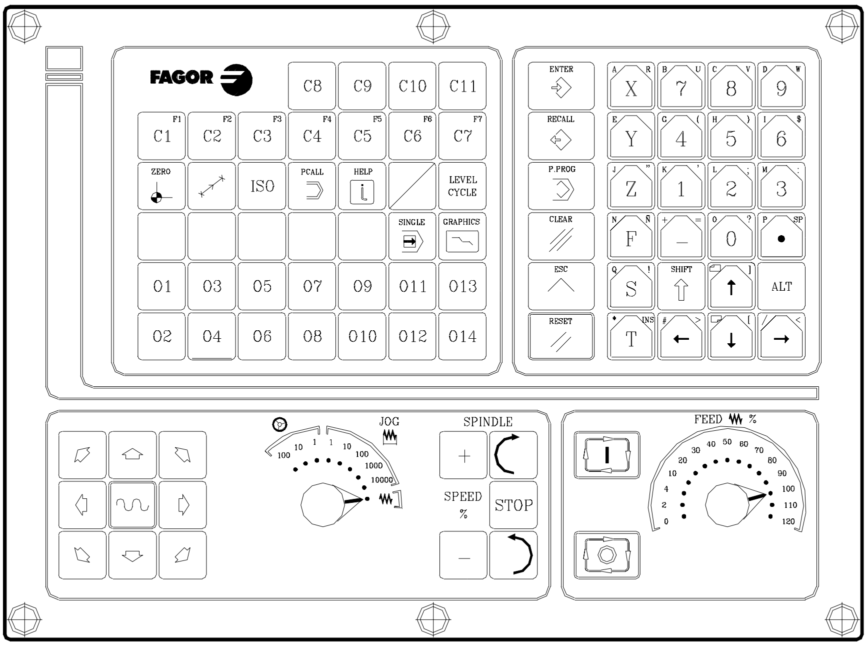



a 97 b 98 c 99 d 100 e 101 f 102
g 103 h 104 i 105 j 106 k 107 l 108
m 109 n 110 ñ 164 o 111 p 112 q 113
r 114 s 115 t 116 u 117 v 118 w 119
x 120 y 121 z 122
65 66 67 68 69 70
71 72 73 74 75 76
77 78 165 79 80 81
82 83 84 85 86 87
88 89 90 32
013 35 61 40 55 41 56 36 57
65522 37 47 91 52 93 53 38 54
61446 63 42 33 49 34 50 44 51
027 62 43 60 45 59 48 58 46
65524 65453 65445 65456 65455
61447 65454 65460 65458 65462
61452 61443 65523
64512 64513 64514 64515 64516 64517 64518
61441 61470 64553 61473 61474 61475 61468 61469 61456
JOG
SPINDLE
61451
FEED %
65521
65520

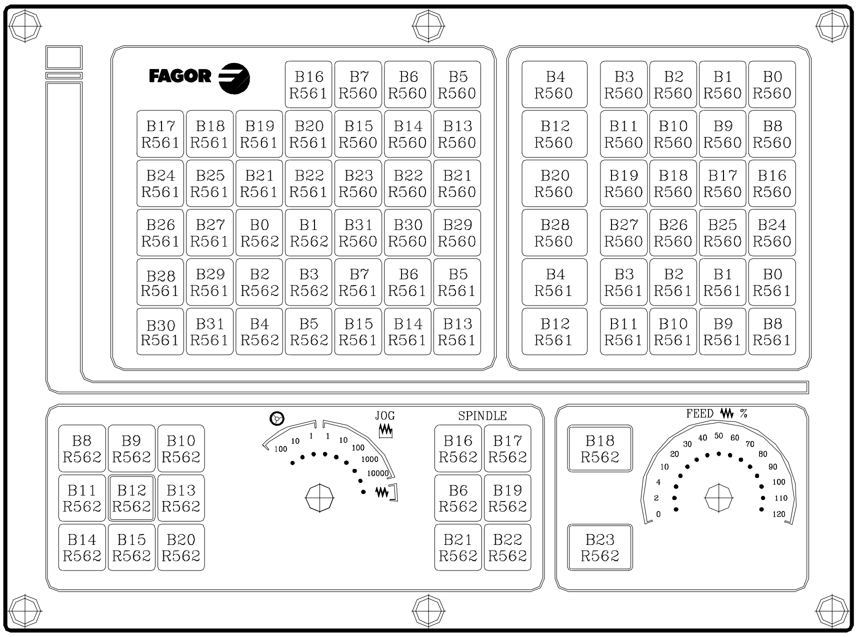

FAGOR
A B C D E F
G H I J K L
M N Ñ O P Q
R S T U V W
X Y Z SP CAPS SHIFT
ENTER = 7 8 9
HELP / 4 5 6
RECALL * 1 2 3
ESC + − 0 .
MAIN MENU CLEAR
P.PROG INS
GRAPHICS EXEC RESET
F1 F2 F3 F4 F5 F6 F7
ZERO ISO PCALL LEVEL CYCLE SINGLE
C+ 4+
C− 4−
JOG
SPINDLE
CSS m/min
%+
%−
FEED %

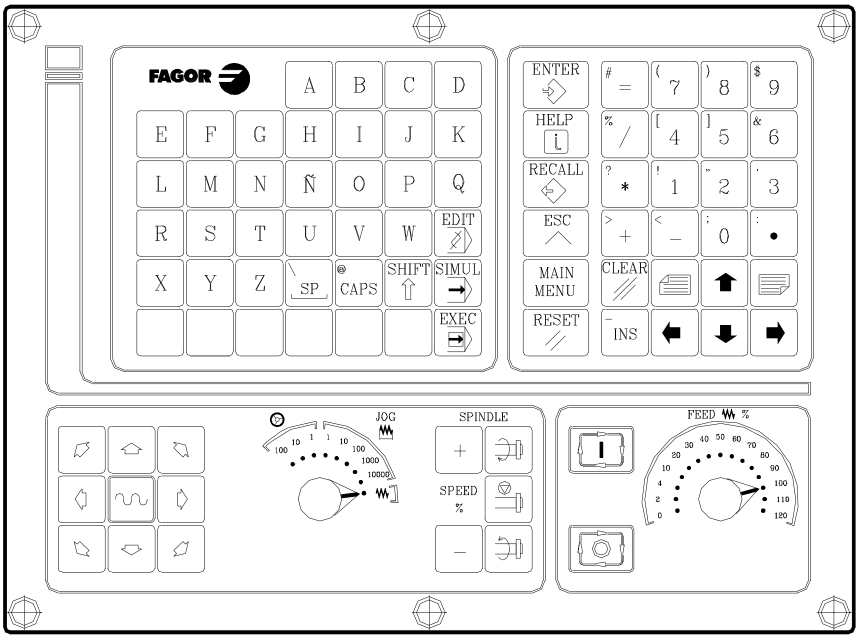
MCO/TCO operator panel

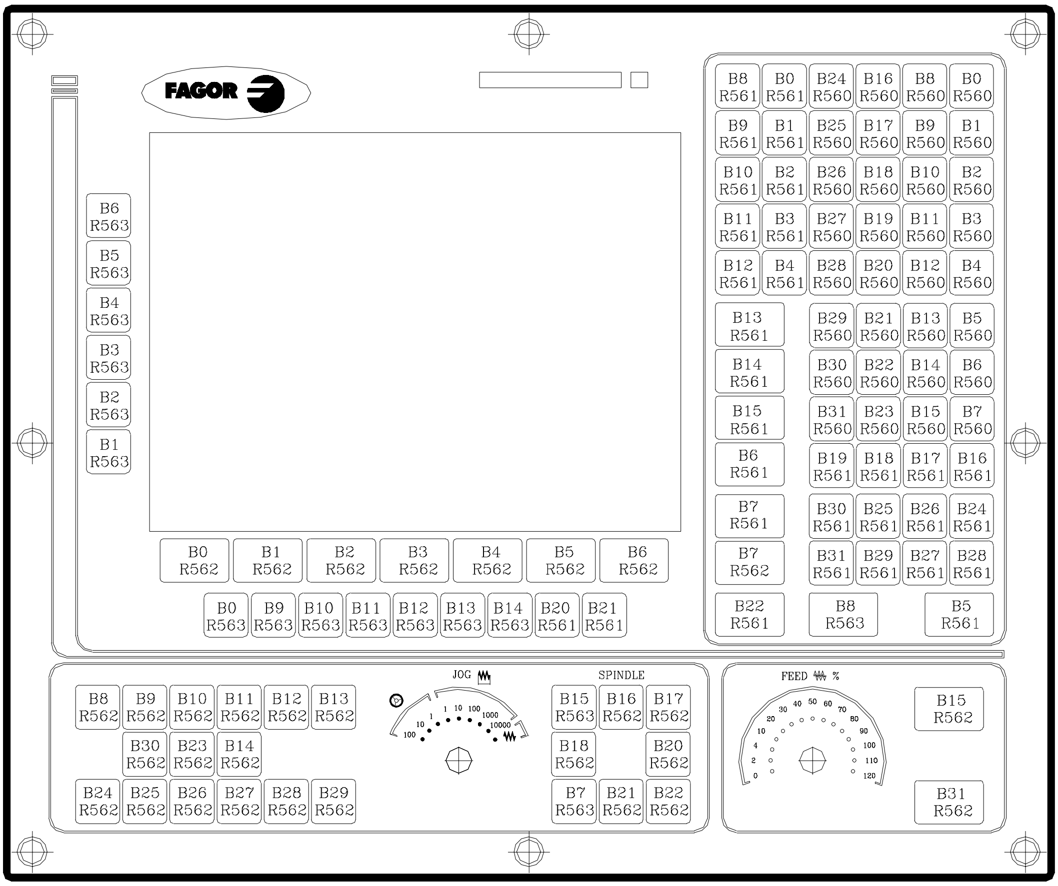


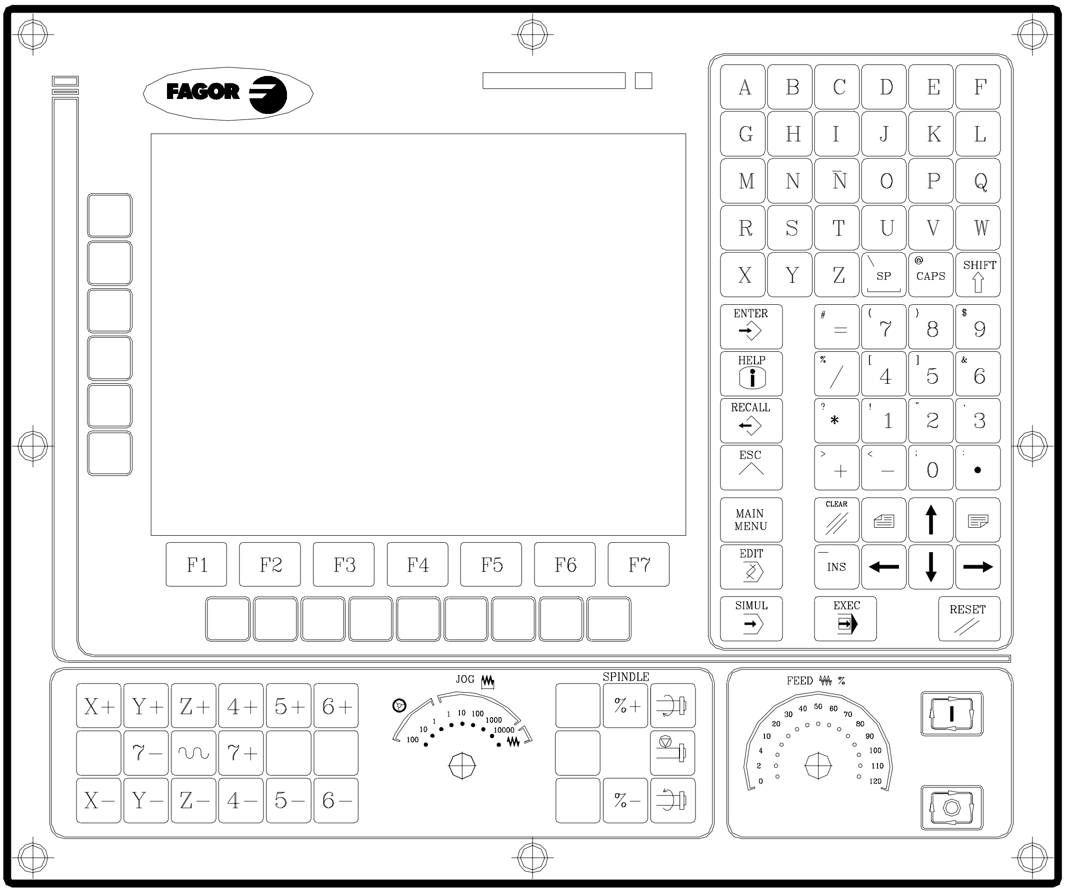


Alphanumeric operator panel (M-T models)









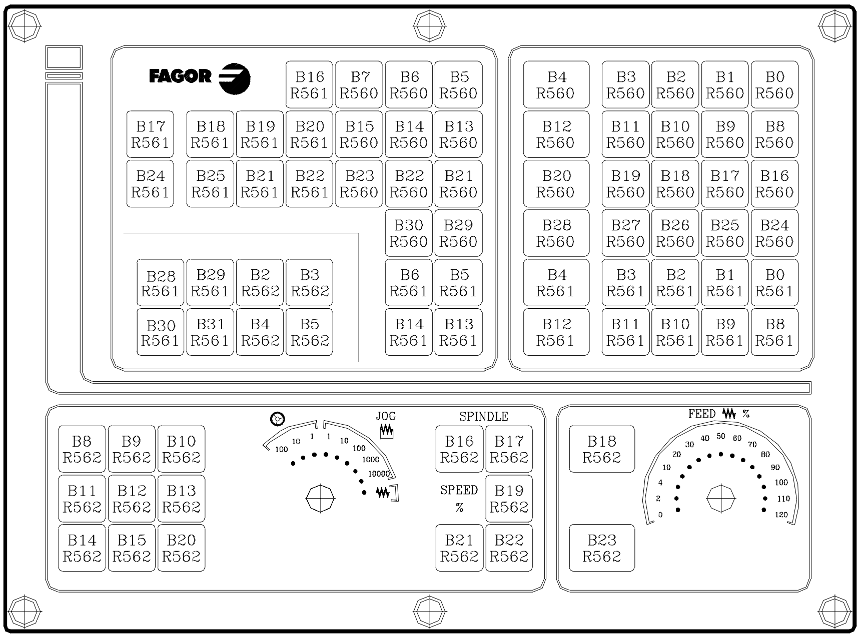
FAGOR

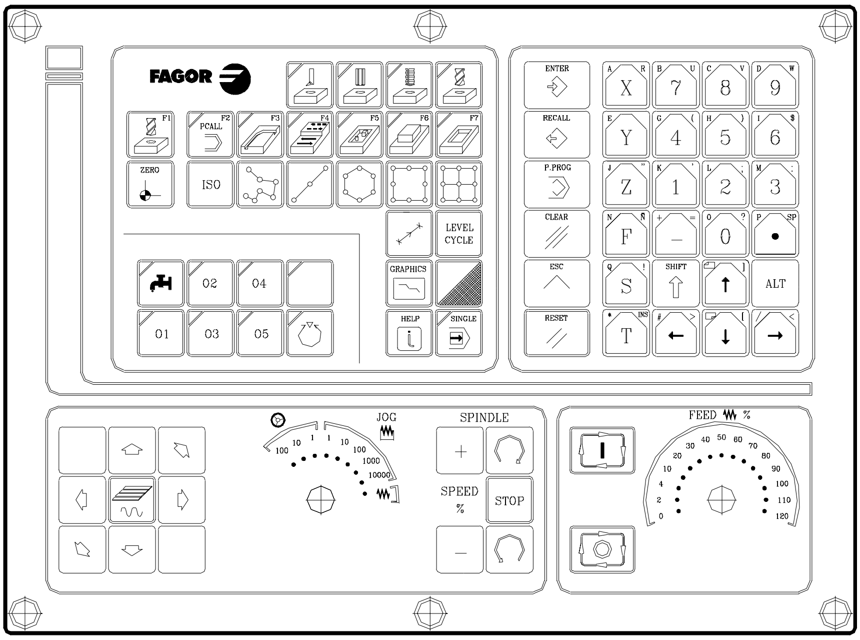
INSTALLATION MANUAL
(SOFT M: 5.3x)
(SOFT T: 6.3x)

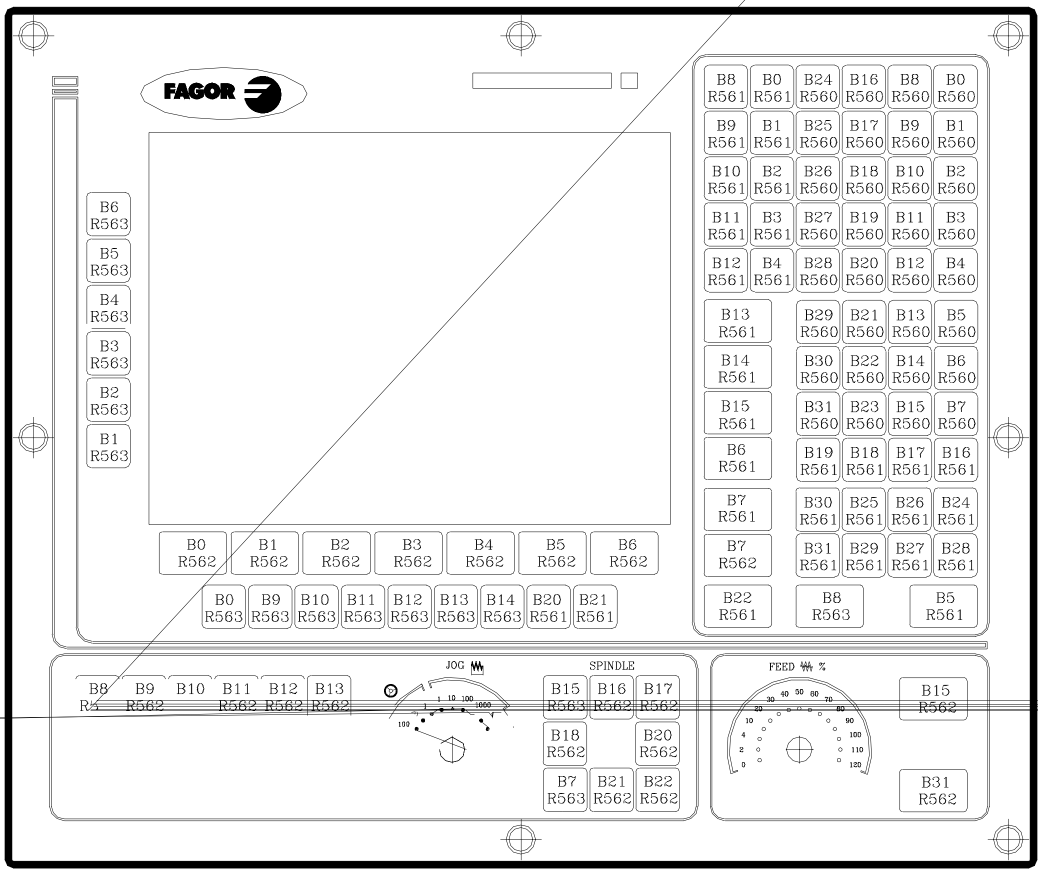
Appendix

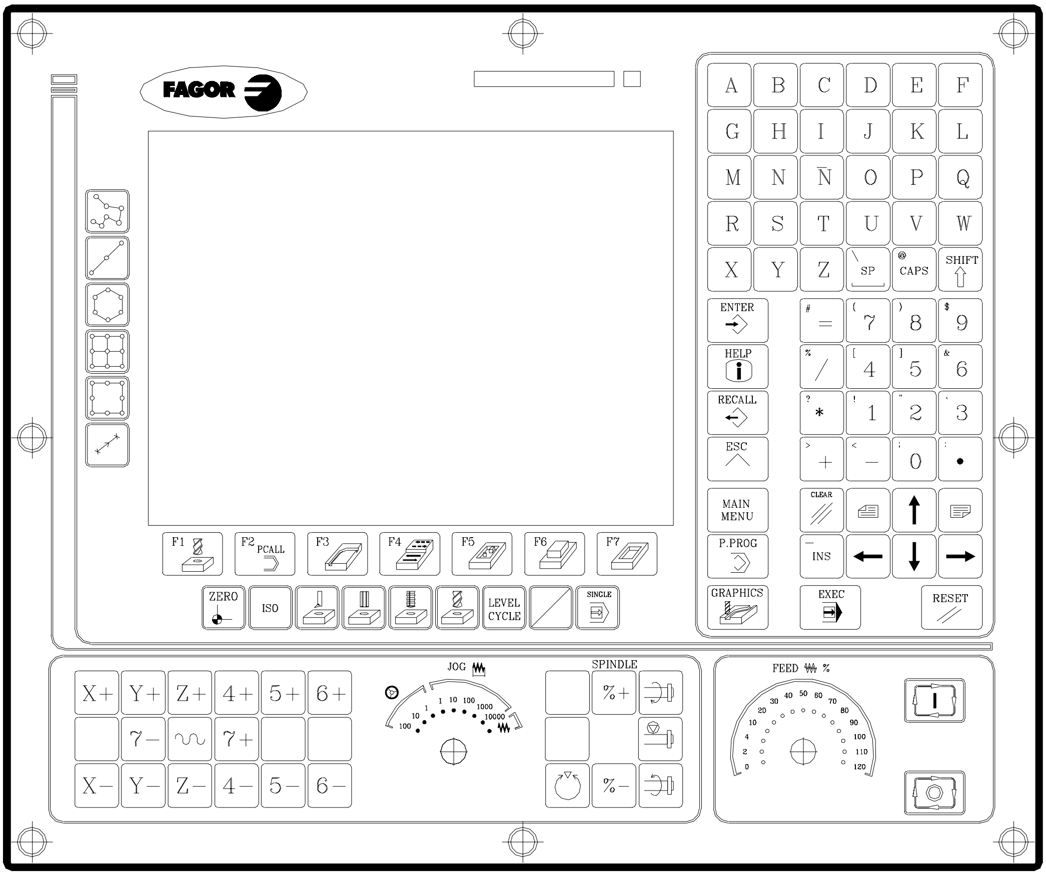
Logic outputs of key status

MC operator panel

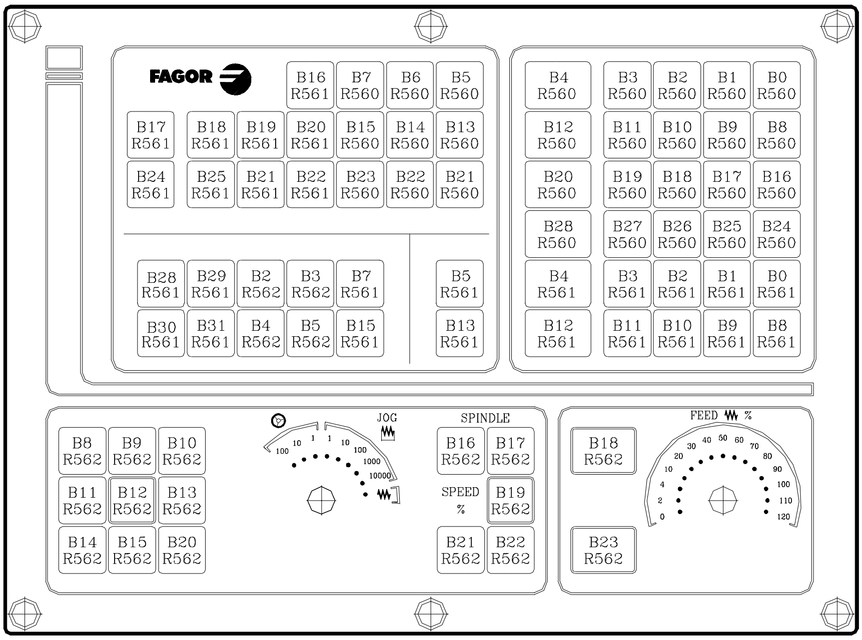


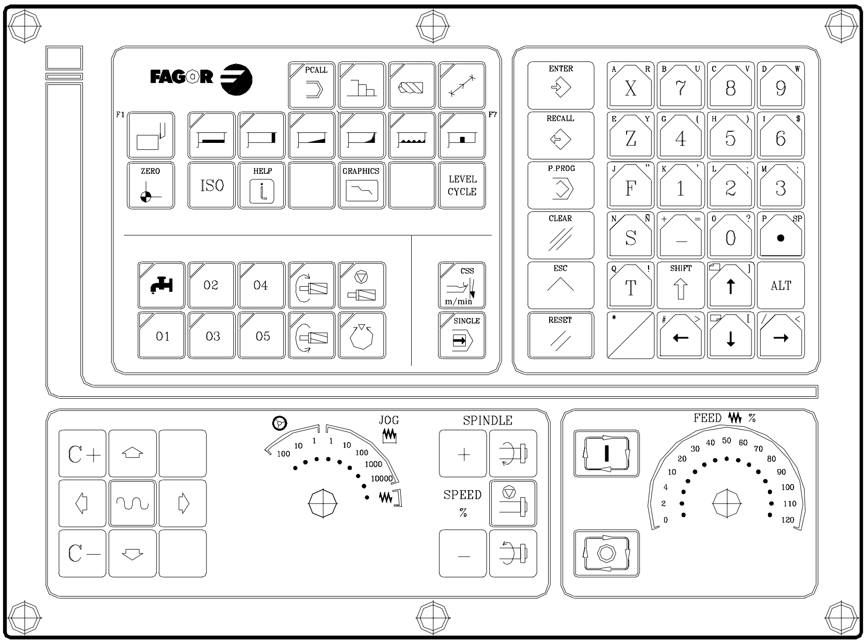


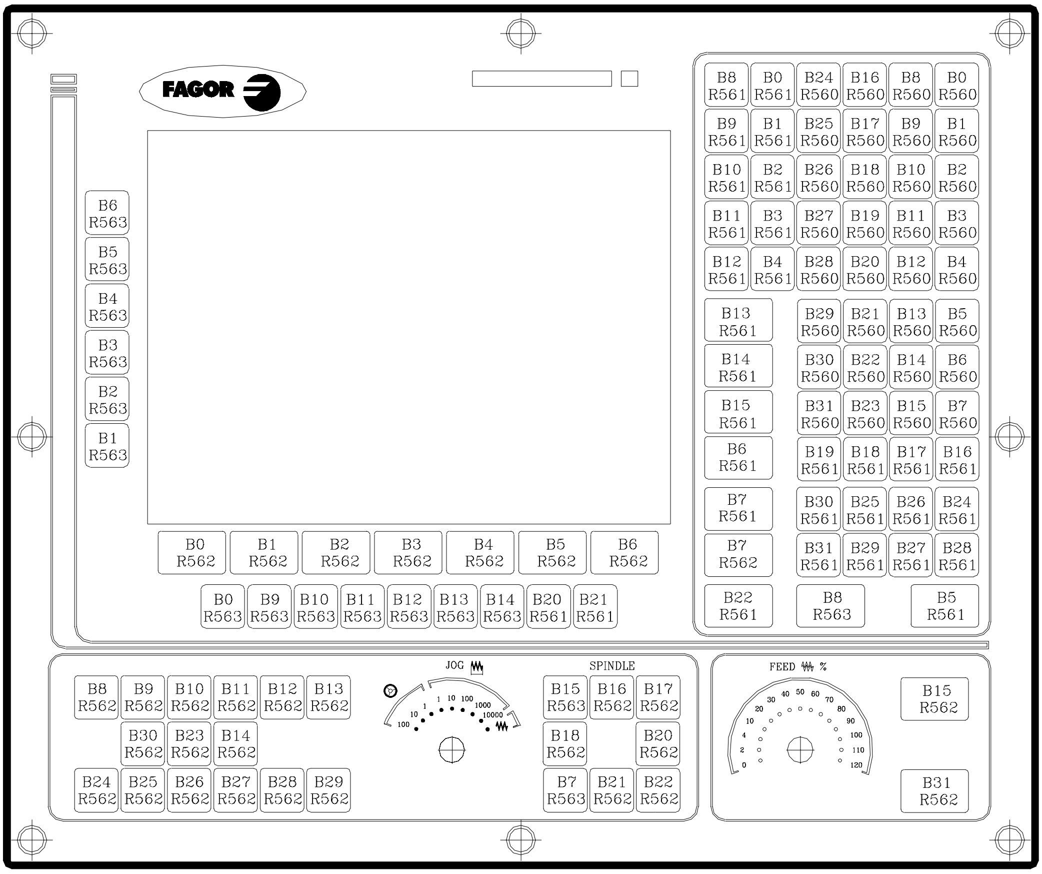


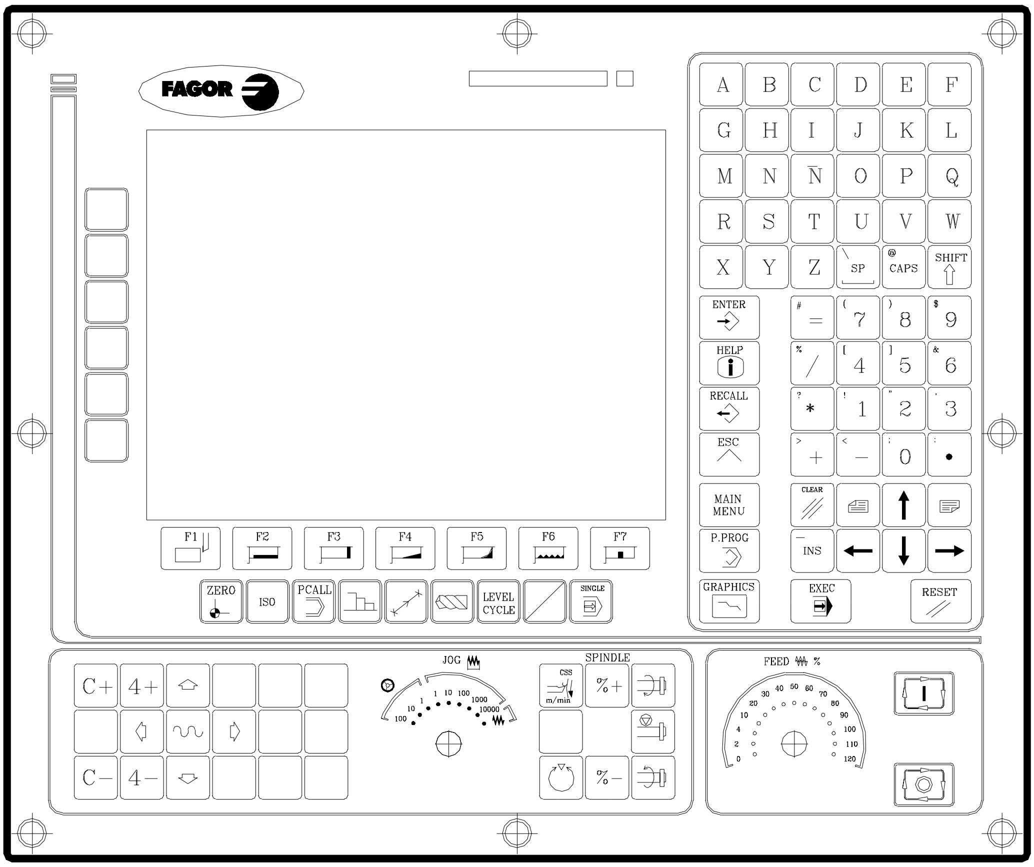


TC operator panel

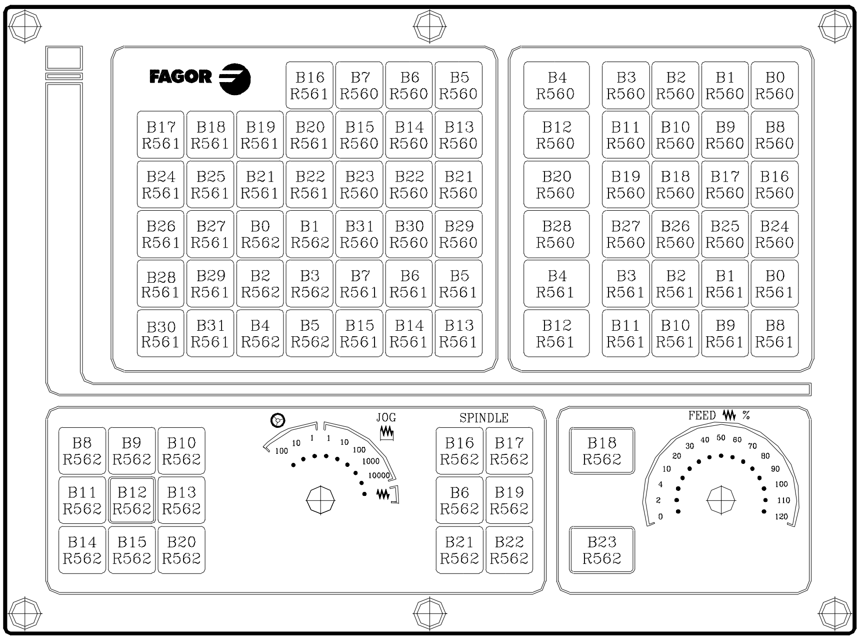


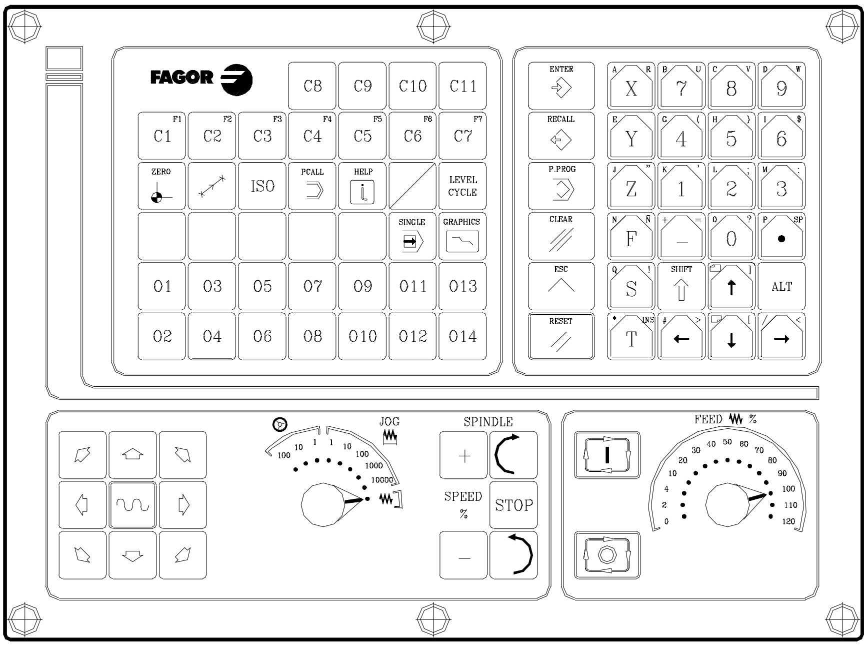




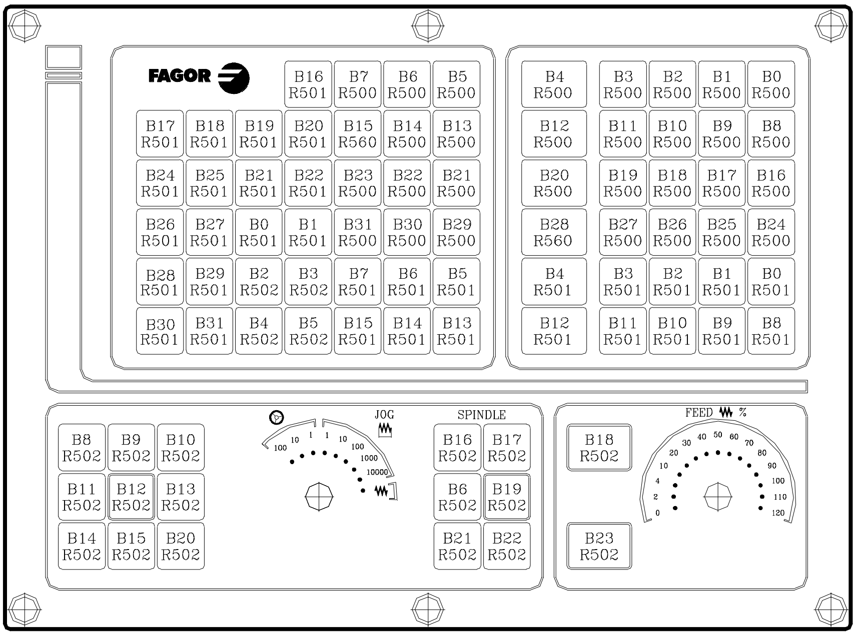


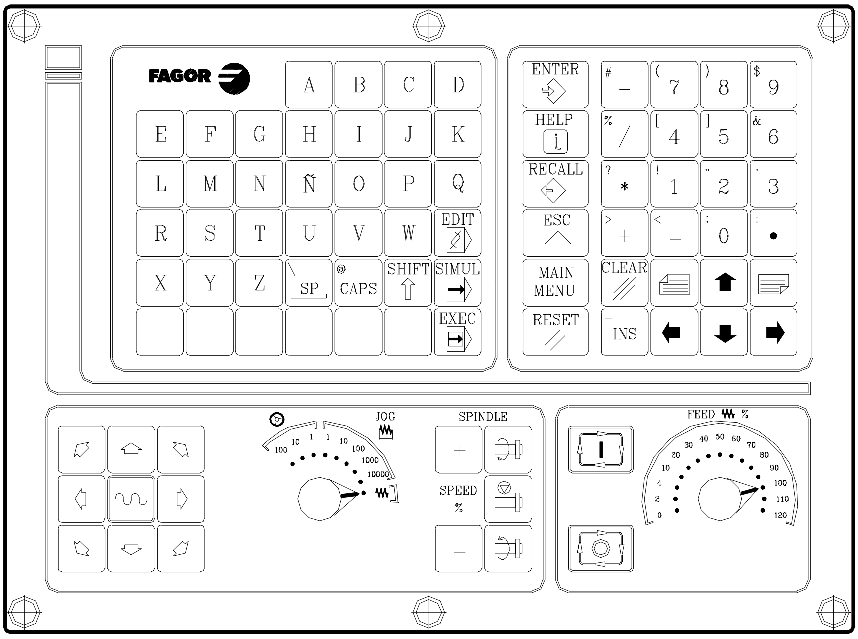
MCO/TCO operator panel

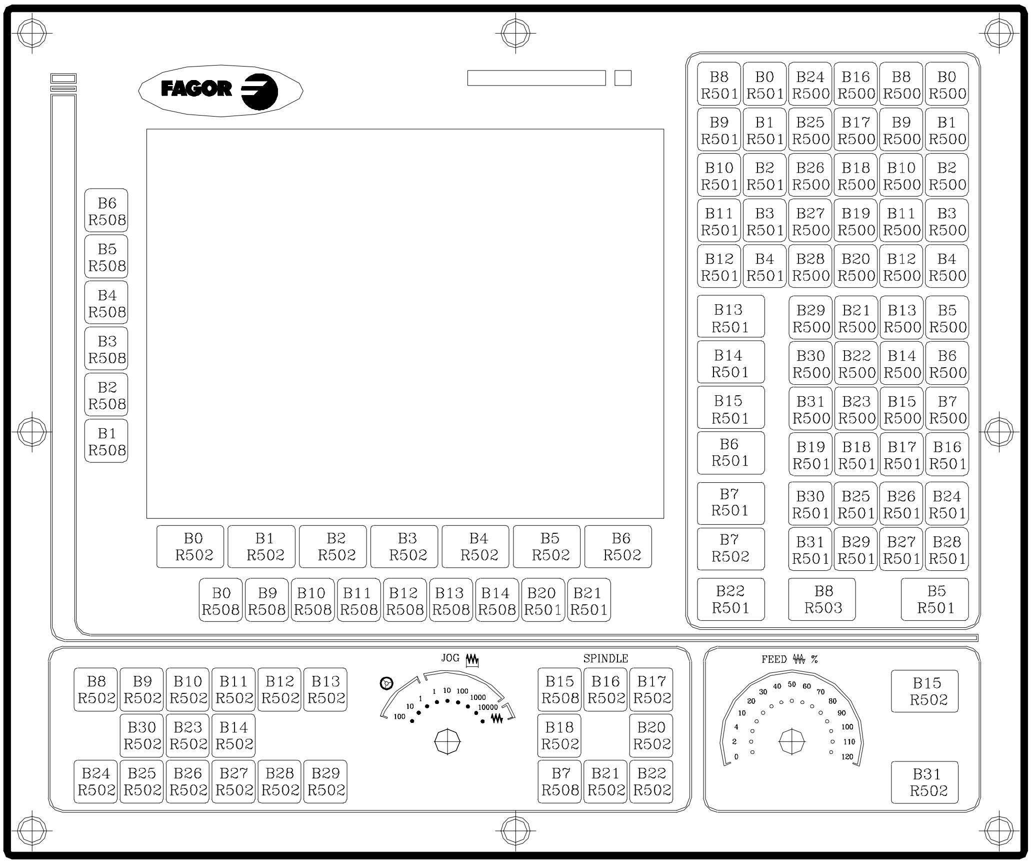


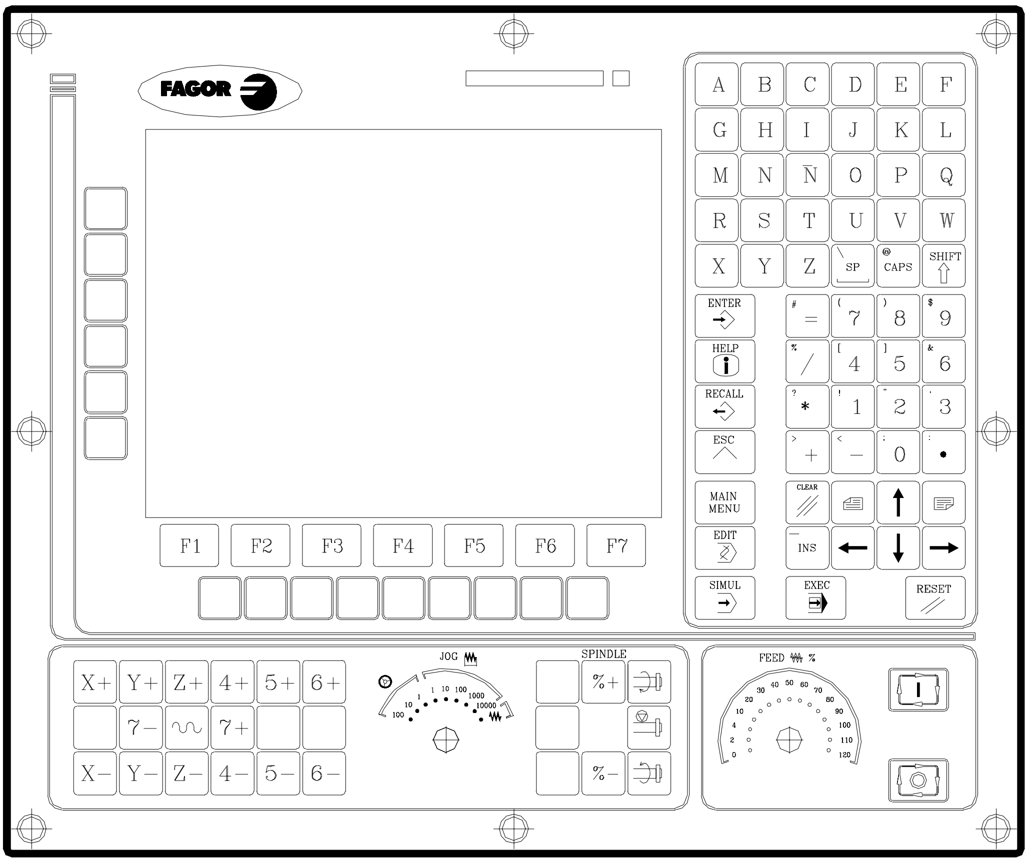


Alphanumeric operator panel (M-T models)









8040 CNC

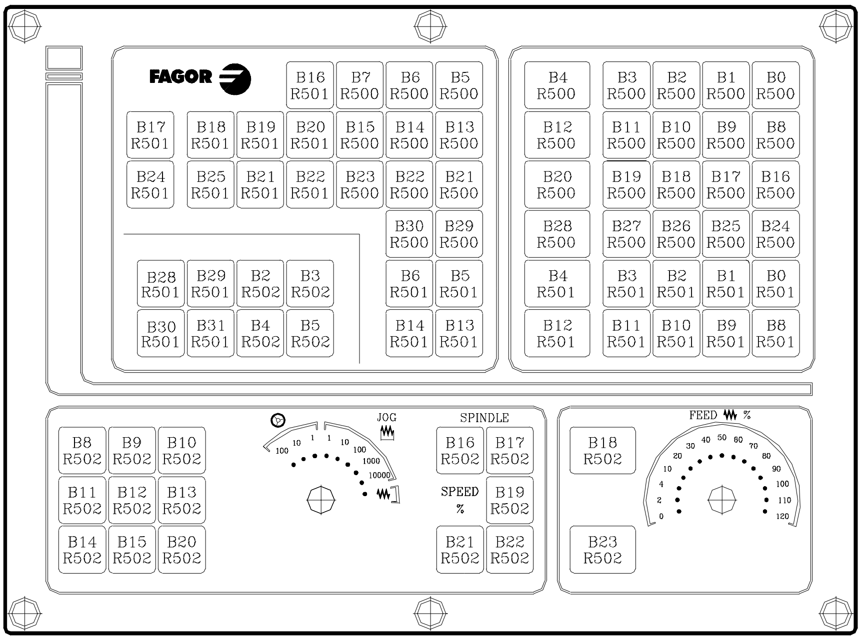
FAGOR

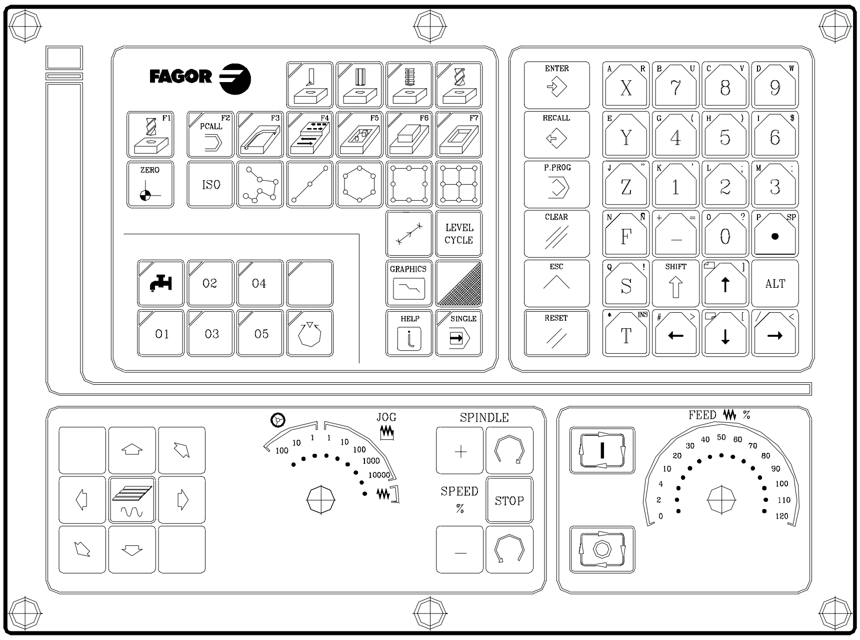
INSTALLATION MANUAL
(SOFT M: 5.3X)
(SOFT T: 6.3X)

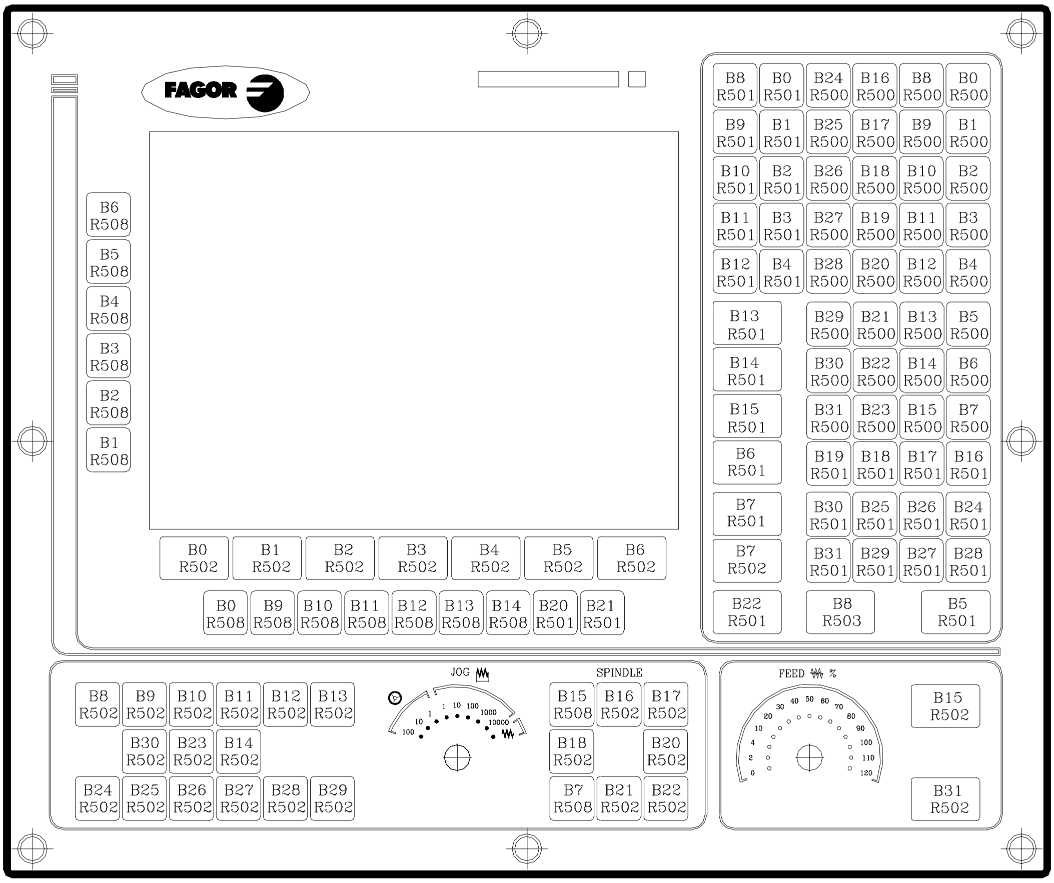
Appendix

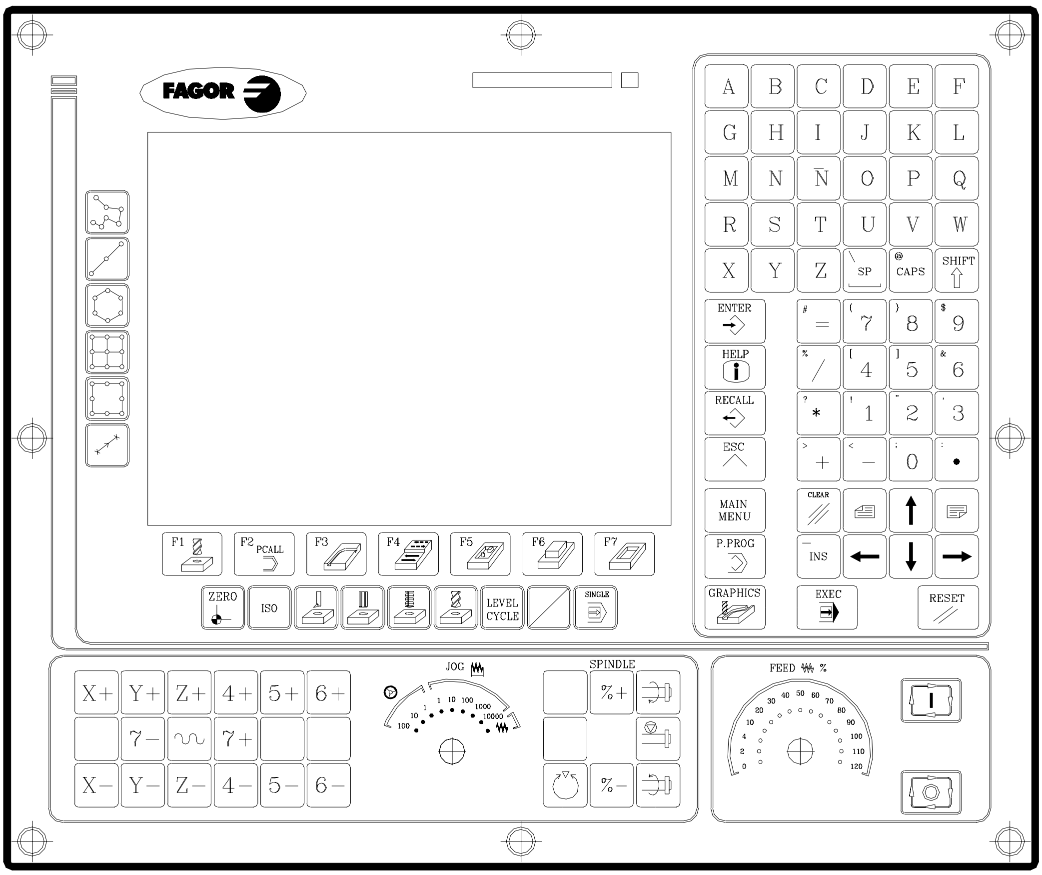
Key inhibiting codes

MC operator panel









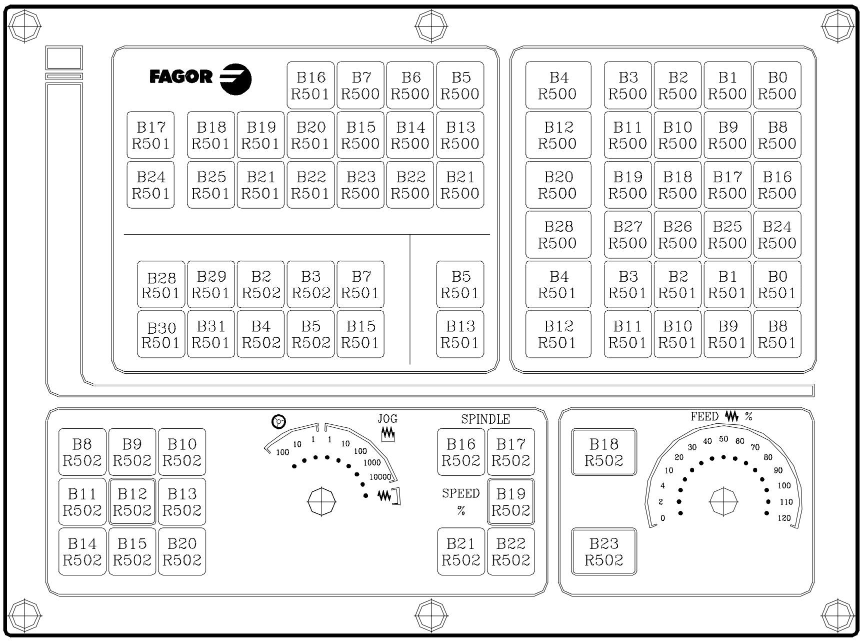
FAGOR

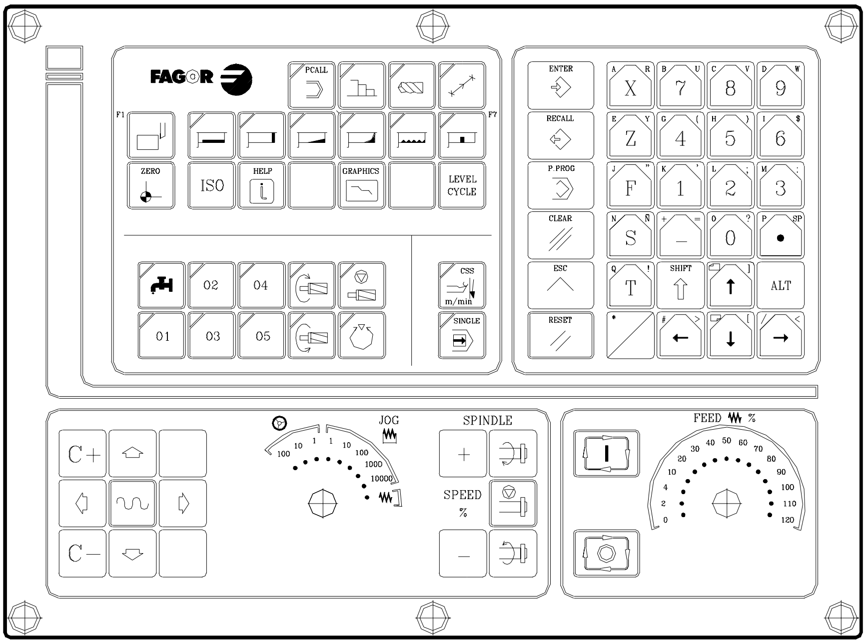
INSTALLATION MANUAL
(SOFT M: 5.3X)
(SOFT T: 6.3X)

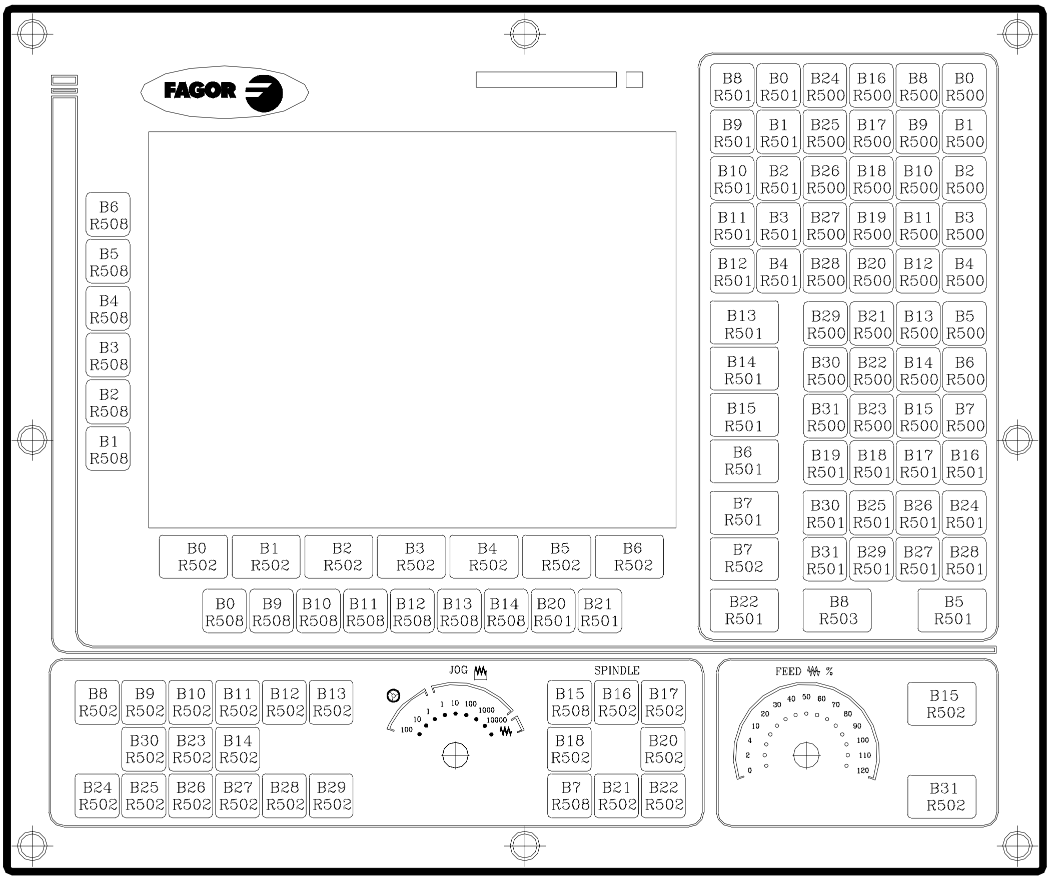
Appendix

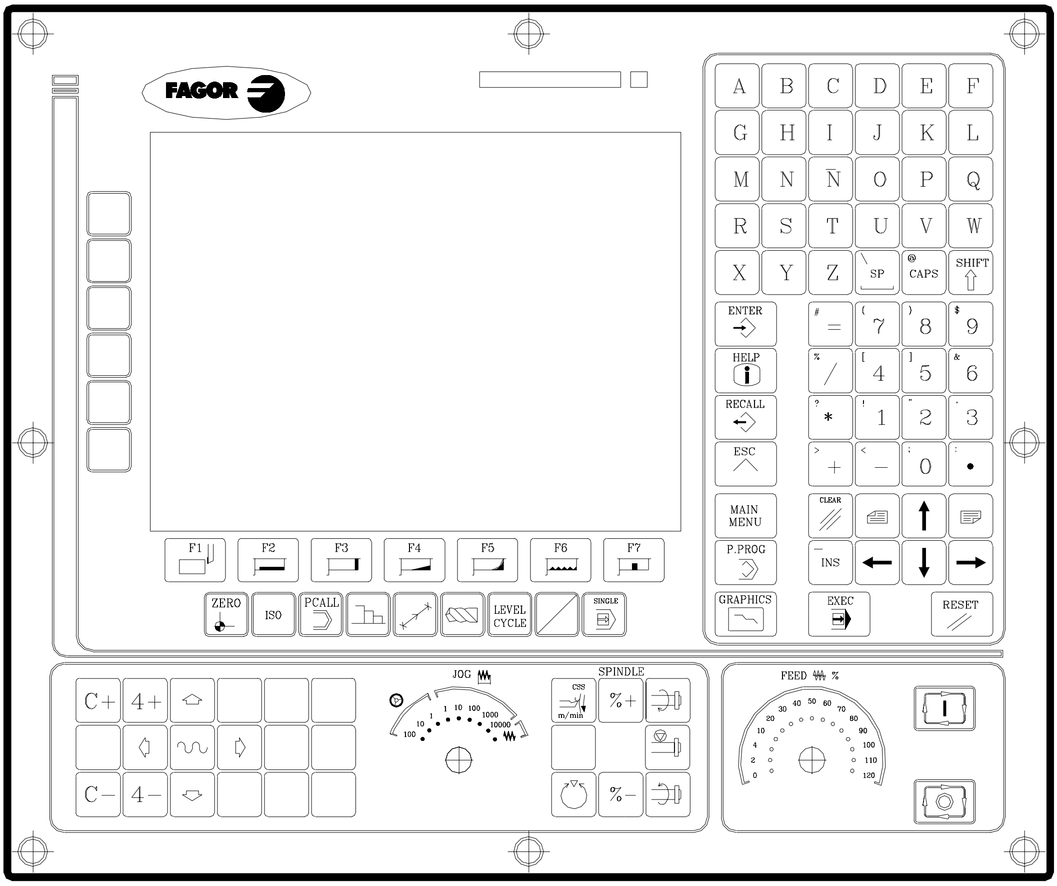
Key inhibiting codes

TC operator panel

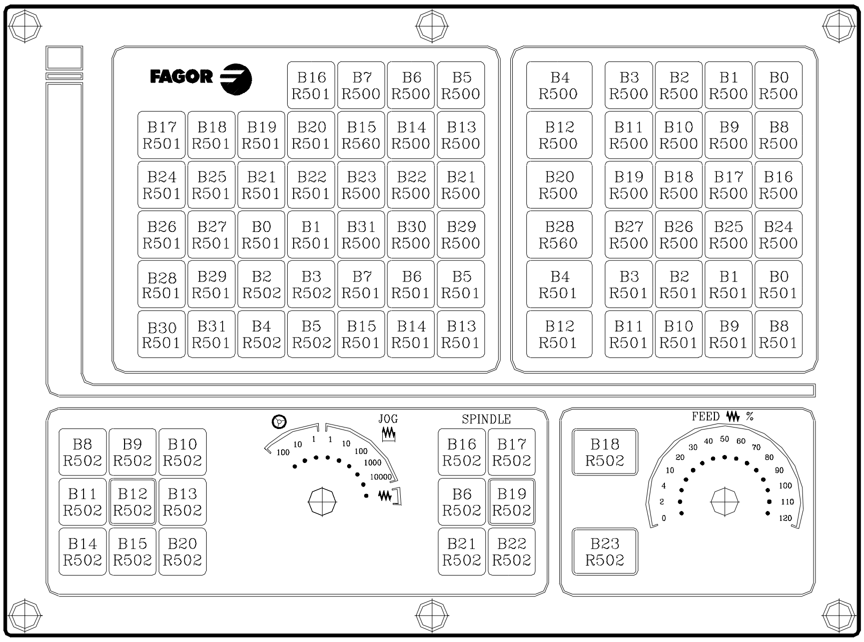


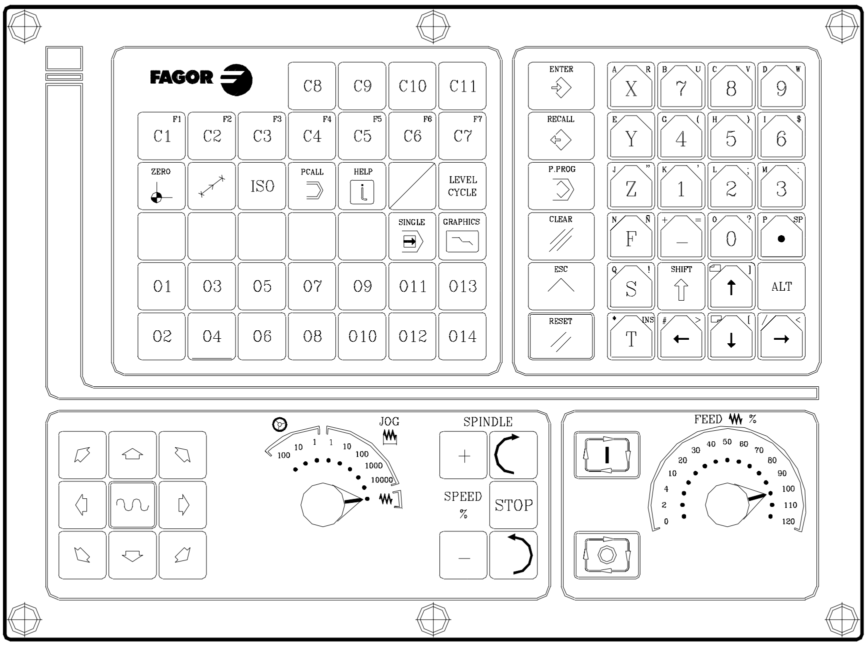






MCO/TCO operator panel





General Machine Parameters

Section 3.1

P 0		P 50		P100		P150	
P 1		P 51		P101		P151	
P 2		P 52		P102		P152	
P 3		P 53		P103		P153	
P 4		P 54		P104		P154	
P 5		P 55		P105		P155	
P 6		P 56		P106		P156	
P 7		P 57		P107		P157	
P 8		P 58		P108		P158	
P 9		P 59		P109		P159	
P 10		P 60		P110		P160	
P 11		P 61		P111		P161	
P 12		P 62		P112		P162	
P 13		P 63		P113		P163	
P 14		P 64		P114		P164	
P 15		P 65		P115		P165	
P 16		P 66		P116		P166	
P 17		P 67		P117		P167	
P 18		P 68		P118		P168	
P 19		P 69		P119		P169	
P 20		P 70		P120		P170	
P 21		P 71		P121		P171	
P 22		P 72		P122		P172	
P 23		P 73		P123		P173	
P 24		P 74		P124		P174	
P 25		P 75		P125		P175	
P 26		P 76		P126		P176	
P 27		P 77		P127		P177	
P 28		P 78		P128		P178	
P 29		P 79		P129		P179	
P 30		P 80		P130		P180	
P 31		P 81		P131		P181	
P 32		P 82		P132		P182	
P 33		P 83		P133		P183	
P 34		P 84		P134		P184	
P 35		P 85		P135		P185	
P 36		P 86		P136		P186	
P 37		P 87		P137		P187	
P 38		P 88		P138		P188	
P 39		P 89		P139		P189	
P 40		P 90		P140		P190	
P 41		P 91		P141		P191	
P 42		P 92		P142		P192	
P 43		P 93		P143		P193	
P 44		P 94		P144		P194	
P 45		P 95		P145		P195	
P 46		P 96		P146		P196	
P 47		P 97		P147		P197	
P 48		P 98		P148		P198	

Machine Parameters of the ___ axis

P 0		P 50		P100		P150	
P 1		P 51		P101		P151	
P 2		P 52		P102		P152	
P 3		P 53		P103		P153	
P 4		P 54		P104		P154	
P 5		P 55		P105		P155	
P 6		P 56		P106		P156	
P 7		P 57		P107		P157	
P 8		P 58		P108		P158	
P 9		P 59		P109		P159	
P 10		P 60		P110		P160	
P 11		P 61		P111		P161	
P 12		P 62		P112		P162	
P 13		P 63		P113		P163	
P 14		P 64		P114		P164	
P 15		P 65		P115		P165	
P 16		P 66		P116		P166	
P 17		P 67		P117		P167	
P 18		P 68		P118		P168	
P 19		P 69		P119		P169	
P 20		P 70		P120		P170	
P 21		P 71		P121		P171	
P 22		P 72		P122		P172	
P 23		P 73		P123		P173	
P 24		P 74		P124		P174	
P 25		P 75		P125		P175	
P 26		P 76		P126		P176	
P 27		P 77		P127		P177	
P 28		P 78		P128		P178	
P 29		P 79		P129		P179	
P 30		P 80		P130		P180	
P 31		P 81		P131		P181	
P 32		P 82		P132		P182	
P 33		P 83		P133		P183	
P 34		P 84		P134		P184	
P 35		P 85		P135		P185	
P 36		P 86		P136		P186	
P 37		P 87		P137		P187	
P 38		P 88		P138		P188	
P 39		P 89		P139		P189	
P 40		P 90		P140		P190	
P 41		P 91		P141		P191	
P 42		P 92		P142		P192	
P 43		P 93		P143		P193	
P 44		P 94		P144		P194	
P 45		P 95		P145		P195	
P 46		P 96		P146		P196	
P 47		P 97		P147		P197	
P 48		P 98		P148		P198	

Machine Parameters of the ___ axis

Section 3.2

P 0		P 50		P100		P150	
P 1		P 51		P101		P151	
P 2		P 52		P102		P152	
P 3		P 53		P103		P153	
P 4		P 54		P104		P154	
P 5		P 55		P105		P155	
P 6		P 56		P106		P156	
P 7		P 57		P107		P157	
P 8		P 58		P108		P158	
P 9		P 59		P109		P159	
P 10		P 60		P110		P160	
P 11		P 61		P111		P161	
P 12		P 62		P112		P162	
P 13		P 63		P113		P163	
P 14		P 64		P114		P164	
P 15		P 65		P115		P165	
P 16		P 66		P116		P166	
P 17		P 67		P117		P167	
P 18		P 68		P118		P168	
P 19		P 69		P119		P169	
P 20		P 70		P120		P170	
P 21		P 71		P121		P171	
P 22		P 72		P122		P172	
P 23		P 73		P123		P173	
P 24		P 74		P124		P174	
P 25		P 75		P125		P175	
P 26		P 76		P126		P176	
P 27		P 77		P127		P177	
P 28		P 78		P128		P178	
P 29		P 79		P129		P179	
P 30		P 80		P130		P180	
P 31		P 81		P131		P181	
P 32		P 82		P132		P182	
P 33		P 83		P133		P183	
P 34		P 84		P134		P184	
P 35		P 85		P135		P185	
P 36		P 86		P136		P186	
P 37		P 87		P137		P187	
P 38		P 88		P138		P188	
P 39		P 89		P139		P189	
P 40		P 90		P140		P190	
P 41		P 91		P141		P191	
P 42		P 92		P142		P192	
P 43		P 93		P143		P193	
P 44		P 94		P144		P194	
P 45		P 95		P145		P195	
P 46		P 96		P146		P196	
P 47		P 97		P147		P197	
P 48		P 98		P148		P198	

P 0		P 50		P100		P150	
P 1		P 51		P101		P151	
P 2		P 52		P102		P152	
P 3		P 53		P103		P153	
P 4		P 54		P104		P154	
P 5		P 55		P105		P155	
P 6		P 56		P106		P156	
P 7		P 57		P107		P157	
P 8		P 58		P108		P158	
P 9		P 59		P109		P159	
P 10		P 60		P110		P160	
P 11		P 61		P111		P161	
P 12		P 62		P112		P162	
P 13		P 63		P113		P163	
P 14		P 64		P114		P164	
P 15		P 65		P115		P165	
P 16		P 66		P116		P166	
P 17		P 67		P117		P167	
P 18		P 68		P118		P168	
P 19		P 69		P119		P169	
P 20		P 70		P120		P170	
P 21		P 71		P121		P171	
P 22		P 72		P122		P172	
P 23		P 73		P123		P173	
P 24		P 74		P124		P174	
P 25		P 75		P125		P175	
P 26		P 76		P126		P176	
P 27		P 77		P127		P177	
P 28		P 78		P128		P178	
P 29		P 79		P129		P179	
P 30		P 80		P130		P180	
P 31		P 81		P131		P181	
P 32		P 82		P132		P182	
P 33		P 83		P133		P183	
P 34		P 84		P134		P184	
P 35		P 85		P135		P185	
P 36		P 86		P136		P186	
P 37		P 87		P137		P187	
P 38		P 88		P138		P188	
P 39		P 89		P139		P189	
P 40		P 90		P140		P190	
P 41		P 91		P141		P191	
P 42		P 92		P142		P192	
P 43		P 93		P143		P193	
P 44		P 94		P144		P194	
P 45		P 95		P145		P195	
P 46		P 96		P146		P196	
P 47		P 97		P147		P197	
P 48		P 98		P148		P198	

P 0		P 50		P100		P150	
P 1		P 51		P101		P151	
P 2		P 52		P102		P152	
P 3		P 53		P103		P153	
P 4		P 54		P104		P154	
P 5		P 55		P105		P155	
P 6		P 56		P106		P156	
P 7		P 57		P107		P157	
P 8		P 58		P108		P158	
P 9		P 59		P109		P159	
P 10		P 60		P110		P160	
P 11		P 61		P111		P161	
P 12		P 62		P112		P162	
P 13		P 63		P113		P163	
P 14		P 64		P114		P164	
P 15		P 65		P115		P165	
P 16		P 66		P116		P166	
P 17		P 67		P117		P167	
P 18		P 68		P118		P168	
P 19		P 69		P119		P169	
P 20		P 70		P120		P170	
P 21		P 71		P121		P171	
P 22		P 72		P122		P172	
P 23		P 73		P123		P173	
P 24		P 74		P124		P174	
P 25		P 75		P125		P175	
P 26		P 76		P126		P176	
P 27		P 77		P127		P177	
P 28		P 78		P128		P178	
P 29		P 79		P129		P179	
P 30		P 80		P130		P180	
P 31		P 81		P131		P181	
P 32		P 82		P132		P182	
P 33		P 83		P133		P183	
P 34		P 84		P134		P184	
P 35		P 85		P135		P185	
P 36		P 86		P136		P186	
P 37		P 87		P137		P187	
P 38		P 88		P138		P188	
P 39		P 89		P139		P189	
P 40		P 90		P140		P190	
P 41		P 91		P141		P191	
P 42		P 92		P142		P192	
P 43		P 93		P143		P193	
P 44		P 94		P144		P194	
P 45		P 95		P145		P195	
P 46		P 96		P146		P196	
P 47		P 97		P147		P197	
P 48		P 98		P148		P198	

Machine Parameters of the ___ axis

Section 3.2

P 0		P 50		P100		P150	
P 1		P 51		P101		P151	
P 2		P 52		P102		P152	
P 3		P 53		P103		P153	
P 4		P 54		P104		P154	
P 5		P 55		P105		P155	
P 6		P 56		P106		P156	
P 7		P 57		P107		P157	
P 8		P 58		P108		P158	
P 9		P 59		P109		P159	
P 10		P 60		P110		P160	
P 11		P 61		P111		P161	
P 12		P 62		P112		P162	
P 13		P 63		P113		P163	
P 14		P 64		P114		P164	
P 15		P 65		P115		P165	
P 16		P 66		P116		P166	
P 17		P 67		P117		P167	
P 18		P 68		P118		P168	
P 19		P 69		P119		P169	
P 20		P 70		P120		P170	
P 21		P 71		P121		P171	
P 22		P 72		P122		P172	
P 23		P 73		P123		P173	
P 24		P 74		P124		P174	
P 25		P 75		P125		P175	
P 26		P 76		P126		P176	
P 27		P 77		P127		P177	
P 28		P 78		P128		P178	
P 29		P 79		P129		P179	
P 30		P 80		P130		P180	
P 31		P 81		P131		P181	
P 32		P 82		P132		P182	
P 33		P 83		P133		P183	
P 34		P 84		P134		P184	
P 35		P 85		P135		P185	
P 36		P 86		P136		P186	
P 37		P 87		P137		P187	
P 38		P 88		P138		P188	
P 39		P 89		P139		P189	
P 40		P 90		P140		P190	
P 41		P 91		P141		P191	
P 42		P 92		P142		P192	
P 43		P 93		P143		P193	
P 44		P 94		P144		P194	
P 45		P 95		P145		P195	
P 46		P 96		P146		P196	
P 47		P 97		P147		P197	
P 48		P 98		P148		P198	

P 0 P 1		P 50 P 51		P100 P101		P150 P151	
P 2 P 3		P 52 P 53		P102 P103		P152 P153	
P 4 P 5		P 54 P 55		P104 P105		P154 P155	
P 6 P 7		P 56 P 57		P106 P107		P156 P157	
P 8 P 9		P 58 P 59		P108 P109		P158 P159	
P 10 P 11		P 60 P 61		P110 P111		P160 P161	
P 12 P 13		P 62 P 63		P112 P113		P162 P163	
P 14 P 15		P 64 P 65		P114 P115		P164 P165	
P 16 P 17		P 66 P 67		P116 P117		P166 P167	
P 18 P 19		P 68 P 69		P118 P119		P168 P169	
P 20 P 21		P 70 P 71		P120 P121		P170 P171	
P 22 P 23		P 72 P 73		P122 P123		P172 P173	
P 24 P 25		P 74 P 75		P124 P125		P174 P175	
P 26 P 27		P 76 P 77		P126 P127		P176 P177	
P 28 P 29		P 78 P 79		P128 P129		P178 P179	
P 30 P 31		P 80 P 81		P130 P131		P180 P181	
P 32 P 33		P 82 P 83		P132 P133		P182 P183	
P 34 P 35		P 84 P 85		P134 P135		P184 P185	
P 36 P 37		P 86 P 87		P136 P137		P186 P187	
P 38 P 39		P 88 P 89		P138 P139		P188 P189	
P 40 P 41		P 90 P 91		P140 P141		P190 P191	
P 42 P 43		P 92 P 93		P142 P143		P192 P193	
P 44 P 45		P 94 P 95		P144 P145		P194 P195	
P 46 P 47		P 96 P 97		P146 P147		P196 P197	
P 48		P 98		P148		P198	

P 0		P 50		P100		P150	
P 1		P 51		P101		P151	
P 2		P 52		P102		P152	
P 3		P 53		P103		P153	
P 4		P 54		P104		P154	
P 5		P 55		P105		P155	
P 6		P 56		P106		P156	
P 7		P 57		P107		P157	
P 8		P 58		P108		P158	
P 9		P 59		P109		P159	
P 10		P 60		P110		P160	
P 11		P 61		P111		P161	
P 12		P 62		P112		P162	
P 13		P 63		P113		P163	
P 14		P 64		P114		P164	
P 15		P 65		P115		P165	
P 16		P 66		P116		P166	
P 17		P 67		P117		P167	
P 18		P 68		P118		P168	
P 19		P 69		P119		P169	
P 20		P 70		P120		P170	
P 21		P 71		P121		P171	
P 22		P 72		P122		P172	
P 23		P 73		P123		P173	
P 24		P 74		P124		P174	
P 25		P 75		P125		P175	
P 26		P 76		P126		P176	
P 27		P 77		P127		P177	
P 28		P 78		P128		P178	
P 29		P 79		P129		P179	
P 30		P 80		P130		P180	
P 31		P 81		P131		P181	
P 32		P 82		P132		P182	
P 33		P 83		P133		P183	
P 34		P 84		P134		P184	
P 35		P 85		P135		P185	
P 36		P 86		P136		P186	
P 37		P 87		P137		P187	
P 38		P 88		P138		P188	
P 39		P 89		P139		P189	
P 40		P 90		P140		P190	
P 41		P 91		P141		P191	
P 42		P 92		P142		P192	
P 43		P 93		P143		P193	
P 44		P 94		P144		P194	
P 45		P 95		P145		P195	
P 46		P 96		P146		P196	
P 47		P 97		P147		P197	
P 48		P 98		P148		P198	

P 0		P 50		P100		P150	
P 1		P 51		P101		P151	
P 2		P 52		P102		P152	
P 3		P 53		P103		P153	
P 4		P 54		P104		P154	
P 5		P 55		P105		P155	
P 6		P 56		P106		P156	
P 7		P 57		P107		P157	
P 8		P 58		P108		P158	
P 9		P 59		P109		P159	
P 10		P 60		P110		P160	
P 11		P 61		P111		P161	
P 12		P 62		P112		P162	
P 13		P 63		P113		P163	
P 14		P 64		P114		P164	
P 15		P 65		P115		P165	
P 16		P 66		P116		P166	
P 17		P 67		P117		P167	
P 18		P 68		P118		P168	
P 19		P 69		P119		P169	
P 20		P 70		P120		P170	
P 21		P 71		P121		P171	
P 22		P 72		P122		P172	
P 23		P 73		P123		P173	
P 24		P 74		P124		P174	
P 25		P 75		P125		P175	
P 26		P 76		P126		P176	
P 27		P 77		P127		P177	
P 28		P 78		P128		P178	
P 29		P 79		P129		P179	
P 30		P 80		P130		P180	
P 31		P 81		P131		P181	
P 32		P 82		P132		P182	
P 33		P 83		P133		P183	
P 34		P 84		P134		P184	
P 35		P 85		P135		P185	
P 36		P 86		P136		P186	
P 37		P 87		P137		P187	
P 38		P 88		P138		P188	
P 39		P 89		P139		P189	
P 40		P 90		P140		P190	
P 41		P 91		P141		P191	
P 42		P 92		P142		P192	
P 43		P 93		P143		P193	
P 44		P 94		P144		P194	
P 45		P 95		P145		P195	
P 46		P 96		P146		P196	
P 47		P 97		P147		P197	
P 48		P 98		P148		P198	

P 0		P 50		P100		P150	
P 1		P 51		P101		P151	
P 2		P 52		P102		P152	
P 3		P 53		P103		P153	
P 4		P 54		P104		P154	
P 5		P 55		P105		P155	
P 6		P 56		P106		P156	
P 7		P 57		P107		P157	
P 8		P 58		P108		P158	
P 9		P 59		P109		P159	
P 10		P 60		P110		P160	
P 11		P 61		P111		P161	
P 12		P 62		P112		P162	
P 13		P 63		P113		P163	
P 14		P 64		P114		P164	
P 15		P 65		P115		P165	
P 16		P 66		P116		P166	
P 17		P 67		P117		P167	
P 18		P 68		P118		P168	
P 19		P 69		P119		P169	
P 20		P 70		P120		P170	
P 21		P 71		P121		P171	
P 22		P 72		P122		P172	
P 23		P 73		P123		P173	
P 24		P 74		P124		P174	
P 25		P 75		P125		P175	
P 26		P 76		P126		P176	
P 27		P 77		P127		P177	
P 28		P 78		P128		P178	
P 29		P 79		P129		P179	
P 30		P 80		P130		P180	
P 31		P 81		P131		P181	
P 32		P 82		P132		P182	
P 33		P 83		P133		P183	
P 34		P 84		P134		P184	
P 35		P 85		P135		P185	
P 36		P 86		P136		P186	
P 37		P 87		P137		P187	
P 38		P 88		P138		P188	
P 39		P 89		P139		P189	
P 40		P 90		P140		P190	
P 41		P 91		P141		P191	
P 42		P 92		P142		P192	
P 43		P 93		P143		P193	
P 44		P 94		P144		P194	
P 45		P 95		P145		P195	
P 46		P 96		P146		P196	
P 47		P 97		P147		P197	
P 48		P 98		P148		P198	

Machine parameters of the auxiliary spindle

Section 3.3.2

P 0		P 50		P100		P150	
P 1		P 51		P101		P151	
P 2		P 52		P102		P152	
P 3		P 53		P103		P153	
P 4		P 54		P104		P154	
P 5		P 55		P105		P155	
P 6		P 56		P106		P156	
P 7		P 57		P107		P157	
P 8		P 58		P108		P158	
P 9		P 59		P109		P159	
P 10		P 60		P110		P160	
P 11		P 61		P111		P161	
P 12		P 62		P112		P162	
P 13		P 63		P113		P163	
P 14		P 64		P114		P164	
P 15		P 65		P115		P165	
P 16		P 66		P116		P166	
P 17		P 67		P117		P167	
P 18		P 68		P118		P168	
P 19		P 69		P119		P169	
P 20		P 70		P120		P170	
P 21		P 71		P121		P171	
P 22		P 72		P122		P172	
P 23		P 73		P123		P173	
P 24		P 74		P124		P174	
P 25		P 75		P125		P175	
P 26		P 76		P126		P176	
P 27		P 77		P127		P177	
P 28		P 78		P128		P178	
P 29		P 79		P129		P179	
P 30		P 80		P130		P180	
P 31		P 81		P131		P181	
P 32		P 82		P132		P182	
P 33		P 83		P133		P183	
P 34		P 84		P134		P184	
P 35		P 85		P135		P185	
P 36		P 86		P136		P186	
P 37		P 87		P137		P187	
P 38		P 88		P138		P188	
P 39		P 89		P139		P189	
P 40		P 90		P140		P190	
P 41		P 91		P141		P191	
P 42		P 92		P142		P192	
P 43		P 93		P143		P193	
P 44		P 94		P144		P194	
P 45		P 95		P145		P195	
P 46		P 96		P146		P196	
P 47		P 97		P147		P197	
P 48		P 98		P148		P198	

Machine parameters of the serial line 1

Section 3.5

P 0		P 8		P 16		P 24	
P 1		P 9		P 17		P 25	
P 2		P 10		P 18		P 26	
P 3		P 11		P 19		P 27	
P 4		P 12		P 20		P 28	
P 5		P 13		P 21		P 29	
P 6		P 14		P 22		P 30	
P 7		P15		P 23		P 31	

Machine parameters of the serial line 2

Section 3.5

P 0		P 8		P 16		P 24	
P 1		P 9		P 17		P 25	
P 2		P 10		P 18		P 26	
P 3		P 11		P 19		P 27	
P 4		P 12		P 20		P 28	
P 5		P 13		P 21		P 29	
P 6		P 14		P 22		P 30	
P 7		P15		P 23		P 31	

Machine parameters of Ethernet

Section 3.6

P 0		P 8		P 16		P 24	
P 1		P 9		P 17		P 25	
P 2		P 10		P 18		P 26	
P 3		P 11		P 19		P 27	
P 4		P 12		P 20		P 28	
P 5		P 13		P 21		P 29	
P 6		P 14		P 22		P 30	
P 7		P15		P 23		P 31	

Machine parameters of the PLC

Section 3.7

P 0		P 25		P 50		P 75	
P 1		P 26		P 51		P 76	
P 2		P 27		P 52		P 77	
P 3		P 28		P 53		P 78	
P 4		P 29		P 54		P 79	
P 5		P 30		P 55		P 80	
P 6		P 31		P 56		P 81	
P 7		P 32		P 57		P 82	
P 8		P 33		P 58		P 83	
P 9		P 34		P 59		P 84	
P 10		P 35		P 60		P 85	
P 11		P 36		P 61		P 86	
P 12		P 37		P 62		P 87	
P 13		P 38		P 63		P 88	
P 14		P 39		P 64		P 89	
P 15		P 40		P 65		P 90	
P 16		P 41		P 66		P 91	
P 17		P 42		P 67		P 92	
P 18		P 43		P 68		P 93	
P 19		P 44		P 69		P 94	
P 20		P 45		P 70		P 95	
P 21		P 46		P 71		P 96	
P 22		P 47		P 72		P 97	
P 23		P 48		P 73		P 98	
P 24		P 49		P 74		P 99	

M FUNCTIONS SETTING CHART

M function	Associated subroutine	Setting bits							
		7	6	5	4	3	2	1	0
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								

M function	Associated subroutine	Setting bits							
		7	6	5	4	3	2	1	0
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								
M	S								

LEADSCREW ERROR COMPENSATION TABLE

___axis		
Point	Position	Error
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E

___axis		
Point	Position	Error
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E

___axis		
Point	Position	Error
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E

___axis		
Point	Position	Error
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E

___axis		
Point	Position	Error
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E

___axis		
Point	Position	Error
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E

___axis		
Point	Position	Error
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E

___axis		
Point	Position	Error
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E

CROSS COMPENSATION TABLES

Moving axis___"MOVAXIS(P32)			Axis to be compensated___ "COMPAXIS(P33)		
Point	Position	Error	Point	Position	Error
P P		E E	P P		E E
P P		E E	P P		E E
P P		E E	P P		E E
P P		E E	P P		E E
P P		E E	P P		E E
P P		E E	P P		E E
P P		E E	P P		E E
P P		E E	P P		E E
P P		E E	P P		E E
P P		E E	P P		E E
P P		E E	P P		E E
P P		E E	P P		E E

Moving axis___"MOVAXIS2(P55)		
Point	Position	Error
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E

Axis to be compensated___"COMAXIS2(P56)		
Point	Position	Error
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E

Moving axis___"MOVAXIS3(P58)		
Point	Position	Error
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E

Axis to be compensated___"COMAXIS3(P59)		
Point	Position	Error
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E
P		E

Cleaning

The dirt accumulated in the unit could act as a layer that hampers the dissipation of the heat generated by the internal electronic circuitry with the risk of overheating and damaging the CNC.

Also, accumulated dirt

To clean the operator panel and the monitor’s front panel, a soft cloth should be used soaked in de-ionized water and home non-abrasive dishwasher soap (liquid, never powder) or with 75º alcohol.

Do not use air at high pressure to clean the unit because it could cause grease to accumulate which in turn may cause electrostatic discharges.

The plastics used on the front panel of the units are resistant to:

- Grease and mineral oil.
- Bases and bleach.
- Dissolved detergents.
- Alcohol.

Warning:

To check the fuses, first unplug the unit from mains

- *If the CNC does not turn on when flipping the power switch, check that the fuses are the right ones and they are in good condition.*

Avoid solvents

- *The action of solvents such as Chlorine hydrocarbons, Benzole, Esters and Ether may damage the plastics used to make the front panel of the unit.*

Do not open this unit

- *Only personnel authorized by Fagor Automation may open this module.*

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

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

Note:

- *Fagor Automation shall not be held responsible for any material or physical damage derived from the violation of these basic safety requirements.*

User notes:

User notes:

Error solving Manual (M model)

Ref 0204-ing

FAGOR

INDEX

PROGRAMMING ERRORS

0001 'Línea vacía.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When trying to enter into a program or execute an empty block or containing the label (block number).
2. Within the «Irregular pocket canned cycle with islands (G66)», when parameter "S" (beginning of the profile) is greater than parameter "E" (end of profile).

Solution The solution for each cause is:

1. The CNC cannot enter into the program or execute an empty line. To enter an empty line in the program, use the «;» symbol at the beginning of that block. The CNC will ignore the rest of the block.
2. The value of parameter "S" (block where the profile definition begins) must be lower than the value of parameter "E" (block where the profile definition ends).

0002 'Improper data'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When editing an axis coordinate after the cutting conditions (F, S, T or D) or the «M» functions.
2. When the marks of the block skip (conditional block /1, /2 or /3) are not at the beginning of the block.
3. When programming a block number greater than 9999 while programming in ISO code.
4. When trying to define the coordinates of the machining starting point in the finishing operation (G68) of the «Irregular pocket canned cycle».
5. While programming in high-level, the value of the RPT instruction exceeds 9999.

Solution The solution for each cause is:

1. Remember the programming order
2. Remember the programming order
   - Block skip (conditional block /1, /2 or /3).
   - Label (N).
   - «G» functions.
   - Axis coordinates. (X, Y, Z…).
   - Machining conditions (F, S, T, D).
   - «M» functions.
3. Correct the syntax of the block. Program the labels between 0 and 9999
4. No point can be programmed within the definition of the finishing cycle (G68) for the «Irregular pocket canned cycle». The CNC selects the point where it will start machining. The programming format is: G68 B...L...Q...I...R...K...V... And then the cutting conditions.
5. Correct the syntax of the block. Program a number of repetitions between 0 and 9999

0003 'Improper data order.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The machining conditions or the tool data have been programmed in the wrong order.

Solution Remember that the programming order is… F...S...T...D......

All the data need not be programmed.

0004 'No more information allowed in the block.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When editing a «G» function after an axis coordinate.

2. When trying to edit some data after a «G» function (or after its associated parameters) which must go alone in the block (or which only admits its own associated data).
3. When assigning a numeric value to a parameter that does not need it.

Solution The solution for each cause is:

1. Remember the programming order
   - Block skip (conditional block /1, /2 or /3).
   - Label (N).
   - «G» functions.
   - Axis coordinates. (X, Y, Z…).
   - Machining conditions (F, S, T, D).
   - «M» functions.
2. There are some «G» functions which carry associated data in the block. Maybe, this type of functions do not let program other type of information after their associated parameters. On the other hand, neither machining conditions, (F, S), tool data (T, D) nor «M» functions may be programmed.
3. There are some «G» functions having certain parameters associated to them which do not need to be defined with values.

0005 'Repeated information'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The same data has been entered twice in a block.

Solution Correct the syntax of the block. The same data cannot be defined twice in a block.

0006 'Improper data format'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While defining the parameters of a machining canned cycle, a negative value has been assigned to a parameter which only admits positive values.

Solution Verify the format of the canned cycle. In some canned cycles, there are parameters which only accept positive values.

0007 'Incompatible G functions.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

When programming in the same block two «G» functions which are incompatible with each other.

When trying to define a canned cycle in a block containing a nonlinear movement (G02, G03, G08, G09, G33).

Solution The solution for each cause is:

1. There are groups of «G» functions which cannot go together in the block because they involve actions incompatible with each other. For example:
   G01/G02: Linear and circular interpolation
   G41/G42: Left-hand or right-hand tool radius compensation.
   This type of functions must be programmed in different blocks.
2. A canned cycle must be defined in a block containing a linear movement. In other words, to define a cycle, a "G00" or a "G01" must be active. Nonlinear movements (G02, G03, G08 and G09) may be defined in the blocks following the profile definition.

0008 'Nonexistent G function'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A nonexistent «G» function has been programmed.

Solution Check the syntax of the block and verify that a different «G» function is not being edited by mistake.

0009 'No more G functions allowed'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A «G» function has been programmed after the machining conditions or after the tool data.

Solution Remember that the programming order is:

- Block skip (conditional block /1, /2 or /3).
- Label (N).
- «G» functions.
- Axis coordinates. (X, Y, Z…).
- Machining conditions (F, S, T, D).
- «M» functions.

0010 'No more M functions allowed'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause More than 7 «M» functions have been programmed in a block.

Solution The CNC does not let program more than 7 «M» functions in a block. To execute any other functions, write them in a separate block. The «M» functions may go alone in a block.

0011 'This G or M function must be alone.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The block contains either a «G» or an «M» function that must go alone in the block.

Solution Write it alone in the block.

0012 'Program F, S, T, D before the M functions.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A machining condition (F, S) or tool data (T, D) has been programmed after the «M» functions.

Solution Remember that the programming order is:

… F...S...T...D...M...

Up to 7 «M» functions may be programmed .

All the data need not be programmed.

0013 'Program G30 D +/-359.9999'

No explanation required

0014 'Do not program labels by parameters.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A label (block number) has been defined with a parameter.

Solution Programming the block number is optional, but it cannot be defined with a parameter It can only be defined with a number between 0 and 9999.

0015 'Number of repetitions not possible.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A repetition has been programmed wrong or the block does not admit repetitions.

Solution High level instructions do not admit a number of repetitions at the end of the block. To do a repetition, assign to the block to be repeated a label (block number) and use the RPT instruction.

0016 'Program: G15 axis.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the function «Longitudinal axis selection (G15)» the parameter for the axis has not been programmed.

Solution Check the syntax of the block. The definition of the "G15" function requires the name of the new longitudinal axis.

0017 'Program: G16 axis-axis.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the function «Main plane selection by two axes (G16)» one of the two parameters for the axes has not been programmed.

Solution Check the syntax of the block. The definition of the "G16" function requires the name of the axes defining the new work plane.

0018 'Program: G22 K(1/2/3/4) S(0/1/2).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the function «Enable/Disable work zones (G22)» the type of enable or disable of the work zone has not been defined or it has been assigned the wrong value.

Solution The parameter for enabling or disabling the work zones "S" must always be programmed and it may take the following values.
- S=0: The work zone is disabled.
- S=1: It is enabled as a no-entry zone.
- S=2: It is enabled as a no-exit zone.

0019 'Program: work zone K1, K2, K3 or K4.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. A "G20", "G21" or "G22" function has been programmed without defining the work zone K1, K2, K3 or K4.
2. The programmed work zone is smaller than 0 or greater than 4.

Solution The solution for each cause is:

1. The programming format for functions "G20", "G21" and "G22" is:

G20 K...X...C±5.5	Definition of lower work zone limits
G21 K...X...C±5.5	Definition of upper work zone limits.
G22 K...S...	Enable/disable work zones.

Where:

K :	Is the work zone.
X...C	Are the axes where the limits are defined.
S	Is the type of work zone enable.

2. The "K" work zone may only have the values of K1, K2, K3 or K4.

0020 'Program G36-G39 with R+5.5.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the "G36" or "G39" function, the "R" parameter has not been programmed or it has been assigned a negative value.

Solution To define "G36" or "G39", parameter "R" must also be defined and with a positive value).

G36	R= Rounding radius.
G39	R= Distance between the end of the programmed path and the point to be chamfered.

0021 'Program: G72 S5.5 or axis (axes).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When programming a general scaling factor (G72) without the scaling factor to apply.
2. When programming a particular scaling factor (G72) to several axes, but the axes have been defined in the wrong order.

Solution Remember that the programming format for this function is:

G72 S5.5"	When applying a general scaling factor (to all axes).
G72 X…C5.5"	When applying a particular scaling factor to one or several axes.

0022 'Program: G73 Q (angle) I J (center).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Pattern rotation (G73)» function have been programmed wrong. The causes may be:

1. The rotation angle has not been defined.
2. Only one of the rotation center coordinates has been defined.
3. The rotation center coordinates have been defined in the wrong order.

Solution The programming format for this function is:

G73 Q (angle) [I J] (center)

The "Q" value must always be programmed.

The "I", "J" values are optional, but if programmed, both must be programmed.

0023 'Block incompatible when defining a profile.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the set of blocks defining a pocket profile, there is a block containing a «G» function that cannot be part of the profile definition.

Solution The "G" functions available in the profile definition of a pocket (2D/3D) are:

- G00: Beginning of the profile.
- G01: Linear interpolation.
- G02/G03: Clockwise/counterclockwise interpolation.
- G06: Circle center in absolute coordinates.
- G08: Arc tangent to previous path.
- G09: Three point arc.
- G36: Controlled corner rounding
- G39: Chamfer.
- G53: Programming with respect to home.
- G70/G71: Inch/metric programming.
- G90/G91: Programming in absolute/incremental coordinates.
- G93: Polar origin preset.

And also, in the 3D pocket profile:

- G16: Main plane selection by two axes.
- G17: Main plane X-Y and longitudinal Z.
- G18: Main plane Z-X and longitudinal Y.
- G19: Main plane Y-Z and longitudinal X.

0024 'High level blocks not allowed when defining a profile.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause Within the set of blocks defining a pocket profile, a high level block has been programmed.

Solution The pocket profile must be defined in ISO code. High level instructions are not allowed (GOTO, MSG, RPT ...).

0025 'Program: G77 axes (2 to 6) or G77 S.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the "axis slaving function (G77)» the parameters for the axes are missing or in "spindle synchronization (G77S) functions the "S" parameter is missing.

Solution In the "axis slaving" function, program at least two axes and in the "spindle synchronization" function, always program the "S" parameter.

0026 'Program: G93 IJ.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Polar origin preset (G93)» function, some of the parameters for the new polar origin have not been programmed.

Solution Remember that the programming format for this function is:

G93 I...J...

The "I", "J" values are optional, but if programmed, both must be programmed and they indicate the new polar origin.

0027 'G49 T X Y Z S, X Y Z A B C ', or, ' X Y Z Q R S.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Incline plane definition (G49)» function, a parameter has been programmed twice.

Solution Check the syntax of the block. The programming formats are:

T X Y Z S   X Y Z A B C   X Y Z Q R S

0028 'G2 or G3 not allowed when programming a canned cycle.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A canned cycle has been attempted to execute while the "G02", "G03" or "G33" functions were active.

Solution To execute a canned cycle, "G00" or "G01" must be active. A "G02" or "G03" function may be programmed previously in the program history. Check that these functions are not active when the canned cycle is defined.

0029 'G60: [A] /X I K/(2) [P Q R S T U V].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Multiple machining in a straight line (G60)» have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

Solution In this type of machining, two of the following parameters must always be programmed:

X Path length.
I Step between machining operations.
K : Number of machining operations.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0030 'G61-2: [A B] /X I J/(2) Y J D (2)/ [P Q R S T U V].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Multiple machining in a parallelogram pattern (G61)» or «Multiple machining in a grid pattern (G62)» cycle have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

Solution This type of machining requires the programming of two parameters of each group (X, I, K) and (Y, J, D).

X/Y Path length.
I/J Step between machining operations.
K /D Number of machining operations.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0031 'G63: X Y /I K/(1) [C P][P Q R S T U V].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Multiple machining in a circle (G63)» cycle have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

Solution This type of machining requires the programming of:

X/Y Distance from the center to the first hole.

And one of the following data:

I Angular step between machining operations.
K : Number of machining operations.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0032 'G64: X Y /I K/(1) [C P][P Q R S T U V.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «multiple machining in an arc (G64)» cycle have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

Solution This type of machining requires the programming of:

X/Y Distance from the center to the first hole.
B Total angular travel.

And one of the following data:

I Angular step between machining operations.
K : Number of machining operations.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0033 'G65: X Y /A I/(1) [C P].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Multiple machining programmed by means of an arc chord (G65)» cycle have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. Some data might be superfluous.

Solution This type of machining requires the programming of:

X/Y Distance from the center to the first hole.

And one of the following data:

A Angle of the matrix of the chord with the abscissa axis (in degrees).
I Chord length.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0034 'G66: [D H][R I][C J][F K] S E [Q].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Irregular pocket canned cycle with islands (G66)» have been programmed wrong. These may be the probable causes:

1. A parameter has been programmed which does not match the calling format.
2. Some mandatory parameter is missing.
3. The parameters of the cycle have not been edited in the correct order.

Solution This machining cycle requires the programming of :

S First block of the description of the geometry of the profiles making up the pocket.
E End block of the description of the geometry of the profiles making up the pocket.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. Also, the following parameters cannot be defined:

H if D has not been defined.
I if R has not been defined.
J if C has not been defined.
K : if F has not been defined.

The (X...C) position where the machining takes place cannot be programmed either.

0035 'G67: [A] B [C] [I] [R] [K] [V].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the roughing (2D/3D pocket) or semi-finishing (3D pocket) operation have been programmed wrong in the «Irregular pocket canned cycle with islands». These may be the probable causes:

1. A parameter has been programmed which does not match the calling format.
2. Some mandatory parameter is missing.
3. The parameters of the cycle have not been edited in the correct order.

Solution This machining cycle requires the programming of :

Roughing operation (2D or 3D pockets)

B Cutting pass
I Total pocket depth.
R Coordinate of the reference plane.

Semi-finishing operation (3D pockets)

B Cutting pass
I Total pocket depth (if no roughing operation has been defined).
R Coordinate of the reference plane (if no roughing operation has been defined).

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place cannot be programmed in this cycle.

0036 ‘G68: [B] [L] [Q] [J] [I] [R] [K].’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters for the finishing operation (2D/3D pocket) have been programmed wrong in the «Irregular pocket cycle with islands. These may be the probable causes:

1. A parameter has been programmed which does not match the calling format.
2. Some mandatory parameter is missing.
3. The parameters of the cycle have not been edited in the correct order.

Solution This machining cycle requires the programming of :

2D pockets

B	Cutting pass (if no roughing operation has been defined).
I	Total pocket depth (if no roughing operation has been defined).
R	Coordinate of the reference plane (if no roughing operation has been defined).

3D pockets

B	Cutting pass
I	Total pocket depth (if no roughing or semi-finishing operation has been defined).
R	Coordinate of the reference plane (if no roughing or semi-finishing operation has been defined).

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place cannot be programmed in this cycle.

0037 ‘G69: I B [C D H J K L R].’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Deep hole drilling cycle with variable peck (G69)». These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.

Solution This type of machining requires the programming of:

I	Machining depth.
B	Drilling peck.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0038 ‘G81-84-85-86-89: I [K].’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters have been programmed wrong in the following cycles: drilling (G81), tapping (G84), reaming (G85) or boring (G86/G89). This could be because parameter “I : Machining depth” is missing in the canned cycle being edited.

Solution This type of machining requires the programming of:

I	Machining depth.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0039 ‘G82: I K.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters have been programmed wrong in the «Drilling cycle with dwell (G82)». This could be because some parameter is missing.

Solution Both parameters must be programmed in this cycle:

I	Machining depth.
K :	Dwell at the bottom.

To program a drilling operation without dwell at the bottom, use function G81.

The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0040 'G83: I J.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters have been programmed wrong in the «Deep hole drilling with constant peck (G83)». This could be because some parameter is missing.

Solution This type of machining requires the programming of:

- I Machining depth.
- J Number of pecks.

The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0041 'G87: I J K B [C] [D] [H] [L] [V].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters have been programmed wrong in the «Rectangular pocket canned cycle (G87)». These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.

Solution This type of machining requires the programming of:

- I Pocket depth.
- J Distance from the center to the edge of the pocket along the abscissa axis.
- K : Distance from the center to the edge of the pocket along the ordinate axis.
- B Defines the machining pass along the longitudinal axis.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0042 'G88: I J B [C] [D] [H] [L] [V].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters have been programmed wrong in the «Circular pocket canned cycle (G88)». These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.

Solution This type of machining requires the programming of:

- I Pocket depth.
- J Pocket radius.
- B Defines the machining pass along the longitudinal axis.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0043 'Incomplete Coordinates.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. During simulation or execution, when trying to make a movement defined with only one coordinate of the end point or without defining the arc radius while a «circular interpolation (G02/G03) is active.
2. During editing, when editing a circular movement (G02/G03) by defining only one coordinate of the end point or not defining the arc radius.

Solution The solution for each cause is:

1. A "G02" or "G03" function may be programmed previously in the program history. In this case, to make a move, both coordinates of the end point and the arc radius must be defined. To make a linear movement, program "G01".
2. To make a circular movement (G02/G03), both coordinates of the end point and the arc radius must be programmed.

0044 'Incorrect Coordinates.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. An attempt has been made to execute a block syntactically incorrect (G1 X20K-15)
2. The "I" parameter is missing in the definition of a machining canned cycle (G81-G89) Machining depth.

Solution The solution for each cause is:

1. Correct the syntax of the block.
2. This type of machining requires the programming of:

   I Machining depth.

   The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message. The (X...C) position where the machining takes place can be programmed in this cycle.

0045 'Polar coordinates not allowed.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause When «Programming with respect to home (G53)», the end point has been defined in polar or cylindrical coordinates or in Cartesian coordinates with an angle.

Solution When programming with respect to home, only Cartesian coordinates may be programmed.

0046 'Axis does not exist.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When editing a block whose execution involves the movement of a nonexistent axis.
2. Sometimes, this error comes up while editing a block that is missing a parameter of the «G» function. This is because some parameters with an axis name have a special meaning inside certain «G» functions. For example: G69 I...B....
   In this case, parameter "B" has a special meaning after "I". If the "I" parameter is left out, the CNC assumes "B" as the position where the machining takes place on that axis. If that axis does not exist, it will issue this error message.

Solution The solution for each cause is:

1. Check that the axis name being edited is correct.
2. Check the block syntax and make sure that all the mandatory parameters have been programmed.

0047 'Program axes.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause No axis has been programmed in a function requiring an axis.

Solution Some instructions require the programming of axes (REPOS, G14, G20, G21…).

0048 'Incorrect order of axes.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The axis coordinates have not been programmed in the correct order or an axis has been programmed twice in the same block.

Solution Remember that the correct programming order for the axes is:

X...Y...Z...U...V...W...A...B...C...

All axes need not be programmed:

0049 'Point incompatible with active plane.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When trying to do a circular interpolation, the end point is not in the active plane.
2. When trying to do a tangential exit in a path that is not in the active plane.

Solution The solution for each cause is:

1. Maybe a plane has been defined with "G16", "G17", "G18" or "G19". In this case, circular interpolations can only be carried out on the main axes defining that plane. To define a circular interpolation in another plane, it must be defined beforehand.

2. Maybe a plane has been defined with “G16”, “G17”, “G18” or “G19”. In this case, corner rounding, chamfers and tangential entries/exits can only be carried out on the main axes defining that plane. To do it in another plane, it must be defined beforehand.

0050 ‘Program positions on active plane.’

No explanation required

0051 ‘Perpendicular axis included in active plane.’

No explanation required

0052 ‘Center of circle programmed incorrectly.’

No explanation required

0053 ‘Program pitch.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Electronic threading cycle (G33)» the parameter for the thread pitch is missing.

Solution Remember that the programming format for this function is:

G33 X...C...L...

Where: “L” is the thread pitch.

0054 ‘Pitch programmed incorrectly.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A helical interpolation has been programmed with the wrong or negative pitch.

Solution Remember that the programming format is:

G02/G03 X...Y...I...J...Z...K...

Where: “K” is the helical pitch (always positive value).

0055 ‘Positioning axes or Hirth axes not allowed’

No explanation required

0056 ‘The axis is already slaved.’

No explanation required

0057 ‘Do not program a slaved axis.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When trying to move an axis alone while being slaved to another one.
2. When trying to slave an axis that is already slaved using the G77 function «Electronic axis slaving».

Solution The solution for each cause is:

1. A slaved axis cannot be moved separately. To move a slaved axis, its master axis must be moved. Both axes will move at the same time.
   Example: If the Y axis is slaved to the X axis, an X axis move must be programmed in order to move the Y axis (together with the X axis).
   To unslave the axes, program “G78”.
2. An axis cannot be slaved to two different axes at the same time. To unslave the axes, program “G78”.

0058 ‘Do not program a GANTRY axis.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When trying to move an axis alone while being slaved to another one as a GANTRY axis
2. When defining an operation on a GANTRY axis. (Definition of work zone limits, planes, etc.).

Solution The solution for each cause is:

1. A GANTRY axis cannot be moved separately. To move a GANTRY axis, its associated axis must be moved. Both axes will move at the same time.
   Example: If the Y axis is a GANTRY axis associated with the X axis, an X axis move must be programmed in order to move the Y axis (together with the X axis).
   GANTRY axes are defined by machine parameter.
2. The axes defined as GANTRY cannot be used in the definition of operations or movements. These operations are defined with the main axis that the GANTRY axis is associated with.

0059 'Eje HIRTH: program only integer values.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A rotation of a HIRTH axis has been programmed with a decimal value.

Solution HIRTH axes do not accept decimal angular values. They must be full degrees.

0060.'Invalid action.'

No explanation required

0061 'ELSE not associated with IF.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. While editing in High level language, when editing the "ELSE" instruction without having previously programmed an "IF".
2. When programming in high level language, an "IF" is programmed without associating it with any action after the condition.

Solution Remember that the programming formats for this instruction are:

(IF (condition) <action1>)
(IF (condition) <action1> ELSE <action2>)

If the condition is true, it executes the < action1>, otherwise, it executes < action2>.

0062 'Program label N(0-9999).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a block number out of the 0-9999 range has been programmed in the "RPT" or "GOTO" instruction.

Solution Remember that the programming format of these instructions is:

(RPT N(block number), N(block number))
(GOTO N(block number))

The block number (label) must be between 0 and 9999.

0063 'Program subroutine number 1 thru 9999.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a subroutine number out of the 0-9999 range has been programmed in the "SUB" instruction.

Solution Remember that the programming format for this instruction is:

(SUB (integer))

The subroutine number must be between 0 and 9999.

0064 'Repeated subroutine.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause There has been an attempt to define a subroutine already existing in another program of the memory.

Solution In the CNC memory, there could not be more than one subroutine with the same identifying number even if they are contained in different programs.

0065 'The main program cannot have a subroutine.'

Detection In execution or while executing programs transmitted via DNC.

Cause The various causes might be:

1. An attempt has been made to define a subroutine in the MDI execution mode.
2. A subroutine has been defined in the main program.

Solution The solution for each cause is:

1. Subroutines cannot be defined from the «MDI execution» option of the menu.
2. Subroutines must be defined after the main program or in a separate program. They cannot be defined before or inside the main program.

0066 'Expecting a message.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level, the "MSG" or "ERROR" instruction has been edited but without the message to be displayed.

Solution Remember that the programming format of these instructions is:

(MSG "message")
(ERROR integer, "error message")

Although it can also be programmed as follows:

(ERROR integer)
(ERROR "error message")

0067 'OPEN is missing.'

Detection In execution or while executing programs transmitted via DNC.

Cause While programming in high level, a "WRITE" instruction has been edited, but the OPEN instruction has not been written previously to tell it where that instruction has to be executed.

Solution The "OPEN" instruction must be edited before the "WRITE" instruction to «tell» the CNC where (in which program) it must execute the "WRITE" instruction.

0068 'Expecting a program number.'

No explanation required

0069 'Program does not exist.'

Detection In execution or while executing programs transmitted via DNC.

Cause Inside the «Irregular pocket with islands cycle (G66)», it has been programmed that the profiles defining the irregular pocket are in another program (parameter "Q"), but that program does not exist.

Solution Parameter "Q" defines which program contains the definition of the profiles that, in turn, define the irregular pocket with islands. If this parameter is programmed, that program number must exist and it must contain the labels defined by parameters "S" and "E".

0070 'Program already exists.'

Detection In execution or while executing programs transmitted via DNC.

Cause This error comes up during execution when using the "OPEN" instruction (While programming in high level language) to create an already existing program.

Solution Change the program number or use parameters A/D in the "OPEN" instruction:

(OPEN P.........,A/D,…)

Where:

A: Appends new blocks after the existing ones.
D: Deletes the existing program and it opens it as a new one.

0071 'Expecting a parameter'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When defining the function «Modification of canned cycle parameters (G79)», the parameter to be modified has not been indicated.
2. While editing the machine parameter table, the wrong parameter number has been entered (maybe the "P" character is missing) or another action is being carried out (moving around in the table) before quitting the table editing mode.

Solution The solution for each cause is:

1. To define the "G79" function, the cycle parameter to be modified must be indicated as well as its new value.
2. Enter the parameter number to be edited or press [ESC] to quit this mode.

0072 'Parameter does not exist.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the "ERROR" instruction has been edited, but the error number to be displayed has been defined either with a local parameter greater than 25 or with a global parameter greater than 299.

Solution The parameters used by the CNC are:

Local:	0-25
Global:	100-299

0073 'Parameter range protected. Cannot be written.'

No explanation required

0074 'Variable not accessible from CNC.'

No explanation required

0075 'Read-only variable.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to assign a value to a read-only variable.

Solution Read-only variables cannot be assigned any values through programming. However, their values can be assigned to a parameter.

0076 'Write-only variable.'

No explanation required

0077 'Analog output not available.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to write to an analog output currently being used by the CNC.

Solution The selected analog output may be currently used by an axis or a spindle. Select another analog output between 1 and 8.

0078 'Program channel 0(CNC),1(PLC) or 2(DNC).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the "KEYSCR" instruction has been programmed, but the source of the keys is missing.

Solution When programming the "KEYSCR" instruction, the parameter for the source of the keys must always be programmed:

(KEYSCR=0) : CNC keyboard
(KEYSCR=1) : PLC
(KEYSCR=2) : DNC

The CNC only lets modifying the contents of this variable if it is «zero»

0079 'Program error number 0 thru 9999.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the "ERROR" instruction has been programmed, but the error number to be displayed is missing.

Solution Remember that the programming format for this instruction is:

(ERROR integer, "error message")

Although it can also be programmed as follows:

(ERROR integer)
(ERROR "error message")

0080 'Operator missing.'

No explanation required

0081 'Incorrect expression.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, an expression has been edited with the wrong format.

Solution Correct the syntax of the block.

0082 'Incorrect operation.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. While programming in high level language, the assignment of a value to a parameter is incomplete.
2. While programming in high level language, the call to a subroutine is incomplete.

Solution Correct (complete) the format to assign a value to a parameter or a call to a subroutine.

0083 'Incomplete operation.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. While programming in high level language, the "IF" instruction has been edited without the condition between brackets.
2. While programming in high level language, the "DIGIT" instruction has been edited without assigning a value to some parameter.

Solution The solution for each cause is:

1. Remember that the programming formats for this instruction are:
   (IF (condition) <action1>)
   (IF (condition) <action1> ELSE <action2>)
   If the condition is true, it executes the < action1>, otherwise, it executes < action2>.
2. Correct the syntax of the block. All the parameters defined within the "DIGIT" instruction must have a value assigned to them.

0084 'Expecting "=".'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

Solution Enter the "=" symbol in the right place.

0085 'Expecting ")".'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

Solution Enter the ")" symbol in the right place.

0086 'Expecting "(".'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

Solution Enter the "(" symbol in the right place.

0087 'Expecting ",".'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.
2. While programming in high level language, an ISO-coded instruction has been programmed.
3. While programming in high level language, an operation has been assigned either to a local parameter greater than 25 or to a global parameter greater 299.

Solution The solution for each cause is:

1. Enter the "," symbol in the right place.
2. A block cannot contain high level language instructions and ISO-coded instructions at the same time.

3. The parameters used by the CNC are:

Local:	0-25.
Global:	100-299.

Other parameters out of this range cannot be used in operations.

0088 'Operation limit exceeded.'

No explanation required

0089 'Logarithm of zero or negative number.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves the calculation of a negative number or a zero.

Solution Only logarithms of numbers greater than zero can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0090 'Square root of a negative number.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves the calculation of the square root of a negative number.

Solution Only the square root of numbers greater than zero can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0091 'Division by zero.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves a division by zero.

Solution Only divisions by numbers other than zero are allowed. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0092 'Base zero with positive exponent.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves elevating zero to a negative exponent (or zero).

Solution Zero can only be elevated to positive exponents greater than zero. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0093 'Negative base with decimal exponent.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves elevating a negative number to a decimal exponent.

Solution Negative numbers can only be elevated to integer exponents. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0094 'ASIN/ACOS range exceeded.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves calculating the arcsine or arccosine of a number out of the ±1 range.

Solution Only the arcsine (ASIN) or arccosine (ACOS) of numbers between ±1 can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0095 'Program row number.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While editing a customizing program, a window has been programmed with the "ODW" instruction, but the vertical position of the window on the screen is missing.

Solution The vertical position of the window on the screen is defined by rows (0-25).

0096 'Program column number.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While editing a customizing program, a window has been programmed with the "ODW" instruction, but the horizontal position of the window on the screen is missing.

Solution The horizontal position of the window on the screen is defined by columns (0-79).

0097 'Program another softkey.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While editing a customizing program, the programming format for the "SK" instruction has not been respected.

Solution Correct the syntax of the block. The programming format is:
(SK1=(text 1), SK2=(text 2)…)

If the "," character is entered after a text, the CNC expects the name of another softkey.

0098 'Program softkeys 1 thru 7.'

Detection While executing in the user channel.

Cause In the block syntax, a softkey has been programmed out of the 1 to 7 range.

Solution Only softkeys within the 1 to 7 range can be programmed.

0099 'Program another window.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While editing a customizing program, the programming format for the "DW" instruction has not been respected.

Solution Correct the syntax of the block. The programming format is:
(DW1=(assignment), DW2=(assignment)…)

If the "," character is entered after an assignment, the CNC expects the name of another window.

0100 'Program windows 0 thru 25.'

Detection While executing in the user channel.

Cause In the block syntax, a window has been programmed out of the 0 to 25 range.

Solution Only windows within the 0 to 25 range can be programmed.

0101 'Program rows 0 thru 20.'

Detection While executing in the user channel.

Cause In the block syntax, a row has been programmed out of the 0 to 20 range.

Solution Only rows within the 0 to 20 range can be programmed.

0102 'Program columns 0 thru 79.'

Detection While executing in the user channel.

Cause In the block syntax, a column has been programmed out of the 0 to 79 range.

Solution Only columns within the 0 to 79 range can be programmed.

0103 'Program pages 0 thru 255.'

Detection While executing in the user channel.

Cause In the block syntax, a page has been programmed out of the 0 to 255 range.

Solution Only pages within the 0 to 255 range can be programmed.

0104 'Program INPUT.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, an "IB" instruction has been edited without associating an "INPUT" to it.

Solution Remember that the programming formats for this instruction are:
(IB (expression) = INPUT “text”, format)
(IB (expression) = INPUT “text”)

0105 ‘Program inputs 0 thru 25.’

Detection While executing in the user channel.

Cause In the block syntax, an input has been programmed out of the 0 to 25 range.

Solution Only inputs within the 0 to 25 range can be programmed.

0106 ‘Program numerical format.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, an “IB” instruction has been edited with non-numeric format.

Solution Remember that the programming format for this instruction is:
(IB (expression) = INPUT “text”, format)

Where «format» must be a signed number with 6 entire digits and 5 decimals at the most.

If the “,” character is entered after the text, the CNC expects the format.

0107 ‘Do not program formats greater than 6.5 .’

Detection While executing in the user channel.

Cause While programming in high level language, an “IB” instruction has been edited in a format with more than 6 entire digits or more than 5 decimals.

Solution Remember that the programming format for this instruction is:
(IB (expression) = INPUT “text”, format)

Where «format» must be a signed number with 6 entire digits and 5 decimals at the most.

0108 ‘This command can only be executed in the user channel.’

Detection During execution.

Cause An attempt has been made to execute a block containing information that can only be executed through the user channel.

Solution There are specific expressions for customizing programs that can only be executed inside the user program.

0109 ‘C. User: do not program geometric help, compensation or cycles.’

Detection While executing in the user channel.

Cause An attempt has been made to execute a block containing geometric aide, tool radius/length compensation or machining canned cycles.

Solution Inside a customizing program the following cannot be programmed:
Neither geometric assistance nor movements.
Neither tool radius nor length compensation.
Canned cycles.

0110 ‘Local parameters not allowed.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause Some functions can only be programmed with global parameters.

Solution Global parameters are the ones included in the 100-299 range.

0111 ‘Block cannot be executed while running another program’

Detection While executing in MDI mode

Cause An attempt has been made to execute a customizing instruction from MDI mode while the user channel program is running.

Solution Customizing instructions can only be executed through the user channel.

0112 ‘WBUF can only be executed in user channel while editing’

Detection During normal execution or execution through the user channel.

Cause An attempt has been made to execute the “WBUF” instruction.

Solution The "WBUF" instruction cannot be executed. It can only be used in the editing stage through the user input.

0113 'Table limits exceeded.'

Detection While editing tables.

Cause The various causes might be:

1. In the tool offset table, an attempt has been made to define a tool offset with a greater number than allowed by the manufacturer.
2. In the parameter tables, an attempt has been made to define a nonexistent parameter.

Solution The tool offset number must be smaller than the one allowed by the manufacturer.

0114 'Offset: D3 R L I K.'

Detection While editing tables.

Cause In the tool offset table, the parameter editing order has not been respected.

Solution Enter the table parameters in the right order.

0115 'Tool: T4 D3 F3 N5 R5(.2).'

Detection While editing tables.

Cause In the tool table, the parameter editing order has not been respected.

Solution Enter the table parameters in the right order.

0116 'Origin: G54-59 axes (1-5).'

Detection While editing tables.

Cause In the Zero offset table, the zero offset to be defined (G54-G59) has not be selected.

Solution Enter the table parameters in the right order. To fill out the zero offset table, first select the offset to be defined (G54-G59) and then the zero offset position for each axis.

0117 'Function: M4 S4 bits(8).'

Detection While editing tables.

Cause In the «M» function table, the parameter editing order has not been respected.

Solution Edit table following the format:
M1234 (associated subroutine) (customizing bits)

0118 'G51 [A] E'

Detection In execution or while executing programs transmitted via DNC.

Cause In the «Look-Ahead (G51)» function, the parameter for the maximum contouring error is missing.

Solution This type of machining requires the programming of:
E : Maximum contouring error.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0119 'Leadscrew: Coordinate-error.'

Detection While editing tables.

Cause In the leadscrew compensation tables, the parameter editing order has not been respected.

Solution Enter the table parameters in the right order.
P123 (position of the axis to be compensated) (leadscrew error at that point)

0120 'Incorrect axis.'

Detection While editing tables.

Cause In the leadscrew compensation tables, an attempt has been made to edit a different axis from the one corresponding to that table.

Solution Each axis has its own table for leadscrew compensation. The table for each axis can only contain the positions for that axis.

0121 'Program P3 = value.'

Detection While editing tables.

Cause In the machine parameter table, the editing format has not been respected.

Solution Enter the table parameters in the right order.
P123 = (parameter value)

0122 'Magazine: P(1-255) = T(1-9999).'

Detection While editing tables.

Cause In the tool magazine table, the editing format has not been respected or some data is missing.

Solution Enter the table parameters in the right order.

0123 'Tool T0 does not exist.'

Detection While editing tables.

Cause In the tool table, an attempt has been made to edit a tool as T0.

Solution No tool can be edited as T0. The first tool must be T1.

0124 'Offset D0 does not exist.'

Detection While editing tables.

Cause In the tool table, an attempt has been made to edit a tool offset as D0.

Solution No tool offset can be edited as D0. The first tool offset must be D1.

0125 'Do not modify the active tool or the next one.'

Detection During execution.

Cause In the tool magazine table, an attempt has been made to change the active tool or the next one.

Solution During execution, neither the active tool nor the next one may be changed.

0126 'Tool not defined.'

Detection While editing tables.

Cause In the tool magazine table, an attempt has been made to assign to the magazine position a tool that is not defined in the tool table.

Solution Define the tool in the tool table.

0127 'Magazine is not RANDOM.'

Detection While editing tables.

Cause There is no RANDOM magazine and, in the tool magazine table, the tool number does not match the tool magazine position.

Solution When the tool magazine is not RANDOM, the tool number must be the same as the magazine position (pocket number).

0128 'The position of a special tool is set.'

Detection While editing tables.

Cause In the tool magazine table, an attempt has been made to place a tool in a magazine position reserved for a special tool.

Solution When a special tool occupies more than one position in the magazine, it has a reserved position in the magazine. No other tool can be placed in this position.

0129 'Next tool only possible in machining centers.'

Detection During execution.

Cause A tool change has been programmed with M06, but the machine is not a machining center. (it is not expecting the next tool).

Solution When the machining is not a machining center, the tool change is done automatically when programming the tool number «T».

0130 'Write 0/1.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values of 0 or 1.

0131 'Write +/-.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values of + or -.

0132 'Write YES/NO.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values of YES or NO.

0133 'Write ON/OFF.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values of ON or OFF.

0134 'Values 0 thru 2.'
0135 'Values 0 thru 3.'
0136 'Values 0 thru 4.'
0137 'Values 0 thru 9.'
0138 'Values 0 thru 29.'
0139 'Values 0 thru 100.'
0140 'Values 0 thru 255.'
0141 'Values 0 thru 9999.'
0142 'Values 0 thru 32767.'
0143 'Values within +/-32767.'
0144 'Values 0 thru 65535.'

Detection While editing machine parameters

Cause The various causes might be:

1. An attempt has been made to assign the wrong value to a parameter.
2. During execution, when inside the program a call has been made to a subroutine (MCALL, PCALL) with a value greater than allowed.

0145 'Format +/- 5.5.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values with the format:

0146 'Word does not exist.'

No explanation required

0147 'Numerical format exceeded.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A data or parameter has been assigned a value greater than the established format.

Solution Correct the syntax of the block. Most of the time, the numeric format will be 5.4 (5 integers and 4 decimals).

0148 'Text too long.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the "ERROR" or "MSG" instruction has been assigned a text with more than 59 characters.

Solution Correct the syntax of the block. The "ERROR" and "MSG" instructions cannot be assigned texts longer than 59 characters.

0149 'Incorrect message.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the text associated with the “ERROR” or “MSG” instruction has been edited wrong.

Solution Correct the syntax of the block. The programming format is:

(MSG “message”)
(ERROR number, “message”)

The message must be between “ ”.

0150 ‘Incorrect number of bits.’

Detection While editing tables.

Cause The various causes might be:

1. In the «M» function table, in the section on customizing bits:
   The number does not have 8 bits.
   The number does not consist of 0’s and 1’s.
2. In the machine parameter table, an attempt has been made to assign the wrong value of bit to a parameter.

Solution The solution for each cause is:

1. The customizing bits must consist of 8 digits of 0’s and 1’s.
2. The parameter only admits 8-bit or 16-bit numbers.

0151 ‘Negative numbers not allowed.’

No explanation required

0152 ‘Incorrect parametric programming.’

Detection During execution.

Cause The parameter has a value that is incompatible with the function it has been assigned to.

Solution This parameter may have taken the wrong value, in the program history. Correct the program so this parameter does not reach the function with that value.

0153 ‘Decimal format not allowed.’

No explanation required

0154 ‘Insufficient memory.’

Detection During execution.

Cause The CNC does not have enough memory to internally calculate the paths.

Solution Sometimes, this error is taken care of by changing the machining conditions.

0155 ‘Help not available.’

No explanation required

0156 ‘Don’t program G33 ,G34, G95 or M19 S with no spindle encoder’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A “G33”, “G34”, “G95”, “G95” or “M19 S” has been programmed without having an encoder on the spindle.

Solution If the spindle does not have an encoder, functions “M19 S”, “G33”, “G34” or “G95” cannot be programmed. Spindle machine parameter “NPULSES (P13)” indicates the number of encoder pulses per turn.

0157 ‘G79 not allowed when there is no active canned cycle.’

Detection During execution.

Cause An attempt has been made to execute the «Modification of canned cycle parameters (G79)» function without any canned cycle being active.

Solution The “G79” function modifies the values of a canned cycle; therefore, there must be an active canned cycle and the “G79” must be programmed in the influence zone of that canned cycle.

0158 ‘Tool T must be programmed with G67 and G68.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Irregular pocket canned cycle with islands (G66)» the tool has not been defined for roughing "G67" (2D/3D pockets) for semi-finishing "G67" (3D pocket) or finishing "G68" (2D/3D pocket).

Solution The irregular pocket canned cycle with islands requires the programming of the roughing tool "G67" (2D/3D pockets), the semi-finishing tool "G67" (3D pocket) and the finishing tool "G68" (2D/3D pocket).

0159 'Inch programming limit exceeded.'

Detection During execution.

Cause An attempt has been made to execute in inches a program edited in millimeters.

Solution Enter function G70 (inch programming) or G71 (mm programming) at the beginning of the program.

0160 'G79 not allowed when executing the canned cycle.'

No explanation required

0161 'G66 must be programmed before G67 and G68.'

Detection During execution.

Cause A roughing operation "G67" (2D/3D pockets), a semi-finishing operation "G67" (3D pocket) or a finishing operation "G68" (2D/3D pocket) has been programmed without having previous programmed the call to an «Irregular pocket canned cycle with islands (G66)».

Solution When working with irregular pockets, before programming the aforementioned cycles, the call to the «Irregular canned cycle with islands (G66)» must be programmed.

0162 'No negative radius allowed with absolute coordinates'

Detection During execution.

Cause While operating with absolute polar coordinates, a movement with a negative radius has been programmed.

Solution Negative radius cannot be programmed when using absolute polar coordinates.

0163 'The programmed axis is not longitudinal.'

Detection During execution.

Cause An attempt has been made to modify the coordinates of the point where the canned cycle is to be executed using the «Modification of the canned cycle parameters (G79)»function.

Solution With "G79", the parameters defining a canned cycle may be modified, except the coordinates of the point where it will be executed. To change those coordinates, program only the new coordinates.

0164 'Wrong password.'

Detection While assigning protections.

Cause [ENTER] has been pressed before selecting the type of code to be assigned a password.

Solution Use the softkeys to select the type of code to which a password is to be assigned.

0165 'Password: use uppercase/lowercase letters or digits.'

Detection While assigning protections.

Cause A bad character has been entered in the password.

Solution The password can only consist of letters (upper and lower case) or digits.

0166 'Only one HIRTH axis per block is allowed.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A movement has been programmed which involves the movement of two HIRTH axes simultaneously.

Solution The CNC does not admit movements involving more than one HIRTH axis at a time. HIRTH axes must move one at a time.

0167 'Rot. axis position: absolute values (G90) within 0-359.9999.'

Detection During execution.

Cause A movement of a positioning-only rotary axis has been programmed. The movement has been programmed in absolute coordinates (G90) and the target coordinate of the movement is not within the 0 to 359.9999 range.

Solution Positioning-only rotary axes: In absolute coordinates, only movements within the 0 to 359.9999 range are possible.

0168 'Rotary axis: Absolute values (G90) within +/-359.9999.'

Detection During execution.

Cause A movement of a rotary axis has been programmed. The movement has been programmed in absolute coordinates (G90) and the target coordinate of the movement is not within the 0 to 359.9999 range.

Solution Rotary axes: In absolute coordinates, only movements within the 0 to 359.9999 range are possible.

0169 'Modal subroutines cannot be programmed.'

Detection While executing in MDI mode

Cause An attempt has been made to call upon a modal subroutine (MCALL).

Solution MCALL modal subroutines cannot be executed from the menu option «MDI execution».

0170 'Program symbols 0 thru 255 in positions 0-639, 0-335. '

No explanation required

0171 'The window must be previously defined.'

Detection During normal execution or execution through the user channel.

Cause An attempt has been made to write in a window (DW) that has not been previously defined (ODW).

Solution It is not possible to write in a window that has not been previously defined. Check that the window to write in (DW) has been previously defined.

0172 'The program is not accessible'

Detection During execution.

Cause An attempt has been made to execute a program that cannot be executed.

Solution The program may be protected against execution. To know whether a program may be executed, check for the "X" character on the attributes column. If this character is missing, the program cannot be executed.

0173 'It is not possible to program angle + angle.'

No explanation required

0174 'Circular (helical) interpolation not possible.'

Detection During execution.

Cause An attempt has been made to execute a helical interpolation while the «LOOK-AHEAD (G51)» function was active.

Solution Helical interpolations are not possible while the «LOOK-AHEAD (G51)» function is active.

0175 'Analog inputs: ANAI(1-8) = +/-5 Volts.'

Detection During execution.

Cause An analog input has taken a value out of the ±5V range.

Solution Analog inputs may only take values within the ±5V range.

0176 'Analog outputs: ANAO(1-8) = +/-10 Volts.'

Detection During execution.

Cause An analog output has been assigned a value out of the ±10V range.

Solution Analog outputs may only take values within the ±10V range.

0177 'A gantry axis cannot be part of the active plane.'

No explanation required

0178 'G96 only possible with analog spindle.'

Detection During execution.

Cause The "G96" function has been programmed but either the spindle speed is not controlled or the spindle does not have an encoder.

Solution To operate with the "G96" function, the spindle speed must be controlled (SPDLTYPE(P0)=0) and the spindle must have an encoder (NPULSES(P13) other than zero).

0179 'Do not program more than 4 axes simultaneously.'

No explanation required

0180 'Program DNC1/2, HD or CARD A (optional).'

Detection While editing or executing.

Cause While programming in high level language, in the "OPEN" and "EXEC" instructions, an attempt has been made to program a parameter other than DNC1/2, HD or CARD A, or the DNC parameter has been assigned a value other than 1 or 2.

Solution Check the syntax of the block.

0181 'Program A (append) or D (delete).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the "OPEN" instruction the A/D parameter is missing.

Solution Check the syntax of the block. The programming format is:
(OPEN P.........,A/D,…)

Where:

A Appends new blocks after the existing ones.
D: Deletes the existing program and it opens it as a new one.

0182 'Option not available.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A «G» function has been defined which is not a software option.

0183 'Cycle does not exist.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the "DIGIT" instruction, a digitizing cycle has been defined which is not available.

Solution The "DIGIT" instruction only admits two types of digitizing:
(DIGIT 1,…) : Grid pattern digitizing cycle.
(DIGIT 2,…) : Arc pattern digitizing cycle.

0184 'T with subroutine: program only T and D.'

No explanation required

0185 'Tool offset does not exist'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause Within the block syntax, a tool offset has been called upon which is greater than the ones allowed by the manufacturer.

Solution Program a new smaller tool offset.

0188 'Function not possible from PLC.'

Detection During execution.

Cause From the PLC channel and using the "CNCEX" instruction, an attempt has been made to execute a function that is incompatible with the PLC channel execution.

Solution The installation manual (chapter 11.1.2) offers a list of the functions and instructions that may be executed through the PLC channel.

0189 'The live tool does not exist.'

No explanation required

0190 ‘Programming not allowed while in tracing mode.’

Detection During execution.

Cause Among the blocks defining the «Tracing and digitizing canned cycles (TRACE)», there is block that contains a «G» function which does not belong in the profile definition.

Solution The «G» functions available in the profile definition are:

G00	G01	G02	G03	G06	G08	G09	G36
G39	G53	G70	G71	G90	G91	G93	

0191 ‘Do not program tracing axes.’

Detection During execution.

Cause An attempt has been made to move an axis that has been defined as a tracing axis using the “G23” function.

Solution The tracing axes are controlled by the CNC. To deactivate the tracing axes, use the “G25” function..

0192 ‘Incorrect active plane and longitudinal axis.’

Detection During execution.

Cause While programming in high level language, an attempt has been made to execute a probing cycle using the “PROBE” instruction, but the longitudinal axis is included in the active plane.

Solution The “PROBE” probing canned cycles are executed on the X, Y, Z axes, the active plane being formed by two of them. The other axis must be perpendicular and it must be selected as the longitudinal axis.

0193 ‘G23 has not been programmed.’

Detection During execution.

Cause Digitizing “G24” has been activated or a tracing contour “G27” has been programmed, but without previously activating the tracing function “G23”.

Solution To digitize or operate with a contour, the tracing function must be activated previously.

0194 ‘Repositioning not allowed.’

Detection During execution.

Cause The axes cannot be repositioned using the “REPOS” instruction because the subroutine has not been activated with one of the interruption inputs.

Solution Before executing the “REPOS” instruction, one of the interruption inputs must be activated.

0195 ‘Axes X, Y or Z slaved or synchronized.’

Detection During execution.

Cause While programming in high level language, an attempt has been made to execute a probing cycle using the “PROBE” instruction, but one of the X, Y or Z axis is slaved or synchronized.

Solution To execute the “PROBE”” instruction, the X, Y, Z axes must not be slaved or synchronized. To unslave the axes, program “G78”.

0196 ‘Axes X, Y and Z must exist.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, an attempt has been made to edit the “PROBE” instruction, but one of the X, Y or Z axis is missing.

Solution To operate with the “PROBE” instruction, the X, Y, Z axes must be defined.

0198 ‘Deflection out of range.’

Detection During execution.

Cause In the tracing cycle “G23”, a nominal probe deflection has been defined which is greater than the value set by machine parameter.

Solution Program a smaller nominal probe deflection.

0199 'Rotary axis preset: values between 0 and 359.9999.'

Detection While presetting coordinates.

Cause An attempt has been made preset the coordinates of a rotary axis with a value out of the 0 to 359.9999 range.

Solution The preset value of rotary axes must be within the 0 to 359.9999 range.

0200 'Program: G52 axis +/-5.5'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause When programming the «Movement against a hard stop (G52)», either the axis to be moved has not been programmed or several axes have been programmed.

Solution When programming "G52", the axis to be moved must be indicated. Only one axis may be programmed at a time.

0201 'Program only one positioning axis in G01.'

No explanation required

0202 'Program G27 only when tracing a profile.'

Detection During execution.

Cause A tracing contour (G27) has been defined, but the tracing function is neither bi-dimensional nor three-dimensional.

Solution The «Definition of a tracing contour (G27)» function must only be defined when tracing or digitizing in two or three dimensions.

0203 'G23-G27 not allowed during INSPECTION.'

No explanation required

0204 'Incorrect tracing method.'

Detection During execution.

Cause While executing a manual tracing "G23", an attempt has been made to jog a «follower» axis with the jog keys or the electronic handwheels.

Solution When executing a manual tracing, the axes selected as followers are moved by hand. The rest may be jogged with the jog keys or the electronic handwheels.

0205 'Incorrect digitizing method.'

Detection During execution.

Cause Point-to-point digitizing has been defined, but the CNC is not in jog mode (it is in either in simulation or execution mode, instead).

Solution To execute point-to-point digitizing, the CNC must be in jog mode.

0206 'Values 0 thru 6.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values between 0 and 6.

0207 'Complete Table.'

Detection While editing tables.

Cause In the tables for «M» functions or tool offsets, an attempt has been made to define more data than those allowed by the manufacturer by means of machine parameters. When loading a table via DNC, the CNC does not delete the previous table, it replaces the existing values and it copies the new data in the free positions of the table.

Solution The maximum number of data that can be defined is limited by the machine parameters:

Maximum number of «M» functions	NMISCFUN(P29).
Maximum number of	NTOOL(P23).
Maximum number of tool offset	NTOFFSET(P27).
Maximum number of magazine positions	NPOCKET(P25).

To load a new table via DNC, the previous table should be deleted.

0208 'Program A from 0 to 255'

Detection During execution.

Cause In the «LOOK-AHEAD (G51)» function, parameter "A" (% of acceleration to be applied) has been programmed with a value greater than 255.

Solution Parameter "A" is optional, but when programmed, it must have a value between 0 and 255.

0209 'Program nesting not allowed.'

Detection During execution.

Cause From a running program, an attempt has been made to execute another program with the "EXEC" instruction which in turn also has an "EXEC" instruction.

Solution Another program cannot be called upon from a program being executed using the "EXEC" instruction.

0210 'No compensation is permitted.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to activate or cancel tool radius compensation (G41, G42, G40) in a block containing a nonlinear movement.

Solution Tool radius compensation must be activated/deactivated in linear movements (G00, G01).

0211 'Do not program a zero offset without cancelling the previous one.'

Detection During execution.

Cause An attempt has been made to define an incline plane using the «Definition of the incline plane (G49)» function while another one was already defined.

Solution To define a new incline plane, the one previously defined must be canceled first. To cancel an incline plane, program "G49" without parameters.

0212 'Programming not permitted while G48-G49 are active.'

Detection During execution.

Cause While programming in high level language, an attempt has been made to execute a probing cycle with the "PROBE" instruction while function "G48" or "G49" was active.

Solution The digitizing cycles "PROBE" are carried out on the X, Y, Z axes. Therefore, neither the "G48" nor the "G49" function may be active when executing them.

0213 'A second spindle is required for G28, G29, G77 or G78.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to select the work spindle with "G28/G29" or synchronize spindles with "G77/G78", but the machine only has one work spindle.

Solution If the machine only has one work spindle, the "G28, G29, G77 and G78" functions cannot be programmed.

0214 'Invalid G function when selecting a profile'

Detection While restoring a profile.

Cause Within the group of blocks selected to restore the profile, there is a block containing a «G» code that does not belong in the profile definition.

Solution The «G» functions available in the profile definition are:

G00	G01	G02	G03	G06	G08	G09
G36	G37	G38	G39	G90	G91	G93

0215 'Invalid G function after first point of profile'

Detection While restoring a profile.

Cause Within the selected blocks for restoring the profile, and after the starting point of a profile, there is a block containing a «G» function that does not belong in the profile definition.

Solution The «G» functions available in the profile definition are:

G00	G01	G02	G03	G06	G08	G09
G36	G37	G38	G39	G90	G91	G93

0216 'Nonparametric assignment after first point of profile'

Detection While restoring a profile.

Cause Within the selected blocks for restoring the profile, and after the starting point of a profile, a nonparametric assignment has been programmed in high level language (a local or global parameter).

Solution The only high level instructions that can be edited are assignments to local parameters (P0 thru P25) and global parameters (P100 thru P299).

0217 'Invalid programming after first point of profile'

Detection While restoring a profile.

Cause Within the selected blocks for restoring the profile, and after the starting point of a profile, there is a high level block that is not an assignment.

Solution The only high level instructions that can be edited are assignments to local parameters (P0 thru P25) and global parameters (P100 thru P299).

0218 'The axis cannot be programmed after first point of profile'

Detection While restoring a profile.

Cause Within the selected blocks for restoring the profile, and after the starting point of a profile, a position has been defined on an axis that does not belong to the active plane. A surface coordinate may have been defined after the starting point of the profile.

Solution The surface coordinate of the profiles is only defined in the starting block of the first profile, the one corresponding to the starting point of the outside profile.

0219 'First point programmed wrong when selecting profile'

Detection While selecting a profile.

Cause The starting point of the profile has been programmed wrong. One of the two coordinates defining its position is missing.

Solution The starting point of a profile must be defined on the two axes forming the active plane.

0226 'A tool cannot be programmed with G48 active'

Detection During execution.

Cause A tool change has been programmed while the «TCP transformation (G48)» function is active.

Solution A tool change cannot take place while TCP transformation is active. To make a tool change, cancel TCP transformation first.

0227 'Program Q between +/-359.9999.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Electronic threading (G33)» function, the entry angle "Q" has been programmed with a value out of the ±359.9999 range.

Solution Program an entry angle within the ±359.9999 range.

0228 'Do not program "Q" with parameter M19TYPE=0.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Electronic threading (G33)» function, an entry angle "Q" has been programmed, but the type of spindle orientation available does not allow this operation.

Solution In order to define an entry angle, spindle machine parameter M19TYPE(P43) must be set to «1».

0229 'Program maximum X'
0230 'Program minimum Y'
0231 'Program maximum Y'
0232 'Program minimum Z'
0233 'Program maximum Z'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, in the "DGWZ" instruction, the indicated limit is missing or it has been defined with a non-numerical value.

Solution Check the syntax of the block.

0234 'Wrong graphic limits'

Detection During execution.

Cause One of the lower limits defined with the "DGWZ" instruction is greater than its corresponding upper limit.

Solution Program the upper limit of the graphics display area greater than the lower ones.

0235 'Do not program the axis in tangential control'

No explanation required

0236 'Do not program the longitudinal axis or the axis of the active plane'

No explanation required

0237 'Program values between +/-359.9999.'

Detection During execution.

Cause A G30 offset has been programmed greater than the maximum allowed. For example G30 D380

Solution The offset must be within ±359.9999.

0238 'Do not program G30 without synchronizing the spindles in speed'

Detection During execution.

Cause An attempt has been made to synchronize the spindles in "G30" offset without having them synchronized in speed.

Solution First, synchronize the spindle in speed using G77S.

0239 'Do not synchronize the spindles while the "C" axis is active'

Detection During execution.

Cause An attempt has been made to synchronize the spindle, but the "C" axis is not active.

Solution Activate the "C" axis first.

0240 'Do not activate the "C" axis while the spindles are synchronized'

Detection During execution.

Cause An attempt has been made to activate the "C" axis while the spindles were synchronized.

Solution First, cancel the spindle synchronization (G78 S).

0241 'Do not program G77 S, G78 S if there is no encoder at the spindle'

Detection During execution.

Cause An attempt has been made to synchronize the spindles (G77 S or G78 S) and one of them does not have an encoder or Sercos feedback.

Solution Both spindles must have an encoder or Sercos feedback.

0242 'Do not synchronize spindles with M19TYPE=0'

Detection During execution.

Cause An attempt has been made to synchronize the spindles (G77 S or G78 S) and one of them has parameter M19TYPE=0.

Solution Both spindles must have parameter M19TYPE=1

0243 'Values 0 thru 15.'

No explanation required

Block preparation and execution errors

1000 'There is no enough path information.'

Detection During execution.

Cause The program contains too many blocks without information about the path to apply tool radius compensation, rounding, chamfer or tangential entry or exit.

Solution In order to carry out these operations, the CNC needs to know in advance the path to follow; therefore, there cannot be more than 48 blocks in a row without information about the path to follow.

1001 'Plane change in rounding/chamfering.'

Detection During execution.

Cause A plane change has been programmed on the path following the definition of a "controlled corner rounding G36" or "chamfer (G39)".

Solution The plane cannot be changed while executing a rounding or a chamfer. The path following the definition of a rounding or chamfer must be in the same plane that the rounding or the chamfer.

1002 'Rounding radius too large.'

Detection During execution.

Cause In the "Controlled corner rounding (G36)" function, the programmed rounding radius is larger than one of the paths where it has been defined.

Solution The rounding radius must be smaller than the paths that define it.

1003 'Rounding in last block.'

Detection During execution.

Cause A "Controlled rounding radius (G36) or "Chamfer (G39) has been defined on the last path of the program or when the CNC does not find information about the path following the definition of the rounding or chamfer.

Solution A rounding or chamfer must be defined between two paths.

1004 'Tangential output programmed wrong'

Detection During execution.

Cause The move following the definition of a tangential output (G38) is a circular path.

Solution

The move following the definition of a tangential output must be a straight path.

1005 'Chamfer programmed wrong.'

Detection During execution.

Cause The move following the definition of a "Chamfer (G39)" is a circular path.

Solution The move following the definition of a chamfer must be a straight path.

1006 'Chamfer value too large.'

Detection During execution.

Cause In the "Chamfer (G39)" function, the programmed chamfer value is larger than one of the paths where it has been defined.

Solution The chamfer size must be smaller than the paths that define it.

1007 'G8 defined wrong.'

Detection During execution.

Cause The various causes might be:

1. When a full circle has been programmed using the function "Arc tangent to previous path (G08)"
2. When the tangent path ends in a point of the previous path or its extension (in a straight line).
3. In an irregular pocket canned cycle with islands, when programming function "G08" in the block following the definition of the beginning of the profile (G00).

Solution The solution for each cause is:

1. Function “G08” does not allow programming full circles.
2. Tangent path must not end in a point of the previous path or in its extension (in a straight line).
3. The CNC does not have information about the previous path and cannot execute the tangent arc.

1008 ‘There is no information about the previous path’

Detection During execution.

Cause An arc tangent to the previous path has been programmed using function “G08”, but there is no information about the previous path.

Solution To do a path tangent to the previous one, there must be information about the previous path and it must be within the 48 blocks preceding the tangent path.

1009 ‘There is no information for tangent arc in pockets with islands.’

Detection During execution.

Cause Within the set of blocks defining the profile of an irregular pocket with islands, a tangent arc has been programmed, but some data is missing or there is not enough information about the previous path.

Solution Check the data that defines the profile.

1010 ‘Wrong plane for tangent path.’

Detection During execution.

Cause A plane change has been programmed between the definition of the function “arc tangent to previous path (G08)” and the previous path.

Solution A plane cannot be changed between two paths

1011 ‘Jog movement out of limits.’

Detection During execution.

Cause After defining an incline plane, the tool positions at a point out of the work limits; the operator tries to move an axis with the JOG keys, the tool does not position within the area defined by the work limits.

Solution Jog the axis that allows to position the tool within the work limits.

1012 ‘G48 cannot be programmed while G43 is active’

Detection During execution.

Cause An attempt has been made to activate TCP (G48) while tool length compensation (G43) was active.

Solution To activate TCP transformation (G48), tool length compensation must be OFF because TCP already applies its own specific tool length compensation.

1013 ‘G43 cannot be programmed while G48 is active’

Detection During execution.

Cause An attempt has been made to activate tool length compensation (G43) while TCP (G48) was active.

Solution To activate tool length compensation (G43) cannot be activated while TCP transformation (G48) is ON because TCP already applies its own specific tool length compensation.

1014 ‘G49 cannot be programmed if it’’s already active’

No explanation required

1015 ‘The tool is not defined in the tool table’

Detection During execution.

Cause A tool change has been defined, but the new tool is not defined in the tool table.

Solution Define the new tool in the tool table.

1016 ‘The tool is not in the tool magazine’

Detection During execution.

Cause A tool change has been defined, but the new tool is not defined in position of the tool magazine table.

Solution Define the new tool in the tool magazine table.

1017 'There is no empty pocket in the tool magazine'

Detection During execution.

Cause A tool change has been defined and there is no empty pocket for the tool that is currently in the spindle.

Solution Perhaps, the new tool has been defined as special in the tool table and there are more than one pockets reserved to it in the magazine. In this case, that position is set for that tool and no other tool can occupy it. To avoid this error, an empty pocket (position) should be left in the tool magazine.

1018 'A tool change has been programmed without M06'

Detection During execution.

Cause An M06 has not been programmed after having looked for a tool and before searching again.

Solution This error occurs when having a machining center (general machine parameter TOFFM06(P28)=YES) that has a cyclic tool changer (general machine parameter CYCATC(P61)=YES). In this case, the tool change must be done with an m06 after searching for a tool and before searching for the next one.

1019 'There is no tool of the same family for replacement.'

Detection During execution.

Cause The real life of the requested tool exceeds its nominal life. The CNC has tried to replace it with another one of the same family, but it has not found any.

Solution Replace the tool or define another one of the same family.

1020 'Do not change the active or pending tool using high level language.'

Detection During execution.

Cause While programming in high level language and using the "TMZT" variable, an attempt has been made to assign the current or next tool to a magazine position.

Solution Use the "T" function to change the active tool or the next one. The "TMZT" variable cannot be used to move the active tool or the next one to the magazine.

1021 'No tool offset has been programmed in the canned cycle.'

Detection During execution.

Cause The "PROBE" canned cycle for tool calibration has been programmed, but no tool offset has been selected.

Solution To execute the "Tool calibration canned cycle (PROBE), a tool offset must be selected where the probing cycle information will be stored.

1022 'Tool radius programmed incorrectly'

No explanation required

1023 'G67. Tool radius too large.'

Detection During execution.

Cause In the "Irregular pocket canned cycle with islands (G66)", a tool has been selected whose radius is too large for the roughing operation "G67" (2D pocket). The tool cannot get in anywhere in the pocket.

Solution Select a tool of a smaller radius.

1024 'G68. Tool radius too large.'

Detection During execution.

Cause In the "Irregular pocket canned cycle with islands (G66)", a tool has been selected whose radius is too large for the finishing operation "G68" (2D pocket). Somewhere in the machining operation, the distance between the outside profile and the profile of an island is smaller than the tool diameter.

Solution Select a tool of a smaller radius.

1025 'A tool with no radius has been programmed'

Detection During execution.

Cause In the "Irregular pocket canned cycle with islands (G66), a (G67/G68) operation has been programmed with no radius.

Solution Correct the tool definition in the tool table or select another one for that operation.

1026 'A step has been programmed that is larger than the tool diameter'

Detection During execution.

Cause In the "Rectangular pocket canned cycle (G87)", in the "circular pocket canned cycle (G68) or in an operation of the "irregular pocket canned cycle with islands (G66)", the "C" parameter has been programmed with a value larger than that of the tool that will be used for that operation.

Solution Correct the syntax of the block. The machining step "C" must be smaller than or equal to the tool diameter.

1027 'A tool cannot be programmed with G48 active'

Detection During execution.

Cause A tool change has been programmed while the «TCP transformation (G48)» function is active.

Solution A tool change cannot take place while TCP transformation is active. To make a tool change, cancel TCP transformation first.

1028 'Do not switch axes over while G23, G48 or G49 is active'

Detection During execution.

Cause An attempt has been made to switch over to an axis or back (G28/G29) while function "G23", "G48" or "G49" was active.

Solution The axes cannot be swapped while function "G23", "G48" or "G49" is active.

1029 'Do not swap axes that are already swapped.'

Detection During execution.

Cause An attempt has been made to swap (G28) an axis that was already swapped with another one.

Solution An axis already swapped with another one cannot be swapped with a third one. It must be switched back first (G29 axis)

1030 'The "M" for the automatic gear change does not fit'

Detection During execution.

Cause Using automatic gear change, 7 "M" functions and the "S" function (involving a gear change) have been programmed. In this case, the CNC cannot include the "M" for automatic gear change in that block.

Solution Program an "M" function or the "S" function in a separate block.

1031 'No subroutine is allowed with automatic gear change.'

Detection During execution.

Cause On machines having an automatic gear change, when programming a spindle speed "S" that involves a gear change and the "M" function of the automatic gear change has a subroutine associated with it.

Solution When having an automatic gear change, the "M" functions corresponding to the gear change cannot have a subroutine associated with it.

1032 'Spindle gear (range) not defined in M19.'

Detection During execution.

Cause "M19" has been programmed, but none of the gear change functions "M41", "M42", "M43" or "M44" are active.

Solution On power-up, the CNC does not assume any range; Therefore, if the gear change function is not generated automatically (spindle parameter AUTOGEAR(P6)=NO), the auxiliary gear change functions ("M41", "M42", "M43" or "M44") must be programmed.

1033 'Wrong gear change.'

Detection During execution.

Cause The various causes might be:

1. When trying to make a gear change and the machine parameters for gears (MAXGEAR1, MAXGEAR2, MAXGEAR3, or MAXGEAR4) are set wrong. All the gears (ranges) have not been used and the unused ones have been set to a maximum speed of zero rpm.
2. When programming a gear change ("M41", "M42", "M43" or "M44") and the PLC has not responded with the relevant active gear signal (GEAR1, GEAR2, GEAR3 or GEAR4).

Solution The solution for each cause is:

1. When not using all four gears, the lower ones must be used starting with "MAXGEAR1" and the unused gears must be assigned the value of the highest one used.
2. Check the PLC program.

1034 '"S" has been programmed, but no gear is active.'

Detection During execution.

Cause An attempt has been made to start the spindle, but no gear is selected.

Solution On power-up, the CNC does not assume any range; Therefore, when programing a spindle speed and the gear change function is not generated automatically (spindle parameter AUTOGEAR(P6)=NO), the auxiliary gear change functions ("M41", "M42", "M43" or "M44") must be programmed.

1035 'Programmed "S" too high'

Detection During execution.

Cause An "S" has been programmed with a higher value than allowed by the last active gear.

Solution Program a lower spindle speed "S" .

1036 '"S" has not been programmed in G95 or in threading'

Detection During execution.

Cause "mm(inches)/revolution (G95)" or "electronic threading (G33)"has been programmed, but no spindle speed has been selected.

Solution An "S" must be programmed to work in mm/rev (G95) or for an electronic threading (G33).

1038 'The spindle has not been oriented'

Detection During execution.

Cause A threading cycle is to be executed without having oriented the active spindle (main or secondary) first.

1040 'Canned cycle does not exist'

Detection While executing in MDI mode

Cause When trying to execute a canned cycle (G8x) after interrupting a program during the execution of a canned cycle (G8x) and then changing the plane.

Solution Do not interrupt the program while executing a canned cycle.

1041 'Mandatory parameter missing in canned cycle'

Detection During execution.

Cause The various causes might be:

1. In the "Irregular canned cycle with islands" some parameter is missing.
   2D POCKETS
   - In the roughing operation "G67", either parameter "I" or "R" is missing.
   - There is no roughing operation and in the finishing operation "G68", either parameter "I" or "R" is missing.

   3D POCKETS
   - In the roughing operation "G67", either parameter "I" or "R" is missing.
   - There is no roughing operation and in the semifinishing operation "G67", either parameter "I" or "R" is missing.

- There is neither roughing nor semifinishing operation and in the finishing operation “G68”, either parameter “I” or “R” is missing.
- In the finishing operation “G68”, parameter “B” is missing.

2. In the “Digitizing canned cycle” some parameter is missing.

Solution Correct the definition of parameters.

Pocket with islands (finishing operation)

In the irregular pocket canned cycle with islands, parameters “I” and “R” must be programmed in the roughing operation. If there is no roughing operation, they must be defined in the finishing operation (2D) or in the semifinishing operation (3D). If there is no semifinishing operation (3D), they must be defined in the finishing operation. In the 3D pocket, parameter “B” must be defined in the finishing operation.

Digitizing cycles

Check the syntax of the block. The programming formats are:
(DIGIT 1,X,Y,Z,I,J,K,B,C,D,F)
(DIGIT 2,X,Y,Z,I,J,K,A,B,C,F)

1042 ‘Wrong parameter value in canned cycle’

Detection During execution.

Cause The various causes might be:

1. In the “Irregular pocket canned cycle with islands”, when a parameter has been defined with a wrong value in the finishing operation “G68”. Perhaps, a parameter that only takes positive values has been assigned a negative value (or zero).
2. In the “Irregular pocket canned cycle with islands”, when in the drilling operation (G69) parameter “B”, “C” or “H” has been defined with a zero value.
3. In the rectangular (G87) or circular (G88) pocket canned cycles, either parameter “C” or a pocket dimension has been defined with a zero value.
4. In the “Deep hole drilling canned cycle with variable peck (G69), parameter “C” has been defined with zero value.
5. In the digitizing canned cycle, a parameter has been assigned a wrong value. Perhaps, a parameter that only takes positive values has been assigned a negative value (or zero).

Solution Correct the definition of parameters.

Pocket with islands (finishing operation)

“Q” parameter	Only takes a value of 0, 1 or 2.
“B” parameter	Only takes values other than zero.
“J” parameter	It must be smaller than the radius of the tool used for that operation.

GRID pattern digitizing.

“B” parameter	Only takes positive values greater than zero.
“C” parameter	Only takes positive values other than zero.
“D” parameter	It only admits values of 0 or 1.

ARC pattern digitizing.

“J” and “C” parameter	Only takes positive values greater than zero.
“K”, “A” and “B” parameter	It only admits positive values.

1043 ‘Wrong depth profile in pocket with islands.’

Detection During execution.

Cause In the “Irregular pocket canned cycle with islands” (3D):

- The depth profiles of two sections of the same contour (simple or composite) cross each other.
- A contour cannot be finished with the programmed tool (spherical path with non-spherical tool).

Solution The depth profiles of two sections of the same profile cannot cross each other. On the other hand, the depth profile must be defined after the plane profile and the same starting point must be used in both profiles. Check that the tip of the selected tool is the best for the programmed depth profile.

1044 ‘Plane profile intersects itself in a pocket with islands’

Detection During execution.

Cause Within the set of profiles that define a pocket with islands, one of the profiles intersects itself.

Solution Check the definition of the profiles. The profile of a pocket with islands cannot intersect itself.

1045 'Error when programming a drilling operation in a pocket with islands.'

Detection During execution.

Cause In the "Irregular pocket canned cycle with islands (G66), a canned cycle has been programmed that is not for drilling.

Solution In the drilling operation, only canned cycle "G81", "G82", "G83" or "G69" may be programmed.

1046 'Wrong tool position before the canned cycle'

Detection During execution.

Cause When calling a canned cycle, the tool is positioned between the reference plane and the final depth coordinate of one of the operations.

Solution When calling a canned cycle, the tool must be positioned above the reference plane.

1047 'Open plane profile in pocket with islands'

Detection During execution.

Cause Within the set of profiles that define a pocket with islands, one of the profiles does not start and end at the same point.

Solution Check the definition of the profiles. The profiles that define the pockets with islands must be closed. The error may occurred because "G01" has not been programmed after the beginning, with "G00", of one of the profiles.

1048 'Part surface coordinate not programmed in pocket with islands'

Detection During execution.

Cause The part surface coordinate of the pocket has not been programmed at the first point of the geometry definition.

Solution The data for the surface coordinate must be defined in the first definition block of the pocket profile (in absolute coordinates).

1049 'Wrong reference plane coordinate in canned cycle'

Detection During execution.

Cause In an operation of the "Irregular pocket canned cycle with islands (G66), the coordinate of the reference plane is located between the part surface coordinate and the final depth coordinate of one of the operations.

Solution The reference plane must be located above the part surface. This error comes up sometimes because the part surface position has been programmed in incremental coordinates. (The pocket surface data must be programmed in absolute coordinates).

1050 'Wrong value to be assigned to a variable'

Detection During execution.

Cause Using parameters, the value assigned to a variable is too high.

Solution Check the program history to make sure that this parameter does not have that value when it reaches the block where this assignment is made.

1051 'Wrong access to PLC variable.'

Detection During execution.

Cause From the CNC, an attempt has been made to read a PLC variable that is not defined in the PLC program.

1052 'Access to a variable with wrong index'

Detection During editing

Cause While programming in high level language, an operation has been carried out either with a local parameter greater than 25 or with a global parameter greater 299.

Solution The parameters used by the CNC are:

Local: 0-25.
Global: 100-299.

Other parameters out of these ranges cannot be used in operations.

1053 'Local parameters not accessible'

Detection While executing in the user channel.

Cause An attempt has been made to execute a block with an operation that uses local parameters.

Solution The program that is executed in the user channel does not allow operations with local parameters (P0 to P25).

1054 'Limit of local parameters exceeded'

Detection During execution.

Cause While programming in high level language, more than 6 nesting levels have been used with the "PCALL" instruction. More than 6 calls have been made in the same loop using the "PCALL" instruction.

Solution Only up to 6 nesting levels are allowed for local parameters within the 15 nesting levels of the subroutines. Calling with a "PCALL" instruction generates a new nesting level for local parameters (and a new one for subroutines).

1055 'Nesting exceeded'

Detection During execution.

Cause While programming in high level language, more than 15 nesting levels have been used with the "CALL", "PCALL" or "MCALL" instruction. More than 15 calls have been made in the same loop using the "CALL", "PCALL" or "MCALL" instruction.

Solution Only 15 nesting levels allowed. Calling with the "CALL", "PCALL" and "MCALL" instructions generates a new nesting level.

1056 'RET not associated with subroutine.'

Detection During execution.

Cause The "RET" instruction has been edited, but the "SUB" instruction has not been edited before.

Solution To using the "RET" instruction (subroutine), the subroutine must begin with the "SUB (subroutine number)".

1057 'Undefined subroutine'

Detection During execution.

Cause A (CALL, PCALL…) has been made to a subroutine that was not defined in the CNC memory.

Solution Check that the name of the subroutine is correct and that the subroutine exists in the CNC memory (not necessarily in the same program where the call is).

1058 'Undefined probing canned cycle'

Detection During execution.

Cause Using the "PROBE" instruction, a probing cycle has been defined which is not available.

Solution The available "PROBE" canned cycles are 1 to 9.

1059 'Jump to an undefined label'

Detection During execution.

Cause While programming in high level language, the "GOTO N..." instruction has been programmed, but the programmed block number (N) does not exist.

Solution When programming the "GOTO N..." instruction, the block it refers to must be defined in the same program.

1060 'Undefined label'

Detection During execution.

Cause The various causes might be:

1. While programming in high level language, the instrucción “RPT N..., N...” instruction has been programmed, but a programmed block number (N) does not exist.
2. When programming “G66 … S...E...” in an “Irregular pocket canned cycle with islands (G66) and one of the data defining the beginning or the end of the profiles is missing.

Solution The solution for each cause is:

1. When programming the “RPT N..., N...” instruction, the blocks it refers to must be defined in the same program.
2. Check the program. Place the label for parameter “S” at the beginning of the profile definition and the label for parameter “E” at the end of the profile definition.

1061 ‘Label cannot be searched’

Detection While executing in MDI mode

Cause While programming in high level language, either an “RPT N..., N...” or “GOTO N...” instruction has been defined.

Solution While operating in MDI mode, “RPT” or “GOTO” type instructions cannot be programmed.

1062 ‘Subroutine in an unavailable program.’

Detection During execution.

Cause A call has been made to a subroutine that it is located in a program being used by the DNC.

Solution Wait for the DNC to finish using the program. If the subroutine is to be used often, it should be stored in a separate program.

1063 ‘The program cannot be opened.’

Detection During execution.

Cause While executing a program in infinite mode, an attempt has been made to execute another infinite program from the current one using the “EXEC” instruction.

Solution Only one infinite program may be executed at a time.

1064 ‘The program cannot be executed’

Detection During execution.

Cause An attempt has been made to execute a program from another with the “EXEC” instruction, but the program does not exist or is protected against execution.

Solution The program to be executed with the “EXEC” instruction must exist in the CNC memory and must be executable.

1065 ‘Beginning of compensation without straight path’

Detection During execution.

Cause The first movement in work plane after activating tool radius compensation (G41/G42) is not a linear movement.

Solution The first movement after activating radius compensation (G41/G42) must be linear.

1066 ‘End of compensation without straight path’

Detection During execution.

Cause The first movement in work plane after deactivating tool radius compensation (G40) is not a linear movement.

Solution The first movement after deactivating radius compensation (G40) must be linear.

1067 ‘Compensation radius too large.’

Detection During execution.

Cause While working with tool radius compensation (G41/G42), an inside radius has been programmed with a smaller radius than that of the tool.

Solution use a tool with a smaller radius. When working with tool radius compensation, the arc radius must larger than that of the tool. Otherwise, the tool cannot machine the programmed path.

1068 'Step on linear path'

Detection During execution.

Cause When operating with tool compensation (G41/G42), the profile has a straight section that cannot be machined because the tool diameter is too large.

Solution use a tool with a smaller radius.

1069 'Circular path defined incorrectly'

No explanation required

1070 'Step on circular path'

Detection During execution.

Cause When operating with tool compensation (G41/G42), the profile has a curved section that cannot be machined because the tool diameter is too large.

Solution use a tool with a smaller radius.

1071 'Plane change in tool radius compensation.'

Detection During execution.

Cause When operating with tool compensation (G41/G42), another work plane has been selected.

Solution To change the work plane, tool radius compensation must be off (G40).

1072 'Tool radius compensation not possible with positioning-only rotary axis.

Detection During execution.

Cause An attempt has been made to move a positioning-only axis with tool radius compensation (G41/G42).

Solution Tool radius compensation not allowed for positioning-only rotary axes. Use "G40" to cancel tool radius compensation.

1076 'Coordinate angle programmed wrong.'

Detection During execution.

Cause When programming in angle-coordinate format, an axis movement has been programmed with an angle perpendicular to that axis. (For example, the main plane is formed by the XY axes and the X axis movement is programmed at a 90º angle).

Solution Check and correct the definition of the movement in the program. If using parameters, check that the parameters have the correct values when arriving to the definition of the movement.

1077 'Either the arc radius is too small or a full circle has been programmed'

Detection During execution.

Cause The various causes might be:

1. When programming a full circle using the "G02/G03 X Y R" format.
2. When programming using the "G02/G03 X Y R" format, the distance to the arc's end point is greater than the diameter of the programmed circle.

Solution The solution for each cause is:

1. This format cannot be used to make full circles. Program the coordinates of the end point different from those of the starting point.
2. The diameter of the circle must be greater than the distance to the arc's end point.

1078 'Negative radius in polar coordinates'

Detection During execution.

Cause Working with incremental polar coordinates, a block is executed where the end position has a negative radius.

Solution Incremental polar coordinate programming allows negative radius, but the (absolute) end point of the radius must be positive,

G74 'There is no subroutine associated with G74'

Detection While executing a home search.

Cause The various causes might be:

1. When trying to search home on all the axes manually, but there is no associated subroutine indicating the home searching sequence (order).
2. "G74" has been programmed, but there is no associated subroutine indicating the home searching sequence (order).

Solution The solution for each cause is:

1. An associated subroutine is required to execute the "G74" function.
2. If "G74" is to be executed from a program, the home searching order must be defined.

1080 'Plane change in tool inspection'

Detection While executing the "tool inspection" option.

Cause the work plane has been chanted and the original one has not been restored before resuming the execution.

Solution The plane that was active before inspecting the tool must be restored before resuming the execution.

1081 'Block not allowed in tool inspection.'

Detection While executing the "tool inspection" option.

Cause An attempt has been made to execute the "RET" instruction.

Solution This instruction cannot be executed in the "tool inspection" option.

1082 'The probe signal has not been received.'

Detection During execution.

Cause The various causes might be:

1. When programming a "PROBE" canned cycle, the probe has moved the maximum safety distance of the cycle without the CNC receiving the probe signal.
2. When programming the "G75" function, it has reached the end point and the CNC has not received the signal from the probe. (Only when general machine parameter PROBERR(P119)=YES).

Solution The solution for each cause is:

1. Check that the probe is connected properly.
   The maximum probing distance (in PROBE cycles) depends on the safety distance "B". To increase this distance, increase the safety distance.
2. If PROBERR(P119)=NO, this error will not be issued when the end point is reached without having received the probe signal (only with "G75").

1083 'Range exceeded'

Detection During execution.

Cause The distance for the axes to travel is very long and the programmed feedrate is too low.

Solution Program a higher speed for that movement.

1084 'Arc programmed wrong'

Detection During execution.

Cause The various causes might be:

1. When the arc programmed using "G02/G03 X Y I J" cannot go through the defined end point.
2. When programming an arc using "G09 X Y I J" the three points are in line or two of them are the same.
3. When trying to do a rounding tangential entry on a path that is not in the active plane.
4. When programming a tangential exit and the next path is tangent (being on its straight extension) to the path preceding the tangential exit.
   If the error comes up in the block calling the "Irregular canned cycle with islands" is because one of the cases mentioned earlier occurs in the set of blocks defining the profiles of a pocket with islands.

Solution The solution for each cause is:

1. Correct the syntax of the block. The coordinates of the end point or of the radius are defined wrong.

2. The three points used to define an arc must be different and cannot be in line.
3. Maybe a plane has been defined with “G16”, “G17”, “G18” or “G19”. In this case, corner rounding, chamfers and tangential entries/exits can only be carried out on the main axes defining that plane. To do it in another plane, it must be defined beforehand.
4. The path after a tangential exit may be tangent, but it cannot be on the extension (in a straight line) of the previous path.

1085 ‘Helical path programmed wrong’

Detection During execution.

Cause When programming an arc using “G02/G03 X Y I J Z K”, the programmed arc is impossible. The desired height cannot be reached with the programmed helical pitch.

Solution Correct the syntax of the block. The height of the interpolation and the coordinates of the end point in the plane must be related taking the helical pitch into account.

1086 ‘The spindle cannot be homed.’

Cause Spindle machine parameter REFEED1(P34) = 0.

1087 ‘Circle with zero radius’

Detection During execution.

Cause The various causes might be:

1. When programming an arc using “G02/G03 X Y I J”, an arc has been programmed with a zero radius.
2. When operating with tool radius compensation, an inside arc has been programmed with the same radius as that of the tool.

Solution The solution for each cause is:

1. Arcs with zero radius are not allowed. Program a radius other than zero.
2. When working with tool radius compensation, the arc radius must larger than that of the tool. Otherwise, the tool cannot machine the programmed path (because to do so, the tool would have to make an arc of zero radius).

1088 ‘Range exceeded in zero offset.’

Detection During execution.

Cause A zero offset has been programmed and the value of the end position is too high.

Solution Check that the values assigned to the zero offsets (G54-G59) are correct. If the zero offsets have been assigned values from the program using parameters, check that the parameter values are correct. If an absolute (G54-G57) and an incremental (G58-G59) zero offset has been programmed, check that the sum of both does not exceed the machine limits.

1089 ‘Range exceeded in zone limit.’

Detection During execution.

Cause When programming zone limits “G20” or “G21” with parameters, the parameter value is greater than the maximum allowed for that function

Solution Check the program history to make sure that this parameter does not have that value when it reaches the block where the limits have been defined.

1090 ‘Point inside the forbidden zone 1.’

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 1 that is defined as “no entry” zone.

Solution In the program history, work zone 1 (defined with G20/G21) has been set as “no entry” zone ” (G22 K1 S1). To cancel this work zone, program “G22 K1 S0”

1091 ‘Point inside the forbidden zone 2.’

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 2 that is defined as “no entry” zone.

Solution In the program history, work zone 2 (defined with G20/G21) has been set as "no entry" zone " (G22 K1 S1). To cancel this work zone, program "G22 K2 S0"

1092 'Insufficient acceleration for the speed programmed in threading.'

Detection During execution.

Cause A thread has been programmed and there isn't enough room to accelerate and decelerate.

Solution Program a lower speed.

1093 'Only one Hirth axis can be moved at a time'

No explanation required

1094 'Probe calibrated wrong'

No explanation required

1095 'Probing axes out of alignment .'

Detection During the probe calibration process

Cause An axis has moved to touch a cube and one of the axis that did not move registers a deflection greater than allowed by machine parameter MINDEFLE(P66). This is because the probing axes are not parallel enough to the axes of the machine.

Solution Correct the parallelism between the probing axes and those of the machine.

1096 'Point inside the forbidden zone 3.'

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 3 that is defined as "no entry" zone.

Solution In the program history, work zone 3 (defined with G20/G21) has been set as "no entry" zone " (G22 K3 S1). To cancel this work zone, program "G22 K3 S0"

1097 'Point inside the forbidden zone 4.'

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 4 that is defined as "no entry" zone.

Solution In the program history, work zone 4 (defined with G20/G21) has been set as "no entry" zone " (G22 K4 S1). To cancel this work zone, program "G22 K4 S0"

1098 'Work zone limits defined wrong'

Detection During execution.

Cause The upper limits (G21) of the defined work zone are the same or smaller than the lower ones (G20) of the same work zone.

Solution Program the upper limits (G21) of the work zone greater than the lower ones (G20).

1099 'Do not program a slaved axis.'

Detection During execution.

Cause When operating in polar coordinates, a movement has been programmed that involves an axis that is slaved to another one.

Solution The movements in polar coordinates are made with the main axes of the work plane; therefore, the axes that define the plane cannot be slaved to each other or to a third one. To unslave the axes, program "G78".

1100 'Travel limits of spindle 1 exceeded'

Detection During execution.

Cause An attempt has been made to exceed the physical turning limits of the spindle. As a result, the PLC activates the spindle mark "LIMIT+S" or "LIMIT-S". ("LIMIT+S2" or "LIMIT-S2" when working with the second spindle).

1101 'Spindle 1 locked'

Detection During execution.

Cause The CNC tries to output the command to the drive when the spindle input SERVOSON is still low. The error may be due to an error in the PLC program where this signal is not properly treated or that the value of the spindle parameter DWELL(P17) is not high enough.

1102 'Following error of spindle 1 out of limit'

Detection During execution.

Cause When the spindle is working in closed loop (M19), its following error is greater than the values indicated by spindle parameter MAXFLWE1(P21) and MAXFLE2(P22) The possible causes for this error are:

Servo drive error
- Faulty drive.
- Enable signals missing.
- Power supply missing.
- Drive adjusted incorrectly.
- The velocity command signal is not received.

Motor error
- Faulty motor.
- Power cables.

Feedback failure
- Defective feedback.
- Defective feedback cable.

Mechanical failure
- Mechanical stiffness.
- Spindle mechanically locked.

CNC error
- Defective CNC.
- Parameters adjusted incorrectly.

1103 'Do not synchronize spindles without homing them first'

Detection During execution.

Cause An attempt has been made to synchronize the spindle without homing them first.

Solution Before activating the synchronization, both spindles must be homed using the "M19" function.

1104 ' Do not program G28 or G29 while spindle synchronization is active'

Detection During execution.

Cause An attempt has been made to swap spindles (G28/G29) while the spindles were synchronized.

Solution First, cancel spindle synchronization (G78S).

1105 'Do not change the ranges (gears) while the spindle are synchronized'

Detection During execution.

Cause While the spindles are synchronized, a gear changing "M" function (M41 to M44) has been executed or the programmed "S" involves a gear change (with automatic gear changer).

Solution First, cancel spindle synchronization (G78S).

1106 'Travel limits of spindle 2 exceeded'

Same as error 1000, but for the second spindle.

1107 'Spindle 2 locked'

Same as error 1001, but for the second spindle.

1108 'Following error of spindle 2 out of limit'

Same as error 1002, but for the second spindle.

1109 'Axis software limit overrun'

No explanation required

1110-1118 'Range of the * axis exceeded'

Detection During execution.

Cause A movement has been defined with parameters and the parameter value is greater than the maximum travel distance of the axis.

Solution Check the program history to make sure that this parameter does not have that value when it reaches the block where this movement is programmed.

1119-1127 'The * axis cannot be synchronized'

Detection During execution.

Cause The various causes might be:

1. When trying to synchronize two axes from the PLC and one axis is already slaved to another one using the "G77" function.
2. When programming or trying to move an axis that is slaved to another one.

1128-1136 'Maximum feedrate of the * axis exceeded'

Detection During execution.

Cause The resulting feedrate of one of the axes after applying an individual scaling factor exceeds the maximum value indicated by axis machine parameter MAXFEED (P42).

1137-1145 'Wrong feedrate parameter of the * axis'

Detection During execution.

Cause "G00" programmed with parameter G00FEED(P38)=0 or "G1 F00" with axis parameter MAXFEED(P42) = 0.

1146-1154 '* axis locked'

Detection During execution.

Cause The CNC tries to output the command to the drive when the spindle input SERVO(n)ON is still low. The error may be due to an error in the PLC program where this signal is not properly treated or that the value of the axis parameter DWELL(P17) is not high enough.

1155-1163 'Maximum axis limits of the * axis exceeded'

Detection During execution.

Cause A coordinate has been programmed that is out of the limits defined by axis parameters LIMIT+(P5) and LIMIT-(P6).

1164-1172 'Work zone 1 of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 1 that is defined as "no exit" zone.

Solution In the program history, work zone 1 (defined with G20/G21) has been set as "no exit" zone " (G22 K1 S2). To cancel this work zone, program "G22 K1 S0"

1173-1181 'Work zone 2 of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 2 that is defined as "no exit" zone.

Solution In the program history, work zone 2 (defined with G20/G21) has been set as "no exit" zone " (G22 K2 S2). To cancel this work zone, program "G22 K2 S0"

1182-1190 'Following error of * axis out of limit'

Detection During execution.

Cause The following error of the axis is greater than the values indicated by axis parameter MAXFLWE1(P21) or maxflwe2(P22). The possible causes for this error are:

Servo drive error

- Faulty drive.
- Enable signals missing.
- Power supply missing.
- Drive adjusted incorrectly.
- The velocity command signal is not received.

Motor error

- Faulty motor.
- Power cables.

Feedback failure

- Defective feedback.
- Defective feedback cable.

Mechanical failure

- Mechanical stiffness.
- Spindle mechanically locked.

CNC error

- Defective CNC.
- Parameters adjusted incorrectly.

1191-1199 'Difference of following errors of the slaved axis * tool large '

Cause The "n" axis is electronically coupled to another one or is a slaved Gantry axis and the difference between the following errors of the "n" axis and the one it is coupled to is greater than the value set by the machine parameter for the "n" axis MAXCOUPE(P45).

1200-1208 'Travel limits of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to exceed the physical travel limits. As a result, the PLC activates the axis mark "LIMIT+1" or "LIMIT-1".

1209-1217 '* axis servo error'

Cause The real feedrate of the axis, after the time indicated by axis parameter FBALTIME(P12), is below 50% or over 200% of the one programmed.

1218-1226 'Work zone 3 of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 3 that is defined as "no exit" zone.

Solution In the program history, work zone 3 (defined with G20/G21) has been set as "no exit" zone " (G22 K3 S2). To cancel this work zone, program "G22 K3 S0"

1227 'Wrong profile intersection in pocket with islands.'

Detection During execution.

Cause In the "Irregular pocket canned cycle with islands (G66)", there are two plane profiles that either have the starting point or a section in common.

Solution Define the profiles again. Two plane profiles cannot start at the same point or have sections in common.

1228-1236 'Work zone 4 of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 4 that is defined as "no exit" zone.

Solution In the program history, work zone 4 (defined with G20/G21) has been set as "no exit" zone " (G22 K4 S2). To cancel this work zone, program "G22 K4 S0"

1237 'Do not change the entry angle inside a thread'

Detection During execution.

Cause A thread joint has been defined and an entry angle "Q" has been programmed between two threads.

Solution When joining threads, only the first one may have an entry angle "Q".

1238 'Range of write-protected parameters. P297, P298'

Detection During execution.

Cause When trying to execute the function: "Definition of incline plane (G49)", parameters P297 and P298 are write-protected with machine parameters ROPARMIN(P51) and ROPARMAX(P52).

Solution While defining an incline plane, the CNC updates parameters P297 and P298. Therefore, these two parameters cannot be write-protected.

1239 'Point inside the forbidden zone 5.'

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 5 that is defined as "no entry" zone.

Solution In the program history, work zone 5 (defined with G20/G21) has been set as "no entry" zone " (G22 K5 S1). To cancel this work zone, program "G22 K5 S0"

1240-1248 'Work zone 5 of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 5 that is defined as "no exit" zone.

Solution In the program history, work zone 5 (defined with G20/G21) has been set as "no exit" zone " (G22 K5 S2). To cancel this work zone, program "G22 K5 S0"

1249 'Variable pitch thread programmed wrong'

Detection During execution.

Cause We are trying to make a variable-pitch thread with the following conditions:

- The "K" increment is positive and equal to or greater than 2L.
- The "K" increment is positive and with one of the calculated pitches, it exceeds the maximum feedrate (parameter MAXFEED) of one of the threading axis.
- The "K" increment is negative and one of the calculated pitches 0 or negative.

1250 'The K value is too large in G34'

Detection During execution.

Cause The ratio between the initial and final pitches of the variable-pitch thread (G34) to be executed

is greater than 32767.

1251 'Two variable-pitch threads cannot be joined in round corner'

Detection During motionless simulation, except when graphics are active

Cause To variable-pitch threads cannot be joined in round corner unless the second one is of the type: G34 … L0 K0.

G34 'G34 without a pitch is only allowed after a variable-pitch thread.

Detection During motionless simulation, except when graphics are active

Cause G34 L0 cannot be programmed after a movement, no G34, or in square corner.

1253 'Retrace function unavailable'

No explanation required

HARDWARE ERRORS

2000 'External emergency activated.'

Detection During execution.

Cause PLC input I1 is set to "0" (maybe the E-stop button) or the PLC mark M5000(/EMERGEN) is set to "0".

Solution Check at the PLC why the inputs are at "0". (Possible lack of power).

2001-2009 '* axis feedback error'

Detection During execution.

Cause The CNC does not receive feedback signal from the axes.

Solution Check that the connections are properly made.

NOTE: This error comes up on differential axes DIFFBACK(P9) =YES and sinusoidal axes SINMAGNI(P10) other than 0 when parameter FBACKAL(P11)=ON Setting parameter FBACKAL(P11)=OFF avoids this error, but this is only temporary solution.

2010 'Spindle feedback error'

Detection During execution.

Cause The CNC does not receive feedback signal from the spindle.

Solution Check that the connections are properly made.

NOTE: This error comes up on differential axes DIFFBACK(P14)=YES when parameter FBACKAL(P15)=ON. Setting parameter FBACKAL(P15)=OFF avoids this error, but this is only temporary solution.

2011 'Maximum temperature exceeded'

Detection Any time.

Cause The CNC's internal temperature has been exceeded. The causes may be:

- Electrical cabinet poorly ventilated.
- Axis board with some defective component.

Solution Turn the CNC and wait until it cools off. If the error persists, a component of the board may be defective. In that case, replace the board. Contact the Service Department.

2012 'There is no voltage at the axis board'

Detection During execution.

Cause 24V are missing at the output supply of the axis board. The fuse may be blown.

Solution Power the outputs of the axis board (24v). If the fuse is blown, replace it.

2013 'There is no voltage at the I/O 1 board.'
2014 'There is no voltage at the I/O 2 board.'
2015 'There is no voltage at the I/O 3 board.'

Detection During execution.

Cause 24V are missing at the output supply of the corresponding I/O board. The fuse may be blown.

Solution Power the outputs of the corresponding I/O board (24v). If the fuse is blown, replace it.

2016 'The PLC is not ready.'

Detection During execution.

Cause The PLC program is not running. These may be the probable causes:

- The PLC program is missing.
- WATCHDOG error.
- The program has been interrupted from monitoring.

Solution Start the PLC program. (Restart the PLC).

2017 'CNC RAM memory error.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the CNC's RAM memory.
Solution Replace the CPU board. Contact the Service Department.

2018 'CNC's EPROM memory error.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the CNC's EPROM memory.
Solution Replace the EPROM. Contact the Service Department.

2019 'PLC's RAM memory error.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the PLC's RAM memory.
Solution Replace the PLC board. Contact the Service Department.

2020 'PLC's EPROM memory error.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the PLC's EPROM memory.
Solution Replace the EPROM. Contact the Service Department.

2021 'CNC's user RAM memory error.' Press any key.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the CNC's user RAM memory.
Solution Contact the Service Department.

2022 'CNC's system RAM memory error.' Press any key.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the CNC's system RAM memory.
Solution Contact the Service Department.

2023 'PLC's RAM memory error.' Press any key.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the PLC's RAM memory.
Solution Contact the Service Department.

2024 'There is no voltage at the tracing board'

Detection During execution.
Cause 24V are missing at the output supply of the tracing board. The fuse may be blown.
Solution Power the outputs of the tracing board. If the fuse is blown, replace it.

2025 'Probe feedback error'

Detection During execution.
Cause The tracing probe is not connected or any of its cables is connected wrong.
Solution Check that the probe is properly connected.

2026 'Probe's maximum travel limit overrun.'

Detection During execution.
Cause The probe has exceeded the maximum deflection allowed by machine parameter.
Solution Decrease the feedrate and check that the probe has not been damaged.

2027 'SERCOS chip RAM memory error.' Press any key.'

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the SERCOS chip RAM memory.
Solution Replace the SERCOS board. Contact the Service Department.

2028 'SERCOS chip version error.' Press any key.'

Detection	During CNC start-up.
Cause	The SERCOS chip version is old.
Solution	Replace the SERCOS chip. Contact the Service Department.

PLC ERRORS

3001 ' (PLC_ERR without description) '

Detection During execution.

Cause Marks ERR1 to ERR64 have been set to "1".

Solution Check at the PLC why these marks are set to "1" and act accordingly.

3002 'WATCHDOG in the main module (PRG).'

Detection Any time.

Cause The various causes might be:

1. The execution of the PLC's main program has exceeded the time set in PLC parameter WAGPRG(P0).
2. The program is in an endless loop.

Solution Increase the time of PLC parameter WAGPRG(P0) or increase the PLC speed.

- Insert CPU TURBO.
- Change PLC parameter CPUTIME(P26) or general parameter LOOPTIME(P72).

3003 'WATCHDOG in the periodic module (PE).'

Detection Any time.

Cause The various causes might be:

1. The execution of the PLC's periodic program has exceeded the time set in PLC parameter WAGPER(P1).
2. The program is in an endless loop.

Solution Increase the time of PLC parameter WAGPER(P1) or increase the PLC speed.

- Insert CPU TURBO.
- Change PLC parameter CPUTIME(P26) or general parameter LOOPTIME(P72).

3004 'Division by zero at the PLC'

Detection Any time.

Cause In the PLC program, there is a line whose execution implies a division by zero.

Solution When working with registers, that register may have already acquired a zero value. Check that the register does not reach the operation with that value.

3005 'PLC error -> '

Detection Any time.

Cause An error has been detected on the PLC board.

Solution Replace the PLC board. Contact the Service Department.

SERVO ERRORS

4000 'Sercos ring error'

Detection During execution.

Cause SERCOS communication has been interrupted. It may be caused by an interruption in the connection ring (optical fiber disconnected or broken) or by a wrong configuration.

1. The identifying wheel does not match the sercosid.
2. Parameter P120 (SERSPD) does not match the transmission speed.
3. The drive version is incompatible with the CNC.
4. There is an error on the SERCOS board.
5. Different transmission speed (baudrate) at the drive and at the CNC.

A drive has been turned off and back on due to a power supply failure. When starting up again, it displays the error *4027 'The drive has started up again'*

An attempt has been made to read or write an non-existent variable or too many variables in a drive through the fast channel.

Solution To check that the connection ring is not interrupted, check that the light goes through the optical fiber. If it is due to a wrong configuration, contact the Service Department.

If the error is due to the fast channel

- Check that all the variables to be read or written through the fast channel actually exist
- Save the SERCOS LOG into a file and see which axis causes the error.
- Set PLC machine parameters "SRD700 and SWR800" of that drive to "0".
- Reset the CNC and verify that no errors come up.
- Set the parameters one by one to the desired value until the failure occurs.
- When locating the parameter, look that variable up in the drive manual to verify that it exists in that version and it may be accessed. If so, the error may come up because it tries read or write too many variables in that drive.

4001 'Undefined class 1 error'

Detection During execution.

Cause The drive has detected an error, but it cannot identify it.

Solution Contact the Service Department.

4002 'Overload (201...203)'
4003 'Overtemperature at the drive (107)'
4004 'Overtemperature at the motor (108)'
4005 'Overtemperature at the heatsink (106)'
4006 'Voltage control error (100...105)'
4007 'Feedback error (600...606)'
4008 'Error at the power bus (213...215)'
4009 'Overcurrent (212)'
4010 'Overvoltage at the power bus (304/306)'
4011 'Undervoltage at the power bus (307)'

Detection During execution.

Cause An error occurred at the drive. The number in brackets indicates the standard error number of the drive. Refer to the drive manual for further information.

Solution These types of error come with the messages 4019, 4021, 4022 or 4023 that indicate in which axis or spindle drive the error came up. Refer to the drive manual to check the error (number in brackets) and act accordingly.

4012 'Drive error'
4013 'Position deviation too high'
4014 'Communications error'
4015 'Travel limit overrun'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4016 'Undefined class 1 error'

Detection During execution.

Cause The drive has detected an error, but it cannot identify it.

Solution Contact the Service Department.

4017 'Drive error'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4018 'Error accessing a SERCOS variable'

Detection During execution.

Cause An attempt has been made to read (or write) a SERCOS variable from the CNC and:

1. That variable does not exist.
2. The maximum/minimum values have been exceeded.
3. The SERCOS variable has a variable length.
4. An attempt has been made to write a read-only variable.

Solution Check that the variable to be associated with an action is of the right type.

4019 'Drive error : Axis '

Detection During execution.

Cause These messages come with errors 4002 – 4011. When one of the mentioned errors occurs, they indicate in which axis it came up.

4020 'SERCOSID parameter value error'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4021 'Spindle drive error :
4022 'Spindle-2 drive error :
4023 'Auxiliary spindle drive error'

Detection During execution.

Cause These messages come with errors 4002 – 4011. When one of the mentioned errors occurs, they indicate in which spindle it came up.

4024 'SERCOS error when searching home'

Detection During execution.

Cause The home search command of SERCOS has been executed incorrectly.

4025 'SERCOS loop time exceeded: Increase P72 (looptime)'

Detection During execution.

Cause The time it takes to calculate the feedrate of the axis is greater than the cycle time established for transmission to the drive.

Solution Increase the value of general machine parameter LOOPTIME (P72). If the error persists, contact the Service Department.

4026 'Error in SERCOS chip RAM memory'

Detection During execution.

Solution Contact the service department to replace the SERCOS board.

4027 'The drive has started up again'

Detection During execution.

Cause A drive has been turned off and back on due to a power supply failure.

4028 'The light does not reach the CNC through the optic fiber'

Detection On power-up.

Cause The signal sent by the CNC through the optical fiber does not return to the CNC.

Solution Check the condition and installation of the fiber optic cables. Check that the light going "OUT" of the CNC is going through the drives and comes "IN"to the CNC.

If the cables are OK, remove the drives from the ring until the error no longer comes up.

4029 'Communication with the drive cannot be established. No response'

Detection On power-up.

Cause A drive is not responding to the signal sent by the CNC due to one of these causes:

- The drive does not recognize the sercos board.
- The drive is locked up
- The switch number has not been properly read.
- The SERCOS transmission speed has been set differently at the drives and at the CNC. General parameter SERSPD at the CNC and QP11 at the drives.

Solution Save the SERCOS LOG into a file.

See the value of axis machine parameter SERCOSID of the axis causing the error.

Check that the ring contains a drive with the switch in that position.

Reset the drive because the drive only reads the switch on power-up.

Check that the CNC and the drives have the same transmission speed. General parameter SERSPD at the CNC and QP11 at the drives.

Check that the drive does not issue sercos board. To do that look at the display of the drive. If it shows hardware errors, change the drive's sercos board.

If there are no errors at that drive, set the switch of the drive to "1", reset it, set the CNC with a single Sercos axis and connect to the CNC. If it still issues the error, change the drive.

4030 'SERCON register writing error'

Detection During execution.

Solution Contact the Service Department.

4050 'ERROR 1: Internal (Fatal error): Internal RAM test failed'
4051 'ERROR 2 : Internal (Fatal error): internal program malfunction'
4052 'ERROR 3 : Power bus drop: No torque. '
4053 'ERROR 4 : The emergency stop cannot stop the motor in the established time frame'
4054 'ERROR 5 : Program code checksum error'
4055 'ERROR 6 : Sercos board error'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4056 'ERROR 100 : Internal +5 V out of range'
4057 'ERROR 101 : Internal -5 V out of range'
4058 'ERROR 102 : Internal +8 V out of range'
4059 'ERROR 103 : Internal -8 V out of range'
4060 'ERROR 104 : Internal +18 V out of range'
4061 'ERROR 105 : Internal -18 V out of range'
4062 'ERROR 106 : Heat-sink overheating'
4063 'ERROR 107 : VeCon card overheating'
4064 'ERROR 108 : Motor overheating'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4065 'ERROR 200 : Overspeed'
4066 'ERROR 201 : Motor overload'
4067 'ERROR 202 : Driver overload'
4068 'ERROR 211 : Internal (Fatal error): DSP program execution error'
4069 'ERROR 212 : Overcurrent'
4070 'ERROR 213 : Undervoltage at the IGBT power driver'
4071 'ERROR 214: Shortcircuit'
4072 'ERROR 215 : Overvoltage at the power bus(Hard) '

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4073 'ERROR 300 : Power supply module overheating'
4074 'ERROR 301 : Power supply module ballast circuit overheating'
4075 'ERROR 302 : Shortcircuit in the power supply module ballast'
4076 'ERROR 303 : Ballast circuit supply voltage out of range'
4077 'ERROR 304 : Overvoltage at the power bus detected by the power supply module'
4078 'ERROR 305 : Protocol error in the interface between the power supply module and the driver'
4079 'ERROR 306 : Overvoltage at the power bus (Soft, trigger prior to hardware)'
4080 'ERROR 307 : Undervoltage of the power bus'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4081 'ERROR 400 : No SERCOS board is detected'
4082 'ERROR 401 : Internal SERCOS error'
4083 'ERROR 403 : MST failure'
4084 'ERROR 404 : MDT failure'
4085 'ERROR 405 : Wrong phase (> 4)'
4086 'ERROR 406 : Wrong phase increase'
4087 'ERROR 407 : Wrong phase decrease'
4088 'ERROR 408 : Phase change without «ready» aknowledgement'
4089 'ERROR 409 : Change to an unitialized phase'
4090 'ERROR 410 : Two drivers have the same ring address'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4091 'ERROR 500 : Inconsistent parameters'
4092 'ERROR 501 : Parameter checksum error'
4093 'ERROR 502 : Wrong parameter value'
4094 'ERROR 503 : The table for default parameter values for each motor is wrong. '
4095 'ERROR 504 : Wrong parameter in SERCOS phase 2'
4096 'ERROR 505 : Different RAM and Flash parameters'
4097 'ERROR 600 : Communication error with the second feedback'
4098 'ERROR 601 : Communication error with the rotor encoder'
4099 'ERROR 602 : motor feedback B signal saturation'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4100 'ERROR 603 : motor feedback A signal saturation '
4101 'ERROR 604 : Saturation of A and/or B signal values'
4102 'ERROR 605 : Week A and/or B signal values'
4103 'ERROR 606 : Too much dispersion of the rotor sensor signals'
4104 'ERROR 700 : RS232 board error'
4105 'ERROR 701 : Internal : Wrong VeCon board identification'
4106 'ERROR 702 : Expansion board identification error'
4107 'ERROR 703 : I/O board identification error'
4108 'ERROR 704 : Analog board identification error'
4109 'ERROR 705 : Power board identification error'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4110 'ERROR 706 : X3 encoder simulation board identification error'
4111 'ERROR 707 : X4 motor feedback board identification error'
4112 'ERROR 801 : Encoder not detected'
4113 'ERROR 802 : Communication error with the encoder'
4114 'ERROR 803 : Unitialized encoder'
4115 'ERROR 804 : Defective encoder'
4116 'ERROR 805 : No encoder has been detected on the motor'
4117 'ERROR 7 : SERCON clock error'
4118 'ERROR 8 : SERCON data error'
4119 'ERROR 203 : Torque overload error'
4120 'ERROR 411 : telegram reception error'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

CAN ERRORS

‘5003 Application error

Cause Internal CANopen error

Solution Contact the Service Department.

5004 CAN bus error

Cause The error type is indicated with a code:

2	Transmission queue full, the message cannot be sent.
128	Bus Off, the bus has been deactivated due to too many errors.
129	CAN warning, there are more than 96 errors at the bus, step prior to the buss off error.
130	Loss of message received or too many messages received. Usually due to wrong speed for the cable length.
131	The CNC has switched to an inoperative state in the bus (internal).

Solution The solution for each cause is:

2	Check the connection between the CNC and first node.
128	Check cables and connections.
129	Check cables and connections.
130	Check machine parameter IOCANSPE (P88).
131	Check cables and connections.

5005 Presence control error detected by the CNC

Cause The CNC detects that the node has reset itself or is connected wrong.

Solution Check cables and connections.

5006 Error because the node has been reset

Cause The node has been reset due to a power supply failure

Solution Check the power supply voltage at the indicated node, the ground connection and the load of the outputs.

5007 Error message corrected

Cause It is activated when an error situation disappears and shows whether there are any more left. If there is none, it resets the node connections.

5022 Internal software error

Cause Internal node software error.

Solution Access the Status screen \ Can \ Versions and reload the software.

5027 Communications error

Cause Node communication error

Solution Contact the Service Department.

5028 Lost messages

Cause The node has lost messages.

Solution Check cables and connections.

5029 Presence control error detected by the node

Cause The presence control done by the CNC node has failed.

Solution Check cables and connections.

5030 Protocol error

Cause The node has received a message that it cannot interpret

Solution Contact the Service Department.

5031 PDO not processed due to its length.

Cause The node has received a process message whose length does not match

Solution Contact the Service Department.

5032 PDO too long

Cause The node has received a process message longer than the one programmed

Solution Contact the Service Department.

5036 Output over-current

Cause Excessive consumption (over current) has been detected in the outputs of the indicated node. As a precaution, the system deactivates all the outputs of this module setting them to zero volts.

Solution Check the consumption and possible short-circuits at the outputs of the module.

5037 Power supply voltage error

Cause A power supply failure has been detected at the indicated node, it has no power or it is under +24V.

Solution Check the supply voltage at the outputs and the consumption of the module's supply voltage.

TABLE DATA ERRORS

'CHECKSUM ERROR: GENERAL PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR: SPINDLE PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:2nd SPINDLE PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:AUX. SPINDLE PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:SERIAL LINE 1 PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:SERIAL LINE 2 PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:HD/ETHERNET PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:USER PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:OEM PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:PLC PARAMETERS Load CARD A? (ENTER/ESC)'

'CHECKSUM ERROR:ZERO OFFSET TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:PASSWORD TABLE Load CARD A? (ENTER/ESC)'

'CHECKSUM ERROR:AXIS * PARAMETERS Load CARD A? (ENTER/ESC)'

'CHECKSUM ERROR:TOOL TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:TOOL OFFSET TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:MAGAZINE TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:M FUNCTION TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:LEADSCREW * TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:CROSS COMP. TABLE * Load CARD A? (ENTER/ESC)'

Detection During CNC start-up.

Cause Certain table data has been lost (possible RAM error) and there is a table saved in CARD A.

Solution Pressing [ENTER] copies the table saved in CARD A to RAM memory. If the error persists, contact the service department.

'ERROR:GENERAL PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SPINDLE PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SPINDLE-2 PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:AUX. SPINDLE PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SERIAL-LINE-1 PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SERIAL-LINE-2 PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'CHECKSUM ERROR:HD/ETHERNET PARAMETERS Initialize? (ENTER/ESC)'
'CHECKSUM ERROR:USER PARAMETERS Initialize? (ENTER/ESC)'
'CHECKSUM ERROR:OEM PARAMETERS Initialize? (ENTER/ESC)'
'ERROR:PLC PARAMETER CHECKSUM Reset? (ENTER/ESC)'

'ERROR:ZERO OFFSET TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:CODE TABLE CHECKSUM Reset? (ENTER/ESC)'

'ERROR:AXIS PARAMETER CHECKSUM Reset? (ENTER/ESC)'

'ERROR:TOOL TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:TOOL OFFSET TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:TOOL MAGAZINE TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:M FUNCTION TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:AXIS LEADSCREW TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:CROSS COMP. TABLE CHECKSUM Reset? (ENTER/ESC)'

Detection During CNC start-up.

Cause Certain table data has been lost (possible RAM error) and there is no table saved in CARD A.

Solution Pressing [ENTER] loads the tables with CNC's default values. If the error persists, contact the Service Department.

'Wrong * leadscrew table. Press key'

Detection During CNC start-up.

Cause There is some erroneous data in the parameters of the leadscrew compensation table.

Solution The definition of the points of the table must meet the following requirements:

- The points of the table must be ordered according to their position on the axis, starting from the most negative or less positive point to be compensated.
- The machine reference point must have no error (zero).
- The error difference between consecutive points cannot be greater than the distance between them.

'Wrong * cross compensation table. Press key'

Detection During CNC start-up.

Cause There is some erroneous data in the parameters of the cross compensation table.

Solution The definition of the points of the table must meet the following requirements:

- The points of the table must be ordered according to their position on the axis, starting from the most negative or less positive point to be compensated.
- The machine reference point must have no error (zero).

'Incorrect cross compensation table parameters'

Detection During CNC start-up.

Cause The parameters indicating the axes involved in the cross compensation are defined wrong.

Solution Maybe a nonexistent axis has been defined or the affected axis (to be compensated) and the one affecting it are the same.

'Wrong axis or spindle parameters sercosid'

Detection During CNC start-up.

Cause The servosid parameters have not been entered correctly.

Solution The rules of sercosid parameters are:

- They must begin with number 1.
- They must be consecutive.
- They cannot be repeated.

ERRORS OF THE MC WORK MODE

9001 'CENTER PUNCHING: F=0'

Detection During execution.
Cause A feedrate "F" has been defined with a wrong value.
Solution Program a positive feedrate "F" other than zero.

9002 'CENTER PUNCHING: S=0'

Detection During execution.
Cause A spindle speed "S" has been defined with a wrong value.
Solution Program a positive spindle speed "S" other than zero.

9003 'CENTER PUNCHING: T=0'

Detection During execution.
Cause The tool number "T" has not been defined.
Solution The tool number "T" must be other than zero.

9004 'CENTER PUNCHING: P=0'

Detection During execution.
Cause The center punching depth "P" has not been defined.
Solution The center punching depth "P" must be other than zero.

9005 'CENTER PUNCHING: ø=0'

Detection During execution.
Cause The point diameter «ø» has not been defined.
Solution The point diameter «ø» must be positive and other than zero.

9006 'CENTER PUNCHING: α=0'

Detection During execution.
Cause The angle of the tip of the drill bit has not been «α».
Solution The angle of the tip of the drill bit «α» must be positive and other than zero.

9007 'DRILLING 1: F=0'

Detection During execution.
Cause A feedrate "F" has been defined with a wrong value.
Solution Program a positive feedrate "F" other than zero.

9008 'DRILLING 1: S=0'

Detection During execution.
Cause A spindle speed "S" has been defined with a wrong value.
Solution Program a positive spindle speed "S" other than zero.

9009 'DRILLING 1: T=0'

Detection During execution.
Cause The tool number "T" has not been defined.
Solution The tool number "T" must be other than zero.

9010 'DRILLING 1: P=0'

Detection During execution.
Cause The center drilling depth "P" has not been defined.
Solution The drilling depth "P" must be other than zero.

9011 'DRILLING 2: F=0'

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9012 ‘DRILLING 2: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9013 ‘DRILLING 2: T=0’

Detection During execution.

Cause The tool number “T” has not been defined.

Solution The tool number “T” must be other than zero.

9014 ‘DRILLING 2: P=0’

Detection During execution.

Cause The center drilling depth “P” has not been defined.

Solution The drilling depth “P” must be other than zero.

9015 ‘DRILLING 2: B=0’

Detection During execution.

Cause The withdrawal distance «B» after each penetration has not been defined.

Solution The distance «B» it withdraws after each penetration must be other than zero.

9016 ‘TAPPING: F=0’

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9017 ‘TAPPING: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9018 ‘TAPPING: T=0’

Detection During execution.

Cause The tool number “T” has not been defined.

Solution The tool number “T” must be other than zero.

9019 ‘TAPPING: P=0’

Detection During execution.

Cause The tapping depth “P” has not been defined.

Solution The tapping depth “P” must be other than zero.

9020 ‘REAMING: F=0’

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9021 ‘REAMING: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9022 ‘REAMING: T=0’

Detection During execution.

Cause The tool number “T” has not been defined.

Solution The tool number “T” must be other than zero.

9023 ‘REAMING: P=0’

Detection During execution.

Cause The reaming depth “P” has not been defined.

Solution The reaming depth “P” must be other than zero.

9024 ‘BOARING: F=0’

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9025 ‘BOARING: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9026 ‘BOARING: T=0’

Detection During execution.

Cause The tool number “T” has not been defined.

Solution The tool number “T” must be other than zero.

9027 ‘BOARING: P=0’

Detection During execution.

Cause The boaring depth “P” has not been defined.

Solution The boring depth “P” must be other than zero.

9028 ‘DRILLING 3: F=0’

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9029 ‘DRILLING 3: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9030 ‘DRILLING 3: T=0’

Detection During execution.

Cause The tool number “T” has not been defined.

Solution The tool number “T” must be other than zero.

9031 ‘DRILLING 3: P=0’

Detection During execution.

Cause The center drilling depth “P” has not been defined.

Solution The drilling depth “P” must be other than zero.

9032 ‘BOARING 2: F=0’

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9033 ‘BOARING 2: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9034 'BOARING 2: T=0'

Detection During execution.
Cause The tool number "T" has not been defined.
Solution The tool number "T" must be other than zero.

9035 'BOARING 2: P=0'

Detection During execution.
Cause The boaring depth "P" has not been defined.
Solution The boring depth "P" must be other than zero.

9036 'RECTANGULAR POCKET 1: F=0'

Detection During execution.
Cause A feedrate "F" has been defined with a wrong value.
Solution Program a positive feedrate "F" other than zero.

9037 'RECTANGULAR POCKET 1: S=0'

Detection During execution.
Cause A spindle speed "S" has been defined with a wrong value.
Solution Program a positive spindle speed "S" other than zero.

9038 'RECTANGULAR POCKET 1: T=0'

Detection During execution.
Cause The tool number "T" has not been defined.
Solution The tool number "T" must be other than zero.

9039 'RECTANGULAR POCKET 1: P=0'

Detection During execution.
Cause The pocket depth "P" has not been defined.
Solution The pocket depth "P" must be other than zero.

9040 'RECTANGULAR POCKET 1: Tool diameter smaller than Δ'

Detection During execution.
Cause The programmed milling step «Δ» is larger than the tool diameter.
Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9041 'RECTANGULAR POCKET 1: Tool diameter larger than the pocket'

Detection During execution.
Cause The tool diameter is larger than one of the pocket's "H" or "L" dimensions.
Solution Choose a tool of smaller diameter to mill the pocket.

9042 'RECTANGULAR POCKET 1: FINISHING tool diameter smaller than δ'

Detection During execution.
Cause The programmed finishing stock «δ» is larger than the tool diameter.
Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9043 'RECTANGULAR POCKET 2: F=0'

Detection During execution.
Cause A feedrate "F" has been defined with a wrong value.
Solution Program a positive feedrate "F" other than zero.

9044 'RECTANGULAR POCKET 2: S=0'

Detection During execution.
Cause A spindle speed "S" has been defined with a wrong value.
Solution Program a positive spindle speed "S" other than zero.

9045 'RECTANGULAR POCKET 2: P=0'

Detection During execution.

Cause The pocket depth "P" has not been defined.

Solution The pocket depth "P" must be other than zero.

9046 'RECTANGULAR POCKET 2: Wrong penetration angle'

Detection During execution.

Cause A penetration angle smaller than 0º and greater than 90º has been programmed

Solution Program a penetration angle «β» and «θ» between 0º and 90º.

9047 'RECTANGULAR POCKET 2: Tool diameter smaller than Δ'

Detection During execution.

Cause The programmed milling step «Δ» is larger than the tool diameter.

Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9048 'RECTANGULAR POCKET 2: Tool diameter larger than the pocket'

Detection During execution.

Cause The tool diameter is larger than one of the pocket's "H" or "L" dimensions.

Solution Choose a tool of smaller diameter to mill the pocket.

9049 'RECTANGULAR POCKET 2: FINISHING tool diameter smaller than δ'

Detection During execution.

Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9050 'CIRCULAR POCKET 1: F=0'

Detection During execution.

Cause A feedrate "F" has been defined with a wrong value.

Solution Program a positive feedrate "F" other than zero.

9051 'CIRCULAR POCKET 1: S=0'

Detection During execution.

Cause A spindle speed "S" has been defined with a wrong value.

Solution Program a positive spindle speed "S" other than zero.

9052 'CIRCULAR POCKET 1: P=0'

Detection During execution.

Cause The pocket depth "P" has not been defined.

Solution The pocket depth "P" must be other than zero.

9053 'CIRCULAR POCKET 1: Wrong penetration angle'

Detection During execution.

Cause A penetration angle smaller than 0º and greater than 90º has been programmed

Solution Program a penetration angle «β» and «θ» between 0º and 90º.

9054 'CIRCULAR POCKET 1: Tool diameter smaller than Δ'

Detection During execution.

Cause The programmed milling step «Δ» is larger than the tool diameter.

Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9055 'CIRCULAR POCKET 1: Tool diameter larger than the pocket'

Detection During execution.

Cause The tool radius is greater than the pocket radius "R".

Solution Choose a tool of smaller diameter to mill the pocket.

9056 'CIRCULAR POCKET 1: FINISHING tool diameter smaller than δ'

Detection During execution.

Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9057 'CIRCULAR POCKET 2: F=0'

Detection During execution.

Cause A feedrate "F" has been defined with a wrong value.

Solution Program a positive feedrate "F" other than zero.

9058 'CIRCULAR POCKET 2: S=0'

Detection During execution.

Cause A spindle speed "S" has been defined with a wrong value.

Solution Program a positive spindle speed "S" other than zero.

9059 'CIRCULAR POCKET 2: P=0'

Detection During execution.

Cause The pocket depth "P" has not been defined.

Solution The pocket depth "P" must be other than zero.

9060 'CIRCULAR POCKET 2: Wrong penetration angle'

Detection During execution.

Cause A penetration angle smaller than 0º and greater than 90º has been programmed

Solution Program a penetration angle «β» and «θ» between 0º and 90º.

9061 'CIRCULAR POCKET 2: Tool radius greater than Ri'

Detection During execution.

Cause A tool has been selected with a radius greater than Ri (inside radius).

Solution Select a tool with a smaller diameter.

9062 'CIRCULAR POCKET 2: Tool diameter smaller than Δ'

Detection During execution.

Cause The programmed milling step «Δ» is larger than the tool diameter.

Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9063 'CIRCULAR POCKET 2: Tool diameter larger than the pocket'

Detection During execution.

Cause The tool radius is greater than the pocket radius "R".

Solution Choose a tool of smaller diameter to mill the pocket.

9064 'CIRCULAR POCKET 2: FINISHING tool diameter smaller than δ'

Detection During execution.

Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9065 'CIRCULAR POCKET 2: Ri > Re'

Detection During execution.

Cause An inside radius (Ri) has been programmed greater than the outside (Re).

9066 'RECTANGULAR BOSS: F=0'

Detection During execution.

Cause A feedrate "F" has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9067 ‘RECTANGULAR BOSS: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9068 ‘RECTANGULAR BOSS: P=0’

Detection During execution.

Cause The boss depth “P” has not been defined.

Solution The boss height “P” must be other than zero.

9069 ‘RECTANGULAR BOSS: Tool diameter smaller than Δ’

Detection During execution.

Cause The programmed milling step «Δ» is larger than the tool diameter.

Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9070 ‘RECTANGULAR BOSS: FINISHING tool diameter smaller than δ’

Detection During execution.

Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9071 ‘CIRCULAR BOSS: F=0’

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9072 ‘CIRCULAR BOSS: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9073 ‘CIRCULAR BOSS: P=0’

Detection During execution.

Cause The boss depth “P” has not been defined.

Solution The boss height “P” must be other than zero.

9074 ‘CIRCULAR BOSS: Tool diameter smaller than Δ’

Detection During execution.

Cause The programmed milling step «Δ» is larger than the tool diameter.

Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9075 ‘CIRCULAR BOSS: FINISHING tool diameter smaller than δ’

Detection During execution.

Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9076 ‘PROFILE POCKET: F=0’

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9077 'PROFILE POCKET: S=0'

Detection During execution.
Cause A spindle speed "S" has been defined with a wrong value.
Solution Program a positive spindle speed "S" other than zero.

9078 'PROFILE POCKET: P=0'

Detection During execution.
Cause The pocket depth "P" has not been defined.
Solution The pocket depth "P" must be other than zero.

9079 'PROFILE POCKET: Wrong penetration angle'

Detection During execution.
Cause A penetration angle smaller than 0º and greater than 90º has been programmed
Solution Program a penetration angle «β» and «θ» between 0º and 90º.

9080 'PROFILE POCKET: Tool diameter smaller than Δ'

Detection During execution.
Cause The programmed milling step «Δ» is larger than the tool diameter.
Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9081 'PROFILE POCKET: FINISHING tool diameter smaller than δ'

Detection During execution.
Cause The programmed finishing stock «δ» is larger than the tool diameter.
Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9082 '3D PROFILE POCKET: F=0'

Detection During execution.
Cause A feedrate "F" has been defined with a wrong value.
Solution Program a positive feedrate "F" other than zero.

9083 '3D PROFILE POCKET: S=0'

Detection During execution.
Cause A spindle speed "S" has been defined with a wrong value.
Solution Program a positive spindle speed "S" other than zero.

9084 '3D PROFILE POCKET: P=0'

Detection During execution.
Cause The pocket depth "P" has not been defined.
Solution The pocket depth "P" must be other than zero.

9085 '3D PROFILE POCKET: Wrong penetration angle'

Detection During execution.
Cause A penetration angle smaller than 0º and greater than 90º has been programmed
Solution Program a penetration angle «β» and «θ» between 0º and 90º.

9086 '3D PROFILE POCKET: Tool diameter smaller than Δ'

Detection During execution.
Cause The programmed milling step «Δ» is larger than the tool diameter.
Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9087 '3D PROFILE POCKET: FINISHING tool diameter smaller than δ'

Detection During execution.
Cause The programmed finishing stock «δ» is larger than the tool diameter.

Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9088 ‘3D PROFILE POCKET: FINISHING tool radius smaller than R'

Detection During execution.

Cause The radius of the finishing tool is smaller than R (finishing tool tip radius).

Solution Select a tool with a larger diameter.

9089 ‘SURFACE MILLING: F=0’

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9090 ‘SURFACE MILLING: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9091 ‘SURFACE MILLING: T=0’

Detection During execution.

Cause The tool number “T” has not been defined.

Solution The tool number “T” must be other than zero.

9092 ‘SURFACE MILLING: P=0’

Detection During execution.

Cause The depth “P” of the surface milling has not been defined.

Solution The surface milling depth “P” must be other than zero.

Errors in the profile milling operation 1.

9093 ‘PROFILE MILLING 1: F=0’

Detection During execution.

Cause A feedrate “F” has been defined with a wrong value.

Solution Program a positive feedrate “F” other than zero.

9094 ‘PROFILE MILLING 1: S=0’

Detection During execution.

Cause A spindle speed “S” has been defined with a wrong value.

Solution Program a positive spindle speed “S” other than zero.

9095 ‘PROFILE MILLING 1: T=0’

Detection During execution.

Cause The tool number “T” has not been defined.

Solution The tool number “T” must be other than zero.

9096 ‘PROFILE MILLING 1: P=0’

Detection During execution.

Cause The milling depth “P” has not been defined.

Solution The milling depth “P” must be other than zero.

9097 ‘PROFILE MILLING 1: Null profile’

Detection During execution.

Cause The profile to be machined has not been defined.

Solution The profile must consist of at least two points besides the entry one and the exit one.

9098 'PROFILE MILLING 2: T=0'

Detection During execution.
Cause The tool number "T" has not been defined.
Solution The tool number "T" must be other than zero.

9099 'PROFILE MILLING 2: F=0'

Detection During execution.
Cause A feedrate "F" has been defined with a wrong value.
Solution Program a positive feedrate "F" other than zero.

9100 'PROFILE MILLING 2: S=0'

Detection During execution.
Cause A spindle speed "S" has been defined with a wrong value.
Solution Program a positive spindle speed "S" other than zero.

9101 'PROFILE MILLING 2: P=0'

Detection During execution.
Cause The milling depth "P" has not been defined.
Solution The milling depth "P" must be other than zero.

9102 'SLOT MILLING: F=0'

Detection During execution.
Cause A feedrate "F" has been defined with a wrong value.
Solution Program a positive feedrate "F" other than zero.

9103 'SLOT MILLING: S=0'

Detection During execution.
Cause A spindle speed "S" has been defined with a wrong value.
Solution Program a positive spindle speed "S" other than zero.

9104 'SLOT MILLING: P=0'

Detection During execution.
Cause The milling depth "P" has not been defined.
Solution The milling depth "P" must be other than zero.

9105 'SLOT MILLING: L=0'

Detection During execution.
Cause The slot length "L" has not been defined.
Solution The pocket depth "P" must be other than zero.

9106 'SLOT MILLING: Tool diameter smaller than Δ'

Detection During execution.
Cause The programmed milling step «Δ» is larger than the tool diameter.
Solution Program a milling step «Δ» smaller than the tool diameter or choose a tool of larger diameter.

9107 'SLOT MILLING: Tool diameter greater than groove'

Detection During execution.
Cause The diameter of the tool is larger than the programmed slot.
Solution Select a tool with a smaller diameter.

9108 'SLOT MILLING: FINISHING tool diameter smaller than δ'

Detection During execution.
Cause The programmed finishing stock «δ» is larger than the tool diameter.
Solution Program a finishing stock «δ» smaller than the tool diameter or choose a tool of larger diameter.

9109 'LINEAR POSITIONING: Wrong I'

Detection During execution.

Cause The «I» distance between positionings has the wrong value and it does not allow an integer number of machining operations.

Solution Check that the data entered is correct.

9110 'ARC POSITIONING: Wrong β'

Detection During execution.

Cause The distance «β» between positionings has the wrong value and it does not allow an integer number of machining operations.

Solution Check that the data entered is correct.

9111 'RECTANGULAR POSITIONING: Wrong Ix/Iy'

Detection During execution.

Cause One of the distances «Ix/Iy» between positionings has the wrong value and it does not allow an integer number of machining operations.

Solution Check that the data entered is correct.

9112 'GRID PATTERN POSITIONING: Wrong Ix/Iy'

Detection During execution.

Cause One of the distances «Ix/Iy» between positionings has the wrong value and it does not allow an integer number of machining operations.

Solution Check that the data entered is correct.

Error solving Manual
(T model)

Ref 0204-ing

FAGOR

INDEX

PROGRAMMING ERRORS

0001 'Línea vacía.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When trying to enter into a program or execute an empty block or containing the label (block number).
2. In the "Pattern repeat canned cycle (G66)", "Roughing canned cycle along the X axis (G68)" or "Roughing canned cycle along the Z axis (G69)" when parameter "S" (beginning of profile) is greater than parameter "E" (end of profile).

Solution The solution for each cause is:

1. The CNC cannot enter into the program or execute an empty line. To enter an empty line in the program, use the «;» symbol at the beginning of that block. The CNC will ignore the rest of the block.
2. The value of parameter "S" (block where the profile definition begins) must be lower than the value of parameter "E" (block where the profile definition ends).

0002 'Improper data'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When editing an axis coordinate after the cutting conditions (F, S, T or D) or the «M» functions.
2. When the marks of the block skip (conditional block /1, /2 or /3) are not at the beginning of the block.
3. When programming a block number greater than 9999 while programming in ISO code.
4. While programming in high-level, the value of the RPT instruction exceeds 9999.

Solution The solution for each cause is:

1. Remember the programming order
2. Remember the programming order
   - Block skip (conditional block /1, /2 or /3).
   - Label (N).
   - «G» functions.
   - Axis coordinates. (X, Y, Z…).
   - Machining conditions (F, S, T, D).
   - «M» functions.
3. Correct the syntax of the block. Program the labels between 0 and 9999
4. Correct the syntax of the block. Program a number of repetitions between 0 and 9999

0003 'Improper data order.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The machining conditions or the tool data have been programmed in the wrong order.

Solution Remember that the programming order is:

… F...S...T...D……

All the data need not be programmed.

0004 'No more information allowed in the block.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When editing a «G» function after an axis coordinate.
2. When trying to edit some data after a «G» function (or after its associated parameters) which must go alone in the block (or which only admits its own associated data).
3. When assigning a numeric value to a parameter that does not need it.

Solution The solution for each cause is:

1. Remember that the programming order is:
   - Block skip (conditional block /1, /2 or /3).
   - Label (N).
   - «G» functions.
   - Axis coordinates. (X, Y, Z…).
   - Machining conditions (F, S, T, D).
   - «M» functions.
2. There are some «G» functions which carry associated data in the block. Maybe, this type of functions do not let program other type of information after their associated parameters. On the other hand, neither machining conditions, (F, S), tool data (T, D) nor «M» functions may be programmed.
3. There are some «G» functions having certain parameters associated to them which do not need to be defined with values.

0005 'Repeated information'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The same data has been entered twice in a block.

Solution Correct the syntax of the block. The same data cannot be defined twice in a block.

0006 'Improper data format'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While defining the parameters of a machining canned cycle, a negative value has been assigned to a parameter which only admits positive values.

Solution Verify the format of the canned cycle. In some canned cycles, there are parameters which only accept positive values.

0007 'Incompatible G functions.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When programming in the same block two «G» functions which are incompatible with each other.
2. When trying to define a canned cycle in a block containing a nonlinear movement (G02, G03, G08, G09, G33).

Solution The solution for each cause is:

1. There are groups of «G» functions which cannot go together in the block because they involve actions incompatible with each other. For example:
   G01/G02: Linear and circular interpolation
   G41/G42: Left-hand or right-hand tool radius compensation.
   This type of functions must be programmed in different blocks.
2. A canned cycle must be defined in a block containing a linear movement. In other words, to define a cycle, a "G00" or a "G01" must be active. Nonlinear movements (G02, G03, G08 and G09) may be defined in the blocks following the profile definition.

0008 'Nonexistent G function'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A nonexistent «G» function has been programmed.

Solution Check the syntax of the block and verify that a different «G» function is not being edited by mistake.

0009 'No more G functions allowed'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A «G» function has been programmed after the machining conditions or after the tool data.

Solution Remember that the programming order is:

- Block skip (conditional block /1, /2 or /3).
- Label (N).
- «G» functions.
- Axis coordinates. (X,Y,Z…).
- Machining conditions (F, S, T, D).
- «M» functions.

0010 'No more M functions allowed'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause More than 7 «M» functions have been programmed in a block.

Solution The CNC does not let program more than 7 «M» functions in a block. To execute any other functions, write them in a separate block. The «M» functions may go alone in a block.

0011 'This G or M function must be alone.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The block contains either a «G» or an «M» function that must go alone in the block.

Solution Write it alone in the block.

0012 'Program F, S, T, D before the M functions.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A machining condition (F, S) or tool data (T, D) has been programmed after the «M» functions.

Solution Remember that the programming order is:

… F...S...T...D...M—

Up to 7 «M» functions may be programmed .

All the data need not be programmed.

0013 'Program G30 D +/-359.9999'

No explanation required

0014 'Do not program labels by parameters.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A label (block number) has been defined with a parameter.

Solution Programming the block number is optional, but it cannot be defined with a parameter It can only be defined with a number between 0 and 9999.

0015 'Number of repetitions not possible.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A repetition has been programmed wrong or the block does not admit repetitions.

Solution High level instructions do not admit a number of repetitions at the end of the block. To do a repetition, assign to the block to be repeated a label (block number) and use the RPT instruction.

0016 'Program: G15 o G15 C.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to execute an operation on the "C" axis, but the axis is not active.

Solution In order to operate with the "C" axis, it must be activated first using the "G15" function.

0017 'Program: G16 axis-axis.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the function «Main plane selection by two axes (G16)» one of the two parameters for the axes has not been programmed.

Solution Check the syntax of the block. The definition of the "G16" function requires the name of the axes defining the new work plane.

0018 'Program: G22 K(1/2/3/4) S(0/1/2).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the function «Enable/Disable work zones (G22)» the type of enable or disable of the work zone has not been defined or it has been assigned the wrong value.

Solution The parameter for enabling or disabling the work zones "S" must always be programmed and it may take the following values.

S=0: The work zone is disabled.
S=1: It is enabled as a no-entry zone.
S=2: It is enabled as a no-exit zone.

0019 'Program: work zone K1, K2, K3 or K4.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. A "G20", "G21" or "G22" function has been programmed without defining the work zone K1, K2, K3 or K4.
2. The programmed work zone is smaller than 0 or greater than 4.

Solution The solution for each cause is:

1. The programming format for functions "G20", "G21" and "G22" is:

G20 K...X...C±5.5	Definition of lower work zone limits
G21 K...X...C±5.5	Definition of upper work zone limits.
G22 K...S—	Enable/disable work zones.

Where:

K :	Is the work zone.
X...C	Are the axes where the limits are defined.
S	Is the type of work zone enable.

2. The "K" work zone may only have the values of K1, K2, K3 or K4.

0020 'Program G36-G39 with R+5.5.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the "G36" or "G39" function, the "R" parameter has not been programmed or it has been assigned a negative value.

Solution To define "G36" or "G39", parameter "R" must also be defined and with a positive value).

G36	R= Rounding radius.
G39	R= Distance between the end of the programmed path and the point to be chamfered.

0021 'Program: G72 S5.5 or axis (axes).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When programming a general scaling factor (G72) without the scaling factor to apply.
2. When programming a particular scaling factor (G72) to several axes, but the axes have been defined in the wrong order.

Solution Remember that the programming format for this function is:

G72 S5.5"	When applying a general scaling factor (to all axes).
G72 X…C5.5"	When applying a particular scaling factor to one or several axes.

0023 'Block incompatible when defining a profile.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the set of blocks defining a profile, there is a block containing a «G» function that cannot be part of the profile definition.

Solution The «G» functions available in the profile definition are:

G00: Beginning of the profile.
G01: Linear interpolation.
G02/G03: Clockwise/counterclockwise interpolation.
G06: Circle center in absolute coordinates.
G08: Arc tangent to previous path.
G09: Three point arc.
G36: Controlled corner rounding
G39: Chamfer.
G53: Programming with respect to home.
G70/G71: Inch/metric programming.
G90/G91: Programming in absolute/incremental coordinates.
G93: Polar origin preset.

0024 'High level blocks not allowed when defining a profile.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause Within the set of blocks defining a profile, a high level block has been programmed.

Solution Profiles must be defined in ISO code. High level instructions are not allowed (GOTO, MSG, RPT ...).

0025 'Program: G77 axes (2 to 6) or G77 S.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the "axis slaving function (G77)» the parameters for the axes are missing or in "spindle synchronization (G77S) functions the "S" parameter is missing.

Solution In the "axis slaving" function, program at least two axes and in the "spindle synchronization" function, always program the "S" parameter.

0026 'Program: G93 IJ.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Polar origin preset (G93)» function, some of the parameters for the new polar origin have not been programmed.

Solution Remember that the programming format for this function is:

G93 I...J...

The "I", "J" values are optional, but if programmed, both must be programmed and they indicate the new polar origin.

0028 'G2 or G3 not allowed when programming a canned cycle.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A canned cycle has been attempted to execute while the "G02", "G03" or "G33" functions were active.

Solution To execute a canned cycle, "G00" or "G01" must be active. A "G02" or "G03" function may be programmed previously in the program history. Check that these functions are not active when the canned cycle is defined.

0029 'G84-85: X Z Q R C [D L M F H] I K.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the canned cycle for "Turning of curved sections (G84)" or for "Facing curved sections (G85)" have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. A parameter has been programmed which does not match the calling format.

Solution The following parameters must be programmed in this cycle:

X-Z : Profile starting point
Q-R : Profile end point
C : Cutting pass
I-K : Distance from the starting point to the arc center.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0030 'G86-87: X Z Q R I B [D L] C [J A].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the canned cycle for "longitudinal threadcutting (G86)" or for "face threadcutting (G87)" have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. A parameter has been programmed which does not match the calling format.

Solution The following parameters must be programmed in this cycle:

X-Z : Starting point of the thread.
Q-R : End point of the thread.
I : Thread depth.
B : Cutting pass
C : Thread pitch.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0031 'G88-G98: X Z Q R [C D K].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the canned cycle for "grooving along X (G88)" or "grooving along Z (G89)" have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. A parameter has been programmed which does not match the calling format.

Solution The following parameters must be programmed in this cycle:

X-Z : Starting point of the groove.
Q-R : End point of the groove.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0032 'G66: X Z I C [A L M H] S E.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the «Pattern repeat canned cycle with islands (G66)» have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. A parameter has been programmed which does not match the calling format.

Solution The following parameters must be programmed in this cycle:

X-Z : Profile starting point
I : Residual stock.
C : Cutting pass
S : Block where the profile geometry description begins.
E : Block where the profile geometry description ends.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0033 'G68-G69: X Z C [D L M F H] S E .'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the canned cycle for "roughing along X (G68)" or "roughing along Z (G69)" have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. A parameter has been programmed which does not match the calling format.

Solution The following parameters must be programmed in this cycle:

X-Z : Profile starting point
C : Cutting pass
S : Block where the profile geometry description begins.
E : Block where the profile geometry description ends.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0034 'G81-G82: X Z Q R C [D L M F H].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the canned cycle for "Turning of straight sections (G81)" or for "Facing straight sections (G82)" have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. A parameter has been programmed which does not match the calling format.

Solution The following parameters must be programmed in this cycle:

X-Z : Profile starting point
Q-R : Profile end point
C : Cutting pass

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0035 'G83: X Z I B [D K H C].'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters have been programmed wrong in the «Axial drilling/tapping cycle (G83)». These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. A parameter has been programmed which does not match the calling format.

Solution The following parameters must be programmed in this cycle:

X-Z : Machining position.
I : Machining depth.
B : Type of operation.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0036 'G60-G61: X Z I B Q A J [D K H C] S.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the canned cycle for "face drilling or tapping (G60)" or for "longitudinal drilling or tapping (G61)" have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.
3. A parameter has been programmed which does not match the calling format.

Solution The following parameters must be programmed in this cycle:

X-Z : Machining position.
I : Machining depth.
B : Type of operation.
Q : Angular position of the first machining operation.
A : Angular step between machining operations.
J : Number of machining operations.
S : Speed and turning direction of the live tool.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0037 'G62-G63: X Z L I Q A J [D] F S.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The parameters of the canned cycle for "longitudinal slot milling (G62)" or "face slot milling (G62)" have been programmed wrong. These may be the probable causes:

1. Some mandatory parameter is missing.
2. The parameters of the cycle have not been edited in the correct order.

Solution The following parameters must be programmed in this cycle:

X-Z : Slot position.
L : Slot length.
I : Slot depth.
Q : Angular position of the first slot.
A : Angular step between slots.
J : Number of slots.

F : Feedrate.
S : Speed and turning direction of the live tool.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0043 'Incomplete Coordinates.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. During simulation or execution, when trying to make a movement defined with only one coordinate of the end point or without defining the arc radius while a «circular interpolation (G02/G03) is active.
2. During editing, when editing a circular movement (G02/G03) by defining only one coordinate of the end point or not defining the arc radius.

Solution The solution for each cause is:

1. A "G02" or "G03" function may be programmed previously in the program history. In this case, to make a move, both coordinates of the end point and the arc radius must be defined. To make a linear movement, program "G01".
2. To make a circular movement (G02/G03), both coordinates of the end point and the arc radius must be programmed.

0044 'Incorrect Coordinates.'

Detection During the execution in programs transmitted via DNC.

Cause An attempt has been made to execute a block syntactically incorrect (G1 X20K-15)

Solution Correct the syntax of the block.

0045 'Polar coordinates not allowed.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause When «Programming with respect to home (G53)», the end point has been defined in polar or cylindrical coordinates or in Cartesian coordinates with an angle.

Solution When programming with respect to home, only Cartesian coordinates may be programmed.

0046 'Axis does not exist.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A block has been edited whose execution involves the movement of a nonexistent axis.

Solution Check that the axis name being edited is correct.

0047 'Program axes.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause No axis has been programmed in a function requiring an axis.

Solution Some instructions require the programming of axes (REPOS, G14, G20, G21…).

0048 'Incorrect order of axes.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The axis coordinates have not been programmed in the correct order or an axis has been programmed twice in the same block.

Solution Remember that the correct programming order for the axes is:

X...Y...Z...U...V...W...A...B...C—

All axes need not be programmed:

0049 'Point incompatible with active plane.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When trying to do a circular interpolation, the end point is not in the active plane.
2. When trying to do a tangential exit in a path that is not in the active plane.

Solution The solution for each cause is:

1. Maybe a plane has been defined with "G16", "G17", "G18" or "G19". In this case, circular interpolations can only be carried out on the main axes defining that

plane. To define a circular interpolation in another plane, it must be defined beforehand.

2. Maybe a plane has been defined with "G16", "G17", "G18" or "G19". In this case, corner rounding, chamfers and tangential entries/exits can only be carried out on the main axes defining that plane. To do it in another plane, it must be defined beforehand.

0050 'Program positions on active plane.'

No explanation required

0051 'Perpendicular axis included in active plane.'

No explanation required

0052 'Center of circle programmed incorrectly.'

No explanation required

0053 'Program pitch.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Electronic threading cycle (G33)» the parameter for the thread pitch is missing.

Solution Remember that the programming format for this function is:

G33 X...C...L...

Where: "L" is the thread pitch.

0054 'Pitch programmed incorrectly.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A helical interpolation has been programmed with the wrong or negative pitch.

Solution Remember that the programming format is:

G02/G03 X...Y...I...J...Z...K...

Where: "K" is the helical pitch (always positive value).

0055 'Positioning axes or Hirth axes not allowed'

No explanation required

0056 'The axis is already slaved.'

No explanation required

0057 'Do not program a slaved axis.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When trying to move an axis alone while being slaved to another one.
2. When trying to slave an axis that is already slaved using the G77 function «Electronic axis slaving».

Solution The solution for each cause is:

1. A slaved axis cannot be moved separately. To move a slaved axis, its master axis must be moved. Both axes will move at the same time.
   Example: If the Y axis is slaved to the X axis, an X axis move must be programmed in order to move the Y axis (together with the X axis).
   To unslave the axes, program "G78".
2. An axis cannot be slaved to two different axes at the same time. To unslave the axes, program "G78".

0058 'Do not program a GANTRY axis.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. When trying to move an axis alone while being slaved to another one as a GANTRY axis
2. When defining an operation on a GANTRY axis. (Definition of work zone limits, planes, etc.).

Solution The solution for each cause is:

1. A GANTRY axis cannot be moved separately. To move a GANTRY axis, its associated axis must be moved. Both axes will move at the same time.
   Example: If the Y axis is a GANTRY axis associated with the X axis, an X axis move must be programmed in order to move the Y axis (together with the X axis).
   GANTRY axes are defined by machine parameter.
2. The axes defined as GANTRY cannot be used in the definition of operations or movements. These operations are defined with the main axis that the GANTRY axis is associated with.

0059 'Eje HIRTH: program only integer values.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A rotation of a HIRTH axis has been programmed with a decimal value.

Solution HIRTH axes do not accept decimal angular values. They must be full degrees.

0060.'Invalid action.'

No explanation required

0061 'ELSE not associated with IF.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. While editing in High level language, when editing the "ELSE" instruction without having previously programmed an "IF".
2. When programming in high level language, an "IF" is programmed without associating it with any action after the condition.

Solution Remember that the programming formats for this instruction are:

(IF (condition) <action1>)
(IF (condition) <action1> ELSE <action2>)

If the condition is true, it executes the < action1>, otherwise, it executes < action2>.

0062 'Program label N(0-9999).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a block number out of the 0-9999 range has been programmed in the "RPT" or "GOTO" instruction.

Solution Remember that the programming format of these instructions is:

(RPT N(block number), N(block number))
(GOTO N(block number))

The block number (label) must be between 0 and 9999.

0063 'Program subroutine number 1 thru 9999.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a subroutine number out of the 0-9999 range has been programmed in the "SUB" instruction.

Solution Remember that the programming format for this instruction is:

(SUB (integer))

The subroutine number must be between 0 and 9999.

0064 'Repeated subroutine.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause There has been an attempt to define a subroutine already existing in another program of the memory.

Solution In the CNC memory, there could not be more than one subroutine with the same identifying number even if they are contained in different programs.

0065 'The main program cannot have a subroutine.'

Detection In execution or while executing programs transmitted via DNC.

Cause The various causes might be:

1. An attempt has been made to define a subroutine in the MDI execution mode.
2. A subroutine has been defined in the main program.

Solution The solution for each cause is:

1. Subroutines cannot be defined from the «MDI execution» option of the menu.
2. Subroutines must be defined after the main program or in a separate program. They cannot be defined before or inside the main program.

0066 'Expecting a message.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level, the "MSG" or "ERROR" instruction has been edited but without the message to be displayed.

Solution Remember that the programming format of these instructions is:

(MSG "message")
(ERROR integer, "error message")

Although it can also be programmed as follows:

(ERROR integer)
(ERROR "error message")

0067 'OPEN is missing.'

Detection In execution or while executing programs transmitted via DNC.

Cause While programming in high level, a "WRITE" instruction has been edited, but the OPEN instruction has not been written previously to tell it where that instruction has to be executed.

Solution The "OPEN" instruction must be edited before the "WRITE" instruction to «tell» the CNC where (in which program) it must execute the "WRITE" instruction.

0068 'Expecting a program number.'

No explanation required

0069 'Program does not exist.'

Detection In execution or while executing programs transmitted via DNC.

Cause In the "Pattern repeat canned cycle (G66)", "Roughing canned cycle along the X axis (G68)" or "Roughing canned cycle along the Z axis (G69)", it has been programmed that the profiles are located in another program (parameter "Q"), but the program does not exist.

Solution Parameter "Q" defines which program contains the profile definitions of the cycles. If this parameter is programmed, that program number must exist and it must contain the labels defined by parameters "S" and "E".

0070 'Program already exists.'

Detection In execution or while executing programs transmitted via DNC.

Cause This error comes up during execution when using the "OPEN" instruction (While programming in high level language) to create an already existing program.

Solution Change the program number or use parameters A/D in the "OPEN" instruction:

(OPEN P———,A/D,…)

Where:

A: Appends new blocks after the existing ones.
D: Deletes the existing program and it opens it as a new one.

0071 'Expecting a parameter'

Detection While editing tables.

Cause The wrong parameter number has been entered (maybe the "P" character is missing) or another action is being carried out (moving around in the table) before quitting the table editing mode.

Solution Enter the parameter number to be edited or press [ESC] to quit this mode.

0072 'Parameter does not exist.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the "ERROR" instruction has been edited, but the error number to be displayed has been defined either with a local parameter greater than 25 or with a global parameter greater than 299.

Solution The parameters used by the CNC are:

Local: 0-25

Global: 100-299

0073 'Parameter range protected. Cannot be written.'

No explanation required

0074 'Variable not accessible from CNC.'

No explanation required

0075 'Read-only variable.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to assign a value to a read-only variable.

Solution Read-only variables cannot be assigned any values through programming. However, their values can be assigned to a parameter.

0076 'Write-only variable.'

No explanation required

0077 'Analog output not available.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to write to an analog output currently being used by the CNC.

Solution The selected analog output may be currently used by an axis or a spindle. Select another analog output between 1 and 8.

0078 'Program channel 0(CNC),1(PLC) or 2(DNC).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the "KEYSCR" instruction has been programmed, but the source of the keys is missing.

Solution When programming the "KEYSCR" instruction, the parameter for the source of the keys must always be programmed:

(KEYSCR=0) : CNC keyboard
(KEYSCR=1) : PLC
(KEYSCR=2) : DNC

The CNC only lets modifying the contents of this variable if it is «zero»

0079 'Program error number 0 thru 9999.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the "ERROR" instruction has been programmed, but the error number to be displayed is missing.

Solution Remember that the programming format for this instruction is:

(ERROR integer, "error message")

Although it can also be programmed as follows:

(ERROR integer)
(ERROR "error message")

0080 'Operator missing.'

No explanation required

0081 'Incorrect expression.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, an expression has been edited with the wrong format.

Solution Correct the syntax of the block.

0082 'Incorrect operation.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. While programming in high level language, the assignment of a value to a parameter is incomplete.
2. While programming in high level language, the call to a subroutine is incomplete.

Solution Correct (complete) the format to assign a value to a parameter or a call to a subroutine.

0083 'Incomplete operation.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the "IF" instruction has been edited without the condition between brackets.

Solution Remember that the programming formats for this instruction are:

(IF (condition) <action1>)

(IF (condition) <action1> ELSE <action2>)

If the condition is true, it executes the < action1>, otherwise, it executes < action2>.

0084 'Expecting "=".'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

Solution Enter the "=" symbol in the right place.

0085 'Expecting ")".'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

Solution Enter the ")" symbol in the right place.

0086 'Expecting "(".'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.

Solution Enter the "(" symbol in the right place.

0087 'Expecting ",".'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause The various causes might be:

1. While programming in high level language, a symbol or data has been entered that does not match the syntax of the block.
2. While programming in high level language, an ISO-coded instruction has been programmed.
3. While programming in high level language, an operation has been assigned either to a local parameter greater than 25 or to a global parameter greater 299.

Solution The solution for each cause is:

1. Enter the "," symbol in the right place.
2. A block cannot contain high level language instructions and ISO-coded instructions at the same time.
3. The parameters used by the CNC are:

   Local: 0-25

   Global: 100-299

   Other parameters out of this range cannot be used in operations.

0088 'Operation limit exceeded.'

No explanation required

0089 'Logarithm of zero or negative number.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves the calculation of a negative number or a zero.

Solution Only logarithms of numbers greater than zero can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0090 'Square root of a negative number.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves the calculation of the square root of a negative number.

Solution Only the square root of numbers greater than zero can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0091 'Division by zero.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves a division by zero.

Solution Only divisions by numbers other than zero are allowed. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0092 'Base zero with positive exponent.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves elevating zero to a negative exponent (or zero).

Solution Zero can only be elevated to positive exponents greater than zero. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0093 'Negative base with decimal exponent.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves elevating a negative number to a decimal exponent.

Solution Negative numbers can only be elevated to integer exponents. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0094 'ASIN/ACOS range exceeded.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An operation has been programmed which involves calculating the arcsine or arccosine of a number out of the ±1 range.

Solution Only the arcsine (ASIN) or arccosine (ACOS) of numbers between ±1 can be calculated. When working with parameters, that parameter may have already acquired a negative value or zero. Check that the parameter does not reach the operation with that value.

0095 'Program row number.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While editing a customizing program, a window has been programmed with the "ODW" instruction, but the vertical position of the window on the screen is missing.

Solution The vertical position of the window on the screen is defined by rows (0-25).

0096 'Program column number.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While editing a customizing program, a window has been programmed with the "ODW" instruction, but the horizontal position of the window on the screen is missing.

Solution The horizontal position of the window on the screen is defined by columns (0-79).

0097 'Program another softkey.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While editing a customizing program, the programming format for the "SK" instruction has not been respected.

Solution Correct the syntax of the block. The programming format is:

(SK1=(text 1), SK2=(text 2)…)

If the "," character is entered after a text, the CNC expects the name of another softkey.

0098 'Program softkeys 1 thru 7.'

Detection While executing in the user channel.

Cause In the block syntax, a softkey has been programmed out of the 1 to 7 range.

Solution Only softkeys within the 1 to 7 range can be programmed.

0099 'Program another window.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While editing a customizing program, the programming format for the "DW" instruction has not been respected.

Solution Correct the syntax of the block. The programming format is:

(DW1=(assignment), DW2=(assignment)…)

If the "," character is entered after an assignment, the CNC expects the name of another window.

0100 'Program windows 0 thru 25.'

Detection While executing in the user channel.

Cause In the block syntax, a window has been programmed out of the 0 to 25 range.

Solution Only windows within the 0 to 25 range can be programmed.

0101 'Program rows 0 thru 20.'

Detection While executing in the user channel.

Cause In the block syntax, a row has been programmed out of the 0 to 20 range.

Solution Only rows within the 0 to 20 range can be programmed.

0102 'Program columns 0 thru 79.'

Detection While executing in the user channel.

Cause In the block syntax, a column has been programmed out of the 0 to 79 range.

Solution Only columns within the 0 to 79 range can be programmed.

0103 'Program pages 0 thru 255.'

Detection While executing in the user channel.

Cause In the block syntax, a page has been programmed out of the 0 to 255 range.

Solution Only pages within the 0 to 255 range can be programmed.

0104 'Program INPUT.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, an "IB" instruction has been edited without associating an "INPUT" to it.

Solution Remember that the programming formats for this instruction are:

(IB (expression) = INPUT "text", format)

(IB (expression) = INPUT "text")

0105 'Program inputs 0 thru 25.'

Detection While executing in the user channel.

Cause In the block syntax, an input has been programmed out of the 0 to 25 range.

Solution Only inputs within the 0 to 25 range can be programmed.

0106 'Program numerical format.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, an "IB" instruction has been edited with non-numeric format.

Solution Remember that the programming format for this instruction is:
(IB (expression) = INPUT "text", format)

Where «format» must be a signed number with 6 entire digits and 5 decimals at the most.

If the "," character is entered after the text, the CNC expects the format.

0107 'Do not program formats greater than 6.5 .'

Detection While executing in the user channel.

Cause While programming in high level language, an "IB" instruction has been edited in a format with more than 6 entire digits or more than 5 decimals.

Solution Remember that the programming format for this instruction is:
(IB (expression) = INPUT "text", format)

Where «format» must be a signed number with 6 entire digits and 5 decimals at the most.

0108 'This command can only be executed in the user channel.'

Detection During execution.

Cause An attempt has been made to execute a block containing information that can only be executed through the user channel.

Solution There are specific expressions for customizing programs that can only be executed inside the user program.

0109 'C. User: do not program geometric help, compensation or cycles.'

Detection While executing in the user channel.

Cause An attempt has been made to execute a block containing geometric aide, tool radius/length compensation or machining canned cycles.

Solution Inside a customizing program the following cannot be programmed:
Neither geometric assistance nor movements.
Neither tool radius nor length compensation.
Canned cycles.

0110 'Local parameters not allowed.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause Some functions can only be programmed with global parameters.

Solution Global parameters are the ones included in the 100-299 range.

0111 'Block cannot be executed while running another program'

Detection While executing in MDI mode

Cause An attempt has been made to execute a customizing instruction from MDI mode while the user channel program is running.

Solution Customizing instructions can only be executed through the user channel.

0112 'WBUF can only be executed in user channel while editing'

Detection During normal execution or execution through the user channel.

Cause An attempt has been made to execute the "WBUF" instruction.

Solution The "WBUF" instruction cannot be executed. It can only be used in the editing stage through the user input.

0113 'Table limits exceeded.'

Detection While editing tables.

Cause The various causes might be:

1. In the tool offset table, an attempt has been made to define a tool offset with a greater number than allowed by the manufacturer.
2. In the parameter tables, an attempt has been made to define a nonexistent parameter.

Solution The tool offset number must be smaller than the one allowed by the manufacturer.

0114 'Offset: D3 X Z R F I K.'

Detection While editing tables.

Cause In the tool offset table, the parameter editing order has not been respected.

Solution Enter the table parameters in the right order.

0115 'Tool: T4 D3 F3 N5 R5(.2).'

Detection While editing tables.

Cause In the tool table, the parameter editing order has not been respected.

Solution Enter the table parameters in the right order.

0116 'Origin: G54-59 axes (1-5).'

Detection While editing tables.

Cause In the Zero offset table, the zero offset to be defined (G54-G59) has not be selected.

Solution Enter the table parameters in the right order. To fill out the zero offset table, first select the offset to be defined (G54-G59) and then the zero offset position for each axis.

0117 'Function: M4 S4 bits(8).'

Detection While editing tables.

Cause In the «M» function table, the parameter editing order has not been respected.

Solution Edit table following the format:
M1234 (associated subroutine) (customizing bits)

0118 'G51 [A] E'

Detection In execution or while executing programs transmitted via DNC.

Cause In the «Look-Ahead (G51)» function, the parameter for the maximum contouring error is missing.

Solution This type of machining requires the programming of:
E : Maximum contouring error.

The rest of the parameters are optional. The parameters must be edited in the order indicated by the error message.

0119 'Leadscrew: Coordinate-error.'

Detection While editing tables.

Cause In the leadscrew compensation tables, the parameter editing order has not been respected.

Solution Enter the table parameters in the right order.
P123 (position of the axis to be compensated) (leadscrew error at that point)

0120 'Incorrect axis.'

Detection While editing tables.

Cause In the leadscrew compensation tables, an attempt has been made to edit a different axis from the one corresponding to that table.

Solution Each axis has its own table for leadscrew compensation. The table for each axis can only contain the positions for that axis.

0121 'Program P3 = value.'

Detection While editing tables.

Cause In the machine parameter table, the editing format has not been respected.

Solution Enter the table parameters in the right order.
P123 = (parameter value)

0122 'Magazine: P(1-255) = T(1-9999).'

Detection While editing tables.

Cause In the tool magazine table, the editing format has not been respected or some data is missing.

Solution Enter the table parameters in the right order.

0123 'Tool T0 does not exist.'

Detection While editing tables.

Cause In the tool table, an attempt has been made to edit a tool as T0.

Solution No tool can be edited as T0. The first tool must be T1.

0124 'Offset D0 does not exist.'

Detection While editing tables.

Cause In the tool table, an attempt has been made to edit a tool offset as D0.

Solution No tool offset can be edited as D0. The first tool offset must be D1.

0125 'Do not modify the active tool or the next one.'

Detection During execution.

Cause In the tool magazine table, an attempt has been made to change the active tool or the next one.

Solution During execution, neither the active tool nor the next one may be changed.

0126 'Tool not defined.'

Detection While editing tables.

Cause In the tool magazine table, an attempt has been made to assign to the magazine position a tool that is not defined in the tool table.

Solution Define the tool in the tool table.

0127 'Magazine is not RANDOM.'

Detection While editing tables.

Cause There is no RANDOM magazine and, in the tool magazine table, the tool number does not match the tool magazine position.

Solution When the tool magazine is not RANDOM, the tool number must be the same as the magazine position (pocket number).

0128 'The position of a special tool is set.'

Detection While editing tables.

Cause In the tool magazine table, an attempt has been made to place a tool in a magazine position reserved for a special tool.

Solution When a special tool occupies more than one position in the magazine, it has a reserved position in the magazine. No other tool can be placed in this position.

0129 'Next tool only possible in machining centers.'

Detection During execution.

Cause A tool change has been programmed with M06, but the machine is not a machining center. (it is not expecting the next tool).

Solution When the machining is not a machining center, the tool change is done automatically when programming the tool number «T».

0130 'Write 0/1.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values of 0 or 1.

0131 'Write +/-.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values of + or -.

0132 'Write YES/NO.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values of YES or NO.

0133 'Write ON/OFF.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values of ON or OFF.

0134 'Values 0 thru 2.'
0135 'Values 0 thru 3.'
0136 'Values 0 thru 4.'
0137 'Values 0 thru 9.'
0138 'Values 0 thru 29.'
0139 'Values 0 thru 100.'
0140 'Values 0 thru 255.'
0141 'Values 0 thru 9999.'
0142 'Values 0 thru 32767.'
0143 'Values within +/-32767.'
0144 'Values 0 thru 65535.'

Detection While editing machine parameters

Cause The various causes might be:

1. An attempt has been made to assign the wrong value to a parameter.
2. During execution, when inside the program a call has been made to a subroutine (MCALL, PCALL) with a value greater than allowed.

0145 'Format +/- 5.5.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values with the format:

0146 'Word does not exist.'

No explanation required

0147 'Numerical format exceeded.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A data or parameter has been assigned a value greater than the established format.

Solution Correct the syntax of the block. Most of the time, the numeric format will be 5.4 (5 integers and 4 decimals).

0148 'Text too long.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the "ERROR" or "MSG" instruction has been assigned a text with more than 59 characters.

Solution Correct the syntax of the block. The "ERROR" and "MSG" instructions cannot be assigned texts longer than 59 characters.

0149 'Incorrect message.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, the text associated with the "ERROR" or "MSG" instruction has been edited wrong.

Solution Correct the syntax of the block. The programming format is:
(MSG "message")
(ERROR number, "message")

The message must be between " ".

0150 'Incorrect number of bits.'

Detection While editing tables.

Cause The various causes might be:

1. In the «M» function table, in the section on customizing bits:
   The number does not have 8 bits.
   The number does not consist of 0's and 1's.

2. In the machine parameter table, an attempt has been made to assign the wrong value of bit to a parameter.

Solution The solution for each cause is:

1. The customizing bits must consist of 8 digits of 0's and 1's.
2. The parameter only admits 8-bit or 16-bit numbers.

0151 'Negative numbers not allowed.'

No explanation required

0152 'Incorrect parametric programming.'

Detection During execution.

Cause The parameter has a value that is incompatible with the function it has been assigned to.

Solution This parameter may have taken the wrong value, in the program history. Correct the program so this parameter does not reach the function with that value.

0153 'Decimal format not allowed.'

No explanation required

0154 'Insufficient memory.'

Detection During execution.

Cause The CNC does not have enough memory to internally calculate the paths.

Solution Sometimes, this error is taken care of by changing the machining conditions.

0155 'Help not available.'

No explanation required

0156 'Don't program G33 ,G34, G95 or M19 S with no spindle encoder'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A "G33", "G34", "G95", "G95" or "M19 S" has been programmed without having an encoder on the spindle.

Solution If the spindle does not have an encoder, functions "M19 S", "G33", "G34" or "G95" cannot be programmed. Spindle machine parameter "NPULSES (P13)" indicates the number of encoder pulses per turn.

0159 'Inch programming limit exceeded.'

Detection During execution.

Cause An attempt has been made to execute in inches a program edited in millimeters.

Solution Enter function G70 (inch programming) or G71 (mm programming) at the beginning of the program.

0162 'No negative radius allowed with absolute coordinates'

Detection During execution.

Cause While operating with absolute polar coordinates, a movement with a negative radius has been programmed.

Solution Negative radius cannot be programmed when using absolute polar coordinates.

0164 'Wrong password.'

Detection While assigning protections.

Cause [ENTER] has been pressed before selecting the type of code to be assigned a password.

Solution Use the softkeys to select the type of code to which a password is to be assigned.

0165 'Password: use uppercase/lowercase letters or digits.'

Detection While assigning protections.

Cause A bad character has been entered in the password.

Solution The password can only consist of letters (upper and lower case) or digits.

0166 'Only one HIRTH axis per block is allowed.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A movement has been programmed which involves the movement of two HIRTH axes simultaneously.

Solution The CNC does not admit movements involving more than one HIRTH axis at a time. HIRTH axes must move one at a time.

0167 'Rot. axis position: absolute values (G90) within 0-359.9999.'

Detection During execution.

Cause A movement of a positioning-only rotary axis has been programmed. The movement has been programmed in absolute coordinates (G90) and the target coordinate of the movement is not within the 0 to 359.9999 range.

Solution Positioning-only rotary axes: In absolute coordinates, only movements within the 0 to 359.9999 range are possible.

0168 'Rotary axis: absolute values (G90) within 0-359.9999.'

Detection During execution.

Cause A movement of a rotary axis has been programmed. The movement has been programmed in absolute coordinates (G90) and the target coordinate of the movement is not within the 0 to 359.9999 range.

Solution Rotary axes: In absolute coordinates, only movements within the 0 to 359.9999 range are possible.

0169 'Modal subroutines cannot be programmed.'

Detection While executing in MDI mode

Cause An attempt has been made to call upon a modal subroutine (MCALL).

Solution MCALL modal subroutines cannot be executed from the menu option «MDI execution».

0170 'Program symbols 0 thru 255 in positions 0-639, 0-335. '

No explanation required

0171 'The window must be previously defined.'

Detection During normal execution or execution through the user channel.

Cause An attempt has been made to write in a window (DW) that has not been previously defined (ODW).

Solution It is not possible to write in a window that has not been previously defined. Check that the window to write in (DW) has been previously defined.

0172 'The program is not accessible'

Detection During execution.

Cause An attempt has been made to execute a program that cannot be executed.

Solution The program may be protected against execution. To know whether a program may be executed, check for the "X" character on the attributes column. If this character is missing, the program cannot be executed.

0173 'It is not possible to program angle + angle.'

No explanation required

0174 'Circular (helical) interpolation not possible.'

Detection During execution.

Cause An attempt has been made to execute a helical interpolation while the «LOOK-AHEAD (G51)» function was active.

Solution Helical interpolations are not possible while the «LOOK-AHEAD (G51)» function is active.

0175 'Analog inputs: ANAI(1-8) = +/-5 Volts.'

Detection During execution.

Cause An analog input has taken a value out of the ±5V range.

Solution Analog inputs may only take values within the ±5V range.

0176 'Analog outputs: ANAO(1-8) = +/-10 Volts.'

Detection During execution.

Cause An analog output has been assigned a value out of the ±10V range.

Solution Analog outputs may only take values within the ±10V range.

0177 'A gantry axis cannot be part of the active plane.'

No explanation required

0178 'G96 only possible with analog spindle.'

Detection During execution.

Cause The "G96" function has been programmed but either the spindle speed is not controlled or the spindle does not have an encoder.

Solution To operate with the "G96" function, the spindle speed must be controlled (SPDLTYPE(P0)=0) and the spindle must have an encoder (NPULSES(P13) other than zero).

0179 'Do not program more than 4 axes simultaneously.'

No explanation required

0180 'Program DNC1/2, HD or CARD A (optional).'

Detection While editing or executing.

Cause While programming in high level language, in the "OPEN" and "EXEC" instructions, an attempt has been made to program a parameter other than DNC1/2, HD or CARD A, or the DNC parameter has been assigned a value other than 1 or 2.

Solution Check the syntax of the block.

0181 'Program A (append) or D (delete).'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the "OPEN" instruction the A/D parameter is missing.

Solution Check the syntax of the block. The programming format is:

(OPEN P———,A/D,…)

Where:

A Appends new blocks after the existing ones.

D: Deletes the existing program and it opens it as a new one.

0182 'Option not available.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause A «G» function has been defined which is not a software option.

0184 'T with subroutine: program only T and D.'

No explanation required

0185 'Tool offset does not exist'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause Within the block syntax, a tool offset has been called upon which is greater than the ones allowed by the manufacturer.

Solution Program a new smaller tool offset.

0186 'The "C" axis does not exist.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to activate the "C" axis, but the machine does not have this feature.

0187 'G66, G68, G69 are not allowed when machining with the "C" axis.'

Detection During execution.

Cause An attempt has been made to execute a "G66", "G67" or "G68" canned cycle while the "C" axis is active.

Solution Cancel the “C” axis to work with these canned cycles.

0188 ‘Function not possible from PLC.’

Detection During execution.

Cause From the PLC channel and using the “CNCEX” instruction, an attempt has been made to execute a function that is incompatible with the PLC channel execution.

Solution The installation manual (chapter 11.1.2) offers a list of the functions and instructions that may be executed through the PLC channel.

0189 ‘The live tool does not exist.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to start the live tool “M45 S—”, but the machine does not have this feature.

0194 ‘Repositioning not allowed.’

Detection During execution.

Cause The axes cannot be repositioned using the “REPOS” instruction because the subroutine has not been activated with one of the interruption inputs.

Solution Before executing the “REPOS” instruction, one of the interruption inputs must be activated.

0195 ‘Axes X, or Z slaved or synchronized.’

Detection During execution.

Cause While programming in high level language, an attempt has been made to execute a probing cycle using the “PROBE” instruction, but one of the X or Z axis is slaved or synchronized.

Solution To execute the “PROBE”¨ instruction, the X-Z axes must not be slaved or synchronized. To unslave the axes, program “G78”.

0196 ‘Axes X and Z must exist.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, an attempt has been made to edit the “PROBE” instruction, but one of the X or Z axis is missing.

Solution To operate with the “PROBE” instruction, the X, Z axes must be defined.

0197 ‘G15 must be programmed before the “C” axis.’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to execute an operation on the “C” axis, but the axis is not active.

Solution In order to operate with the “C” axis, it must be activated first using the “G15” function.

0199 ‘Rotary axis preset: values between 0 and 359.9999.’

Detection While presetting coordinates.

Cause An attempt has been made preset the coordinates of a rotary axis with a value out of the 0 to 359.9999 range.

Solution The preset value of rotary axes must be within the 0 to 359.9999 range.

0200 ‘Program: G52 axis +/-5.5’

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause When programming the «Movement against a hard stop (G52)», either the axis to be moved has not been programmed or several axes have been programmed.

Solution When programming “G52”, the axis to be moved must be indicated. Only one axis may be programmed at a time.

0201 ‘Program only one positioning axis in G01.’

No explanation required

0206 'Values 0 thru 6.'

Detection While editing machine parameters

Cause An attempt has been made to assign the wrong value to a parameter.

Solution The parameter only admits values between 0 and 6.

0207 'Complete Table.'

Detection While editing tables.

Cause In the tables for «M» functions or tool offsets, an attempt has been made to define more data than those allowed by the manufacturer by means of machine parameters. When loading a table via DNC, the CNC does not delete the previous table, it replaces the existing values and it copies the new data in the free positions of the table.

Solution The maximum number of data that can be defined is limited by the machine parameters:

Maximum number of «M» functions	NMISCFUN(P29).
Maximum number of	NTOOL(P23).
Maximum number of tool offset	NTOFFSET(P27).
Maximum number of magazine positions	NPOCKET(P25).

To load a new table via DNC, the previous table should be deleted.

0208 'Program A from 0 to 255'

Detection During execution.

Cause In the «LOOK-AHEAD (G51)» function, parameter "A" (% of acceleration to be applied) has been programmed with a value greater than 255.

Solution Parameter "A" is optional, but when programmed, it must have a value between 0 and 255.

0209 'Program nesting not allowed.'

Detection During execution.

Cause From a running program, an attempt has been made to execute another program with the "EXEC" instruction which in turn also has an "EXEC" instruction.

Solution Another program cannot be called upon from a program being executed using the "EXEC" instruction.

0210 'No compensation is permitted.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to activate or cancel tool radius compensation (G41, G42, G40) in a block containing a nonlinear movement.

Solution Tool radius compensation must be activated/deactivated in linear movements (G00, G01).

0213 'A second spindle is required for G28, G29, G77 or G78.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause An attempt has been made to select the work spindle with "G28/G29" or synchronize spindles with "G77/G78", but the machine only has one work spindle.

Solution If the machine only has one work spindle, the "G28, G29, G77 and G78" functions cannot be programmed.

0214 'Invalid G function when selecting a profile'

Detection While restoring a profile.

Cause Within the group of blocks selected to restore the profile, there is a block containing a «G» code that does not belong in the profile definition.

Solution The «G» functions available in the profile definition are:

G00	G01	G02	G03	G06	G08	G09
G36	G37	G38	G39	G90	G91	G93

0215 'Invalid G function after first point of profile'

Detection While restoring a profile.

Cause Within the selected blocks for restoring the profile, and after the starting point of a profile, there is a block containing a «G» function that does not belong in the profile definition.

Solution The «G» functions available in the profile definition are:

G00	G01	G02	G03	G06	G08	G09
G36	G37	G38	G39	G90	G91	G93

0216 'Nonparametric assignment after first point of profile'

Detection While restoring a profile.

Cause Within the selected blocks for restoring the profile, and after the starting point of a profile, a nonparametric assignment has been programmed in high level language (a local or global parameter).

Solution The only high level instructions that can be edited are assignments to local parameters (P0 thru P25) and global parameters (P100 thru P299).

0217 'Invalid programming after first point of profile'

Detection While restoring a profile.

Cause Within the selected blocks for restoring the profile, and after the starting point of a profile, there is a high level block that is not an assignment.

Solution The only high level instructions that can be edited are assignments to local parameters (P0 thru P25) and global parameters (P100 thru P299).

0218 'The axis cannot be programmed after first point of profile'

Detection While restoring a profile.

Cause Within the selected blocks for restoring the profile, and after the starting point of a profile, a position has been defined on an axis that does not belong to the active plane. A surface coordinate may have been defined after the starting point of the profile.

Solution The surface coordinate of the profiles is only defined in the starting block of the first profile, the one corresponding to the starting point of the outside profile.

0219 'First point programmed wrong when selecting profile'

Detection While selecting a profile.

Cause The starting point of the profile has been programmed wrong. One of the two coordinates defining its position is missing.

Solution The starting point of a profile must be defined on the two axes forming the active plane.

0227 'Program Q between +/-359.9999.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Electronic threading (G33)» function, the entry angle "Q" has been programmed with a value out of the ±359.9999 range.

Solution Program an entry angle within the ±359.9999 range.

0228 'Do not program "Q" with parameter M19TYPE=0.'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause In the «Electronic threading (G33)» function, an entry angle "Q" has been programmed, but the type of spindle orientation available does not allow this operation.

Solution In order to define an entry angle, spindle machine parameter M19TYPE(P43) must be set to «1».

0229 'Program maximum Z'
0230 'Program inside R'
0231 'Program outside R'

Detection While editing at the CNC or while executing a program transmitted via DNC.

Cause While programming in high level language, in the "DGWZ" instruction, the indicated limit is missing or it has been defined with a non-numerical value.

Solution Check the syntax of the block.

0234 'Wrong graphic limits'

Detection During execution.

Cause One of the lower limits defined with the "DGWZ" instruction is greater than its corresponding upper limit.

Solution Program the upper limit of the graphics display area greater than the lower ones.

0235 'Do not program the axis in tangential control'

No explanation required

0236 'Do not program the longitudinal axis or the axis of the active plane'

No explanation required

0237 'Program values between +/-359.9999.'

Detection During execution.

Cause A G30 offset has been programmed greater than the maximum allowed. For example G30 D380

Solution The offset must be within ±359.9999.

0238 'Do not program G30 without synchronizing the spindles in speed'

Detection During execution.

Cause An attempt has been made to synchronize the spindles in "G30" offset without having them synchronized in speed.

Solution First, synchronize the spindle in speed using G77S.

0239 'Do not synchronize the spindles while the "C" axis is active'

Detection During execution.

Cause An attempt has been made to synchronize the spindle, but the "C" axis is not active.

Solution Activate the "C" axis first.

0240 'Do not activate the "C" axis while the spindles are synchronized'

Detection During execution.

Cause An attempt has been made to activate the "C" axis while the spindles were synchronized.

Solution First, cancel the spindle synchronization (G78 S).

0241 'Do not program G77 S, G78 S if there is no encoder at the spindle'

Detection During execution.

Cause An attempt has been made to synchronize the spindles (G77 S or G78 S) and one of them does not have an encoder or Sercos feedback.

Solution Both spindles must have an encoder or Sercos feedback.

0242 'Do not synchronize spindles with M19TYPE=0'

Detection During execution.

Cause An attempt has been made to synchronize the spindles (G77 S or G78 S) and one of them has parameter M19TYPE=0.

Solution Both spindles must have parameter M19TYPE=1

BLOCK PREPARATION AND EXECUTION ERRORS

1000 'There is no enough path information.'

Detection During execution.

Cause The program contains too many blocks without information about the path to apply tool radius compensation, rounding, chamfer or tangential entry or exit.

Solution In order to carry out these operations, the CNC needs to know in advance the path to follow; therefore, there cannot be more than 48 blocks in a row without information about the path to follow.

1001 'Plane change in rounding/chamfering.'

Detection During execution.

Cause A plane change has been programmed on the path following the definition of a "controlled corner rounding G36" or "chamfer (G39)".

Solution The plane cannot be changed while executing a rounding or a chamfer. The path following the definition of a rounding or chamfer must be in the same plane that the rounding or the chamfer.

1002 'Rounding radius too large.'

Detection During execution.

Cause In the "Controlled corner rounding (G36)" function, the programmed rounding radius is larger than one of the paths where it has been defined.

Solution The rounding radius must be smaller than the paths that define it.

1003 'Rounding in last block.'

Detection During execution.

Cause A "Controlled rounding radius (G36) or "Chamfer (G39) has been defined on the last path of the program or when the CNC does not find information about the path following the definition of the rounding or chamfer.

Solution A rounding or chamfer must be defined between two paths.

1004 'Tangential output programmed wrong'

Detection During execution.

Cause The move following the definition of a tangential output (G38) is a circular path.

Solution The move following the definition of a tangential output must be a straight path.

1005 'Chamfer programmed wrong.'

Detection During execution.

Cause The move following the definition of a "Chamfer (G39)" is a circular path.

Solution The move following the definition of a chamfer must be a straight path.

1006 'Chamfer value too large.'

Detection During execution.

Cause In the "Chamfer (G39)" function, the programmed chamfer value is larger than one of the paths where it has been defined.

Solution The chamfer size must be smaller than the paths that define it.

1007 'G8 defined wrong.'

Detection During execution.

Cause The various causes might be:

1. When a full circle has been programmed using the function "Arc tangent to previous path (G08)"
2. When the tangent path ends in a point of the previous path or its extension (in a straight line).
3. In an irregular pocket canned cycle with islands, when programming function "G08" in the block following the definition of the beginning of the profile (G00).

Solution The solution for each cause is:

1. Function “G08” does not allow programming full circles.
2. Tangent path must not end in a point of the previous path or in its extension (in a straight line).
3. The CNC does not have information about the previous path and cannot execute the tangent arc.

1008 ‘There is no information about the previous path’

Detection During execution.

Cause An arc tangent to the previous path has been programmed using function “G08”, but there is no information about the previous path.

Solution To do a path tangent to the previous one, there must be information about the previous path and it must be within the 48 blocks preceding the tangent path.

1010 ‘Wrong plane for tangent path.’

Detection During execution.

Cause A plane change has been programmed between the definition of the function “arc tangent to previous path (G08)” and the previous path.

Solution A plane cannot be changed between two paths

1011 ‘No radius has been programmed for G15.’

Detection During execution.

Cause The Z-C plane has been selected as a new work plane, but the radius of the cylinder to be machined has not been defined.

Solution In order to work in the Z-C plane, first define the radius of the cylinder on which to machine using function “G15 R—”.

1015 ‘The tool is not defined in the tool table’

Detection During execution.

Cause A tool change has been defined, but the new tool is not defined in the tool table.

Solution Define the new tool in the tool table.

1016 ‘The tool is not in the tool magazine’

Detection During execution.

Cause A tool change has been defined, but the new tool is not defined in position of the tool magazine table.

Solution Define the new tool in the tool magazine table.

1017 ‘There is no empty pocket in the tool magazine’

Detection During execution.

Cause A tool change has been defined and there is no empty pocket for the tool that is currently in the spindle.

Solution Perhaps, the new tool has been defined as special in the tool table and there are more than one pockets reserved to it in the magazine. In this case, that position is set for that tool and no other tool can occupy it. To avoid this error, an empty pocket (position) should be left in the tool magazine.

1018 ‘A tool change has been programmed without M06’

Detection During execution.

Cause An M06 has not been programmed after having looked for a tool and before searching again.

Solution This error occurs when having a machining center (general machine parameter TOFFM06(P28)=YES) that has a cyclic tool changer (general machine parameter CYCATC(P61)=YES). In this case, the tool change must be done with an m06 after searching for a tool and before searching for the next one.

1019 ‘There is no tool of the same family for replacement.’

Detection During execution.

Cause The real life of the requested tool exceeds its nominal life. The CNC has tried to replace it with another one of the same family, but it has not found any.

Solution Replace the tool or define another one of the same family.

1020 'Do not change the active or pending tool using high level language.'

Detection During execution.

Cause While programming in high level language and using the "TMZT" variable, an attempt has been made to assign the current or next tool to a magazine position.

Solution Use the "T" function to change the active tool or the next one. The "TMZT" variable cannot be used to move the active tool or the next one to the magazine.

1021 'No tool offset has been programmed in the canned cycle.'

Detection During execution.

Cause The "PROBE" canned cycle for tool calibration has been programmed, but no tool offset has been selected.

Solution To execute the "Tool calibration canned cycle (PROBE), a tool offset must be selected where the probing cycle information will be stored.

1022 'Tool radius programmed incorrectly'

No explanation required

1028 'Do not switch axes over or back while G15 is active'

Detection During execution.

Cause An attempt has been made to switch over to an axis or back (G28/G29) while function "G15" was active.

1029 'Do not swap axes that are already swapped.'

Detection During execution.

Cause An attempt has been made to swap (G28) an axis that was already swapped with another one.

Solution An axis already swapped with another one cannot be swapped with a third one. It must be switched back first (G29 axis)

1030 'The "M" for the automatic gear change does not fit'

Detection During execution.

Cause Using automatic gear change, 7 "M" functions and the "S" function (involving a gear change) have been programmed. In this case, the CNC cannot include the "M" for automatic gear change in that block.

Solution Program an "M" function or the "S" function in a separate block.

1031 'No subroutine is allowed with automatic gear change.'

Detection During execution.

Cause On machines having an automatic gear change, when programming a spindle speed "S" that involves a gear change and the "M" function of the automatic gear change has a subroutine associated with it.

Solution When having an automatic gear change, the "M" functions corresponding to the gear change cannot have a subroutine associated with it.

1032 'Spindle gear (range) not defined in M19.'

Detection During execution.

Cause "M19" has been programmed, but none of the gear change functions "M41", "M42", "M43" or "M44" are active.

Solution On power-up, the CNC does not assume any range; Therefore, if the gear change function is not generated automatically (spindle parameter AUTOGEAR(P6)=NO), the auxiliary gear change functions ("M41", "M42", "M43" or "M44") must be programmed.

1033 'Wrong gear change.'

Detection During execution.

Cause The various causes might be:

1. When trying to make a gear change and the machine parameters for gears (MAXGEAR1, MAXGEAR2, MAXGEAR3, or MAXGEAR4) are set wrong. All

the gears (ranges) have not been used and the unused ones have been set to a maximum speed of zero rpm.

2. When programming a gear change (“M41”, “M42”, “M43” or “M44”) and the PLC has not responded with the relevant active gear signal (GEAR1, GEAR2, GEAR3 or GEAR4).

Solution The solution for each cause is:

1. When not using all four gears, the lower ones must be used starting with “MAXGEAR1” and the unused gears must be assigned the value of the highest one used.
2. Check the PLC program.

1034 ‘”S” has been programmed, but no gear is active.’

Detection During execution.

Cause An attempt has been made to start the spindle, but no gear is selected.

Solution On power-up, the CNC does not assume any range; Therefore, when programing a spindle speed and the gear change function is not generated automatically (spindle parameter AUTOGEAR(P6)=NO), the auxiliary gear change functions (“M41”, “M42”, “M43” or “M44”) must be programmed.

1035 ‘Programmed “S” too high’

Detection During execution.

Cause An “S” has been programmed with a higher value than allowed by the last active gear.

Solution Program a lower spindle speed “S” .

1036 ‘”S” has not been programmed in G95 or in threading’

Detection During execution.

Cause “mm(inches)/revolution (G95)” or “electronic threading (G33)”has been programmed, but no spindle speed has been selected.

Solution An “S” must be programmed to work in mm/rev (G95) or for an electronic threading (G33).

1037 ‘No “S” has been programmed for G96.’

Detection During execution.

Cause The “Constant Surface Speed (G96)” function has been programmed, but no cutting speed has been defined, a previous one does not exist or no spindle range (gear) is selected.

Solution In order to work at constant surface speed (G96), a cutting speed “S” must be already programmed and a spindle range must be active.

1038 ‘The spindle has not been oriented’

Detection During execution.

Cause An attempt has been made to execute a threading cycle (G86 or G87) as a thread repair without already having oriented the active spindle (main or secondary).

1039 ‘No “F” has been programmed in G94.’

Detection During execution.

Cause An attempt has been made to execute a live tool cycle (G60, G61, G62 and G63) and there is feedrate selected in G94 (mm/min).

Solution First, select the feedrate “F” in mm/min (G94).

1040 ‘Canned cycle does not exist’

Detection While executing in MDI mode

Cause When trying to execute a canned cycle (G8x) after interrupting a program during the execution of a canned cycle (G8x) and then changing the plane.

Solution Do not interrupt the program while executing a canned cycle.

1042 ‘Wrong parameter value in canned cycle’

Detection During execution.

Cause When defining a canned cycle, a parameter has been defined with the wrong value. Perhaps, a parameter that only takes positive values has been assigned a negative value (or zero).

Solution Correct the definition of parameters:

- In the “Pattern repeat cycle”:
  - Parameter “C” only takes positive values greater than zero.
  - Parameter “A” only admits values of 0 or 1.
  - Parameter “J” only takes positive values greater than zero.
- In the canned cycle for “roughing along the Z axis” or “roughing along the X axis”, parameter “C” only takes positive values greater than zero.
- In the “ Axial drilling/tapping canned cycle”:
  - Parameter “I” only admits values other than zero.
  - Parameter “B” only takes positive values or zero.
- In the canned cycle for “facing curved sections” or “turning curved sections”, parameter “C” only takes positive values greater than zero.
- In the “Face threading canned cycle” or “Longitudinal threading canned cycle”, parameter “I”, “B”, “E” or “C” has been defined with a zero value.
- In the canned cycle for “grooving along the Z axis” or “grooving along the X axis”, parameter “C” only takes positive values greater than zero.
- In the canned cycle for “Face drilling/tapping” or “Longitudinal drilling/tapping”:
  - Parameter “I” only admits values other than zero.
  - Parameter “B” only takes positive values or zero.
  - Parameter “J” only takes positive values greater than zero.
- In the canned cycle for “slot milling on the face” or “slot milling on the side”, the slot dimension cannot be zero and parameters “I” and “J” only take positive values greater than zero.

1043 ‘Wrong tool for programmed profile.’

Detection During execution.

Cause The selected tool cannot machine anywhere on the profile.

Solution Choose a more appropriate tool to machine the profile.

1044 ‘A profile has been programmed that intersects itself.’

Detection During execution.

Cause In the set of profiles, there is one that intersects itself.

Solution Check the definition of the profiles. A profile cannot intersect itself.

1045 ‘Wrong cutter geometry angle.’

Detection During execution.

Cause The cutter’s geometry angles have been assigned a wrong value.

Solution Correct the tool geometry data.

1046 ‘Wrong tool position before the canned cycle’

Detection During execution.

Cause The canned cycle calling point is defined wrong.

Solution The canned cycle calling point must be off the part and at a distance greater than the one defined as finishing stock on both axes. (Cycles that do not have a finishing stock will use the safety distance).

1047 ‘Wrong location code in canned cycle’

Detection During execution.

Cause The location code (shape) of the tool is not the right one.

Solution Choose a tool with the right location code (shape).

1048 ‘Wrong cutter width’

Detection During execution.

Cause A grooving operation has been defined with a cutter of zero width.

Solution Check the definition of the cutter dimensions (NOSEW). The cutter width must be other than zero.

1049 'Incompatible tool position and location code in profile cycle.

Detection During execution.

Cause The canned cycle calling point is defined wrong or the tool location code (shape) is the right one to execute the machining operation.

Solution The canned cycle calling point must be off the part and at a distance greater than the one defined as finishing stock on both axes. Besides, the tool's location code must allow executing the profile without running into the part.

1050 'Wrong value to be assigned to a variable'

Detection During execution.

Cause Using parameters, the value assigned to a variable is too high.

Solution Check the program history to make sure that this parameter does not have that value when it reaches the block where this assignment is made.

1051 'Wrong access to PLC variable.'

Detection During execution.

Cause From the CNC, an attempt has been made to read a PLC variable that is not defined in the PLC program.

1052 'Access to a variable with wrong index'

Detection During editing

Cause While programming in high level language, an operation has been carried out either with a local parameter greater than 25 or with a global parameter greater 299.

Solution The parameters used by the CNC are:

Local: 0-25.
Global: 100-299.

Other parameters out of these ranges cannot be used in operations.

1053 'Local parameters not accessible'

Detection While executing in the user channel.

Cause An attempt has been made to execute a block with an operation that uses local parameters.

Solution The program that is executed in the user channel does not allow operations with local parameters (P0 to P25).

1054 'Limit of local parameters exceeded'

Detection During execution.

Cause While programming in high level language, more than 6 nesting levels have been used with the "PCALL" instruction. More than 6 calls have been made in the same loop using the "PCALL" instruction.

Solution Only up to 6 nesting levels are allowed for local parameters within the 15 nesting levels of the subroutines. Calling with a "PCALL" instruction generates a new nesting level for local parameters (and a new one for subroutines).

1055 'Nesting exceeded'

Detection During execution.

Cause While programming in high level language, more than 15 nesting levels have been used with the "CALL", "PCALL" or "MCALL" instruction. More than 15 calls have been made in the same loop using the "CALL", "PCALL" or "MCALL" instruction.

Solution Only 15 nesting levels allowed. Calling with the "CALL", "PCALL" and "MCALL" instructions generates a new nesting level.

1056 'RET not associated with subroutine.'

Detection During execution.

Cause The "RET" instruction has been edited, but the "SUB" instruction has not been edited before.

Solution To using the "RET" instruction (subroutine), the subroutine must begin with the "SUB (subroutine number)".

1057 'Undefined subroutine'

Detection During execution.

Cause A (CALL, PCALL…) has been made to a subroutine that was not defined in the CNC memory.

Solution Check that the name of the subroutine is correct and that the subroutine exists in the CNC memory (not necessarily in the same program where the call is).

1058 'Undefined probing canned cycle'

Detection During execution.

Cause Using the "PROBE" instruction, a probing cycle has been defined which is not available.

Solution The available "PROBE" canned cycles are 1 to 4.

1059 'Jump to an undefined label'

Detection During execution.

Cause While programming in high level language, the "GOTO N—" instruction has been programmed, but the programmed block number (N) does not exist.

Solution When programming the "GOTO N—" instruction, the block it refers to must be defined in the same program.

1060 'Undefined label'

Detection During execution.

Cause The various causes might be:

1. While programming in high level language, the instrucción "RPT N—, N—" instruction has been programmed, but a programmed block number (N) does not exist.
2. In the "Pattern repeat canned cycle (G66)", "Roughing canned cycle along the X axis (G68)" or "Roughing canned cycle along the Z axis (G69)" a profile definition has been programmed, but one of the two data defining the beginning "S" or end "E" of the profiles is missing.

Solution The solution for each cause is:

1. When programming the "RPT N—, N—" instruction, the blocks it refers to must be defined in the same program.
2. Check the program. Place the label for parameter "S" at the beginning of the profile definition and the label for parameter "E" at the end of the profile definition.

1061 'Label cannot be searched'

Detection While executing in MDI mode

Cause While programming in high level language, either an "RPT N—, N—" or "GOTO N— " instruction has been defined.

Solution While operating in MDI mode, "RPT" or "GOTO" type instructions cannot be programmed.

1062 'Subroutine in an unavailable program.'

Detection During execution.

Cause A call has been made to a subroutine that it is located in a program being used by the DNC.

Solution Wait for the DNC to finish using the program. If the subroutine is to be used often, it should be stored in a separate program.

1063 'The program cannot be opened.'

Detection During execution.

Cause While executing a program in infinite mode, an attempt has been made to execute another infinite program from the current one using the "EXEC" instruction.

Solution Only one infinite program may be executed at a time.

1064 'The program cannot be executed'

Detection During execution.

Cause An attempt has been made to execute a program from another with the “EXEC” instruction, but the program does not exist or is protected against execution.

Solution The program to be executed with the “EXEC” instruction must exist in the CNC memory and must be executable.

1065 ‘Beginning of compensation without straight path’

Detection During execution.

Cause The first movement in work plane after activating tool radius compensation (G41/G42) is not a linear movement.

Solution The first movement after activating radius compensation (G41/G42) must be linear.

1066 ‘End of compensation without straight path’

Detection During execution.

Cause The first movement in work plane after deactivating tool radius compensation (G40) is not a linear movement.

Solution The first movement after deactivating radius compensation (G40) must be linear.

1067 ‘Compensation radius too large.’

Detection During execution.

Cause While working with tool radius compensation (G41/G42), an inside radius has been programmed with a smaller radius than that of the tool.

Solution use a tool with a smaller radius. When working with tool radius compensation, the arc radius must larger than that of the tool. Otherwise, the tool cannot machine the programmed path.

1068 ‘Step on linear path’

Detection During execution.

Cause When operating with tool compensation (G41/G42), the profile has a straight section that cannot be machined because the tool diameter is too large.

Solution use a tool with a smaller radius.

1069 ‘Circular path defined incorrectly’

No explanation required

1070 ‘Step on circular path’

Detection During execution.

Cause When operating with tool compensation (G41/G42), the profile has a curved section that cannot be machined because the tool diameter is too large.

Solution use a tool with a smaller radius.

1071 ‘Plane change in tool radius compensation.’

Detection During execution.

Cause When operating with tool compensation (G41/G42), another work plane has been selected.

Solution To change the work plane, tool radius compensation must be off (G40).

1072 ‘Tool radius compensation not possible with positioning-only rotary axis.

Detection During execution.

Cause An attempt has been made to move a positioning-only axis with tool radius compensation (G41/G42).

Solution Tool radius compensation not allowed for positioning-only rotary axes. Use “G40” to cancel tool radius compensation.

1076 ‘Coordinate angle programmed wrong.’

Detection During execution.

Cause When programming in angle-coordinate format, an axis movement has been programmed with an angle perpendicular to that axis. (For example, the main plane is formed by the XZ axes and the X axis movement is programmed at a 90º angle).

Solution Check and correct the definition of the movement in the program. If using parameters, check that the parameters have the correct values when arriving to the definition of the movement.

1077 'Either the arc radius is too small or a full circle has been programmed'

Detection During execution.

Cause The various causes might be:

1. When programming a full circle using the "G02/G03 X Z R" format.
2. When programming using the "G02/G03 X Z R" format, the distance to the arc's end point is greater than the diameter of the programmed circle.

Solution The solution for each cause is:

1. This format cannot be used to make full circles. Program the coordinates of the end point different from those of the starting point.
2. The diameter of the circle must be greater than the distance to the arc's end point.

1078 'Negative radius in polar coordinates'

Detection During execution.

Cause Working with incremental polar coordinates, a block is executed where the end position has a negative radius.

Solution Incremental polar coordinate programming allows negative radius, but the (absolute) end point of the radius must be positive,

G74 'There is no subroutine associated with G74'

Detection While executing a home search.

Cause The various causes might be:

1. When trying to search home on all the axes manually, but there is no associated subroutine indicating the home searching sequence (order).
2. "G74" has been programmed, but there is no associated subroutine indicating the home searching sequence (order).

Solution The solution for each cause is:

1. An associated subroutine is required to execute the "G74" function.
2. If "G74" is to be executed from a program, the home searching order must be defined.

1080 'Plane change in tool inspection'

Detection While executing the "tool inspection" option.

Cause the work plane has been chanted and the original one has not been restored before resuming the execution.

Solution The plane that was active before inspecting the tool must be restored before resuming the execution.

1081 'Block not allowed in tool inspection.'

Detection While executing the "tool inspection" option.

Cause An attempt has been made to execute the "RET" instruction.

Solution This instruction cannot be executed in the "tool inspection" option.

1082 'The probe signal has not been received.'

Detection During execution.

Cause The various causes might be:

1. When programming a "PROBE" canned cycle, the probe has moved the maximum safety distance of the cycle without the CNC receiving the probe signal.
2. When programming the "G75" function, it has reached the end point and the CNC has not received the signal from the probe. (Only when general machine parameter PROBERR(P119)=YES).

Solution The solution for each cause is:

1. Check that the probe is connected properly.

   The maximum probing distance (in PROBE cycles) depends on the safety distance "B". To increase this distance, increase the safety distance.

2. If PROBERR(P119)=NO, this error will not be issued when the end point is reached without having received the probe signal (only with “G75”).

1083 ‘Range exceeded’

Detection During execution.

Cause The distance for the axes to travel is very long and the programmed feedrate is too low.

Solution Program a higher speed for that movement.

1084 ‘Arc programmed wrong’

Detection During execution.

Cause The various causes might be:

1. When the arc programmed using “G02/G03 X Y I J” cannot go through the defined end point.
2. When programming an arc using “G09 X Y I J” the three points are in line or two of them are the same.
3. When trying to do a rounding tangential entry on a path that is not in the active plane.
4. When programming a tangential exit and the next path is tangent (being on its straight extension) to the path preceding the tangential exit.

   If the error comes up in the block calling upon the “Pattern repeat canned cycle (G66)”, “Roughing canned cycle along X (G68)” or “Roughing canned cycle along Z (G69)” is because in the set of blocks that define the profiles, one of the cases mentioned earlier occurs.

Solution The solution for each cause is:

1. Correct the syntax of the block. The coordinates of the end point or of the radius are defined wrong.
2. The three points used to define an arc must be different and cannot be in line.
3. Maybe a plane has been defined with “G16”, “G17”, “G18” or “G19”. In this case, corner rounding, chamfers and tangential entries/exits can only be carried out on the main axes defining that plane. To do it in another plane, it must be defined beforehand.
4. The path after a tangential exit may be tangent, but it cannot be on the extension (in a straight line) of the previous path.

1085 ‘Helical path programmed wrong’

Detection During execution.

Cause When programming an arc using “G02/G03 X Y I J Z K”, the programmed arc is impossible. The desired height cannot be reached with the programmed helical pitch.

Solution Correct the syntax of the block. The height of the interpolation and the coordinates of the end point in the plane must be related taking the helical pitch into account.

1086 ‘The spindle cannot be homed.’

Cause Spindle machine parameter REFEED1(P34) = 0.

1087 ‘Circle with zero radius’

Detection During execution.

Cause The various causes might be:

1. When programming an arc using “G02/G03 X Z I K”, an arc has been programmed with a zero radius.
2. When operating with tool radius compensation, an inside arc has been programmed with the same radius as that of the tool.

Solution The solution for each cause is:

1. Arcs with zero radius are not allowed. Program a radius other than zero.
2. When working with tool radius compensation, the arc radius must larger than that of the tool. Otherwise, the tool cannot machine the programmed path (because to do so, the tool would have to make an arc of zero radius).

1088 ‘Range exceeded in zero offset.’

Detection During execution.

Cause A zero offset has been programmed and the value of the end position is too high.

Solution Check that the values assigned to the zero offsets (G54-G59) are correct. If the zero offsets have been assigned values from the program using parameters, check that the parameter values are correct. If an absolute (G54-G57) and an incremental (G58-G59) zero offset has been programmed, check that the sum of both does not exceed the machine limits.

1089 'Range exceeded in zone limit.'

Detection During execution.

Cause When programming zone limits "G20" or "G21" with parameters, the parameter value is greater than the maximum allowed for that function

Solution Check the program history to make sure that this parameter does not have that value when it reaches the block where the limits have been defined.

1090 'Point inside the forbidden zone 1.'

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 1 that is defined as "no entry" zone.

Solution In the program history, work zone 1 (defined with G20/G21) has been set as "no entry" zone " (G22 K1 S1). To cancel this work zone, program "G22 K1 S0"

1091 'Point inside the forbidden zone 2.'

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 2 that is defined as "no entry" zone.

Solution In the program history, work zone 2 (defined with G20/G21) has been set as "no entry" zone " (G22 K1 S1). To cancel this work zone, program "G22 K2 S0"

1092 'Insufficient acceleration for the speed programmed in threading.'

Detection During execution.

Cause A thread has been programmed and there isn't enough room to accelerate and decelerate.

Solution Program a lower speed.

1093 'Only one Hirth axis can be moved at a time'

No explanation required

1094 'Probe calibrated wrong'

No explanation required

1095 'Probing axes out of alignment .'

Detection During the probe calibration process

Cause An axis has moved to touch a cube and one of the axis that did not move registers a deflection greater than allowed by machine parameter MINDEFLE(P66). This is because the probing axes are not parallel enough to the axes of the machine.

Solution Correct the parallelism between the probing axes and those of the machine.

1096 'Point inside the forbidden zone 3.'

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 3 that is defined as "no entry" zone.

Solution In the program history, work zone 3 (defined with G20/G21) has been set as "no entry" zone " (G22 K3 S1). To cancel this work zone, program "G22 K3 S0"

1097 'Point inside the forbidden zone 4.'

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 4 that is defined as "no entry" zone.

Solution In the program history, work zone 4 (defined with G20/G21) has been set as “no entry” zone ” (G22 K4 S1). To cancel this work zone, program “G22 K4 S0”

1098 ‘Work zone limits defined wrong’

Detection During execution.

Cause The upper limits (G21) of the defined work zone are the same or smaller than the lower ones (G20) of the same work zone.

Solution Program the upper limits (G21) of the work zone greater than the lower ones (G20).

1099 ‘Do not program a slaved axis.’

Detection During execution.

Cause When operating in polar coordinates, a movement has been programmed that involves an axis that is slaved to another one.

Solution The movements in polar coordinates are made with the main axes of the work plane; therefore, the axes that define the plane cannot be slaved to each other or to a third one. To unslave the axes, program “G78”.

1100 ‘Travel limits of spindle 1 exceeded’

Detection During execution.

Cause An attempt has been made to exceed the physical turning limits of the spindle. As a result, the PLC activates the spindle mark “LIMIT+S” or “LIMIT-S”. (“LIMIT+S2” or “LIMIT-S2” when working with the second spindle).

1101 ‘Spindle 1 locked’

Detection During execution.

Cause The CNC tries to output the command to the drive when the spindle input SERVOSON is still low. The error may be due to an error in the PLC program where this signal is not properly treated or that the value of the spindle parameter DWELL(P17) is not high enough.

1102 ‘Following error of spindle 1 out of limit’

Detection During execution.

Cause When the spindle is working in closed loop (M19), its following error is greater than the values indicated by spindle parameter MAXFLWE1(P21) and MAXFLE2(P22) The possible causes for this error are:

Servo drive error
- Faulty drive.
- Enable signals missing.
- Power supply missing.
- Drive adjusted incorrectly.
- The velocity command signal is not received.

Motor error
- Faulty motor.
- Power cables.

Feedback failure
- Defective feedback.
- Defective feedback cable.

Mechanical failure
- Mechanical stiffness.
- Spindle mechanically locked.

CNC error
- Defective CNC.
- Parameters adjusted incorrectly.

1103 ‘Do not synchronize spindles without homing them first’

Detection During execution.

Cause An attempt has been made to synchronize the spindle without homing them first.

Solution Before activating the synchronization, both spindles must be homed using the “M19” function.

1104 ‘ Do not program G28 or G29 while spindle synchronization is active’

Detection During execution.

Cause An attempt has been made to swap spindles (G28/G29) while the spindles were synchronized.

Solution First, cancel spindle synchronization (G78S).

1105 ‘Do not change the ranges (gears) while the spindle are synchronized’

Detection During execution.

Cause While the spindles are synchronized, a gear changing “M” function (M41 to M44) has been executed or the programmed “S” involves a gear change (with automatic gear changer).

Solution First, cancel spindle synchronization (G78S).

1106 ‘Travel limits of spindle 2 exceeded’

Same as error 1000, but for the second spindle.

1107 ‘Spindle 2 locked’

Same as error 1001, but for the second spindle.

1108 ‘Following error of spindle 2 out of limit’

Same as error 1002, but for the second spindle.

1109 ‘Axis software limit overrun’

No explanation required

1110-1118 ‘Range of the * axis exceeded’

Detection During execution.

Cause A movement has been defined with parameters and the parameter value is greater than the maximum travel distance of the axis.

Solution Check the program history to make sure that this parameter does not have that value when it reaches the block where this movement is programmed.

1119-1127 ‘The * axis cannot be synchronized’

Detection During execution.

Cause The various causes might be:

1. When trying to synchronize two axes from the PLC and one axis is already slaved to another one using the “G77” function.
2. When programming or trying to move an axis that is slaved to another one.

1128-1136 ‘Maximum feedrate of the * axis exceeded’

Detection During execution.

Cause The resulting feedrate of one of the axes after applying an individual scaling factor exceeds the maximum value indicated by axis machine parameter MAXFEED (P42).

1137-1145 ‘Wrong feedrate parameter of the * axis’

Detection During execution.

Cause “G00” programmed with parameter G00FEED(P38)=0 or “G1 F00” with axis parameter MAXFEED(P42) = 0.

1146-1154 ‘* axis locked’

Detection During execution.

Cause The CNC tries to output the command to the drive when the spindle input SERVO(n)ON is still low. The error may be due to an error in the PLC program where this signal is not properly treated or that the value of the axis parameter DWELL(P17) is not high enough.

1155-1163 ‘Maximum axis limits of the * axis exceeded’

Detection During execution.

Cause A coordinate has been programmed that is out of the limits defined by axis parameters LIMIT+(P5) and LIMIT-(P6).

1164-1172 'Work zone 1 of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 1 that is defined as "no exit" zone.

Solution In the program history, work zone 1 (defined with G20/G21) has been set as "no exit" zone " (G22 K1 S2). To cancel this work zone, program "G22 K1 S0"

1173-1181 'Work zone 2 of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 2 that is defined as "no exit" zone.

Solution In the program history, work zone 2 (defined with G20/G21) has been set as "no exit" zone " (G22 K2 S2). To cancel this work zone, program "G22 K2 S0"

1182-1190 'Following error of * axis out of limit'

Detection During execution.

Cause The following error of the axis is greater than the values indicated by axis parameter MAXFLWE1(P21) or maxflwe2(P22). The possible causes for this error are:

Servo drive error
- Faulty drive.
- Enable signals missing.
- Power supply missing.
- Drive adjusted incorrectly.
- The velocity command signal is not received.

Motor error
- Faulty motor.
- Power cables.

Feedback failure
- Defective feedback.
- Defective feedback cable.

Mechanical failure
- Mechanical stiffness.
- Spindle mechanically locked.

CNC error
- Defective CNC.
- Parameters adjusted incorrectly.

1191-1199 'Difference of following errors of the slaved axis * tool large '

Cause The "n" axis is electronically coupled to another one or is a slaved Gantry axis and the difference between the following errors of the "n" axis and the one it is coupled to is greater than the value set by the machine parameter for the "n" axis MAXCOUPE(P45).

1200-1208 'Travel limits of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to exceed the physical travel limits. As a result, the PLC activates the axis mark "LIMIT+1" or "LIMIT-1".

1209-1217 '* axis servo error'

Cause The real feedrate of the axis, after the time indicated by axis parameter FBALTIME(P12), is below 50% or over 200% of the one programmed.

1218-1226 'Work zone 3 of the * axis exceeded'

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 3 that is defined as "no exit" zone.

Solution In the program history, work zone 3 (defined with G20/G21) has been set as "no exit" zone " (G22 K3 S2). To cancel this work zone, program "G22 K3 S0"

1228-1236 ‘Work zone 4 of the * axis exceeded’

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 4 that is defined as “no exit” zone.

Solution In the program history, work zone 4 (defined with G20/G21) has been set as “no exit” zone ” (G22 K4 S2). To cancel this work zone, program “G22 K4 S0”

1237 ‘Do not change the entry angle inside a thread’

Detection During execution.

Cause A thread joint has been defined and an entry angle “Q” has been programmed between two threads.

Solution When joining threads, only the first one may have an entry angle “Q”.

1238 ‘Range of write-protected parameters. P297, P298’

Detection During execution.

Cause Parameters P297 and P298 are write-protected by means of machine parameters ROPARMIN(P51) and ROPARMAX(P52).

1239 ‘Point inside the forbidden zone 5.’

Detection During execution.

Cause An attempt has been made to move an axis to a point located inside the work area 5 that is defined as “no entry” zone.

Solution In the program history, work zone 5 (defined with G20/G21) has been set as “no entry” zone ” (G22 K5 S1). To cancel this work zone, program “G22 K5 S0”

1240-1248 ‘Work zone 5 of the * axis exceeded’

Detection During execution.

Cause An attempt has been made to move an axis to a point located out of the work area 5 that is defined as “no exit” zone.

Solution In the program history, work zone 5 (defined with G20/G21) has been set as “no exit” zone ” (G22 K5 S2). To cancel this work zone, program “G22 K5 S0”

1249 ‘Variable pitch thread programmed wrong’

Detection During execution.

Cause We are trying to make a variable-pitch thread with the following conditions:

- The “K” increment is positive and equal to or greater than 2L.
- The “K” increment is positive and with one of the calculated pitches, it exceeds the maximum feedrate (parameter MAXFEED) of one of the threading axis.
- The “K” increment is negative and one of the calculated pitches 0 or negative.

1250 ‘The K value is too large in G34’

Detection During execution.

Cause The ratio between the initial and final pitches of the variable-pitch thread (G34) to be executed

is greater than 32767.

1251 ‘Two variable-pitch threads cannot be joined in round corner’

Detection During motionless simulation, except when graphics are active

Cause To variable-pitch threads cannot be joined in round corner unless the second one is of the type: G34 … L0 K0.

G34 ‘G34 without a pitch is only allowed after a variable-pitch thread.

Detection During motionless simulation, except when graphics are active

Cause G34 L0 cannot be programmed after a movement, no G34, or in square corner.

1253 ‘Retrace function unavailable’

No explanation required

HARDWARE ERRORS

2000 'External emergency activated.'

Detection During execution.

Cause PLC input I1 is set to "0" (maybe the E-stop button) or the PLC mark M5000(/EMERGEN) is set to "0".

Solution Check at the PLC why the inputs are at "0". (Possible lack of power).

2001-2009 '* axis feedback error'

Detection During execution.

Cause The CNC does not receive feedback signal from the axes.

Solution Check that the connections are properly made.

NOTE: This error comes up on differential axes DIFFBACK(P9) =YES and sinusoidal axes SINMAGNI(P10) other than 0 when parameter FBACKAL(P11)=ON Setting parameter FBACKAL(P11)=OFF avoids this error, but this is only temporary solution.

2010 'Spindle feedback error'

Detection During execution.

Cause The CNC does not receive feedback signal from the spindle.

Solution Check that the connections are properly made.

NOTE: This error comes up on differential axes DIFFBACK(P14)=YES when parameter FBACKAL(P15)=ON. Setting parameter FBACKAL(P15)=OFF avoids this error, but this is only temporary solution.

2011 'Maximum temperature exceeded'

Detection Any time.

Cause The CNC's internal temperature has been exceeded. The causes may be:

- Electrical cabinet poorly ventilated.
- Axis board with some defective component.

Solution Turn the CNC and wait until it cools off. If the error persists, a component of the board may be defective. In that case, replace the board. Contact the Service Department.

2012 'There is no voltage at the axis board'

Detection During execution.

Cause 24V are missing at the output supply of the axis board. The fuse may be blown.

Solution Power the outputs of the axis board (24v). If the fuse is blown, replace it.

2013 'There is no voltage at the I/O 1 board.'
2014 'There is no voltage at the I/O 2 board.'
2015 'There is no voltage at the I/O 3 board.'

Detection During execution.

Cause 24V are missing at the output supply of the corresponding I/O board. The fuse may be blown.

Solution Power the outputs of the corresponding I/O board (24v). If the fuse is blown, replace it.

2016 'The PLC is not ready.'

Detection During execution.

Cause The PLC program is not running. These may be the probable causes:

- The PLC program is missing.
- WATCHDOG error.
- The program has been interrupted from monitoring.

Solution Start the PLC program. (Restart the PLC).

2017 ‘CNC RAM memory error.’

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the CNC’s RAM memory.
Solution Replace the CPU board. Contact the Service Department.

2018 ‘CNC’s EPROM memory error.’

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the CNC’s EPROM memory.
Solution Replace the EPROM. Contact the Service Department.

2019 ‘PLC’s RAM memory error.’

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the PLC’s RAM memory.
Solution Replace the PLC board. Contact the Service Department.

2020 ‘PLC’s EPROM memory error.’

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the PLC’s EPROM memory.
Solution Replace the EPROM. Contact the Service Department.

2021 ‘CNC’s user RAM memory error.’ Press any key.’

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the CNC’s user RAM memory.
Solution Contact the Service Department.

2022 ‘CNC’s system RAM memory error.’ Press any key.’

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the CNC’s system RAM memory.
Solution Contact the Service Department.

2023 ‘PLC’s RAM memory error.’ Press any key.’

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the PLC’s RAM memory.
Solution Contact the Service Department.

2026 ‘Probe’s maximum travel limit overrun.’

Detection During execution.
Cause The probe has exceeded the maximum deflection allowed by machine parameter.
Solution Decrease the feedrate and check that the probe has not been damaged.

2027 ‘SERCOS chip RAM memory error.’ Press any key.’

Detection While starting the CNC or during diagnoses.
Cause A defect has been found in the SERCOS chip RAM memory.
Solution Replace the SERCOS board. Contact the Service Department.

2028 ‘SERCOS chip version error.’ Press any key.’

Detection During CNC start-up.
Cause The SERCOS chip version is old.
Solution Replace the SERCOS chip. Contact the Service Department.

PLC ERRORS

3001 ' (PLC_ERR without description) '

Detection During execution.

Cause Marks ERR1 to ERR64 have been set to "1".

Solution Check at the PLC why these marks are set to "1" and act accordingly.

3002 'WATCHDOG in the main module (PRG).'

Detection Any time.

Cause The various causes might be:

1. The execution of the PLC's main program has exceeded the time set in PLC parameter WAGPRG(P0).
2. The program is in an endless loop.

Solution Increase the time of PLC parameter WAGPRG(P0) or increase the PLC speed.

- Insert CPU TURBO.
- Change PLC parameter CPUTIME(P26) or general parameter LOOPTIME(P72).

3003 'WATCHDOG in the periodic module (PE).'

Detection Any time.

Cause The various causes might be:

1. The execution of the PLC's periodic program has exceeded the time set in PLC parameter WAGPER(P1).
2. The program is in an endless loop.

Solution Increase the time of PLC parameter WAGPER(P1) or increase the PLC speed.

- Insert CPU TURBO.
- Change PLC parameter CPUTIME(P26) or general parameter LOOPTIME(P72).

3004 'Division by zero at the PLC'

Detection Any time.

Cause In the PLC program, there is a line whose execution implies a division by zero.

Solution When working with registers, that register may have already acquired a zero value. Check that the register does not reach the operation with that value.

3005 'PLC error -> '

Detection Any time.

Cause An error has been detected on the PLC board.

Solution Replace the PLC board. Contact the Service Department.

SERVO ERRORS

4000 'Sercos ring error'

Detection During execution.

Cause SERCOS communication has been interrupted. It may be caused by an interruption in the connection ring (optical fiber disconnected or broken) or by a wrong configuration.

1. The identifying wheel does not match the sercosid.
2. Parameter P120 (SERSPD) does not match the transmission speed.
3. The drive version is incompatible with the CNC.
4. There is an error on the SERCOS board.
5. Different transmission speed (baudrate) at the drive and at the CNC.

A drive has been turned off and back on due to a power supply failure. When starting up again, it displays the error ***4027 'The drive has started up again'***

An attempt has been made to read or write an non-existent variable or too many variables in a drive through the fast channel.

Solution To check that the connection ring is not interrupted, check that the light goes through the optical fiber. If it is due to a wrong configuration, contact the Service Department.

If the error is due to the fast channel

- Check that all the variables to be read or written through the fast channel actually exist
- Save the SERCOS LOG into a file and see which axis causes the error.
- Set PLC machine parameters "SRD700 and SWR800" of that drive to "0".
- Reset the CNC and verify that no errors come up.
- Set the parameters one by one to the desired value until the failure occurs.
- When locating the parameter, look that variable up in the drive manual to verify that it exists in that version and it may be accessed. If so, the error may come up because it tries read or write too many variables in that drive.

4001 'Undefined class 1 error'

Detection During execution.

Cause The drive has detected an error, but it cannot identify it.

Solution Contact the Service Department.

4002 'Overload (201...203)'
4003 'Overtemperature at the drive (107)'
4004 'Overtemperature at the motor (108)'
4005 'Overtemperature at the heatsink (106)'
4006 'Voltage control error (100...105)'
4007 'Feedback error (600...606)'
4008 'Error at the power bus (213...215)'
4009 'Overcurrent (212)'
4010 'Overvoltage at the power bus (304/306)'
4011 'Undervoltage at the power bus (307)'

Detection During execution.

Cause An error occurred at the drive. The number in brackets indicates the standard error number of the drive. Refer to the drive manual for further information.

Solution These types of error come with the messages 4019, 4021, 4022 or 4023 that indicate in which axis or spindle drive the error came up. Refer to the drive manual to check the error (number in brackets) and act accordingly.

4012 'Drive error'
4013 'Position deviation too high'
4014 'Communications error'
4015 'Travel limit overrun'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4016 'Undefined class 1 error'

Detection During execution.

Cause The drive has detected an error, but it cannot identify it.

Solution Contact the Service Department.

4017 'Drive error'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4018 'Error accessing a SERCOS variable'

Detection During execution.

Cause An attempt has been made to read (or write) a SERCOS variable from the CNC and:

1. That variable does not exist.
2. The maximum/minimum values have been exceeded.
3. The SERCOS variable has a variable length.
4. An attempt has been made to write a read-only variable.

Solution Check that the variable to be associated with an action is of the right type.

4019 'Drive error : Axis '

Detection During execution.

Cause These messages come with errors 4002 – 4011. When one of the mentioned errors occurs, they indicate in which axis it came up.

4020 'SERCOSID parameter value error'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4021 'Spindle drive error :
4022 'Spindle-2 drive error :
4023 'Auxiliary spindle drive error'

Detection During execution.

Cause These messages come with errors 4002 – 4011. When one of the mentioned errors occurs, they indicate in which spindle it came up.

4024 'SERCOS error when searching home'

Detection During execution.

Cause The home search command of SERCOS has been executed incorrectly.

4025 'SERCOS loop time exceeded: Increase P72 (looptime)'

Detection During execution.

Cause The time it takes to calculate the feedrate of the axis is greater than the cycle time established for transmission to the drive.

Solution Increase the value of general machine parameter LOOPTIME (P72). If the error persists, contact the Service Department.

4026 'Error in SERCOS chip RAM memory'

Detection During execution.

Solution Contact the service department to replace the SERCOS board.

4027 'The drive has started up again'

Detection During execution.

Cause A drive has been turned off and back on due to a power supply failure.

4028 'The light does not reach the CNC through the optic fiber'

Detection On power-up.

Cause The signal sent by the CNC through the optical fiber does not return to the CNC.

Solution Check the condition and installation of the fiber optic cables. Check that the light going "OUT" of the CNC is going through the drives and comes "IN"to the CNC.

If the cables are OK, remove the drives from the ring until the error no longer comes up.

4029 'Communication with the drive cannot be established. No response'

Detection On power-up.

Cause A drive is not responding to the signal sent by the CNC due to one of these causes:

- The drive does not recognize the sercos board.
- The drive is locked up
- The switch number has not been properly read.
- The SERCOS transmission speed has been set differently at the drives and at the CNC. General parameter SERSPD at the CNC and QP11 at the drives.

Solution Save the SERCOS LOG into a file.

See the value of axis machine parameter SERCOSID of the axis causing the error.

Check that the ring contains a drive with the switch in that position.

Reset the drive because the drive only reads the switch on power-up.

Check that the CNC and the drives have the same transmission speed. General parameter SERSPD at the CNC and QP11 at the drives.

Check that the drive does not issue sercos board. To do that look at the display of the drive. If it shows hardware errors, change the drive's sercos board.

If there are no errors at that drive, set the switch of the drive to "1", reset it, set the CNC with a single Sercos axis and connect to the CNC. If it still issues the error, change the drive.

4030 'SERCON register writing error'

Detection During execution.

Solution Contact the Service Department.

4050 'ERROR 1: Internal (Fatal error): Internal RAM test failed'
4051 'ERROR 2 : Internal (Fatal error): internal program malfunction'
4052 'ERROR 3 : Power bus drop: No torque. '
4053 'ERROR 4 : The emergency stop cannot stop the motor in the established time frame'
4054 'ERROR 5 : Program code checksum error'
4055 'ERROR 6 : Sercos board error'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4056 'ERROR 100 : Internal +5 V out of range'
4057 'ERROR 101 : Internal -5 V out of range'
4058 'ERROR 102 : Internal +8 V out of range'
4059 'ERROR 103 : Internal -8 V out of range'
4060 'ERROR 104 : Internal +18 V out of range'
4061 'ERROR 105 : Internal -18 V out of range'
4062 'ERROR 106 : Heat-sink overheating'
4063 'ERROR 107 : VeCon card overheating'
4064 'ERROR 108 : Motor overheating'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4065 'ERROR 200 : Overspeed'
4066 'ERROR 201 : Motor overload'
4067 'ERROR 202 : Driver overload'
4068 'ERROR 211 : Internal (Fatal error): DSP program execution error'
4069 'ERROR 212 : Overcurrent'
4070 'ERROR 213 : Undervoltage at the IGBT power driver'
4071 'ERROR 214: Shortcircuit'
4072 'ERROR 215 : Overvoltage at the power bus(Hard) '

Detection During execution.
Cause An error occurred at the drive.
Solution See the drive manual.

4073 'ERROR 300 : Power supply module overheating'
4074 'ERROR 301 : Power supply module ballast circuit overheating'
4075 'ERROR 302 : Shortcircuit in the power supply module ballast'
4076 'ERROR 303 : Ballast circuit supply voltage out of range'
4077 'ERROR 304 : Overvoltage at the power bus detected by the power supply module'
4078 'ERROR 305 : Protocol error in the interface between the power supply module and the driver'
4079 'ERROR 306 : Overvoltage at the power bus (Soft, trigger prior to hardware)'
4080 'ERROR 307 : Undervoltage of the power bus'

Detection During execution.
Cause An error occurred at the drive.
Solution See the drive manual.

4081 'ERROR 400 : No SERCOS board is detected'
4082 'ERROR 401 : Internal SERCOS error'
4083 'ERROR 403 : MST failure'
4084 'ERROR 404 : MDT failure'
4085 'ERROR 405 : Wrong phase (> 4)'
4086 'ERROR 406 : Wrong phase increase'
4087 'ERROR 407 : Wrong phase decrease'
4088 'ERROR 408 : Phase change without «ready» aknowledgement'
4089 'ERROR 409 : Change to an unitialized phase'
4090 'ERROR 410 : Two drivers have the same ring address'

Detection During execution.
Cause An error occurred at the drive.
Solution See the drive manual.

4091 'ERROR 500 : Inconsistent parameters'
4092 'ERROR 501 : Parameter checksum error'
4093 'ERROR 502 : Wrong parameter value'
4094 'ERROR 503 : The table for default parameter values for each motor is wrong. '
4095 'ERROR 504 : Wrong parameter in SERCOS phase 2'
4096 'ERROR 505 : Different RAM and Flash parameters'
4097 'ERROR 600 : Communication error with the second feedback'
4098 'ERROR 601 : Communication error with the rotor encoder'
4099 'ERROR 602 : motor feedback B signal saturation'

Detection During execution.
Cause An error occurred at the drive.
Solution See the drive manual.

4100 'ERROR 603 : motor feedback A signal saturation '
4101 'ERROR 604 : Saturation of A and/or B signal values'
4102 'ERROR 605 : Week A and/or B signal values'
4103 'ERROR 606 : Too much dispersion of the rotor sensor signals'
4104 'ERROR 700 : RS232 board error'
4105 'ERROR 701 : Internal : Wrong VeCon board identification'
4106 'ERROR 702 : Expansion board identification error'
4107 'ERROR 703 : I/O board identification error'
4108 'ERROR 704 : Analog board identification error'
4109 'ERROR 705 : Power board identification error'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

4110 'ERROR 706 : X3 encoder simulation board identification error'
4111 'ERROR 707 : X4 motor feedback board identification error'
4112 'ERROR 801 : Encoder not detected'
4113 'ERROR 802 : Communication error with the encoder'
4114 'ERROR 803 : Unitialized encoder'
4115 'ERROR 804 : Defective encoder'
4116 'ERROR 805 : No encoder has been detected on the motor'
4117 'ERROR 7 : SERCON clock error'
4118 'ERROR 8 : SERCON data error'
4119 'ERROR 203 : Torque overload error'
4120 'ERROR 411 : telegram reception error'

Detection During execution.

Cause An error occurred at the drive.

Solution See the drive manual.

CAN ERRORS

‘5003 Application error

Cause Internal CANopen error

Solution Contact the Service Department.

5004 CAN bus error

Cause The error type is indicated with a code:

2	Transmission queue full, the message cannot be sent.
128	Bus Off, the bus has been deactivated due to too many errors.
129	CAN warning, there are more than 96 errors at the bus, step prior to the buss off error.
130	Loss of message received or too many messages received. Usually due to wrong speed for the cable length.
131	The CNC has switched to an inoperative state in the bus (internal).

Solution The solution for each cause is:

2	Check the connection between the CNC and first node.
128	Check cables and connections.
129	Check cables and connections.
130	Check machine parameter IOCANSPE (P88).
131	Check cables and connections.

5005 Presence control error detected by the CNC

Cause The CNC detects that the node has reset itself or is connected wrong.

Solution Check cables and connections.

5006 Error because the node has been reset

Cause The node has been reset due to a power supply failure

Solution Check the power supply voltage at the indicated node, the ground connection and the load of the outputs.

5007 Error message corrected

Cause It is activated when an error situation disappears and shows whether there are any more left. If there is none, it resets the node connections.

5022 Internal software error

Cause Internal node software error.

Solution Access the Status screen \ Can \ Versions and reload the software.

5027 Communications error

Cause Node communication error

Solution Contact the Service Department.

5028 Lost messages

Cause The node has lost messages.

Solution Check cables and connections.

5029 Presence control error detected by the node

Cause The presence control done by the CNC node has failed.

Solution Check cables and connections.

5030 Protocol error

Cause The node has received a message that it cannot interpret

Solution Contact the Service Department.

5031 PDO not processed due to its length.

Cause The node has received a process message whose length does not match

Solution Contact the Service Department.

5032 PDO too long

Cause The node has received a process message longer than the one programmed

Solution Contact the Service Department.

5036 Output over-current

Cause Excessive consumption (over current) has been detected in the outputs of the indicated node. As a precaution, the system deactivates all the outputs of this module setting them to zero volts.

Solution Check the consumption and possible short-circuits at the outputs of the module.

5037 Power supply voltage error

Cause A power supply failure has been detected at the indicated node, it has no power or it is under +24V.

Solution Check the supply voltage at the outputs and the consumption of the module's supply voltage.

TABLE DATA ERRORS

'CHECKSUM ERROR: GENERAL PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR: SPINDLE PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:2nd SPINDLE PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:AUX. SPINDLE PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:SERIAL LINE 1 PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:SERIAL LINE 2 PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:HD/ETHERNET PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:USER PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:OEM PARAMETERS Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:PLC PARAMETERS Load CARD A? (ENTER/ESC)'

'CHECKSUM ERROR:ZERO OFFSET TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:PASSWORD TABLE Load CARD A? (ENTER/ESC)'

'CHECKSUM ERROR:AXIS * PARAMETERS Load CARD A? (ENTER/ESC)'

'CHECKSUM ERROR:TOOL TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:TOOL OFFSET TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:MAGAZINE TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:M FUNCTION TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:LEADSCREW * TABLE Load CARD A? (ENTER/ESC)'
'CHECKSUM ERROR:CROSS COMP. TABLE * Load CARD A? (ENTER/ESC)'

Detection During CNC start-up.

Cause Certain table data has been lost (possible RAM error) and there is a table saved in CARD A.

Solution Pressing [ENTER] copies the table saved in CARD A to RAM memory. If the error persists, contact the service department.

'ERROR:GENERAL PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SPINDLE PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SPINDLE-2 PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:AUX. SPINDLE PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SERIAL-LINE-1 PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'ERROR:SERIAL-LINE-2 PARAMETER CHECKSUM Reset? (ENTER/ESC)'
'CHECKSUM ERROR:HD/ETHERNET PARAMETERS Initialize? (ENTER/ESC)'
'CHECKSUM ERROR:USER PARAMETERS Initialize? (ENTER/ESC)'
'CHECKSUM ERROR:OEM PARAMETERS Initialize? (ENTER/ESC)'
'ERROR:PLC PARAMETER CHECKSUM Reset? (ENTER/ESC)'

'ERROR:ZERO OFFSET TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:CODE TABLE CHECKSUM Reset? (ENTER/ESC)'

'ERROR:AXIS PARAMETER CHECKSUM Reset? (ENTER/ESC)'

'ERROR:TOOL TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:TOOL OFFSET TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:TOOL MAGAZINE TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:M FUNCTION TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:AXIS LEADSCREW TABLE CHECKSUM Reset? (ENTER/ESC)'
'ERROR:CROSS COMP. TABLE CHECKSUM Reset? (ENTER/ESC)'

Detection During CNC start-up.

Cause Certain table data has been lost (possible RAM error) and there is no table saved in CARD A.

Solution Pressing [ENTER] loads the tables with CNC's default values. If the error persists, contact the Service Department.

'Wrong * leadscrew table. Press key'

Detection During CNC start-up.

Cause There is some erroneous data in the parameters of the leadscrew compensation table.

Solution The definition of the points of the table must meet the following requirements:

- The points of the table must be ordered according to their position on the axis, starting from the most negative or less positive point to be compensated.
- The machine reference point must have no error (zero).
- The error difference between consecutive points cannot be greater than the distance between them.

'Wrong * cross compensation table. Press key'

Detection During CNC start-up.

Cause There is some erroneous data in the parameters of the cross compensation table.

Solution The definition of the points of the table must meet the following requirements:

- The points of the table must be ordered according to their position on the axis, starting from the most negative or less positive point to be compensated.
- The machine reference point must have no error (zero).

'Incorrect cross compensation table parameters'

Detection During CNC start-up.

Cause The parameters indicating the axes involved in the cross compensation are defined wrong.

Solution Maybe a nonexistent axis has been defined or the affected axis (to be compensated) and the one affecting it are the same.

'Wrong axis or spindle parameters sercosid'

Detection During CNC start-up.

Cause The servosid parameters have not been entered correctly.

Solution The rules of sercosid parameters are:

- They must begin with number 1.
- They must be consecutive.
- They cannot be repeated.

ERRORS OF THE TC WORK MODE

9001 'Cycle without ROUGHING or FINISHING'

Detection While executing a drilling, facing, taper turning, rounding or profile cycle.

Cause Neither the roughing nor the finishing tools have been seleted.

Solution Select the roughing tool (If T=0, there is no roughing) and the finishing tool (if T=0, there is no finishing).

9002 'ROUGHING: Value of ∆=0'

Detection While executing a drilling, facing, taper turning, rounding or profile cycle.

Cause The tapping depth «∆» has not been defined.

Solution Program a cutting pass greater than zero.

9003 'ROUGHING: Value of F=0'

Detection While executing a turning, facing, taper turning, rounding or profile cycle.

Cause No roughing feedrate "F" has been selected.

Solution Program a positive feedrate "F" other than zero.

9004 'ROUGHING: Value of S=0'

Detection While executing a drilling, facing, taper turning, rounding or profile cycle.

Cause No roughing spindle speed "S" has been selected.

Solution Program a positive spindle speed "S" other than zero.

9005 'FINISHING: Value of F=0'

Detection While executing a drilling, facing, taper turning, rounding or profile cycle.

Cause No finishing feedrate "F" has been selected.

Solution Program a positive feedrate "F" other than zero.

9006 'FINISHING: Value of S=0'

Detection While executing a drilling, facing, taper turning, rounding or profile cycle.

Cause No finishing spindle speed "S" has been selected.

Solution Program a positive spindle speed "S" other than zero.

9007 'GEOMETRY: Value of Zi=Zf'

Detection While executing a turning, facing, taper turning or grooving cycle.

Cause The Z coordinate of the starting and end points are the same.

Solution The Z coordinate of the starting and end points must be different.

9008 'GEOMETRY: Value of X=Ø'

Detection While executing a turning, facing, taper turning or grooving cycle.

Cause The coordinates of the initial and final diameter are the same.

Solution The X coordinate of the starting and end points must be different from that of the final diameter.

9009 'FINISHING: Wrong finishing stock'

Detection While executing a turning, facing or grooving cycle.

Cause The finishing stock is greater than the total machining depth.

Solution The finishing stock must be smaller than the total machining depth.

9010 'GEOMETRY: Wrong angle value'

Detection While executing a taper turning cycle.

Cause The taper angle is smaller than 0º or greater than 90º.

Solution The taper angle must be between 0º and 90º.

9011 'GEOMETRY: Wrong radius value'

Detection While executing a rounding cycle.
Cause No rounding radius has been defined.
Solution Program a rounding radius other than zero.

9012 'THREADING: Value of T=0'

Detection While executing a threading cycle.
Cause The tool number has not been defined.
Solution The tool number must be other than zero.

9013 'THREADING: Value of S=0'

Detection While executing a threading cycle.
Cause The spindle speed "S" has not been defined.
Solution Program a positive spindle speed "S" other than zero.

9014 'THREADING: Value of P=0'

Detection While executing a threading cycle.
Cause The thread pitch has not been defined.
Solution Program a thread pitch greater than zero.

9015 'THREADING: Value of H=0'

Detection While executing a threading cycle.
Cause The thread depth has not been defined.
Solution Program a thread depth other than zero.

9016 'THREADING: Value of Δ=0'

Detection While executing a threading cycle.
Cause The tapping depth «Δ» has not been defined.
Solution Program a cutting pass greater than zero.

9017 'THREADING: Value of σ > (Zf-Zi)'

Detection While executing a threading cycle.
Cause The distance to the end of the thread is greater than the length of the thread.
Solution Program a distance to the end of the thread smaller than the length of the thread.

9018 'GEOMETRY: Value of Xi=Xf'

Detection While executing a threading cycle.
Cause The X coordinate of the starting and end points are the same.
Solution The X coordinate of the starting and end points must be different.

9019 'THREADING: Value of σ > (Xf-Xi)'

Detection While executing a threading cycle.
Cause The distance to the end of the thread is greater than the length of the thread.
Solution Program a distance to the end of the thread smaller than the length of the thread.

9020 'ROUGHING: Wrong location (shape) code'

Detection While executing a grooving cycle.
Cause No groove roughing can be done with the selected location code.
Solution Select a tool with the right location code.

9021 'FINISHING: Wrong location (shape) code'

Detection While executing a grooving cycle.
Cause No groove finishing can be done with the selected location code.
Solution Select a tool with the right location code.

9022 'GEOMETRY: Null profile'

Detection While executing a level-1 profile cycle.

Cause No profile has been defined because the starting point and the end point are the same.

Solution The starting point and the end point of the profile must be different.

9023 'DRILLING: Value of T=0'

Detection While executing a drilling or multiple drilling cycle.

Cause The tool number has not been defined.

Solution The tool number must be other than zero.

9024 'DRILLING: Value of Δ=0'

Detection While executing a drilling or multiple drilling cycle.

Cause The tapping depth «Δ» has not been defined.

Solution Program a cutting pass greater than zero.

9025 'DRILLING: Value of L=0'

Detection While executing a drilling or multiple drilling cycle.

Cause The drilling depth has not been defined.

Solution Program a drilling depth other than zero.

9026 'DRILLING: Value of F=0'

Detection While executing a drilling or multiple drilling cycle.

Cause The feedrate "F" has not been defined.

Solution Program a positive feedrate "F" other than zero.

9027 'DRILLING: Value of S=0'

Detection While executing a drilling or multiple drilling cycle.

Cause The spindle speed "S" has not been defined.

Solution Program a positive spindle speed "S" other than zero.

9028 'TAPPING: Value of T=0'

Detection While executing a tapping or multiple tapping cycle.

Cause The tool number has not been defined.

Solution The tool number must be other than zero.

9029 'TAPPING: Value of L=0'

Detection While executing a tapping or multiple tapping cycle.

Cause The threading depth has not been defined.

Solution Program a tapping depth other than zero.

9030 'TAPPING: Value of F=0'

Detection While executing a tapping or multiple tapping cycle.

Cause The feedrate "F" has not been defined.

Solution Program a positive feedrate "F" other than zero.

9031 'TAPPING: Value of S=0'

Detection While executing a tapping or multiple tapping cycle.

Cause The spindle speed "S" has not been defined.

Solution Program a positive spindle speed "S" other than zero.

9032 'GEOMETRY: The final diameter is not external'

Detection While executing a turning or grooving cycle.

Cause An outside turning has been defined, but the final diameter is greater than that of the starting diameter.

Solution In an outside turning, the final diameter must be smaller than the starting one.

9033 ‘GEOMETRY: The final diameter is not internal’

Detection While executing a turning or grooving cycle.

Cause An inside turning has been defined, but the final diameter is smaller than that of the starting diameter.

Solution In an inside turning, the final diameter must be greater than the starting one.

9034 ‘GEOMETRY: Wrong quadrant’

Detection While executing a taper turning cycle.

Cause A taper has been defined in the wrong quadrant.

Solution Select the right quadrant using the relevant icon.

9035 ‘A negative safety distance is not allowed in this cycle’

Detection While executing a taper turning, rounding or profiling cycle.

Cause A negative safety distance has been defined.

Solution In taper canned cycles, the safety distance must be positive.

9036 ‘ROUGHING: Wrong tool for GROOVING.’

Detection While executing a grooving cycle.

Cause The geometry of the selected tool is not right for groove roughing.

Solution Select a tool with the right geometry.

9037 ‘FINISHING: Wrong tool for GROOVING.’

Detection While executing a grooving cycle.

Cause The geometry of the selected tool is not right for groove finishing.

Solution Select a tool with the right geometry.

9038 ‘GEOMETRY: Wrong angle for GROOVING.’

Detection While executing a grooving cycle.

Cause The angle of the groove walls is lower than 0º or higher than 90º.

Solution The angle of the groove walls must be between 0º and 90º.
‘GEOMETRY: The sides of the groove intersect each other.’

9039 ‘GEOMETRY: The sides of the groove intersect each other.’

Detection During execution.

Cause The two walls of the groove intersect each other.

Solution Check the cycle data. The walls of the groove cannot intersect each other.

9040 ‘MULTIPLE CYCLE: Value of β=0’

Detection While executing a multiple drilling, multiple tapping or slot multiple milling cycle.

Cause The angular step between machining has not been defined.

Solution Program an angular step other than zero.

9041 ‘MULTIPLE CYCLE: Value of N=0’

Detection While executing a multiple drilling, multiple tapping or slot multiple milling cycle.

Cause The number of machining operations has not been defined.

Solution The minimum number of machining operations is 1.

9042 ‘MULTIPLE SLOT MILLING CYCLE: Value of T=0’

Detection While executing a multiple slot milling cycle.

Cause The tool number has not been defined.

Solution The tool number must be other than zero.

9043 ‘MULTIPLE SLOT MILLING CYCLE: Value of F=0’

Detection While executing a multiple slot milling cycle.

Cause The feedrate “F” has not been defined.

Solution Program a positive feedrate “F” other than zero.

9044 ‘MULTIPLE SLOT MILLING CYCLE: Value of S=0’

Detection While executing a multiple slot milling cycle.

Cause No live tool turning speed “St” has been defined.

Solution Program a positive turning speed “St” other than zero.

9045 ‘MULTIPLE SLOT MILLING CYCLE: Value of I=0’

Detection While executing a multiple slot milling cycle.

Cause The slot milling depth has not been defined.

Solution Program a slot milling depth other than zero.

9046 ‘MULTIPLE SLOT MILLING CYCLE: Value of L=0’

Detection While executing a multiple slot milling cycle.

Cause The slot milling length has not been defined.

Solution The slot length must be other than zero.

9048 ‘C AXIS PROFILE CYCLE: Value of T=0’

Detection While executing a profile cycle on the C axis.

Cause The tool number has not been defined.

Solution The tool number must be other than zero.

9049 ‘C AXIS PROFILE CYCLE: Value of Δ=0’

Detection While executing a profile cycle on the C axis.

Cause The tapping depth «Δ» has not been defined.

Solution Program a cutting pass greater than zero.

9050 ‘C AXIS PROFILE CYCLE: Value of I=0’

Detection While executing a profile cycle on the C axis.

Cause The total machining depth has not been defined.

Solution Program a machining depth other than zero.

9051 ‘C AXIS PROFILE CYCLE: Value of F=0’

Detection While executing a profile cycle on the C axis.

Cause The feedrate “F” has not been defined.

Solution Program a positive feedrate “F” other than zero.

9052 ‘C AXIS PROFILE CYCLE: Value of S=0’

Detection While executing a profile cycle on the C axis.

Cause No live tool turning speed “St” has been defined.

Solution Program a positive turning speed “St” other than zero.

9053 ‘Value of Smax=0’

Detection While executing any cycle.

Cause The maximum spindle speed «Smax» has not been defined.

Solution Program a maximum positive spindle speed “Smax” other than zero.

9054 ‘DRILLING: Value of KΔ>=1’

Detection While executing a drilling or multiple drilling cycle.

Cause The gear ratio «KΔ» with a value greater than 1.

Solution Program a gear ratio «KΔ» with a value between 0 and 1.

9055 ‘GEOMETRY: The tool is too big for the groove.’

Detection While executing a grooving cycle.

Cause The diameter of the selected tool is greater than the width of the programmed groove.

Solution Select another tool with a smaller diameter.

9056 'GEOMETRY: Value of Z=R.'

Detection While executing a grooving cycle.

Cause When milling slots with Z axis penetration (levels 2 and 4), a slot has been programmed without penetration.

Solution Program the external coordinate "Z" and the depth coordinate "R" with different values.