CNC 8060 8065

Execution channels

(Ref: 1402)

FAGOR AUTOMATION

MACHINE SAFETY

It is up to the machine manufacturer to make sure that the safety of the machine is enabled in order to prevent personal injury and damage to the CNC or to the products connected to it. On start-up and while validating CNC parameters, it checks the status of the following safety elements. If any of them is disabled, the CNC shows a warning message.

- Feedback alarm for analog axes.
- Software limits for analog and sercos linear axes.
- Following error monitoring for analog and sercos axes (except the spindle) both at the CNC and at the drives.
- Tendency test on analog axes.

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC resulting from any of the safety elements being disabled.

HARDWARE EXPANSIONS

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC resulting from any hardware manipulation by personnel unauthorized by Fagor Automation.

If the CNC hardware is modified by personnel unauthorized by Fagor Automation, it will no longer be under warranty.

COMPUTER VIRUSES

FAGOR AUTOMATION guarantees that the software installed contains no computer viruses. It is up to the user to keep the unit virus free in order to guarantee its proper operation.

Computer viruses at the CNC may cause it to malfunction. An antivirus software is highly recommended if the CNC is connected directly to another PC, it is part of a computer network or floppy disks or other computer media is used to transmit data.

FAGOR AUTOMATION shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC due a computer virus in the system.

If a computer virus is found in the system, the unit will no longer be under warranty.

All rights reserved. No part of this documentation may be transmitted, transcribed, stored in a backup device or translated into another language without Fagor Automation's consent. Unauthorized copying or distributing of this software is prohibited.

The information described in this manual may be changed due to technical modifications. Fagor Automation reserves the right to make any changes to the contents of this manual without prior notice.

All the trade marks appearing in the manual belong to the corresponding owners. The use of these marks by third parties for their own purpose could violate the rights of the owners.

It is possible that CNC can execute more functions than those described in its associated documentation; however, Fagor Automation does not guarantee the validity of those applications. Therefore, except under the express permission from Fagor Automation, any CNC application that is not described in the documentation must be considered as "impossible". In any case, Fagor Automation shall not be held responsible for any personal injuries or physical damage caused or suffered by the CNC if it is used in any way other than as explained in the related documentation.

The content of this manual and its validity for the product described here has been verified. Even so, involuntary errors are possible, thus no absolute match is guaranteed. Anyway, the contents of the manual is periodically checked making and including the necessary corrections in a future edition. We appreciate your suggestions for improvement.

The examples described in this manual are for learning purposes. Before using them in industrial applications, they must be properly adapted making sure that the safety regulations are fully met.

INDEX

About the product 5
Version history 9
Safety conditions 11
Warranty terms 15
Material returning terms 17
CNC maintenance 19

CHAPTER 1 INTRODUCTION.

1.1 Basic notions about channels. 22

PARTE I. CONFIGURATION

CHAPTER 2 SETTING A MULTI-CHANNEL SYSTEM.

2.1 Setting the machine parameters. 28
2.1.1 Setting the number of channels and their behavior. 31
2.1.2 Distributing the channels and spindles between the different channels. 33
2.1.3 Allowing to change channel axes and spindles. 38
2.1.4 Arithmetic parameter configuration. 39
2.1.5 Channel operation and display. 40
2.1.6 Subroutines related to the –M– functions. 42
2.2 Setting the PLC program. 43

CHAPTER 3 LOGIC CNC INPUTS AND OUTPUTS.

3.1 General consulting and modifiable signals. 46
3.1.1 Consultation signals. 46
3.1.2 Modifiable signals. 48
3.2 Consultation and modifiable signals associated with the –M– function. 50
3.2.1 Consultation signals. 50
3.3 Consultation and modifiable signals associated with the –H– function. 52
3.3.1 Consultation signals. 52
3.4 Consultation and modifiable signals associated with the –S– function. 53
3.4.1 Consultation signals. 53
3.5 Consulting and modifiable signals of the spindle. 55
3.5.1 Consultation signals. 55
3.5.2 Modifiable signals. 57
3.6 Consulting and modifiable signals of the spindle. 61
3.6.1 Consultation signals. 61
3.6.2 Modifiable signals. 62
3.7 Consulting and modifiable signals of the tool manager. 63
3.7.1 Consultation signals. 63
3.7.2 Modifiable signals. 64

CHAPTER 4 CNC-PLC COMMUNICATION.

4.1 Auxiliary –M– functions. 68
4.2 Auxiliary –H– functions. 70
4.3 Auxiliary –S– function. 72
4.4 Transferring auxiliary functions -M-, -H-, -S-. 73
4.4.1 Synchronized transfer. 74
4.4.2 Non-synchronized transfer. 75
4.5 Displaying PLC errors and messages. 76

FAGOR

CNC 8060
CNC 8065

(REF: 1402)

PARTE II. OPERATION AND PROGRAMMING

CHAPTER 5 OPERATION AND INTERFACE.

5.1 The general status bar. ... 80
5.2 Changing channels. The channel selector switch. ... 82
5.3 Channel synchronization window. ... 83
5.4 The user tables. ... 84

CHAPTER 6 PROGRAMMING IN A SYSTEM WITH CHANNELS.

6.1 Executing a program in the indicated channel. ... 86
6.2 Executing a block in the indicated channel. ... 87
6.3 Axis swapping. ... 88
6.3.1 Set a new axes configuration. ... 89
6.3.2 Add a new axis to the configuration of the channel. ... 91
6.3.3 Remove an axis from the configuration of the channel. ... 93
6.3.4 Rename the axes of a channel. ... 94
6.3.5 Cancel the renaming of the axes. ... 96
6.3.6 Variables related to the axis configuration of the channel. ... 97
6.4 Spindle swapping. ... 98
6.4.1 Set a new spindle configuration. ... 99
6.4.2 Add a spindle to the configuration. ... 100
6.4.3 Remove a spindle from the configuration. ... 101
6.4.4 Rename the spindles of a channel. ... 102
6.4.5 Cancel the renaming of the spindles. ... 103
6.4.6 Variables related to the spindle configuration of the channel. ... 104
6.5 Communication and synchronization. ... 105
6.5.1 Synchronization of channels interrupting the execution of all the channels involved. ... 106
6.5.2 Synchronization of channels without interrupting the execution of all the channels involved.107
6.5.3 Variables related to channel synchronization. ... 109

CHAPTER 7 MULTI-SPINDLE SYSTEM. SPINDLE CONTROL.

7.1 The master spindle of the channel. ... 112
7.1.1 Manual selection of a master spindle ... 114
7.2 Spindle speed. ... 115
7.3 G96/G97. Turning speed or constant surface speed. ... 116
7.3.1 Variables related to spindle speed. ... 117
7.4 G192. Turning speed limit. ... 121
7.4.1 Variables associated with the turning speed limit. ... 122
7.5 M03/M04/M05. Spindle start and stop. ... 123
7.6 M41-M44. Gear change. ... 125
7.7 M19. Spindle orientation. ... 127
7.7.1 Variables related to spindle orientation. ... 129

FAGOR

CNC 8060
CNC 8065

(REF: 1402)

ABOUT THE PRODUCT

BASIC CHARACTERISTICS.

Basic characteristics.	8060 M	8060 T	8065 M	8065 T
PC-based system.	Closed system		Open system	
Operating system.	Windows XPe		Windows XP	
Number of axes.	3 a 5		3 to 28	
Number of spindles.	1	2	1 to 4	
Number of tool magazines.	1	2	1 to 4	
Number of execution channels.	1	2	1 to 4	
Number of handwheels.	1 a 3		1 to 12	
Type of servo system.	Analog / Sercos Digital		Analog / Digital Sercos / Digital Mechatrolink	
Communications.	RS485 / RS422 / RS232 Ethernet			
Integrated PLC. PLC execution time. Digital inputs / Digital outputs. Marks / Registers. Timers / Counters. Symbols.	 < 1ms/K 1024 / 1024 8192 / 1024 512 / 256 Unlimited			
Block processing time.	< 1.5 ms		< 1 ms	

Remote modules.	RIOW	RIO5	RIO70	RCS-S
Communication with the remote modules.	CANopen	CANopen	CANfagor	Sercos
Digital inputs per module.	8	16 / 32	16	- - -
Digital outputs per module.	8	24 / 48	16	- - -
Analog inputs per module.	4	4	8	- - -
Analog outputs per module.	4	4	4	4
Inputs for PT100 temperature sensors.	2	2	- - -	- - -
Feedback inputs.	- - -	- - -	4 (*)	4 (**)

(*) Differential TTL / Sinusoidal 1 Vpp (**) TTL / Differential TTL / Sinusoidal 1 Vpp / SSI protocol

Customizing (on an open system only).
PC-based open system, fully customizable. INI configuration files. FGUIM visual configuration tool. Visual Basic®, Visual C++®, etc. Internal databases in Microsoft® Access. OPC compatible interface

SOFTWARE OPTIONS.

Bear in mind that some of the features described in this manual depend on the software options that are installed. The information of the following table is informative only; when purchasing the software options, only the information provided in the ordering handbook is valid.

Software options (·M· model).

	8060 M	8065 M		8065 M Power	
		Basic	Pack 1	Basic	Pack 1
Open system. Access to the administrator mode.	- - -	- - -	- - -	Option	Option
Number of execution channels	1	1	1	1	1 to 4
Number of axes	3 a 5	3 to 6	5 to 8	5 to 12	8 to 28
Number of spindles	1	1	1 to 2	1 to 4	1 to 4
Number of tool magazines	1	1	1	1 to 2	1 to 4
Limited to 4 interpolated axes	Option	Option	Option	Option	Option
IEC 61131 language	- - -	- - -	Option	Option	Option
HD graphics	Option	Option	Option	Standard	Standard
Conversational IIP	Option	Option	Option	Option	Option
Dual-purpose machines (M-T)	- - -	- - -	- - -	Option	Standard
"C" axis	Option	Standard	Standard	Standard	Standard
Dynamic RTCP	Option	- - -	Option	Option	Standard
HSSA machining system.	Option	Standard	Standard	Standard	Standard
Probing canned cycles	Option	Option	Standard	Standard	Standard
Tandem axes	Option	- - -	Option	Standard	Standard
Synchronism and cams	- - -	- - -	- - -	Option	Standard
Tangential control	- - -	- - -	Standard	Standard	Standard
Volumetric compensation (up to 10 m^3).	- - -	- - -	- - -	Option	Option
Volumetric compensation (more than 10 m^3).	- - -	- - -	- - -	Option	Option
ProGTL3 language.	Option	Option	Option	Option	Option
Part-program translator.	Option	Option	Option	Option	Option

Software options (·T· model).

	8060 T	8065 T		8065 T Power	
		Basic	Pack 1	Basic	Pack 1
Open system. Access to the administrator mode.	- - -	- - -	- - -	Option	Option
Number of execution channels	2	1	1 to 2	1 to 2	1 to 4
Number of axes	3 a 5	3 to 5	5 to 7	5 to 12	8 to 28
Number of spindles	2	2	2	3 to 4	3 to 4
Number of tool magazines	2	1	1 to 2	1 to 2	1 to 4
Limited to 4 interpolated axes	Option	Option	Option	Option	Option
IEC 61131 language	- - -	- - -	Option	Option	Option
HD graphics	Option	Option	Option	Standard	Standard
Conversational IIP	Option	Option	Option	Option	Option
Dual-purpose machines (T-M)	- - -	- - -	- - -	Option	Standard
"C" axis	Option	Option	Standard	Standard	Standard
Dynamic RTCP	- - -	- - -	- - -	Option	Standard
HSSA machining system.	Option	Option	Standard	Standard	Standard
Probing canned cycles	Option	Option	Standard	Standard	Standard
Tandem axes	Option	- - -	Option	Standard	Standard
Synchronism and cams	Option	- - -	Option	Option	Standard
Tangential control	- - -	- - -	- - -	Option	Standard
Volumetric compensation (up to 10 m^3).	- - -	- - -	- - -	Option	Option
Volumetric compensation (more than 10 m^3).	- - -	- - -	- - -	Option	Option
ProGTL3 language.	- - -	- - -	- - -	- - -	- - -
Part-program translator.	Option	- - -	- - -	- - -	- - -

VERSION HISTORY

Here is a list of the features added to each manual reference.

Ref. 1402

Software V01.00 (8060)
8060. First version.

SAFETY CONDITIONS

Read the following safety measures in order to prevent harming people or damage to this product and those products connected to it. Fagor Automation shall not be held responsible of any physical damage or defective unit resulting from not complying with these basic safety regulations.

Before start-up, verify that the machine that integrates this CNC meets the 89/392/CEE Directive.

PRECAUTIONS BEFORE CLEANING THE UNIT

If the CNC does not turn on when actuating the start-up switch, verify the connections.

Do not get into the inside of the unit.	Only personnel authorized by Fagor Automation may manipulate the inside of this unit.
Do not handle the connectors with the unit connected to AC power.	Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

PRECAUTIONS DURING REPAIR

In case of a malfunction or failure, disconnect it and call the technical service.

Do not get into the inside of the unit.	Only personnel authorized by Fagor Automation may manipulate the inside of this unit.
Do not handle the connectors with the unit connected to AC power.	Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

PRECAUTIONS AGAINST PERSONAL DAMAGE

Interconnection of modules.	Use the connection cables provided with the unit.
Use proper cables.	To prevent risks, use the proper cables for mains, Sercos and Bus CAN recommended for this unit. In order to avoid electrical shock at the central unit, use the proper power (mains) cable. Use 3-wire power cables (one for ground connection).
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. In order to avoid electrical shock, before turning the unit on verify that the ground connection is properly made.
Do not work in humid environments.	8060. In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 50 ºC (122 ºF). 8065. In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45 ºC (113 ºF).
Do not work in explosive environments.	In order to avoid risks or damages, 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 by the CNC when installed in other environments (residential or homes).

Install the unit in the right place.

It is recommended, whenever possible, to install the CNC away from coolants, chemical product, blows, etc. that could damage it.

This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as:

- Powerful loads connected to the same AC power line as this equipment.
- Nearby portable transmitters (Radio-telephones, Ham radio transmitters).
- Nearby radio/TV transmitters.
- Nearby arc welding machines.
- Nearby High Voltage power lines.

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.

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

Use the proper power supply.

Use an external regulated 24 Vdc power supply for the keyboard and the remote modules.

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.

Use shielded cables connecting all their meshes to the corresponding pin.

Ambient conditions.

8060. The storage temperature must be between +5 ºC and +55 ºC (41 ºF and 131 ºF).

8065. The storage temperature must be between +5 ºC and +45 ºC (41 ºF and 113 ºF).

The storage temperature must be between -25 ºC and 70 ºC (-13 ºF and 158 ºF).

Central unit enclosure.

Make sure that the needed gap is kept between the central unit and each wall of the enclosure.

Use a DC fan to improve enclosure ventilation.

Main AC power switch.

This switch must be easy to access and at a distance between 0.7 and 1.7 m (2.3 and 5.6 ft) off the floor.

PROTECTIONS OF THE UNIT ITSELF

Remote modules.

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

SAFETY SYMBOLS

Symbols that may appear on the manual.



Danger or prohibition symbol.
It indicates actions or operations that may hurt people or damage products.



Warning symbol.
It indicates situations that certain operations could cause and the suggested actions to prevent them.



Obligation symbol.
It indicates actions and operations that must be carried out.



Information symbol.
It indicates notes, warnings and advises.

Symbols that the product may carry.



Ground protection symbol.
It indicates that that point must be under voltage.

WARRANTY TERMS

INITIAL WARRANTY

All products manufactured or marketed by FAGOR carry a 12-month warranty for the end user which could be controlled by the our service network by means of the warranty control system established by FAGOR for this purpose.

In order to prevent the possibility of having the time period from the time a product leaves our warehouse until the end user actually receives it run against this 12-month warranty, FAGOR has set up a warranty control system based on having the manufacturer or agent inform FAGOR of the destination, identification and on-machine installation date, by filling out the document accompanying each FAGOR product in the warranty envelope. This system, besides assuring a full year of warranty to the end user, enables our service network to know about FAGOR equipment coming from other countries into their area of responsibility.

The warranty starting date will be the one appearing as the installation date on the above mentioned document. FAGOR offers the manufacturer or agent 12 months to sell and install the product. This means that the warranty starting date may be up to one year after the product has left our warehouse so long as the warranty control sheet has been sent back to us. This translates into the extension of warranty period to two years since the product left our warehouse. If this sheet has not been sent to us, the warranty period ends 15 months from when the product left our warehouse.

This warranty covers all costs of material and labour involved in repairs at FAGOR carried out to correct malfunctions in the equipment. FAGOR undertakes to repair or replace their products within the period from the moment manufacture begins until 8 years after the date on which it disappears from the catalogue.

It is entirely up to FAGOR to determine whether the repair is or not under warranty.

EXCLUDING CLAUSES

Repairs will be carried out on our premises. Therefore, all expenses incurred as a result of trips made by technical personnel to carry out equipment repairs, despite these being within the above-mentioned period of warranty, are not covered by the warranty.

Said warranty will be applied whenever the equipment has been installed in accordance with instructions, has not be mistreated, has not been damaged by accident or by negligence and has not been tampered with by personnel not authorised by FAGOR. If, once servicing or repairs have been made, the cause of the malfunction cannot be attributed to said elements, the customer is obliged to cover the expenses incurred, in accordance with the tariffs in force.

Other warranties, implicit or explicit, are not covered and FAGOR AUTOMATION cannot be held responsible for other damages which may occur.

WARRANTY ON REPAIRS

In a similar way to the initial warranty, FAGOR offers a warranty on standard repairs according to the following conditions:

PERIOD	12 months.
CONCEPT	Covers parts and labor for repairs (or replacements) at the network's own facilities.
EXCLUDING CLAUSES	The same as those applied regarding the chapter on initial warranty. If the repair is carried out within the warranty period, the warranty extension has no effect.

When the customer does not choose the standard repair and just the faulty material has been replaced, the warranty will cover just the replaced parts or components within 12 months.

For sold parts the warranty is 12 moths length.

SERVICE CONTRACTS

The SERVICE CONTRACT is available for the distributor or manufacturer who buys and installs our CNC systems.

MATERIAL RETURNING TERMS

When sending the central nit or the remote modules, pack them in its original package and packaging material. If the original packaging material is not available, pack it as follows:

1. Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).
2. Attach a label indicating the owner of the unit, person to contact, type of unit and serial number. In case of malfunction also indicate symptom and a brief description of the problem.
3. Wrap the unit in a polyethylene roll or similar material to protect it. When sending a central unit with monitor, protect especially 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.

CNC MAINTENANCE

CLEANING

The accumulated dirt inside the unit may act as a screen preventing the proper dissipation of the heat generated by the internal circuitry which could result in a harmful overheating of the unit and, consequently, possible malfunctions. Accumulated dirt can sometimes act as an electrical conductor and short-circuit the internal circuitry, especially under high humidity conditions.

To clean the operator panel and the monitor, a smooth cloth should be used which has been dipped into de-ionized water and /or non abrasive dish-washer soap (liquid, never powder) or 75º alcohol. Do not use highly compressed air to clean the unit because it could generate electrostatic discharges.

The plastics used on the front panel are resistant to grease and mineral oils, bases and bleach, dissolved detergents and alcohol. Avoid the action of solvents such as chlorine hydrocarbons, venzole, esters and ether which can damage the plastics used to make the unit's front panel.

PRECAUTIONS BEFORE CLEANING THE UNIT

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

- Do not handle the connectors with the unit connected to AC power. Before handling these connectors (I/O, feedback, etc.), make sure that the unit is not connected to main AC power.
- Do not get into the inside of the unit. Only personnel authorized by Fagor Automation may manipulate the inside of this unit.
- If the CNC does not turn on when actuating the start-up switch, verify the connections.

INTRODUCTION.

1

This manual is directed to the machine manufacturer as well as to the CNC user and it is meant to be a guide to better understand how to work with channels. It contains the necessary information to configure the CNC in multi-channel mode and a description of how to use it and program it.

Configuration of a multi-channel system.

The channels are configured through the machine parameter tables. These parameters set the number of channels, the axes and spindles of each channel, the possibility to exchange them between the different channels, etc.

There is only one PLC program for the whole system. The particularities of each channel will be dealt with inside the program itself. Each channel has its own marks and registers to communicate with the CNC. Refer to the installation manual for further detail.

Configuration of a multi-spindle system.

The CNC can control up to four spindles that may be distributed indistinctly through the different channels. The spindles are configured and distributed between the channels via machine parameters.

The PLC program manages the spindles. Each spindle has its own group of marks and registers.

The part-program and the programming commands.

This manual only shows the programming functions that are directly related to a multi-channel system. These functions deal with such matters as swapping axes or spindles, synchronizing channels, etc.

The rest of the functions, that are also valid both in a multi-channel CNC and in a single channel CNC, are described in the programming manual.

1.1 Basic notions about channels.

About a multi-channel system.

A multi-channel CNC system may have up to four channels; each of which constitutes a different work environment that may act upon part of the CNC system or on the whole CNC system.

The difference between a multi-channel system and several independent CNC's is that the channels can not only act independently, but also together; in other words, they can communicate, synchronize with each other and carry out coordinated actions.

What is a channel?

As mentioned earlier, each channel constitutes a different work environment inside the CNC. Each channel can execute a different program, be in an different work mode and have its own data.

If necessary, the channels can communicate and synchronize with each other and carry out actions that are coordinated with each other. They can also share information through variables and arithmetic parameters.

A channel can have a group of axes and spindles that act independently or in parallel with the rest of the channels. A channel can also be configured without assigning axes or spindles to it at first. Later on, it will be possible to add or remove axes and spindles through a program in execution or MDI.

A channel may be governed from the PLC, from the CNC or from both. Likewise, a channel may be configured as –hidden– so it cannot be selected from the interface and the screen does not show any information on its axes.

The active channel.

It is the channel selected with the channel selector switch. It is the channel being displayed and receiving the commands, e.g. [START], [STOP] and [RESET].

Channel grouping.

Two or more channels may be configured forming a group that will have the following characteristics.

- All the channels are in the same work mode, JOG or automatic.
- The reset command in any of the channels of the group affects all of them.
- Any error in any of the channels of the group interrupts the execution in all of them.

The groups of channels are defined via machine parameters.

Configuring the axes and spindles of a channel.

A channel may be configured at first with one, several, or no axis and no spindle, according to the machine parameter settings. The axes and spindles of each channel must be chosen among the ones available in the system. An axis or spindle cannot be in several channels at the same time; although it could happen that it is not assigned to any channel at first.

Modifying the configuration of the axes and the spindles of a channel

Via a program in execution or MDI, a channel may release or request axes and spindles. This possibility is determined by machine parameter AXISEXCH, which establishes whether an axis or spindle can change channels or whether this change is permanent or not.

A permanent change is maintained after the end of the program, after a reset and on power-up. The original configuration may be restored either by validating the general parameters and restarting or by a part-program that undoes the changes.



It also restores the machine parameter settings if a checksum error occurs when powering up the CNC.

Main or master spindle of the channel.

It is the spindle that receives the commands when no specific spindle is mentioned. All the actions directed to a spindle through the operator panel will be directed to the master spindle.

On CNC power-up and after a reset, it assumes as master spindle the first spindle defined by the machine parameters of the channel (original master). If this spindle is parked or "handed out" to another channel, it assumes as master spindle the next one defined by the machine parameters and so on. If the channel does not have spindles of the original configuration (the one defined by the machine parameters) because they are parked or "handed out", it assumes as master spindle the first one of the current configuration that is not parked.

In general, whenever a channel has a single spindle, it will be its master spindle. If a channel has several spindles, at first the master spindle will be the one configured by machine parameters. A new master spindle may be selected with the #MASTER instruction.

Tool magazine and tool change.

The CNC may have up to four different tool magazines. The number of magazines is independent from the number of spindles and channels available. A magazine is not associated with any particular channel or spindle; i.e. a magazine may be shared by several channels and a channel can request tools from different magazines.

The only limitation will be the one imposed by the mechanics of the machine; in other words, by the physical accessibility of the machine to the tool magazines.

All the magazines can carry out tool changes simultaneously. However, one magazine can only be involved in a tool change process. If from one channel, one wishes to pick up or leave a tool in a magazine already involved in a tool change, the tool manager will wait for the tool change to be done before attending to the new request.

1.

INTRODUCTION.

Basic notions about channels.

FAGOR

CNC 8060
CNC 8065

(REF: 1402)

CONFIGURATION

I

SETTING A MULTI-CHANNEL SYSTEM.

2

Machine parameters.

A multi-channel system is configured mainly through the machine parameter tables. These tables are the only ones for the whole CNC table; it may be accessed from any channel and it is possible to set all the machine parameters.

General machine parameters and for the channels.

Some of these parameters are common to the whole CNC system whereas the rest are only for each channel. These parameters must be set first, because they define the number of channels, axes and spindles of the CNC. This creates the parameter tables for these elements.

For each channel defined, a sub-table is shown with its parameters. They set which are the axes and spindles that configure the channel.

Machine parameter for axes (spindle).

These parameters determine, for each axis and spindle, whether it is possible to change channels or not. The configuration of axes and spindles of a channel may be modified via part-program or MDI..

Machine parameter HMI.

These parameters determine how to handle the operation and display of the different channels.

PLC program.

There is only one PLC program for the whole CNC system. The particularities of each channel will be dealt with in the program itself. The PLC may be accessed from any channel.

2.1 Setting the machine parameters.

Setting the number of channels and their behavior.

The first step for configuring a multi-channel system is to define the number of channels and the possible channel groups. The characteristics of each channel must also be defined, such as the type of channel and whether it is a hidden channel or not.

Parameter.	Meaning.
NCHANNEL	Number of channels.
GROUPID	Group the channel belongs to.
CHTYPE	Channel type.
HIDDENCH	Hidden channel.

Channel type.

The channel type determines whether the channel is governed from the CNC, from the PLC or from both. A PLC channel may be interesting, for example, for a system to load and unload a tool magazine that is controlled like an axis.

PLC channels cannot be set in jog mode or execute part-programs or blocks in MDI; however the axes that make it up may be displayed in the tables using the corresponding softkey. If during setup, a PLC channel must be displayed, it must be defined as CNC+PLC type during setup and once the setup is completed, it should be set back as PLC type.

Group of channels.

A group of channels is set whenever the number defined in parameter GROUPID is other than zero. All the channels with the same parameter GROUPID (other than 0) make up a group.

Hidden channels.

A hidden channel cannot be displayed nor selected in the various work modes. Sometimes, it may be interesting to define a channel as hidden once the setup has been completed. It is also a good idea to define as hidden an exclusive PLC channel, once the setup is completed.

Distributing the channels and spindles between the different channels.

First, define the number and the name of the axes and spindles that make up the system.

Parameter.	Meaning.
NAXIS	Number of axes of the system.
AXISNAME	Name the axes of the system.
NSPDL	Number of spindles of the system
SPDLNAME	Name of the spindles of the system.

Once the axes and spindles of the system have been defined, they must be distributed between the different channels. The axes and spindles of each channel must be chosen among the ones available in the system. An axis or spindle cannot be in several channels at the same time; although it could happen that it is not assigned to any channel at first.

Likewise, a channel may have at first, one, several, or no axis or no spindle associated with it.

Parameter.	Meaning.
CHNAXIS	Number of axes of the channel.
CHAXISNAME	Name the axes of the channel.
GEOCONFIG	Geometrical configuration of the axes of the channel.
CHNSPDL	Number of spindles of the channel.
CHSPDLNAME	Name of the spindles of the channel.

The order of the axes in the channel sets the main work planes, those selected with functions G17, G18 and G19. At the lathe model, parameter GEOCONFIG also affects when setting the main work planes.

Allowing to change channel axes and spindles.

The configuration of a channel may be modified via part-program or MDI, either adding or removing axes as well as spindles. Doing this requires defining for each axis and spindle whether it may change channels or not and whether the change is permanent or not.

When the changes in the channel are permanent, the original configuration (the one defined in the machine parameters) may be restored either by validating the machine parameters or by undoing the changes, for example using a part-program. It must be borne in mind that validating the machine parameters restores the configuration of all the channels.

Parameter.	Meaning.
AXISEXCH	Channel change permission.

Arithmetic parameter configuration.

The CNC has three types of arithmetic parameters, namely local, global and common.

Local arithmetic parameters.

Local parameters may only be accessed from the program or subroutine where they have been programmed. There are seven groups or levels of local parameters in each channel. The maximum range of local parameters is P0 to P99, the typical range being P0 to P25.

When the parameters are used in the block calling a subroutine may also be referred to by the letters A-Z (except Ñ) so "A" is the same as P0 and "Z" the same as P25.

Parameter.	Meaning.
MAXLOCP	Maximum local arithmetic parameter.
MINLOCP	Minimum local arithmetic parameter.

Global arithmetic parameters.

Global parameters may be accessed from any program or subroutine of the channel. There is a group of global parameters in each channel. The maximum range of global parameters is P100 to P9999, the typical range being P100 to P299.

Parameter.	Meaning.
MAXGLBP	Maximum global arithmetic parameter.
MINGLBP	Minimum global arithmetic parameter.

Common arithmetic parameters.

The common parameters may be accessed from any channel. The value of these parameters is shared by all the channels. The maximum range of common parameters is P10000 to P19999, the typical range being P10000 to P10999.

Parameter.	Meaning.
MAXCOMP	Maximum common arithmetic parameter.
MINCOMP	Minimum common arithmetic parameter.

Channel operation and display.



The way to access the different channels is handled through the change key. This key may be configured either to access the channels sequentially or to show the list of available channels on the softkey menu.

It is also possible to change channels by clicking on the icons of the status bar.

Parameter.	Meaning.
CHANGEKEY	Customizing the change key.
FUNCTION	Function of the change key. Next page of the current mode or next channel.
MENU	Set up the system menu.

Subroutines related to the –M– functions.

The –M– function defining table is general for all the channels. To have different procedures in the subroutines associated with certain –M– functions (for example M06), the code of each channel may be differentiated within the subroutine using the variable (V.)G.CNCHANNEL.

Parameter.	Meaning.
MPROGNAME	Subroutine associated with function M.

Channel kinematics.

The CNC offers a number of pre-defined kinematics that may be easily configured by machine parameters. Besides these kinematics, the OEM can integrate 6 additional kinematics.

One kinematics may be active per channel. A kinematics may be configured by between 3 and 8 axes. All the axes making up the kinematics must belong to the same channel and must occupy the first positions in the following order.

Axis order.	Meaning.
1st axis	First main axis of the plane (abscissa).
2nd axis	2nd main axis of the plane (ordinate).
3rd axis	Longitudinal axis.
4th axis	Four axes of the kinematics.
5th axis	Fifth axis of the kinematics.
6th axis	Sixth axis of the kinematics.
7th axis	Seventh axis of the kinematics.
8th axis	Eighth axis of the kinematics.
9th axis and the next ones.	Rest of the axes.

The first 3 axes must be linear; spindle compensation will be applied on to them. The rest of the axes may be either rotary or linear, depending on the type of kinematics.

2.1.1 Setting the number of channels and their behavior.

Some of these parameters correspond to the common general parameters whereas others belong to the general parameters of each channel.

Parameter.	Parameter type.
NCHANNEL	General machine parameter.
GROUPID	General machine parameter of the channel.
CHTYPE	General machine parameter of the channel.
HIDDENCH	General machine parameter of the channel.

NCHANNEL
Number of CNC channels.
Possible values: From 1 to 4.
Default value: 1.
Associated variable: (V.)MPG.NCHANNEL

The use of channels is oriented to machines like dual-spindle lathes where each channel has one of the spindles and two axes; machines with feeders, where the machine and the feeder will be different channels; tool magazine loading and unloading system controlled as an axis.

The CNC can have a single execution channel (single channel system) or several (multi-channel system). Each channel is a different work environment that can act upon a part of or the whole CNC system. The difference between a multi-channel system and several independent CNC's is that the channels can not only act independently, but also together; in other words, they can communicate, synchronize with each other and carry out coordinated actions.

The axes and spindles of a channel.

A channel can have a group of axes and spindles that act independently or in parallel with the rest of the channels. It is also possible to configure a channel without assigning axes or spindles to it at first; then, later on, axes and spindles may be added to it or removed from it via part-program or in the MDI/MDA.mode.

Operation of a channel.

In order to be able to move an axis or spindle, it must be assigned to a channel. Each channel can only control its axes and spindles, although via part-program or MDI/MDA it can command movements to axes or spindles of other channels.

Each channel can execute a different program, be in an different work mode and have its own data. The channels can share information through variables and arithmetic parameters and, if necessary, they may be synchronized via part-program.

GROUPID
Group the channel belongs to.
Possible values: From 0 to 2.
Default value: 0 (it does not belong to any group).
Associated variable: (V.)[ch].MPG.GROUPID

Two or more channels may be configured to form a group. The channels of the same group behave as follows.

- Each channel may be in a different work mode, except in jog and automatic modes. Toggling between the jog mode and automatic mode of a channel will affect all the channels of the group that are in any of these modes; the channels that are in a different mode will not be affected.
- A reset in any of the channels of the group affects all of them.
- Any error in any of the channels of the group interrupts the execution in all of them.

CHTYPE
Type of channel.
Possible values: CNC / PLC / CNC+PLC.
Default value: CNC.
Associated variable: (V.)[ch].MPG.CHTYPE

A channel may be governed from the CNC, from the PLC or from both.

Channels governed by the PLC have neither jog mode nor MDI/MDA mode. The automatic and EDISIMU modes are available, but programs cannot be executed or simulated.

If during setup, it is necessary to display these work modes or execute or simulate programs, set the channel as being governed from the CNC+PLC and once setup is completed, set it back as a PLC channel.

HIDDENCH
Hidden channel.
Possible values: Yes / No.
Default value: No.
Associated variable: (V.)[ch].MPG.HIDDENCH

Hidden channels are not displayed and cannot be selected.

A hidden channel cannot be reset from the operator panel; to reset it, either group it with another channel or reset it from the PLC mark RESETIN.

2.1.2 Distributing the channels and spindles between the different channels.

Some of these parameters correspond to the common general parameters whereas others belong to the general parameters of each channel.

Parameter.	Parameter type.
NAXIS	General machine parameter.
AXISNAME	General machine parameter.
NSPDL	General machine parameter.
SPDLNAME	General machine parameter.
CHNAXIS	General machine parameter of the channel.
CHAXISNAME	General machine parameter of the channel.
GEOCONFIG	General machine parameter of the channel.
CHNSPDL	General machine parameter of the channel.
CHSPDLNAME	General machine parameter of the channel.

CONFIGURING THE AXES OF THE SYSTEM

NAXIS
Number of axes governed by the CNC.
Possible values: From 1 to 28.
Default value: 3.
Associated variable: (V.)MPG.NAXIS

This parameter sets the number of axes of the system, whether they are servo-controlled or not. When counting axes, we will ignore the spindles.

Bear in mind that the number of axes does not depend on the number of channels. A channel may have one, several or no axes associated with it.

AXISNAME n
Name of the axes.
Possible values: X, X1··X9, ·· , C, C1··C9.
Default value: Starting from AXISNAME1; X, Y, Z...
Associated variable: (V.)MPG.AXISNAMEn

The axis name is defined by 1 or 2 characters. The first character must be one of the letters X - Y - Z - U - V - W - A - B - C. The second character is optional and will be a numerical suffix between 1 and 9. This way, the name of the spindles may be within the range X, X1…X9,...C, C1…C9. For example X, X1, Y3, Z9, W, W7, C...

When defining the axes, bear in mind that the order in which they are defined determines their logic number. The first axis of the table will be logic axis -1- and so on. As with the axis name, the logic number permits identifying the axis in PLC variables, marks, etc.

AXISNAME n	Logic order.
AXISNAME 1	Logic number ·1·.
AXISNAME 2	Logic number ·2·.
AXISNAME 3	Logic number ·3·.

CONFIGURING THE SPINDLES OF THE SYSTEM

NSPDL
Number of spindles governed by the CNC.
Possible values: From 0 to 4.
Default value: 1.
Associated variable: (V.)MPG.NSPDL

This parameter sets the number of spindles of the system, whether they are servo-controlled or not.

Bear in mind that the number of spindles does not depend on the number of channels. A channel may have one, several or no spindles associated with it.

SPDLNAME n
Name of the spindles.
Possible values: S, S1··S9.
Default value: Starting from SPDLNAME1; S, S1...
Associated variable: (V.)MPG.SPDLNAMEn

The spindle name is defined by 1 or 2 characters. The first character must be the letter -S- . The second character is optional and will be a numerical suffix between 1 and 9. This way, the name of the spindles may be within the range S, S1 ... S9.

When defining the spindles, bear in mind that the order in which they are defined determines their logic number. The logic numbering of the spindles continues from the last logic axis; hence, in a 3-axis system, the first spindle of the table will be logic spindle ·4· and so on.

AXISNAME	SPDLNAME	Logic order.
AXISNAME 1		Logic number ·1·.
AXISNAME 2		Logic number ·2·.
AXISNAME 3		Logic number ·3·.
	SPDLNAME 1	Logic number ·4·.

CONFIGURING THE AXES OF THE CHANNEL.

CHNAXIS
Number of axes of the channel.
Possible values: From 0 to 28.
Default value: 3.
Associated variable: (V.)[ch].MPG.CHNAXIS

This parameter sets the number of axes of the channel, whether they are servo-controlled or not.

A channel may have initially associated with it one, several or no of the axes defined in the system. In any case, the number of axes assigned to the channel cannot be higher than the number of axes of the system, defined by parameter NAXIS. The sum of the axes assigned to the channels cannot exceed the number of axes of the system either.

It is possible to change the configuration of the axes of a channel via part-program, (defining a new configuration, adding or removing axes) using the instructions #SET AX, #FREE AX and #CALL AX.

CHAXISNAME n
Name of the axes.
Possible values: Any axes defined in AXISNAME.
Default value: Starting from CHAXISNAME1; X, Y, Z...
Associated variable: (V.)[ch].MPG.CHAXISNAMEn

Any axis defined by parameter AXISNAME may belong to the channel.

When defining the axes, bear in mind that the order in which they are defined determines their index in the channel. The index of the first axis of the table will be ·1· and so on. As with

the axis name, the index in the channel permits identifying the axis in PLC variables, marks, etc.

CHAXISNAME n	Index in the channel.
CHAXISNAME 1	Index ·1·.
CHAXISNAME 2	Index ·2·.
CHAXISNAME 3	Index ·3·.

The order of the axes and work planes (mill model).

The order of the axes in the channel sets the main work planes, those selected with functions G17, G18 and G19. Function G20 may be used to form any work plane with the axes of the channel.

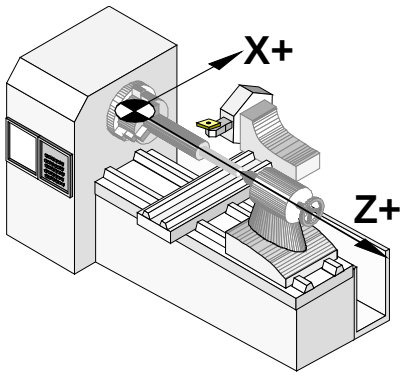
Plane.	Abscissa axis.	Ordinate axis.	Longitudinal axis.
G17	CHAXISNAME 1	CHAXISNAME 2	CHAXISNAME 3
G18	CHAXISNAME 3	CHAXISNAME 1	CHAXISNAME 2
G19	CHAXISNAME 2	CHAXISNAME 3	CHAXISNAME 1

The order of the axes and work planes (lathe model).

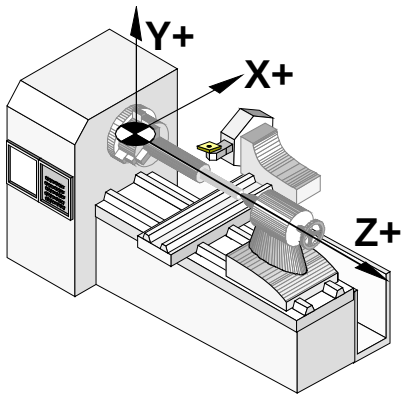
The order of the axes of the channel and parameter GEOCONFIG set the main work planes. Refer to the parameter for further detail.

GEOCONFIG
Geometrical configuration of the axes of the channel.
Possible values: Plane / Trihedron.
Default value: Trihedron.
Associated variable: (V.)[ch].MPG.GEOCONFIG

Not being used at the mill model. On the lathe model, this parameter indicates the axis configuration of the machine, trihedron or plane.



Configuration of "plane" type axes.



Configuration of trihedron type axes.

Configuration of "Trihedron" type axes.

This configuration has three axes forming a Cartesian XYZ type trihedron like on a milling machine. There may be more axes, besides those forming the trihedron; that may be part of the thihedron or be auxiliary axes, rotary axes, etc.

With this configuration, the planes behave in the same way as on a milling machine except that the usual work plane will be G18 (if it has been configured like that in parameter IPLANE). The order of the axes in the channel sets the main work planes, those selected with functions G17, G18 and G19. Function G20 may be used to form any work plane with the axes of the channel.

Plane.	Abscissa axis.	Ordinate axis.	Longitudinal axis.
G17	CHAXISNAME 1	CHAXISNAME 2	CHAXISNAME 3
G18	CHAXISNAME 3	CHAXISNAME 1	CHAXISNAME 2
G19	CHAXISNAME 2	CHAXISNAME 3	CHAXISNAME 1

The CNC displays the ·G· functions associated with the work planes.

Configuration of "plane" type axes.

This configuration has two axes forming the usual work plane on a lathe. There may be more axes, but they cannot be part of the trihedron; there must be auxiliary, rotary, etc.

With this configuration, the work plane is always G18 and will be formed by the first two axes defined in the channel. If the X (firs axis of the channel) and Z (second axis of the channel) have been defined, the work plane will be the ZX (Z as abscissa and X as ordinate).

The work plane is always G18; machine parameter IPLANE is not applied and it is not possible to change planes via part-program. The ·G· functions associated with the work planes have the following effects.

Function.	Meaning.
G17	It does not change planes and shows a warning about it.
G18	It has no effect (except when function G20 is active).
G19	It does not change planes and shows a warning about it.
G20	It is permitted if it does not change the main plane; i.e. it can only be used to change the longitudinal axis.

The CNC does not display the ·G· functions associated with the work planes because it is always the same plane.

Configuration of "plane" type axes. Arc programming.

Programming the arc center I K depends on the active work plane.

- With function G18, in circular interpolations, the arc center I is associated with the first axis of the channel (usually X) and K with the second axis of the channel (usually Z).
- With function G20, in circular interpolations, the arc center I is associated with the abscissa axis (usually Z) and K with the ordinate axis of the channel (usually X).

Configuration of "plane" type axes. The longitudinal axis.

In this configuration, the second axis of the channel is considered as longitudinal axis. If the X (first axis of the channel) and Z (second axis of the channel) axes have been defined, the work plane will be the ZX and Z will be the longitudinal axis. Tool length compensation is applied on this longitudinal axis when using milling tools. With lathe tools, tool length compensation is applied on all the axes where a tool offset has been defined.

When using milling tools on a lathe, the longitudinal compensation axis may be changed by means of the #TOOLAX instruction or the G20 function.

Configuration of "plane" type axes. Axis swapping.

The axes may be swapped, but it must be borne in mind that the previous behavior stays the same for the first and second axes of the channel resulting from the swap.

CONFIGURING THE SPINDLES OF THE CHANNEL.

CHNSPDL
Number of spindles of the channel.
Possible values: From 0 to 4.
Default value: 1.
Associated variable: (V.)[ch].MPG.CHNSPDL

This parameter sets the number of spindles of the channel, whether they are servo-controlled or not.

A channel may have initially one, several or no spindles associated with it. In any case, the number of spindles assigned to the channel cannot be higher than the number of spindles of the system, defined by parameter NSPDL. The sum of the spindles assigned to the channels cannot exceed the number of spindles of the system either.

It is possible to change the configuration of the spindles of a channel via part-program, (defining a new configuration, adding or remove spindles) using the instructions #SET SP, #FREE SP and #CALL SP.

CHSPDLNAME n
Name of the spindles.

Possible values: Any spindle defined in SPDLNAME.
Default value: Starting from CHSPDLNAME1; S, S1...
Associated variable: (V.)[ch].MPG.SPDLNAMEn

Any spindle defined by parameter SPDLNAME may belong to the channel.

On CNC power-up and after a reset, the channel assumes as master spindle the first spindle defined by the machine parameters of the channel (original master). If this spindle is parked or "handed out" to another channel, the channel assumes as master spindle the next spindle defined by the machine parameters and so on.

When defining the spindles, bear in mind that the order in which they are defined determines their index in the channel. The index of the first spindle of the table will be ·1· and so on. As with the spindle name, the index in the channel permits identifying the spindle in PLC variables, marks, etc.

CHSPDLNAME	Index in the channel.
CHSPDLNAME 1	Index ·1·.
CHSPDLNAME 2	Index ·2·.
CHSPDLNAME 3	Index ·3·.

2.1.3 Allowing to change channel axes and spindles.

AXISEXCH
Channel change permission.
Parameter valid for linear and rotary axes and spindles.
Parameter valid for analog drive, Sercos and simulated.
Possible values: No / Temporary / Maintained.
Default value: No.
Associated variable: (V.)[ch].MPA.AXISEXCH.xn

Initially, each channel has some axes and spindles assigned to it. The CNC can change channel axes and spindles or just change the configuration of a channel by modifying the position of its axes and spindles or eliminating some of them.

In order for the CNC to be able to change the channel axes and spindles, they must have permission. Parameter AXISEXCH sets whether the axis or the spindle has permission to change channels and if it does, whether the change is temporary or permanent; in other words, whether the change is maintained after an M02, M30, a reset or after restarting the CNC.

AXISEXCH.	Meaning.
No.	• The CNC can modify the position of the axis or spindle in its channel or even remove it from the channel. • The CNC cannot change the axis or the spindle of the channel. • The CNC does not maintain the changes when restarting the part-program, after a reset or after restarting the CNC. The axis or spindle returns to its original position, set by machine parameters.
Temporary.	• The CNC can modify the position of the axis or spindle in its channel or even remove it from the channel. • The CNC can change the axis or the spindle of the channel. • The CNC does not maintain the changes when restarting the part-program, after a reset or after restarting the CNC. The axis or spindle returns to its channel and original position, set by machine parameters.
Maintained.	• The CNC can modify the position of the axis or spindle in its channel or even remove it from the channel. • The CNC can change the axis or the spindle of the channel. • The CNC maintains the changes when restarting the part-program, after a reset or after restarting the CNC. The axis or spindle stays in its new channel, but it can change the position to let the original axes of the channel return.

The original configuration (the one defined in the machine parameters) of a channel with axes and spindles of the type "AXISEXCH = Maintained" may be restored either by validating the machine parameters or by undoing the changes, for example using a part-program. It must be borne in mind that validating the machine parameters restores the configuration of all the channels.



The CNC will also restores the machine parameter settings if a checksum error occurs when powering up the CNC.

2.1.4 Arithmetic parameter configuration.

MAXLOCP
Maximum local arithmetic parameter.
Possible values: From 0 to 99.
Default value: 25.
Associated variable: (V.)MPG.MAXLOCP

See general machine parameter MINLOCP.

MINLOCP
Minimum local arithmetic parameter.
Possible values: From 0 to 99.
Default value: 0.
Associated variable: (V.)MPG.MINLOCP

Parameters MINLOCP and MAXLOCP define the group of local arithmetic parameters to be used. Local parameters may only be accessed from the program or subroutine where they have been programmed. There are seven groups of local parameters in each channel.

MAXGLBP
Maximum global arithmetic parameter.
Possible values: From 100 to 9999.
Default value: 299.
Associated variable: (V.)MPG.MAXGLBP

Check general machine parameter MINGLBP.

MINGLBP
Minimum global arithmetic parameter.
Possible values: From 100 to 9999.
Default value: 100.
Associated variable: (V.)MPG.MINGLBP

Parameters MAXGLBP and MINGLBP define the group of global arithmetic parameters to be used. Global parameters may be accessed from any program or subroutine called upon from the channel. There is a group of global parameters in each channel. The value of these parameters is shared by the program and the subroutines.

MAXCOMP
Maximum arithmetic parameter common to all the channels.
Possible values: From 10000 to 19999.
Default value: 10025.
Associated variable: (V.)MPG.MAXCOMP

See general machine parameter MINCOMP.

MINCOMP
Minimum arithmetic parameter common to all the channels.
Possible values: From 10000 to 19999.
Default value: 10000.
Associated variable: (V.)MPG.MINCOMP

Parameters MAXCOMP and MINCOMP the group of local arithmetic parameters common to all the channels to be used. The common parameters may be accessed from any channel. The value of these parameters is shared by all the channels.

2.1.5 Channel operation and display.

The following parameters correspond to the HMI parameters.

CHANGEKEY
Customizing the change key.

This table shows the parameters to configure the change key. The table has the following machine parameters.

CHANGEKEY	
FUNCTION	Function of the change key.
SYSMENUMODE	Behavior of the system menu.
SYSHMENU	Horizontal system menu.
SYSVMENU	Vertical system menu.

FUNCTION
Function of the change key.

Possible values: Next page / Next channel / Menu
Default value: Next page.
Parameter included in the CHANGEKEY table.

Depending on the function selected, it is possible to carry out one of the following tasks.

Value.	Meaning.
Next page.	The key selects the next page of the active work mode.
Next Channel.	The key selects the next channel.
Menu.	The key shows the list of channels and pages on the softkey menus.

SYSMENUMODE
Behavior of the system menu.

Possible values: Volatile / Fixed.
Default value: Volatile.
Parameter included in the CHANGEKEY table.

This parameter determines when the system menu is disabled.

Value.	Meaning.
Volatile.	The softkey menu is disabled when selecting a menu option or when changing the active component.
Fixed.	The softkey menu remains until the change key is pressed again.

SYSHMENU
Horizontal system-menu.

Possible values: Disabled / Pages / Channels / Components.
Default value: Disabled.
Parameter included in the CHANGEKEY table.

See parameter SYSHMENU.

SYSVMENU
Vertical system-menu.
Possible values: Disabled / Pages / Channels / Components.
Default value: Disabled.
Parameter included in the CHANGEKEY table.

This parameter sets the options that will appear on each softkey-menu.

Value.	Meaning.
Disabled.	The menu will be disabled.
Screens.	The menu shows the various pages or screens of the active work mode.
Channels.	The menu shows the available channels.
Components.	The menu shows the components or work modes of the CNC.

2.1.6 Subroutines related to the –M– functions.

The following parameters correspond to the –M– function table.

MPROGNAME
Name of subroutine associated with M function.
Possible values: any text with up to 64 characters.
Default value: Without subroutine.
Associated variable: (V.)MPM.MPROGNAME[pos]
Parameter included in the DATA table.

The subroutines associated with the M functions must be located in the "C:\CNC8070 \MTB \SUB" folder.

When associating a subroutine with an M function, the execution of the M function executes the associated subroutine without executing the M function itself. To send the M function to the PLC, it must be programmed in the subroutine.

The type of synchronization of the M functions that have an associated subroutine must be "without synchronization" or "After-After". The CNC executes the associated subroutine after executing the programmed movement (if any).

To have different procedures in the subroutines associated with certain M functions, the code of each channel may be differentiated within the subroutine using the variable (V.)G.CNCHANNEL..

SYNCHTYPE
Type of synchronization.
Possible values: Without synchronization / Before-Before / Before-After / After-After.
Default value: Before - Before.
Associated variable: (V.)MPM.SYNCHTYPE[pos]
Parameter included in the DATA table.

Since the M functions may be programmed together with the movement of the axes, in the same block, it must be indicated when the function is to be sent out to the PLC and when it will be checked that it has been already executed (synchronization).

Type of synchronism.	Meaning.
Without synchronization.	M function not synchronized.
Before - Before.	The M function is sent to the PLC and synchronized before the movement.
Before - After.	The M function is sent to the PLC before the movement and synchronized after the movement.
After - After.	The M function is sent to the PLC and synchronized after the movement.

The M functions may be sent and/or synchronized before or after the movement.

- If an M function is used to turn on a lamp, it will be set without synchronization because there is no need to check that the lamp has indeed turned on.
- Functions M03 and M04 to start up the spindle, they should be executed and synchronized before the movement.
- Function M5 to stop the spindle, should be executed and synchronized after the movement.

2.2 Setting the PLC program.

There is a group of marks and registers for each channel to create the PLC program. See chapter *"3 Logic CNC inputs and outputs."*.

Transferring –M– and –H– functions.

The M and H functions are exchanged by channel. When using channels, the marks and registers of these functions must indicate the channel number they refer to.

Transferring -S- functions.

The exchange of S functions is independent from the channel. When using several spindles, the marks and registers of these functions refer to the spindle number.

Multiple spindles.

The CNC may have up to four spindles. When using channels, the spindles may be distributed indistinctly between them.

All the spindles of a channel may be controlled independently; in other words, each spindle may be given a different command.

In order to be able to manage spindles from the PLC regardless of the channel it belongs to, there is a group of PLC marks and registers for each spindle.

Spindle synchronization.

Via PLC it is possible to synchronize the machining operation in a channel with a particular spindle even if it is in another channel. For example, to program the feedrate depending on the speed of a particular spindle.

From the PLC, it is possible to make a channel synchronize with a spindle that belongs to another channel to carry out electronic threading operation (G33) and set the feedrate per revolution (G95).

2.

SETTING A MULTI-CHANNEL SYSTEM.
Setting the PLC program.

FAGOR

CNC 8060
CNC 8065

(REF: 1402)

LOGIC CNC INPUTS AND OUTPUTS.

3

CNC's physical inputs and outputs are the set of system inputs and outputs that are governed by the PLC and communicate with the outside world through CNC connectors. The CNC also has a number of logic inputs and outputs to exchange the internal data with the marks and registers of the PLC. This way, the PLC has access to some internal CNC data. Each of these logic inputs and outputs may be referred to using its associated mnemonic. The mnemonics that begin with a "_" sign indicate that the signal is active low (0 V.); the rest of the signals are active high.

CNCREADY	_ALARM
AUXEND	_EMERGEN
MANUAL	_STOP

Specific signals for a multi-channel system.

Here is a summary of the specific signals of a multi-channel, multi-spindle and/or multi-magazine system. Refer to the installation manual for further detail about each one of them.

The PLC has more signals, e.g. the general signals of the CNC, of the axes, of the keys, etc. but since they have nothing to do with configuring a multi-channel system, they are not mentioned in this documentation. Refer to the installation manual for further detail about each one of them.

The specific logic signals for configuring a system with channels are grouped as follows:

- General signals.
- Signals related to the –M– auxiliary functions.
- Signals related to the –H– auxiliary functions.
- Channel synchronizing signals.

The specific logic signals for configuring a system with several spindles are grouped as follows:

- Signals related to the –S– auxiliary functions.
- Spindle signals.

The specific logic signals for configuring a system with several tool magazines are grouped as follows:

- Tool manager signals.

3.1 General consulting and modifiable signals.

Here is a table that summarizes the mnemonics associated with general signals. The following tables show the mnemonics for each mark (M) or register (R) in each channel.

3.1.1 Consultation signals.

The general signals are specific for each channel.

M/R	Channel ·1·	Channel ·2·	Channel ·3·	Channel ·4·
M	READYC1	READYC2	READYC3	READYC4
This mark indicates the status of the CNC channel. The channel deactivates this mark when the CNC is in an error state (red status window) and activates it if otherwise.				
M	STARTC1 START	STARTC2	STARTC3	STARTC4
The CNC channel activates this mark to indicate to the PLC that the [START] key has been pressed at the operator panel.				
M	RESETOUTC1 RESETOUT	RESETOUTC2	RESETOUTC3	RESETOUTC4
When pressing the [RESET] key or when the PLC activates the RESETIN mark, the CNC assumes the initial conditions and activates this mark. This mark stays active for a time period set in parameter MINAENDW.				
M	FHOUTC1 FHOUT	FHOUTC2	FHOUTC3	FHOUTC4
The CNC channel activates this mark when the execution of the part-program is interrupted.				
M	_ALARMC1 _ALARM	_ALARMC2	_ALARMC3	_ALARMC4
The CNC channel deactivates this mark when there is an alarm or an emergency generated by the CNC channel. The CNC channel activates this mark again when the CNC channel message is removed and the cause of the alarm has been eliminated.				
M	MANUALC1 MANUAL	MANUALC2	MANUALC3	MANUALC4
The CNC channel activates the mark when the jog mode is selected.				
M	AUTOMATC1 AUTOMAT	AUTOMATC2	AUTOMATC3	AUTOMATC4
The CNC channel activates the mark when the automatic mode is selected.				
M	MDIC1 MDI	MDIC2	MDIC3	MDIC4
The CNC channel activates the mark when the MDI mode is selected.				
M	SBOUTC1 SBOUT	SBOUTC2	SBOUTC3	SBOUTC4
The CNC activates this mark when the "single block" mode is selected.				
M	INCYCEC1 INCYCE	INCYCEC2	INCYCEC3	INCYCEC4
The CNC activates this mark when executing a block or moving an axis.				
M	RAPIDC1 RAPID	RAPIDC2	RAPIDC3	RAPIDC4
The CNC channel activates this mark when executing a rapid traverse movement (G0).				
M	ZEROC1 ZERO	ZEROC2	ZEROC3	ZEROC4
The CNC channel activates this mark when searching home (G74).				

M/R	Channel ·1·	Channel ·2·	Channel ·3·	Channel ·4·
M	PROBEC1 PROBE	PROBEC2	PROBEC3	PROBEC4
The CNC channel activates this mark when executing a probing movement (G100).				
M	THREADC1 THREAD	THREADC2	THREADC3	THREADC4
The CNC channel activates this mark when executing an electronic threading block (G33).				
M	TAPPINGC1 TAPPING	TAPPINGC2	TAPPINGC3	TAPPINGC4
The CNC channel activates this mark when executing a tapping canned cycle. The mark stays active during the possible dwell programmed at the bottom of the thread.				
M	RIGIDC1 RIGID	RIGIDC2	RIGIDC3	RIGIDC4
The CNC channel activates this mark when executing an rigid tapping block (G63).				
M	CSSC1 CSS	CSSC2	CSSC3	CSSC4
The CNC channel activates this mark when constant surface speed is selected (G96).				
M	INTERENDC1 INTEREND	INTERENDC2	INTERENDC3	INTERENDC4
The CNC channel activates this mark when the theoretical movement of the axes (when it no longer outputs the velocity command).				
M	INPOSC1 INPOS	INPOSC2	INPOSC3	INPOSC4
The CNC channel activates this mark to indicate that all its active axes and spindles have reached position, except the independent axes programmed from the PLC. This mark remains active during the movement of independent axes.				
M	BLKSEARCHC1 BLKSEARCH	BLKSEARCHC2	BLKSEARCHC3	BLKSEARCHC4
The CNC channel activates this mark to indicate that the "block search" option is active in automatic mode.				
M	ADVINPOSC1 ADVINPOS	ADVINPOSC2	ADVINPOSC3	ADVINPOSC4
The CNC channel activates this signal a certain amount of time before the axes reach position. This amount of time is set by parameter ANTIME; if this parameter has been set to 0, the mark is always active.				
M	CAXISC1 CAXIS	CAXISC2	CAXISC3	CAXISC4
The CNC channel activates this mark when the spindle is working as a "C" axis. This mark stays active as long as any of functions #CAX, #FACE or #CYL is active.				
M	RETRAENDC1	RETRAENDC2	RETRAENDC3	RETRAENDC4
The CNC channel activates this mark to cancel the retrace function and indicate to the PLC that it must deactivate the RETRACE mark.				
M	TANGACTIVC1	TANGACTIVC2	TANGACTIVC3	TANGACTIVC
The CNC activates this mark to let the PLC know that a tangential control is active in the channel. This mark is not initialized when tangential control is frozen (suspended).				
M	DINDISTC1C2 DINDISTC1C3 DINDISTC1C4	DINDISTC2C1 DINDISTC2C3 DINDISTC2C4	DINDISTC3C1 DINDISTC3C2 DINDISTC3C4	DINDISTC4C1 DINDISTC4C2 DINDISTC4C3
These marks are related to the dynamic machining distribution between channels (#DINDIST instruction), for the option to distribute passes between channels. During the roughing operation of the cycle, the CNC channel activates these marks to indicate which channel has the cycle been programmed in and which are the channels involved in the distribution of the passes. The first channel indicated in the mnemonic refers to the channel that executes the cycle; the second one refers to the channel involved in the distribution of passes.				

3.1.2 Modifiable signals.

The general signals are specific for each channel.

M/R	Channel ·1·	Channel ·2·	Channel ·3·	Channel ·4·
M	_EMERGENC1 _EMERGEN	_EMERGENC2	_EMERGENC3	_EMERGENC4
If the PLC deactivates this mark, the CNC stops the axes and the spindle and displays the corresponding error message. While this mark is deactivated, the CNC does not allow executing programs and aborts any axis or spindle movement.				
M	_STOPC1 _STOP	_STOPC2	_STOPC3	_STOPC4
If the PLC deactivates this mark, the CNC channel interrupts the execution of the part-program; but it keeps the spindle turning. The movement of the independent axes is not affected by this mark. Nor is affected by the [STOP] key of the operator panel.				
M	_XFERINHC1 _XFERINH	_XFERINHC2	_XFERINHC3	_XFERINHC4
If the PLC deactivates this mark, the CNC channel interrupts the execution of the program at the end of the block being executed and inhibits the execution of the next block. If the axis needs more braking distance than it has with the block being executed, the CNC can continue executing more blocks until the axis comes to a complete stop respecting the dynamics of the machine. When the PLC activates this mark, the CNC resumes the execution of the program.				
M	_FEEDHOLC1 _FEEDHOL	_FEEDHOLC2	_FEEDHOLC3	_FEEDHOLC4
If the PLC deactivates this mark, the CNC channel interrupts momentarily the movement of the axes; but keeping the spindle turning. When the PLC activates this mark again, the axes resume their motion. The movement of the independent axes is not affected by this mark.				
M	CYSTARTC1 CYSTART	CYSTARTC2	CYSTARTC3	CYSTARTC4
The PLC must activate this mark in order for the program execution to begin.				
M	SBLOCKC1 SBLOCK	SBLOCKC2	SBLOCKC3	SBLOCKC4
If the PLC activates this mark, the CNC switches to "single block" execution mode.				
M	MANRAPIDC3 MANRAPID	MANRAPIDC2	MANRAPIDC3	MANRAPIDC3
If the PLC turns this mark on, the CNC selects the rapid traverse for continuous jog movements.				
M	OVRCANC1 OVRCAN	OVRCANC2	OVRCANC3	OVRCANC4
If the PLC activates this mark, the CNC applies 100 % override to the axis feedrate regardless of the value currently selected.				
M	RESETINC1 RESETIN	RESETINC2	RESETINC3	RESETINC4
With an up-flank, the CNC channel assumes the machining conditions selected by machine parameter and it activates the RESETOUT mark.				
M	AUXENDC1 AUXEND	AUXENDC2	AUXENDC3	AUXENDC4
Auxiliary mark that the PLC uses when executing auxiliary functions S and M with synchronization.				
M	BLKSKIP1C1 BLKSKIP1	BLKSKIP1C2	BLKSKIP1C3	BLKSKIP1C4
The PLC activates this mark to let the CNC channel know that the block skip condition "/" is met. Therefore, blocks with that condition will not be executed.				
M	M01STOPC1 M01STOP	M01STOPC2	M01STOPC3	M01STOPC4
The PLC activates this mark to "tell" the CNC channel not to ignore the conditional stops (M01).				

M/R	Channel ·1·	Channel ·2·	Channel ·3·	Channel ·4·
M	PLCABORTC1 PLCABORT	PLCABORTC2	PLCABORTC3	PLCABORTC4
If the PLC activates this mark, the CNC channel aborts the CNCEX command launched from the PLC, but without initializing the conditions of the channel and keeping the history of the channnel.				
M	PRGABORTC1 PRGABORT	PRGABORTC2	PRGABORTC3	PRGABORTC4
If the PLC activates this mark, the CNC channel aborts the current execution of the part-program; but without affecting the spindle, the rest of the history is initialized. Then, the CNC resumes the execution of the program from the label indicated in the #ABORT instruction that is active in the part-program.				
M	RETRACEC1	RETRACEC2	RETRACEC3	RETRACEC4
If the PLC activates this mark while executing a program in automatic mode, the retrace function is activated in the selected channel. If the PLC deactivates this mark, the CNC finishes the RETRACE function.				
M	EXRAPIDC1 EXRAPID	EXRAPIDC2	EXRAPIDC3	EXRAPIDC4
If the PLC activates this mark, the CNC channel enables rapid traverse during the execution of a program for the programmed movements.				
M	FLIMITACCHC1 FLIMITACCH	FLIMITACCHC2	FLIMITACCHC3	FLIMITACCHC4
If the PLC activates this mark, the CNC activates the feedrate safety limits (parameter FLIMIT) on all the axes of the channel during the execution of the block. If the PLC deactivates this mark, the CNC restores the programmed feedrate.				
M	INT1C1 INT1	INT1C2	INT1C3	INT1C4
When the PLC activates one of these marks, the channel interrupts the execution of the program and executes the corresponding interruption subroutine associated with parameters INT1SUB to INT4SUB. The CNC cancels the mark when it starts executing the subroutine or when it rejects the execution of the subroutine.				

3.2 Consultation and modifiable signals associated with the –M– function.

3.2.1 Consultation signals.

MFUN1···MFUN7

There is a register for each channel. The mnemonics for each channel are the following. Here is an example of the mnemonics for MFUN1; it is the same for the rest of the registers.

MFUN1C1 (can also be programmed as MFUN1)

MFUN1C2 MFUN1C3 MFUN1C4

The channel uses these registers to indicate to the PLC the M auxiliary functions selected for execution. Each one of them indicates the number of one of the M functions programmed in the block.

Each channel can have up to 7 M functions in a block. If all the registers are not used, the hexadecimal value $FFFFFFFF is assigned to the unused ones (those with the highest numbers).

This way, if functions M100 and M135 are programmed in the first channel and functions M88 and M75 in the second channel, the CNC will transfer the following data.

MFUN1C1	MFUN2C1	MFUN3C1 - MFUN7C1
100	135	$FFFFFFFF

MFUN1C2	MFUN2C2	MFUN3C2 - MFUN7C2
88	75	$FFFFFFFF

If, then, the M88 function is executed in the first channel, then:

MFUN1C1	MFUN2C1	MFUN3C1 - MFUN7C1
88	$FFFFFFFF	$FFFFFFFF

Commands MFUNC1* - MFUNC4*. Checking if a function has been programmed in the channel.

In order to know whether a particular function is programmed in the block currently being executed, all the registers may be checked one by one or the following commands may be used to check them all at the same time.

MFUNC1* For channel 1.

MFUNC2* For channel 2.

MFUNC3* For channel 3.

MFUNC4* For channel 4.

Example for detecting M04 in channel 1. If it has been programmed, it will return a "1", and a "0" if otherwise.

```
CPS MFUNC1* EQ 4 = …
```

MSTROBE

There is a mark for each channel. The mnemonics for each channel are the following.

MSTROBEC1 (can also be programmed as MSTROBE)

MSTROBEC2 MSTROBEC3 MSTROBEC4

The CNC channel sets this mark high (=1) to indicate to the PLC that it must execute the auxiliary M functions indicated in registers MFUN1 through MFUN7 of the channel.

DMxx

This mark is associated with some M auxiliary functions.

The marks associated with functions M00, M01, M02, M06, M08, M09, M30 have a mark for each channel. Here is an example of the mnemonics for DM00; it is the same for the rest of the marks (DM01, DM02, DM06, DM08, DM09, DM30).

DM00C1 (can also be programmed as DM00)

DM00C2 DM00C3 DM00C4

The marks associated with functions M03, M04, M05, M19, M41, M42, M43, M44 have a mark for each spindle. Here is an example of the mnemonics for DM03; it is the same for the rest of the marks (DM04, DM05, DM19, DM41, DM42, DM43, DM44).

DM03SP1 (can also be programmed as DM03)

DM03SP2 DM03SP3 DM03SP4

The CNC indicates in these marks the status of the spindle auxiliary M functions. The mark is set to (=1) if the function is active and to (=0) if otherwise.

3.3 Consultation and modifiable signals associated with the –H– function.

3.3.1 Consultation signals.

HFUN1···HFUN7

There is a register for each channel. The mnemonics for each channel are the following. Here is an example of the mnemonics for MFUN1; it is the same for the rest of the registers.

HFUN1C1 (can also be programmed as HFUN1)

HFUN1C2 HFUN1C3 HFUN1C4

The channel uses these registers to indicate to the PLC the H auxiliary functions selected for execution. Each one of them indicates the number of one of the H functions programmed in the block.

Each channel can have up to 7 H functions in a block. If all the registers are not used, the hexadecimal value $FFFFFFFF is assigned to the unused ones (those with the highest numbers).

This way, if functions H10 and H13 are programmed in the first channel and functions H8 and H10 in the second channel, the CNC will transfer the following data.

HFUN1C1	HFUN2C1	HFUN3C1 - HFUN7C1
10	13	$FFFFFFFF

HFUN1C2	HFUN2C2	HFUN3C2 - HFUN7C2
8	10	$FFFFFFFF

Commands HFUNC1* - HFUNC4*. Checking if a function has been programmed in the channel.

In order to know whether a particular function is programmed in the block currently being executed, all the registers may be checked one by one or the following commands may be used to check them all at the same time.

HFUNC1* For channel 1.

HFUNC2* For channel 2.

HFUNC3* For channel 3.

HFUNC4* For channel 4.

Example for detecting H77 in channel 1. If it has been programmed, it will return a "1", and a "0" if otherwise.

```
CPS HFUNC1* EQ 77 = …
```

HSTROBE

There is a mark for each channel. The mnemonics for each channel are the following.

HSTROBEC1 (can also be programmed as HSTROBE)

HSTROBEC2 HSTROBEC3 HSTROBEC4

The CNC channel sets this mark high (=1) to indicate to the PLC that it must execute the auxiliary H functions indicated in registers HFUN1 through HFUN7.

3.4 Consultation and modifiable signals associated with the –S– function.

3.4.1 Consultation signals.

SFUN1···SFUN4

There is a register for each spindle. The mnemonics for each spindle are the following:

SFUN1 SFUN2 SFUN3 SFUN4

These registers indicate the programmed speed for each spindle. These registers refer to the spindle number; they are independent from the channel where the spindle is. The CNC only uses the registers of the spindles whose SPDLTIME parameter has a value other than zero.

Each one of them indicates the value of one of the S functions programmed. If all the registers are not used, the CNC assigns $FFFFFFFF to the unused ones (those with the highest numbers).

This way, if a block contains functions S1000 and S1=550 and the value of SPDLTIME parameter of both spindles is other than zero, the CNC will transfer the following information to the PLC.

SFUN1	SFUN2	SFUN3	SFUN4
1000	550	$FFFFFFFF	$FFFFFFFF

SSTROBE

There is a mark for each spindle. The mnemonics for each spindle are the following:

SSTROBE1 (can also be programmed as SSTROBE)

SSTROBE2 SSTROBE3 SSTROBE4

The CNC channel sets this mark high (=1) to indicate to the PLC that a new spindle speed has been selected. The CNC channel only uses this mark on spindles whose SPDLTIME parameter has a value other than zero.

SPN1···SPN7

There is a register for each channel. The mnemonics for each channel are the following. Here is an example of the mnemonics for SPN1; it is the same for the rest of the registers.

SPN1C1 SPN1C2 SPN1C3 SPN1C4

The channel uses these registers to indicate to the PLC which spindle of the channel each auxiliary M function selected for execution is addressed to.

Each channel can have up to 7 M functions in a block. If all the registers are not used, the hexadecimal value $FFFFFFFF is assigned to the unused ones (those with the highest numbers).

This way, if the next block is programmed in the first channel, the CNC will pass on to the PLC the following information.

M3.S1 S1=1000 M4.S2 S2=500

Clockwise rotation of spindle S1 at 1000 rpm and counterclockwise rotation of spindle S2 at 500 rpm.

MFUN1C1	MFUN2C1	MFUN3C1 - MFUN7C1
3	4	$FFFFFFFF

SPN1C1	SPN2C1	SPN3C1 - SPN7C1
1	2	$FFFFFFFF

If a function is programmed in the block without mentioning the spindle, it will assume the master spindle of the channel.

Commands SP1FUN* - SP4FUN*. Checking if a spindle receives a function from any channel.

In order to know whether a particular spindle has received a particular function or not, it is possible to check all the registers one by one or use the following commands to check all of them at the same time.

SP1FUN*	For the spindle 1.
SP2FUN*	For the spindle 2.
SP3FUN*	For the spindle 3.
SP4FUN*	For the spindle 4.

Example to detect if the first spindle has received an M5 function from any channel. If it has been programmed, it will return a "1", and a "0" if otherwise.

```
CPS HFUNC1* EQ 77 = …
```

3.5 Consulting and modifiable signals of the spindle.

3.5.1 Consultation signals.

REVOK

There is a mark for each spindle. The mnemonics for each spindle are the following:

REVOK1 (can also be programmed as REVOK)

REVOK2 REVOK3 REVOK4

It indicates whether the actual (real) spindle rpm match the ones programmed (=1) or not (=0). In other words, whether they are within the percentages set by (a.m.p.) UPSPDLIM and LOSPDLIM.

- When the spindle is stopped, M5, REVOK is (=1).
- With M3 and M4, the CNC sets this mark high (=1) when the actual spindle rpm match the ones programmed.
- When working in closed loop (M19 or G63), the CNC sets this mark low (=0) while the spindle is moving and high (=1) when the spindle is in position.

The REVOK signal may be used to handle the Feedhold signal and avoid machining at lower or higher rpm than desired.

SYNCMASTER

There is a mark for each spindle. The mnemonics for each spindle are the following:

SYNCMASTER1 SYNCMASTER2 SYNCMASTER3 SYNCMASTER4

This mark is activated at the master spindle and indicates that a synchronization has begun with the #SYNC instruction. When activating a synchronization, the CNC activates the ENABLE signal at both spindles and waits for the SERVOON signal (if DWELLis used).

When a spindle synchronization is active, the PLCCNTL, INHIBIT and SPDLEREV signals of both master and slave are ignored. Likewise, while threading, only the feedback and reference signal of the main spindle are taken into account.

SYNCHRON

There is a mark for each spindle. The mnemonics for each spindle are the following:

SYNCHRON1 SYNCHRON2 SYNCHRON3 SYNCHRON4

This mark is activated at the slave spindle and indicates that a synchronization has begun with the #SYNC instruction. When activating a synchronization, the CNC activates the ENABLE signal at both spindles and waits for the SERVOON signal (if DWELLis used).

When a spindle synchronization is active, the PLCCNTL, INHIBIT and SPDLEREV signals of both master and slave are ignored. Likewise, while threading, only the feedback and reference signal of the main spindle are taken into account.

SYNCHRONP

There is a mark for each spindle. The mnemonics for each spindle are the following:

SYNCRONP1 SYNCRONP2 SYNCRONP3 SYNCRONP4

This mark is activated at the slave spindle and indicates that a position synchronization has begun. This mark may be used to distinguish between synchronization in position or in velocity and to know which mark, SYNSPEED or SYNCPOSI, to attend to.

SYNSPEED

There is a mark for each spindle. The mnemonics for each spindle are the following:

SYNSPEED1 SYNSPEED2 SYNSPEED3 SYNSPEED4

This mark is activated at the slave spindle when it is synchronized in speed.

This mark is set to (=0) when exceeding the maximum speed error allowed, whose default value is set by machine parameter DSYNCVELW.

SYNCPOSI

There is a mark for each spindle. The mnemonics for each spindle are the following:

SYNCPOSI1 SYNCPOSI2 SYNCPOSI3 SYNCPOSI4

This mark is activated at the slave spindle when it is synchronized in position.

This mark is set to (=0) when exceeding the maximum position error allowed, whose default value is set by machine parameter DSYNCPOSW.

GEAROK

There is a mark for each spindle. The mnemonics for each channel are the following.

GEAROK1 (can also be programmed as GEAROK)

GEAROK2 GEAROK3 GEAROK4

The spindle activates this mark when the parameter set selected at the CNC and at the PLC are the same. In order for both parameter sets to coincide, funciton M41 must be active at the CNC and the GEAR1 mark at the PLC, M42 with GEAR2 and so on.

If both parameter sets do not coincide, the CNC executes no action. Include this mark in the PLC routine to define the actions to carry out when both parameter sets do not coincide, such as stop the spindle or interrupt the execution of the part-program.

3.5.2 Modifiable signals.

GEAR1
GEAR2
GEAR3
GEAR4

There is a mark for each spindle. The mnemonics for each spindle are the following: Here is an example of the mnemonics for GEAR1; it is the same for the rest of the marks.

GEAR1SP1 (can also be programmed as GEAR1)

GEAR1SP2 GEAR1SP3 GEAR1SP4

The PLC uses these marks to let the CNC know which spindle range (gear) is currently selected (=1). When requesting a gearchange, the CNC informs the PLC about it using auxiliary functions: M41, M42, M43 or M44. The gear change concludes when the PLC receives the confirmation signal AUXEND.

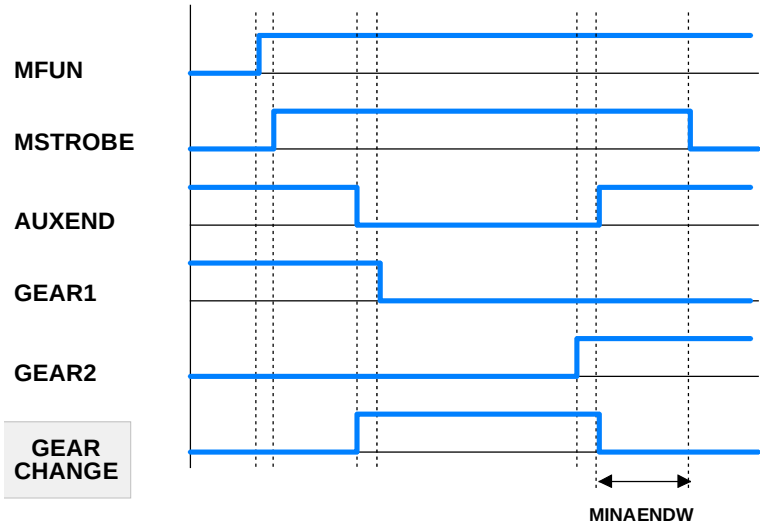
The CNC assumes the parameter set of the new gear when the spindle speed reaches the speed set in parameter SZERO and when the PLC receives the confirmation of one of marks GEAR1 through GEAR4.

On Sercos spindles, when the range change involves a gear change (NPULSES, INPUTREV, OUTPUTREV), functions M41 to M44 also change the gear at the drive.

When the parameter set selected at the CNC and at the PLC are the same, the spindle activates the GEAROK mark. In order for both parameter sets to coincide, funciton M41 must be active at the CNC and the GEAR1 mark at the PLC, M42 with GEAR2 and so on.

Example of a GEAR1 GEAR2 change.

If Gear 2 (M42) is requested while gear 1 is active.



1 The CNC indicates to the PLC the gear requested with MFUN1=42 and sets the MSTROBE mark to (=1).

2 When detecting the request, the PLC sets an internal indicator.

```
DFU MSTROBE AND CPS MFUN* EQ 42 = SET M1002
```

3 The change begins and lets the CNC know by setting AUXEND (=0).

```
NOT M1002 AND <rest of conditions> \
= AUXEND \
= (starts the gear change)
```

During the gear change, the CNC is informed that gear 1 is dropped and gear 2 is now selected. The active gear indicator, GEAR1 through GEAR4 must be set before activating the AUXEND signal.

```
I21 = GEAR1
I22 = GEAR2
```

4 Once the gear change is over, it cancels the indicator (M1002) and it lets the CNC know by setting AUXEND high (=1).

(GEAR change completed) = RES M1002

The AUXEND mark must be kept high (=1) longer than the time period set by g.m.p. MINAENDW so the CNC cancels the MSTROBE mark and concludes the gear change.

PLCCNTL SANALOG

There is one signal for each spindle. The mnemonics for each spindle are the following:

PLCCNTL1 (can also be programmed as PLCCNTL)

PLCCNTL2 PLCCNTL3 PLCCNTL4

SANALOG1 (can also be programmed as SANALOG)

SANALOG2 SANALOG3 SANALOG4

These signals are used with analog spindles, position-Sercos, velocity-Sercos and Mechatrolink.

When the PLC sets the PLCCNTL mark high (=1), the spindle decelerates with a ramp until it stops and it is then controlled by the PLC. The SANALOG register sets the spindle command voltage to be applied. The spindle control via PLC is used, for example, for oscillating the spindle during a gear change.

- Analog spindle.

  10 V of velocity command correspond to SANALOG = 32767. In other words:

  For 4V, program SANALOG = (4x32767)/10 = 13107

  For 4V, program SANALOG = (-4x32767)/10 = -13107

- Velocity-Sercos spindle.

  The command in SANALOG will be given in 0.0001 rpm.

- Sercos spindle in position.

  The command in SANALOG will be given in 0.0001 degrees.

- Mechatrolink spindle.

  The command in SANALOG will be given in hundredths of a Hertz.

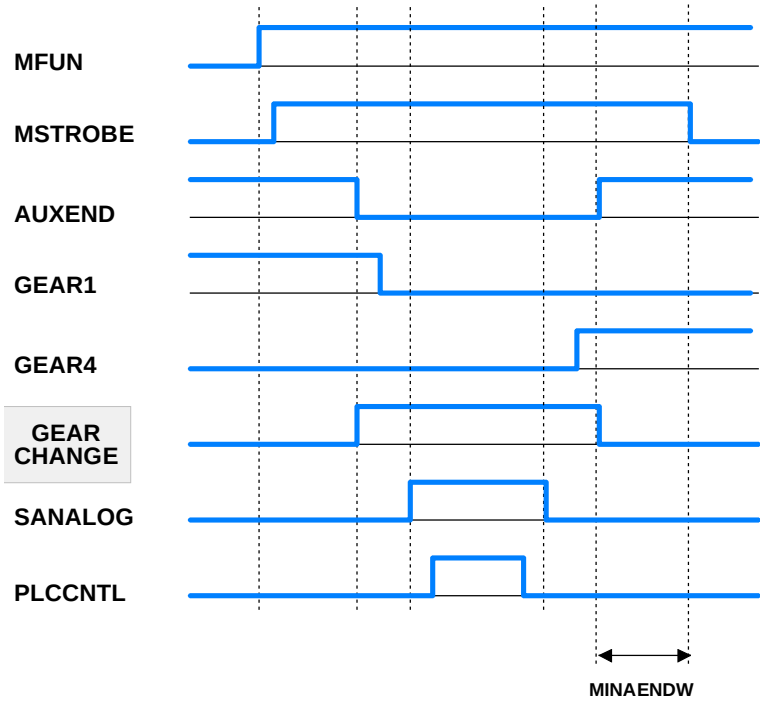
The command indicated in SANALOG is not applied with a ramp; therefore, it is up to the PLC program to apply the command gradually when necessary.

The spindle reference point is not lost when the spindle is controlled via PLC. There is no need to home the spindle again when its control is switched back to the CNC.

The PLC has no priority over a spindle synchronization. When attempting to control a synchronized spindle (either master or slave) using PLCCNTL, a warning will be issued indicating that it is not possible. Also, if the gear change of a synchronized spindle involves a command from the PLC, this change will not be possible.

Example similar to the one used for GEAR1 through GEAR4

The spindle oscillation during a gear change is controlled by the PLC. Gear 4 is requested while gear 1 is active.



The example for the GEAR1 through GEAR4 signals describes how to detect and carry out the gear change. This example shows how to control the spindle oscillation during a gear change.

The PLC sets SANALOG to the value corresponding to the residual velocity command and activates the PLCCNTL mark to indicate that the spindle is controlled by the PLC.

When done, the PLCCNTL mark must be set low (=0) and the SANALOG signal must be set to "0".

SPDLEREV

There is a mark for each spindle. The mnemonics for each spindle are the following:

SPDLEREV1 (can also be programmed as SPDLEREV)

SPDLEREV2 SPDLEREV3 SPDLEREV4

When the PLC sets this mark high (=1), the CNC reverses the spindle turning direction. To do this, it decelerates and accelerates applying the ramps set by machine parameters. This mark is only taken into account when the spindle works in open loop.

If an M2 or M4 function is executed while the SPDLEREV mark is high (=1), the spindle will turn in the opposite direction to the one assigned to the function.

If the PLC activates or cancels the SPDLEREV mark when the spindle is controlled by the PLCA (PLCCNNTL mark active), the CNC does not generate ramps to invert the SANALOG command.

PLCM3
PLCM4
PLCM5

The PLC offers a mark for each spindle. The mnemonics for each spindle are the following: Here is an example of the mnemonics for PLCM3; it is the same for the rest of the spindles.

PLCM3SP1 (can also be programmed as PLCM3)

PLCM3SP2 PLCM3SP3 PLCM3SP4

The PLC turns this mark on to let the CNC know that it must execute the corresponding M function at the indicated spindle. The spindle must belong to a channel, M functions cannot be sent to spindles that are not assigned to any channel. The PLC may change the spindle

speed using the variable "(V.)PLC.S.sn", but without generating a gear change even if the gear change is automatic (parameter AUTOGEAR).

These M functions are treated the same way as those executed from the CNC. When the CNC activates one of these marks, the CNC activates the MSTROBE mark and writes the corresponding M function into the MFUN register. Once the M function has been synchronized at the PLC (AUXEND signal), the CNC starts sending the velocity command to that spindle, updates (refreshes) the M function history and deactivates the mark at the PLC.

The CNC will admit M functions even if the channel is in execution (executing a program, jogging an axis, etc.) as long as the channel status is other than "In error" and "Not ready" and the channel is not doing a reset or validating machine parameters. If during tool inspection, the PLC changes the turning direction of a spindle using these marks, the change will be identified when repositioning and will appear as pending to reposition.

The CNC ignores the PLC requests in the following cases, when the CNC ignores the mark set by the PLC and it erases it so the request does not remain pending.

- When the spindle is working as a C axis.
- When the spindle is threading (rigid tapping, normal tapping or electronic threading).
- When the CNC status is "In error" or "Not ready", is doing a reset or validating parameters.

If the PLC activates these marks while synchronizing another M function of the program being executed or while homing the spindle, the PLC keeps this mark activated until the CNC can attend to it.

SLIMITACSPDL

There is a mark for each spindle. The mnemonics for each spindle are the following:

SLIMITACSPDL1 (can also be programmed as SLIMITACSPDL)

SLIMITACSPDL2 SLIMITACSPDL3 SLIMITACSPDL4

If the PLC activates this mark, the CNC activates the speed safety limits (parameter SLIMIT) on the indicated spindle during the execution of the current block. If the PLC deactivates this mark, the CNC restores the programmed speed.

3.6 Consulting and modifiable signals of the spindle.

3.6.1 Consultation signals.

FREE

There is a mark for each channel. The mnemonics for each channel are the following.

FREEC1 FREEC2 FREEC3 FREEC4

The CNC channel sets this signal high (=1) to indicate to the PLC that it is ready to accept a new block, sent using the CNCEX command.

WAITOUT

There is a mark for each channel. The mnemonics for each channel are the following.

WAITOUTC1 WAITOUTC2 WAITOUTC3 WAITOUTC4

It is applied to channel synchronization. The CNC channel sets this signal high (=1) to indicate to the PLC that it is waiting for a synchronization signal. Synchronization signals may be executed from the part-program using the #WAIT or #MEET instructions.

SYNC

There is a register for each channel. The mnemonics for each channel are the following.

SYNC1 SYNC2 SYNC3 SYNC4

This register is used when using, from one channel, a particular spindle for synchronization even if the spindle is in another channel. For example, in the case of dual-turret lathe with a single spindle.

- With the G33 function, when threading with a particular spindle.
- With the G95 function, when programming the feedrate as a function of the turning speed of a particular spindle.

To do that, the PLC indicates in channel register SYNC the spindle to be used, only for synchronization. The SYNC register will take values 1 through 4; when assigning a 0 value, it will use the master spindle of the channel.

The CNC will check the contents of this register at the beginning of the block. If the PLC modifies this register during the execution of the block, the change will not effective until the beginning of the next block.

3.6.2 Modifiable signals.

NOWAIT

There is a mark for each channel. The mnemonics for each channel are the following.

NOWAITC1 NOWAITC2 NOWAITC3 NOWAITC4

It is applied to channel synchronization. The PLC sets this mark high (=1) to cancel all the synchronizations with the CNC channel.

For example, with the NOWAITC1 signal set to (=1), the waits programmed in any channel with the #WAIT, instruction and that refer to a mark of channel 1, they finish immediately and the program execution resumes.

3.7 Consulting and modifiable signals of the tool manager.

3.7.1 Consultation signals.

Communication between the manager and the PLC.

M/R	Channel ·1·	Channel ·2·	Channel ·3·	Channel ·4·
R	TMOPERATIONC1 TMOPERATION	TMOPERATIONC2	TMOPERATIONC3	TMOPERATIONC4
This register indicates the type of operation to be carried out by the tool manager.				
M	TMOPSTROBEC1 TMOPSTROBE	TMOPSTROBEC2	TMOPSTROBEC3	TMOPSTROBEC4
The CNC channel activates this mark on to let the PLC know that it must execute the operation indicated by TMOPERATION.				
R	MZIDC1	MZIDC2	MZIDC3	MZIDC4
This register indicates which magazine contains the tool requested by the channel. When two magazines are involved in a tool change, the lower portion of this register indicates the destination magazine for the tool and the higher portion the source magazine for the tool.				

M/R	Magazine ·1·	Magazine ·2·	Magazine ·3·	Magazine ·4·
R	LEAVEPOSMZ1 LEAVEPOS	LEAVEPOSMZ2	LEAVEPOSMZ3	LEAVEPOSMZ4
This register indicates the magazine position to leave the tool.				
R	TAKEPOSMZ1 TAKEPOsS	TAKEPOSMZ2	TAKEPOSMZ3	TAKEPOSMZ4
This register indicates the magazine position of the tool to be taken.				
R	NEXTPOSMZ1 NEXTPOS	NEXTPOSMZ2	NEXTPOSMZ3	NEXTPOSMZ4
This register indicates the magazine position occupied by the next tool.				

Manager in error state.

M/R	Magazine ·1·	Magazine ·2·	Magazine ·3·	Magazine ·4·
M	TMINEMZ1 TMINEM	TMINEMZ2	TMINEMZ3	TMINEMZ4
The CNC activates this mark to inform the PLC that an emergency has occurred at the tool manager.				

Tool monitoring.

M/R	Channel ·1·	Channel ·2·	Channel ·3·	Channel ·4·
M	TWORNOUTC1 TWORNOUT	TWORNOUTC2	TWORNOUTC3	TWORNOUTC4
The CNC channel activates this mark to "tell" the PLC that the tool has been rejected because it is worn out (real life > maximum life span).				

3.7.2 Modifiable signals.

Communication between the manager and the PLC.

M/R	Channel ·1·	Channel ·2·	Channel ·3·	Channel ·4·
M	SPDLTOGRC1 SPDLTOGR	SPDLTOGRC2	SPDLTOGRC3	SPDLTOGRC4
The PLC must turn this mark on after taking the tool from the spindle to ground. Use it with a magazine that admits ground tools.				
M	GRTOSPDLC1 GRTOSPDL	GRTOSPDLC2	GRTOSPDLC3	GRTOSPDLC4
The PLC must turn this mark on after taking the tool from the ground to the spindle. Use it with a magazine that admits ground tools.				

M/R	Magazine ·1·	Magazine ·2·	Magazine ·3·	Magazine ·4·
M	MZTOCH1MZ1 MZTOCH1	MZTOCH1MZ2	MZTOCH1MZ3	MZTOCH1MZ4
The PLC must activates this mark after taking the tool from the magazine to the changer arm 1. Use it with an asynchronous magazine or synchronous with arm.				
M	CH1TOSPDLMZ1 CH1TOSPDL	CH1TOSPDLZ2	CH1TOSPDLMZ3	CH1TOSPDLMZ4
The PLC must activates this mark after taking the tool from the changer arm ·1· to the spindle. Use it with an asynchronous magazine or synchronous with arm.				
M	SPDLTOCH1MZ1 SPDLTOCH1	SPDLTOCH1MZ2	SPDLTOCH1MZ3	SPDLTOCH1MZ4
The PLC must activates this mark after taking the tool from the spindle to the changer arm ·1·. Use it with an asynchronous magazine with one changer arm.				
M	SPDLTOCH2MZ1 SPDLTOCH2	SPDLTOCH2MZ2	SPDLTOCH2MZ3	SPDLTOCH2MZ4
The PLC must activates this mark after taking the tool from the spindle to the changer arm ·2·. Use it with an asynchronous magazine or synchronous with arm.				
M	CH1TOMZ1 CH1TOMZ	CH1TOMZ2	CH1TOMZ3	CH1TOMZ4
The PLC must activates this mark after taking the tool from the changer arm 1 to the magazine. Use it with an asynchronous magazine or synchronous with arm.				
M	CH2TOMZ1 CH2TOMZ	CH2TOMZ2	CH2TOMZ3	CH2TOMZ4
The PLC must activates this mark after taking the tool from the changer arm 2 to the magazine. Use it with an asynchronous magazine or synchronous with arm.				
M	MZTOSPDLMZ1 MZTOSPDL	MZTOSPDLMZ2	MZTOSPDLMZ3	MZTOSPDLMZ4
The PLC must turn this mark on after taking the tool from the magazine to the spindle. Use it with a synchronous magazine (without arm).				
M	SPDLMTOMZ1 SPDLMTOMZ	SPDLMTOMZ2	SPDLMTOMZ3	SPDLMTOMZ4
The PLC must turn this mark on after taking the tool from the spindle to the magazine. Use it with a synchronous magazine (without arm).				
M	MZROTMZ1 MZROT	MZROTMZ2	MZROTMZ3	MZROTMZ4
The PLC must activate this mark when the turret has rotated. Use it with a turret type magazine.				
M	TCHANGEOKMZ1 TCHANGEOK	TCHANGEOKMZ2	TCHANGEOKMZ3	TCHANGEOKMZ4
The PLC must activate this mark when the tool change has ended (M06).				
M	MZPOSMZ1 MZPOS	MZPOSZ2	MZPOSMZ3	MZPOSMZ4
The PLC must indicate the current magazine position in this register.				

Manager in error state.

M/R	Magazine ·1·	Magazine ·2·	Magazine ·3·	Magazine ·4·
M	SETTMEMZ1 SETTMEM	SETTMEMZ2	SETTMEMZ3	SETTMEMZ4
The PLC activates this mark to activate the tool manager emergency.				
M	RESTMEMZ1 RESTMEM	RESTMEMZ2	RESTMEMZ3	RESTMEMZ4
The PLC activates this mark to cancel the tool manager emergency.				

Tool monitoring.

M/R	Channel ·1·	Channel ·2·	Channel ·3·	Channel ·4·
M	CUTTINGONC1 CUTTINGON	CUTTINGONC2	CUTTINGONC3	CUTTINGONC4
When associating a maximum life span to a tool (monitoring), the CNC checks this mark to find out whether the tool is machining (mark activated) or not (mark deactivated).				
M	TREJECTC1 TREJECT	TREJECTC2	TREJECTC3	TREJECTC4
If the PLC activates this mark, the CNC interprets that the tool must be rejected.				

3.

LOGIC CNC INPUTS AND OUTPUTS.

Consulting and modifiable signals of the tool manager.

FAGOR

CNC 8060
CNC 8065

(REF: 1402)

CNC-PLC COMMUNICATION.

4

–M– and –H– functions with channels.

The M and H functions are exchanged by channel. When using several channels, the marks and registers of these functions must indicate the channel number they refer to. If no channel number is indicated, the marks and registers refer to the first channel.

–S– functions with multiple spindles.

The exchange of S functions is independent from the channel. When using several spindles, the marks and registers of these functions refer to the spindle number. The spindle number is determined by its logic number.

4.1 Auxiliary –M– functions.

The CNC may have up to 4 channels and each channel can execute a part-program in parallel with the rest. This means that each channel can execute seven auxiliary functions simultaneously. The auxiliary functions executed from each channel are treated independently; to do that, each channel has its own marks and registers.

Since each channel may have four spindles, it is possible to program in the same block 6 non-spindle-related M functions, the startup of all four spindles M3 / M4 and a speed for each of them involving an automatic gear change. This means that, because some functions are generated automatically, it may exceed the maximum of seven auxiliary functions per block. In this case, the CNC will send the M functions out to the PLC in two stages.

Registers used in the communication between the channels and the PLC.

Each channel has 32-bit registers MFUN1 to MFUN7 to indicate to the PLC which auxiliary M functions are programmed in the execution block.

MFUN1C1 - MFUN7C1	for the first channel.
MFUN1C2 - MFUN7C2	for the second channel.
MFUN1C3 - MFUN7C3	for the third channel.
MFUN1C4 - MFUN7C4	for the fourth channel.

Each one of them indicates the number of one of the M functions programmed in the block. If all the registers are not used, the CNC assigns $FFFFFFFF to the unused ones (those with the highest numbers).

This way, if functions M100 and M135 are programmed in the first channel and functions M88 and M75 in the second channel, the CNC will transfer the following data.

MFUN1C1	MFUN2C1	MFUN3C1 - MFUN7C1
100	135	$FFFFFFFF

MFUN1C2	MFUN2C2	MFUN3C2 - MFUN7C2
88	75	$FFFFFFFF

Commands MFUNC1* - MFUNC4*. Checking if a function has been programmed in the channel.

To know whether a particular M function is programmed in the execution block, use one of the following methods:

- Check all the MFUN registers one by one until that particular M function is found or until one of them has the value of $FFFFFFFF.
- Use one of the following commands to check all the MFUN registers of the channel at the same time.

MFUNC1*	For channel 1.
MFUNC2*	For channel 2.
MFUNC3*	For channel 3.
MFUNC4*	For channel 4.

Example for detecting M04 in channel 1. If it has been programmed, it will return a "1", and a "0" if otherwise.

```
CPS MFUNC1* EQ 4 = ...
```

Transferring M functions to the PLC and synchronizing the execution

Since the M functions may be programmed together with the movement of the axes, in the same block, it must be indicated when the function is to be sent out to the PLC and when it will be checked that it has been already executed (synchronization). In the table of machine parameters for the –M– functions, parameter SYNCHTYPE indicates when the function is

sent and when the PLC execution is synchronized. The sending and synchronizing types may be the following:

Synchronization.	Meaning.
Without synchronization.	M function not synchronized.
Before - Before.	The M function is sent to the PLC and synchronized before the movement.
Before - After.	The M function is sent to the PLC before the movement and synchronized after the movement.
After - After.	The M function is sent to the PLC and synchronized after the movement.

M functions with different types of synchronization may be programmed in the same block. Each one of them will be sent out to the PLC at the right moment. The transfer of auxiliary M functions is described later on in this chapter. See *"4.4 Transferring auxiliary functions -M-, -H-, -S-."* on page 73.

The functions may be set as follows:

M11 not synchronized.

M12 is sent and synchronized before the movement.

M13 is sent before and synchronized after the movement.

M14 is sent and synchronized after the movement.

When executing a block like this:

```
X100 F1000 M11 M12 M13 M14
```

The functions are transferred as follows:

1. It sends the M11 M12 and M13 functions out to the PLC.
2. waits for the PLC to execute the M12.
3. The CNC moves the axis to X100.
4. It sends the M14 function to the PLC.
5. It waits until the PLC executes the M13 and M14.

4.2 Auxiliary –H– functions.

Up to 7 M and 7 H functions may be programmed in a block. The treatment of the auxiliary H functions is similar to the M functions without synchronization.

The CNC may have up to 4 channels and each channel can execute a part-program in parallel with the rest. This means that each channel can execute seven auxiliary functions simultaneously. The auxiliary functions executed from each channel are treated independently; to do that, each channel has its own marks and registers.

Registers used in the communication between the channels and the PLC.

Each channel has 32-bit registers HFUN1 to HFUN7 to indicate to the PLC which auxiliary H functions are programmed in the execution block.

HFUN1C1 - HFUN7C1	for the first channel.
HFUN1C2 - HFUN7C2	for the second channel.
HFUN1C3 - HFUN7C3	for the third channel.
HFUN1C4 - HFUN7C4	for the fourth channel.

Each one of them indicates the number of one of the H functions programmed in the block. If all the registers are not used, the CNC assigns $FFFFFFFF to the unused ones (those with the highest numbers).

This way, if functions H10 and H13 are programmed in the first channel and functions H8 and H10 in the second channel, the CNC will transfer the following data.

HFUN1C1	HFUN2C1	HFUN3C1 - HFUN7C1
10	13	$FFFFFFFF

HFUN1C2	HFUN2C2	HFUN3C2 - HFUN7C2
8	10	$FFFFFFFF

Commands HFUNC1* - HFUNC4*. Checking if a function has been programmed in the channel.

To know whether a particular H function is programmed in the execution block, use one of the following methods:

- Check all the HFUN registers one by one until that particular H function is found or until one of them has the value of $FFFFFFFF.
- Use one of the following commands to check all the HFUN registers of the channel at the same time.

HFUNC1*	For channel 1.
HFUNC2*	For channel 2.
HFUNC3*	For channel 3.
HFUNC4*	For channel 4.

Example for detecting H77 in channel 1. If it has been programmed, it will return a "1", and a "0" if otherwise.

```
CPS HFUNC1* EQ 77 = …
```

Transferring H functions to the PLC and synchronizing the execution

The H functions are not synchronized and are sent out to the PLC at the beginning of block execution. The transfer of auxiliary H functions is described later on in this chapter. See *"4.4 Transferring auxiliary functions -M-, -H-, -S-."* on page 73.

When executing a block like this:

```
X100 F1000 H11 H12
```

The functions are transferred as follows:

1. Functions H11 and H12 are sent out to the PLC
2. It does not wait for confirmation and the CNC moves the axis to X100.

4.3 Auxiliary –S– function.

The auxiliary S function indicates the spindle turning speed with M03 and M04 or the angular position with M19. All the spindles of a channel may be controlled independently; in other words, each spindle may be given a different command.

When using channels, the spindles may be distributed indistinctly between them. In this case, a channel can control a spindle of another channel. The marks and registers refer to the spindle regardless of the channel they belong to.

Registers used in the communication between the channels and the PLC.

The CNC indicates to the PLC which S functions are programmed in the execution block using 32-bit registers SFUN1 through SFUN4. These registers refer to the spindle number; they are independent from the channel where the spindle is. The spindle number is determined by its logic number that is set by the order they were defined in the machine parameter SPDLNAME.

SFUN1	First spindle.
SFUN2	Second spindle.
SFUN3	Third spindle.
SFUN4	Fourth spindle.

Each one of them indicates the value of one of the S functions programmed. If all the registers are not used, the CNC assigns $FFFFFFFF to the unused ones (those with the highest numbers).

This way, if a block contains functions S1000 and S1=550, the CNC will transfer the following information to the PLC:

SFUN1	SFUN2	SFUN3	SFUN4
1000	550	$FFFFFFFF	$FFFFFFFF

Commands SP1FUN* - SP4FUN*. Check if an auxiliary function has been programmed for a spindle.

Considering the possible channels/spindles combinations, these functions are available to make it easier to manage the auxiliary M functions associated with each spindle. Each one indicates if any M3, M4, etc. type M function has been programmed in any channel.

SP1FUN*	For the spindle 1.
SP2FUN*	For the spindle 2.
SP3FUN*	For the spindle 3.
SP4FUN*	For the spindle 4.

Checks if the M5 function has been sent to spindle 1 from a channel.

```
CPS SP1FUN* EQ 5 = ...
```

Transferring S functions to the PLC and synchronizing the execution

The S function with M03 and M04 is always executed at the beginning of the block and the CNC waits for confirmation before going on executing the program. When working with M19, the CNC treats the spindle like a regular linear axis. It only sends the M19 out to the PLC.

The transfer of the S function is described later on in this chapter. See *"4.4 Transferring auxiliary functions -M-, -H-, -S-."* on page 73.

4.4 Transferring auxiliary functions -M-, -H-, -S-.

The M and H functions are transferred per channel. Transferring S functions does not depend on the channel. When executing a block that contains M, H, S functions, the following information is transferred to the PLC.

Transferring –M– functions.

The CNC assigns the numbers of the M functions programmed in the block to registers MFUN1 through MFUN7. Some M functions have an associated function (DMxx) that is activated when sending the M to the PLC.

M00	M01	M02	M03	M04
M05	M06	M08	M09	M19
M30	M41	M42	M43	M44

The CNC activates the general logic output MSTROBE to "tell" the PLC that it must execute them. This mark is kept active for a time period indicated by parameter MINAENDW.

Depending on the type of synchronization, the CNC will either wait or not for the general input AUXEND to be activated indicating the end of the PLC execution. The type of synchronization is defined in the machine parameters

The CNC cancels the general logic output MSTROBE to conclude the execution.

Transferring -H- functions.

The CNC assigns the numbers of the H functions programmed in the block to registers HFUN1 through HFUN7.

The CNC activates the general logic output HSTROBE to "tell" the PLC that it must execute them. This mark is kept active for a time period indicated by parameter MINAENDW.

After this time period, the CNC considers its execution completed because there is no synchronization.

When sending several blocks in a row just having H functions, the CNC waits twice the time indicated in parameter MINAENDW.

```
N10 H60
N20 H30 H18
N30 H40
```

Transferring -S- functions.

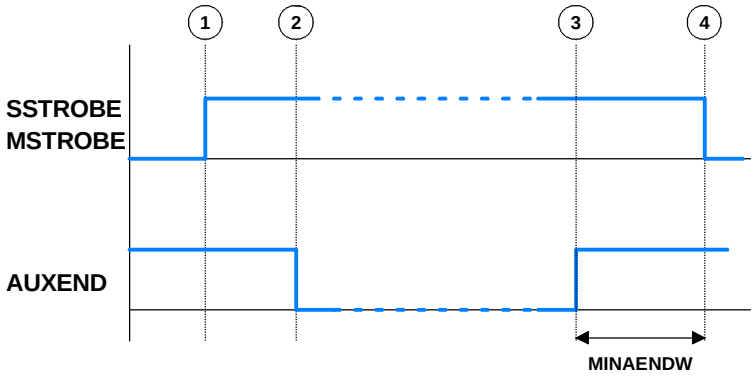
The CNC assigns the values of the S programmed in each spindle to registers SFUN1 through SFUN4.

The CNC activates the general logic output SSTROBE to "tell" the PLC that it must execute it. The CNC waits for the general input AUXEND to be activated indicating the end of the PLC execution.

The CNC cancels the general logic output SSTROBE to conclude the execution.

4.4.1 Synchronized transfer.

This type of transfer takes place with the S function and with the M functions set with synchronization.



When the PLC is requested to execute several M or S functions at the same time, the corresponding SSTROBE or MSTROBE signals are activated; but the CNC waits for a single "AUXEND" signal to end all of them.

Transferring –M– functions.

1 The CNC indicates in registers MFUN1 to MFUN7 of the channel the M functions programmed in the block and it activates the MSTROBE mark so the PLC executes them.

2 The PLC must deactivate the AUXEND mark to let the CNC know that the execution has begun.

3 Once the required auxiliary functions have been executed, the PLC must activate the AUXEND mark to let the CNC know that the execution has ended.

The AUXEND mark must be kept active longer than the time period established by parameter MINAENDW.

4 After this time, the CNC deactivates the MSTROBE mark thus ending the execution of the functions.

Transferring -S- functions.

1 The CNC indicates in register SFUN1 to SFUN4 the S value programmed in the block and activates the SSTROBE mark so the PLC executes it.

2 The PLC must deactivate the AUXEND mark to let the CNC know that the execution has begun.

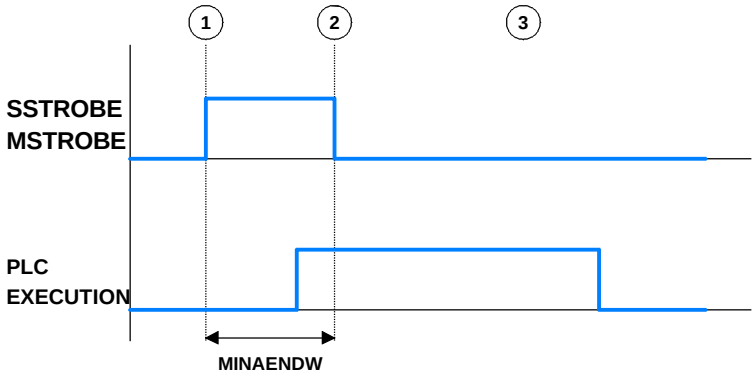
3 After selecting the requested S, the PLC must activate the AUXEND mark to let the CNC know that the execution has ended.

The AUXEND mark must be kept active longer than the time period established by parameter MINAENDW.

4 After this time, the CNC deactivates the MSTROBE mark thus ending the execution of the function.

4.4.2 Non-synchronized transfer.

This type of transfer takes place with the H function and with the M functions set without synchronization.



Transferring -M- functions

1 The CNC indicates in registers MFUN1 to MFUN7 of the channel the M functions programmed in the block and it activates the MSTROBE mark so the PLC executes them.

2 The CNC keeps the MSTROBE mark active for a time period indicated by parameter MINAENDW.

3 After this time, the CNC goes on executing the program regardless of the time required by the PLC to execute that function.

Transferring -H- functions

1 The CNC indicates in registers HFUN1 to HFUN7 of the channel the H functions programmed in the block and it activates the HSTROBE mark so the PLC executes them.

2 The CNC keeps the HSTROBE mark active for a time period indicated by parameter MINAENDW.

3 After this time, the CNC goes on executing the program regardless of the time required by the PLC to execute that function.

Considerations for transferring these functions.

The value of parameter MINAENDW should be the same or longer than the PLC program execution period parameter PRGFREQ in order to ensure that the PLC detects that signal.

When sending non-synchronized H or M functions corresponding to consecutive blocks of the same program, the CNC waits between blocks for a time period indicated by MINANEDW so the PLC can read all the functions.

4.5 Displaying PLC errors and messages.

The PLC has 1024 marks for displaying messages and another 1024 marks for displaying errors at the CNC. When the mark is active, the message or the error is active. These errors and messages are common to all the channels.

MSG1 - MSG1024 for displaying messages.

ERR1 - ERR1024 for displaying errors.

The PLC has a table for defining the messages and the errors. For more information on how to edit this table, refer to the operating manual.

PLC messages.

When activating one of the MSG marks, the status bar of the CNC window for PLC messages shows the message number and its associated text. If the message has a file with additional information associated with it, an access icon will appear to the left of the message.



The message does not have a file with additional information.



The message has a file with additional information.

When there are more than one message activated, it always shows the one with the highest priority (the one with the lowest number). The PLC-messages window shows the "+" sign meaning that there are more messages activated by the PLC. To display the whole list, press [CTRL] + [M].

If the "Show" field of the message is selected, the CNC screen shows the additional information file and if there isn't one, a blue window with the text of the message. If the "Show" field is not selected, to show the additional information file, you must expand the list of messages, select a message and press [ENTER] or click on the message. To close the additional data window, press [ESC].

PLC errors.

When activating one of the ERR marks, the CNC interrupts the execution of the part-program and it displays a window in the middle of the screen showing the error number and its associated text. If the error has the "Emergen" field selected, the error will open the emergency relay of the CNC.



If the error has a file with additional information associated with it, an access icon will appear to the right of the error number. If the error has the "Show" field selected, the CNC shows the additional information file directly on the screen. If the "Show" field is not selected, the additional information file will be displayed when pressing the [HELP] key or when clicking on the icon mentioned earlier. To close the additional data window, press [ESC].

External inputs should be used to activate and deactivate error marks, thus preventing the CNC from receiving those errors at every new PLC cycle scan.

OPERATION AND PROGRAMMING

II

FAGOR

OPERATION AND INTERFACE.

5

The channels are managed from the interface like with a channel selector switch. There is an active channel at all times and it is the one being displayed. All the actions directed to a channel through the keyboard or operator panel will be directed to the active channel. When changing channels, it displays it and it becomes the active channel.

Channels that make up a group.

Two or more channels may be configured forming a group that will have the following characteristics.

- Each channel may be in a different work mode, except in jog and automatic modes. Toggling between the jog mode and automatic mode of a channel will affect all the channels of the group that are in any of these modes; the channels that are in a different mode will not be affected.
- A reset in any of the channels of the group affects all of them.
- Any error in any of the channels of the group interrupts the execution in all of them.

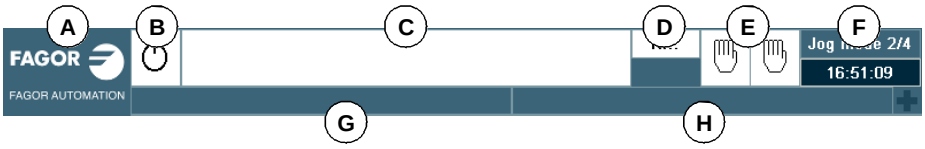
Operation of the operator panel.

By default, the actions directed to a channel through the operator panel will be directed to the active channel. However, this behavior may be different when set accordingly via PLC program.

- The feedrate override switch acts upon all the axes of the system at the same time; in other words, the feedrate override percentage change affects all the channels of the system.
- The spindle control keys (override, stop, etc.) act upon the master spindle of the active channel.
- The [START] or [STOP] keys only affect the active channel.
- The [RESET] key is only applied to the active channel and to the channels that are grouped with it.

5.1 The general status bar.

As mentioned earlier, the CNC can have four channels. The general status bar at the top of the screen shows the number of channels, which one is the active channel and the operating mode of each of them.



A Icon (customizable) identifying the manufacturer.

Clicking with the mouse or pressing on a touch-screen, the CNC shows the task window (same as pressing the keystroke sequence [CTRL]+[A]) that shows the list of the work modes and hotkeys of the CNC.

B Icon showing the status of the program of the active channel: The background color will be different depending on the status of the program.

Icon.	Meaning.
	Programmed stopped. Background color: White.
	Program in execution. Background color: Green.
	Program interrupted. Background color: Dark green.
	Program in error. Background color: Red.

C Program selected in the active channel for execution.

D Number of the block in execution. The bottom icon indicates that the Single-block execution mode is active.

E Information about the channels. Number of channels available and active channel (indicated in blue). Icons show which operating mode each channel is in. The channel synchronization window may be expanded using the [ALT]+[S] keys.

Clicking with the mouse or pressing a touch-screen to access the desired channel, doing it on the icon of the active channel, has the same effect as the [ESC] key.

Icon.	Meaning.
	Execution mode.
	Jog mode.
	MDI/MDA mode.

F Active work mode (automatic, manual, etc.) selected screen number and total number of screens available. System clock.

By clicking on the active work mode, the CNC shows the list of available pages and which ones are visible.

G Active CNC message.

H PLC messages.

Messages active at the CNC.

For each channel, the CNC shows the last message activated by the program that is running. The window shows the last message of the active channel. If there are messages in other channels, it will highlight the "+" sign next to the message window. To display the list of active messages, press the key combination [CTRL]+[O] or click on the CNC message line. The list of messages shows, next to each message, the channel where it is active.

PLC messages.

If the PLC activates two or more messages, the CNC displays the message with the highest priority and it will show the "+" sign indicating that there are more messages activated by the PLC. To display the list of active messages, press the key combination [CTRL]+[M] or click on the PLC message line.

On the message list and next to each message, a symbol will appear to indicate whether the message has an additional information file associated with it or not. To display a message, select it with the cursor and press [ENTER]. If the message has an additional information file, it will be displayed on the screen.

5.2 Changing channels. The channel selector switch.



The way to access the different channels is handled through the change key. This key may be configured either to access the channels sequentially or to show the list of available channels on the softkey menu.

It is also possible to change channels by clicking on the icons of the status bar.

It cannot change channels.

Every time the key is pressed, the CNC shows the next channel. It is a rotating change, so when pressed at the last channel, it shows the first one.

Key set up to show the system menu.

The system menu shows, on one of the softkey menus, the list of available channels. Pressing the corresponding softkey accesses the desired menu.

The other softkey menu may be disabled or show one of the following options:

- The menu shows the various pages or screens of the active work mode.
- The menu shows the components or work modes of the CNC.

Depending on how the system menu is configured, the system menu will be disabled in one of the following ways.

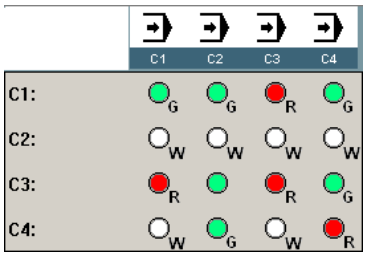
- The menu is disabled when pressing the [ESC] key, the previous menu key, when selecting one of its options or when changing the active component.
- The softkey menu remains until the change key is pressed again.

5.3 Channel synchronization window.

The channel synchronization window is available in all work modes. This window may be expanded using the key combination [ALT]+[S]. The synchronization is carried out using marks in the programs. The window shows for each channel whether it is waiting for synchronization marks or not and the status of those marks in the channel that originates them.

The different color LED's of the window show the status of the synchronization marks of each channel. On the left, the channels waiting for the marks and on top the channels that originate them.

LED.	Meaning.
White	The channel is not expecting any synchronism mark.
Green	The channel is expecting a synchronism mark. The mark is active (set to ·1·) in the channel that originates it.
Red	The channel is expecting a synchronism mark. The mark is not active (set to ·0·) in the channel that originates it.



Channel 1 (CH1) is expecting synchronism marks from the rest of the channels. The marks of channels 2 and 4 are active. The mark of channel 3 is not active.

Channel 2 is not expecting any synchronism mark.

(In the graphic, the white LED's are identified with the letter -W-, the green ones with the letter -G- and the red ones with the letter -R-).

5.4 The user tables.



Some tables are displayed per channel; in other words, they show the data of the active channel. These tables have a softkey for displaying the same table in each channel.

Zero offset and fixture offset tables.

The tables are common to the whole system. Each zero offset contains all the axes of the system; however, in channel, only the zero offsets of its axes are displayed. The corresponding vertical softkey may be used to display the axes of the rest of the channels.

When applying an offset from a channel, it is only applied to the axes that belong to the channel at the time.

Arithmetic parameter table.

Global arithmetic parameters.

There is one table per channel; by default, the CNC shows the table of the active channel. The corresponding vertical softkey may be used to display the table of the rest of the channels.

Local arithmetic parameters.

There are seven tables per channel; by default, the CNC shows the tables of the active channel. The corresponding vertical softkey may be used to display the table of the rest of the channels.

Common arithmetic parameters.

There is one table for the whole system.

PROGRAMMING IN A SYSTEM WITH CHANNELS.

6

This manual only shows the programming functions that are directly related to the multi-channel version. The rest of the functions, that are also valid both in a multi-channel CNC and in a single channel CNC, are described in the programming manual.

- Executing a program in the specified channel.

  From the automatic mode, each channel can execute its own program. Via part-program or in MDI/MDA mode, it is possible to command the execution of a program in a particular channel. When selecting the program to execute, it is possible to indicate its location.

- Executing a block in the indicated channel.

  Via part-program or MDI, it is possible to command the execution of a block in a particular channel.

- Axis swapping.

  Initially, each channel has some axes assigned to it as set by the machine parameters. While executing a program, a channel may release its axes, request new axes or reorder the existing ones.

- Spindle swapping.

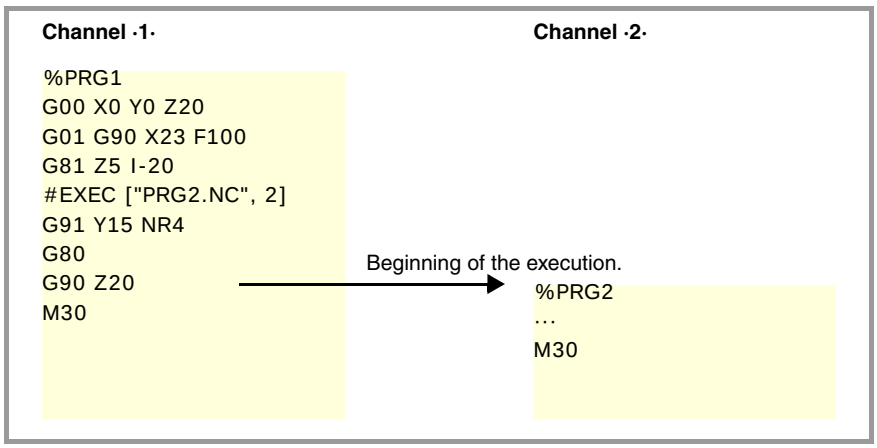
  Initially, each channel has some spindles assigned to it as set by the machine parameters. While executing a program, a channel may release its spindles or request new spindles.

- Communication and synchronization.

  These functions deal with such matters as swapping axes or spindles, synchronizing channels, etc.

6.1 Executing a program in the indicated channel.

With the #EXEC instruction, it is possible, from a program in execution, to begin the execution of a second program in another channel. The execution of the program starts in the indicated channel in parallel (at the same time) with the block following the #EXEC instruction. If the channel where the program to be executed is busy, the CNC waits for the operation in progress to end.



Programming format.

The programming format is: Optional parameters are indicated between angle brackets.

#EXEC ["{prg}"<,{channel}>]

{prg}	Part-program location.
{channel}	Optional. Channel where the block is to be executed.

```
#EXEC ["PRG1.NC",2]
    (It executes in channel 2 the indicated program)
#EXEC ["MYPRG.NC"]
    (It executes the program as a subroutine).
#EXEC ["C:\CNC8070\USERS\PRG\EXAMPLE.NC",3]
    (It executes in channel 3 the indicated program)
```

Program location (path).

The program to be executed may be defined by either writing the full path or without it. When indicating the whole path, the CNC only looks for the program in the indicated folder. If the path has not been indicated, the CNC looks for the program in the following folders and in this order.

1. Directory selected with the #PATH instruction.
2. Directory of the program that executes the #EXEC instruction.
3. Directory defined by machine parameter SUBPATH.

Channel where the block is to be executed.

Programming the channel is optional. If the channel is not indicated or it coincides with the channel where the #EXEC instruction is executed, the second program will be executed as a subroutine. In this case, functions M02 and M30 will carry out all the associated actions (initialization, sending to the PLC, etc.) except the one for finishing the program. After executing function M02 or M30, it goes on executing the blocks programmed after the #EXEC instruction.

Considerations.

A program containing the #EXEC instruction may be executed, simulated, syntax checked or searched for a particular block. In all the cases, programs called upon using the #EXEC instruction are executed in the same conditions as the original program

6.2 Executing a block in the indicated channel.

With the #EXBLK instruction, it is possible, from a program in execution or via MDI, to execute a block in another channel.

If the channel where the block to be executed is busy, the CNC waits for the operation in progress to end. After executing the block, the channel goes back to the previous work mode.

Programming format.

The programming format is: Optional parameters are indicated between angle brackets.

```
#EXBLK [{block}<,{channel}>]
```

{block}	Block to be executed.
{channel}	Optional. Channel where the block is to be executed.

```
#EXBLK [G01 X100 F550, 2]
    (The block is executed in channel ·2·)
#EXBLK [T1 M6]
    (The block is executed in the current channel)
```

Channel where the block is to be executed.

Programming the channel is optional. If the channel is not indicated and the instruction is executed from the program, the block is executed in its own channel. If the instruction is executed in MDI and the channel is not indicated, the block is executed in the active channel.

6.3 Axis swapping.

Initially, each channel has some axes assigned to it as set by the machine parameters. While executing a program, the CNC can change channel axes or just change the configuration of a channel by modifying the position of its axes or eliminating some of them.

Not all the axes can change channels or modify their position in the channel. Parameter AXISEXCH sets whether the axis has permission to change channels and if it does, whether the change is temporary or permanent; in other words, whether the change is maintained after an M02, M30, a reset or after restarting the CNC. See *"2.1.3 Allowing to change channel axes and spindles."* on page 38.

The original configuration (the one defined in the machine parameters) of a channel where permanent changes have been made may be restored either by validating the machine parameters or by undoing the changes, for example using a part-program. It must be borne in mind that validating the machine parameters restores the configuration of all the channels.



The CNC will also restores the machine parameter settings if a checksum error occurs when powering up the CNC.

Commands for modifying the axis configuration via program.

The following instructions are used to modify the configuration of a channel.

Instruction.	Meaning.
#SET AX	Set a new axes configuration.
#CALL AX	Add a new axis to the configuration of the channel.
#FREE AX	Remove an axis from the configuration of the channel.
#RENAME AX	Rename the axes of a channel.

Changing the configuration of the axes cancels the active polar origin, the pattern rotation, the mirror image and the scaling factor.

6.3.1 Set a new axes configuration.

The instruction #SET AX may be used to define a new axis configuration in the channel or to change the order of the current axes in the channel. This instruction is the same as programming #FREE AX of all the axes and then #CALL AX of the new axes.

When defining a new configuration, the CNC behaves as follows.

- The axes existing in the channel and that were not programmed in the instruction are free. The CNC removes them from the configuration of the channel and does not assign them to another channel.
- The CNC adds the new axes to the configuration as if they are free. If any axis is in another channel, the channel requesting the axis waits until the axis is set free.
- If any of the axes already existed in the configuration, the CNC places it in its new position.

When defining a new configuration, the axes are placed in the channel in the same order they were programmed in the instruction. Optionally, one or several offsets may be applied to the defined axes.

Programming.

When programming this instruction, you must define the new axis configuration of the channel. The axes are placed in the channel in the same order they were programmed in the instruction. Optionally, one or several offsets may be applied to the defined axes.

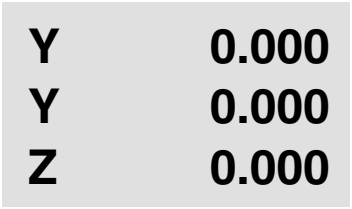
Programming format.

The programming format is the following; the parameters to be set appear between curly brackets and the optional parameters between angle brackets.

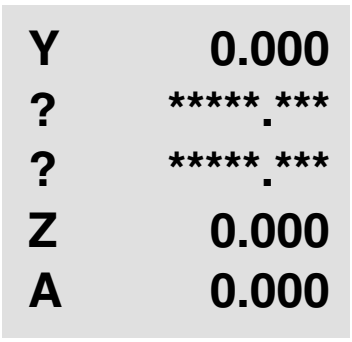
#SET AX [{xn|0}<,{xn|0}>] <{offset}>

{xn|0} Name of the axis that is part of the configuration; with ·0·, position with no axis.

{offset} Optional. One or several offsets to apply to the axes.



#SET AX [X,Y,Z]



#SET AX [Y,0,0,Z,A]

Offset setting.

One or several offsets may be applied when defining a new configuration. If when defining a new configuration only the order of the axes in the channel is swapped, the offsets are ignored.

The offsets that may be applied to the axes are identified with the following commands. To apply several offsets, program the relevant commands separated by a blank space.

Command.	Meaning.
ALL	Include all the offsets.
LOCOF	Include the offset of the reference search.
FIXOF	Include the fixture offset.

Command.	Meaning.
ORGOF	Include zero offset.
MEASOF	Include measurement offset.
MANOF	Include the offset of the manual operations.

```
#SET AX [X,Y,Z] ALL
#SET AX [X,Y,V1,0,A] ORGOF FIXOF
```

6.3.2 Add a new axis to the configuration of the channel.

The instruction #CALL AX may be used to add one or several axes to the channel being possible to define the position to be placed in the channel. When adding new axes to the channel, the CNC behaves as follows.

- The CNC adds the new axes to the configuration as if they are free. If any axis is in another channel, the channel requesting the axis waits until the axis is set free.
- If the axis already exists in the configuration, the CNC places it in the new position.

When adding an axis to the channel without defining its position, the CNC places it at the end of the channel. Optionally, one or several offsets may be applied to the defined axes.

Programming.

When programming this instruction, you must define the axes to be added to the channel and the position where they will be placed in the channel. Optionally, one or several offsets may be applied to the defined axes.

Programming format.

The programming format is the following; the parameters to be set appear between curly brackets and the optional parameters between angle brackets.

```
#CALL AX [{xn},<{pos}><,{xn},{pos}>] <{offset}>
```

{xn}	Axis name.
{pos}	Optional. Position of the axis in the channel.
{offset}	Optional. One or several offsets to apply to the axes.



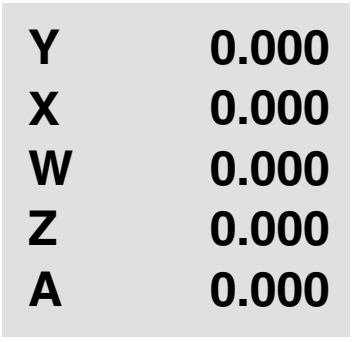
#SET AX [Y,0,0,Z]

Initial channel configuration.



#CALL AX [A]

Add the A axis to the channel.



#CALL AX [X,2,W,3]

Add the X axis in position ·2· and the W axis in position ·3·.

Offset setting.

When adding an axis to the channel configuration, one or several offsets may be applied to the axis. The offsets that may be applied to the axes are identified with the following commands. To apply several offsets, program the relevant commands separated by a blank space.

Command.	Meaning.
ALL	Include all the offsets.
LOCOF	Include the offset of the reference search.
FIXOF	Include the fixture offset.
ORGOF	Include zero offset.
MEASOF	Include measurement offset.
MANOF	Include the offset of the manual operations.

```
#CALL AX [X] ALL
#CALL AX [V1,4,Y] ORGOF FIXOF
```

6.3.3 Remove an axis from the configuration of the channel.

The instruction #FREE AX may be used to remove one or several axes from the channel. After removing an axis, the position is free, but the order of the axes that remain in the channel does not change.

Programming.

When programming this instruction, you must define the axes to be removed from the configuration. The ALL command may be used to remove all the axes of the channel.

Programming format.

The programming format is the following; the parameters to be set appear between curly brackets and the optional parameters between angle brackets.

```
#FREE AX ALL
#FREE AX [{xn}<,{xn}>]
```

{xn}	Axis name.
ALL	Command to remove all the axes from the channel

X	0.000
U	0.000
Z	0.000
A	0.000

#SET AX [X,U,Z,A]

Initial channel configuration.

X	0.000
?	*****.***
Z	0.000
A	0.000

#FREE AX [U]

Remove the U axis from the channel.

#FREE AX ALL

Remove all the axes from the channel.

6.3.4 Rename the axes of a channel.

The instruction #RENAME AX may be used to rename one or several axes of the channel. For each programmed axis pair, the first axis takes the name of the second one; if the second axis is present in the configuration, it takes the name of the first axis.

Programming.

When programming this instruction, you must define one or several pairs of axes.

Programming format.

The programming format is the following; the parameters to be set appear between curly brackets and the optional parameters between angle brackets.

```
#RENAME AX [{xn1},{xn2}] <[{xn1},{xn2}]>
```

{xn1} Axis whose name is to be changed.

{xn2} new axis name.

X	0.000
U	0.000
Z	0.000
A	0.000

#SET AX [X,U,Z,A]

Initial channel configuration.

X	0.000
W1	0.000
Z	0.000
A	0.000

#RENAME AX [U,W1]

Rename the U axis as W1.

X	0.000
W1	0.000
A	0.000
Z	0.000

#RENAME AX [Z,A]

Rename the Z axis as A. Since the A axis already exists in the configuration, it is renamed as Z.

Considerations.

Parameter RENAMECANCEL indicates whether the CNC keeps or cancel the names of the axes and spindles (#RENAME) after executing M02 or M30, after a reset or at the beginning of a new part-program in the same channel.

After turning the CNC off and on, the axes and spindles always keep their new name, except after a checksum error or after validating the machine parameters that involve restoring the original configuration of channels, axes or spindles. In either case, the axes and the spindles will recover their original names.

When a channel releases (frees) an axis (instruction #SET or #FREE), the axis always recovers its original name.

Even if the #RENAME is kept (parameter RENAMECANCEL), the CNC cancels it if the channel recovers an axis with the same name after a reset or after a new program starts. This occurs if the #RENAME uses the name of an axis whose channel change permission is temporary or no_exchange (parameter AXISEXCH), that is not in the channel at the time.

Accessing the variables of a renamed axis.

After changing the name of an axis, the new name of the axis must be used to access its variables from the part-program or MDI. The access to the variables from the PLCA or from an interface does not change; the original name of the axis remains unchanged.

6.3.5 Cancel the renaming of the axes.

The #RENAME AX OFF instruction may be used to change the names of the indicated axes regardless of what parameter RENAMECANCEL indicates.

Programming.

When programming this instruction, define the renamed axes to be canceled; if no axis is defined, it cancels the name change of all the axes of the channel.

Programming format.

The programming format is the following; the parameters to be set appear between curly brackets.

```
#RENAME AX OFF [{Xn}, {Xn}, ...]
```

{xn} Renamed axis.

```
#RENAME AX OFF [X]
    (Cancel the renaming of the X axis).
#RENAME AX OFF
    (Cancel the renaming of all the axes).
```

6.3.6 Variables related to the axis configuration of the channel.

(V.)[ch].G.AXIS

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.
Report variable (to be used from the scripts).

Number of axes of the channel.

Syntax.

·ch· Channel number.

V.[2].G.AXIS	Channel ·2·.

(V.)[ch].G.NAXIS

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.
Report variable (to be used from the scripts).

Number of axes of the channel including the empty positions of the yielded axes.

Syntax.

·ch· Channel number.

V.[2].G.NAXIS	Channel ·2·.

(V.)[ch].A.ACTCH.xn
(V.)[ch].A.ACTCH.sn
(V.)[ch].SP.ACTCH.sn

Current channel of the axis or of the spindle.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.A.ACTCH.Z	Z axis.
V.A.ACTCH.S	Spindle S.
V.SP.ACTCH.S	Spindle S.
V.SP.ACTCH	Master spindle.
V.A.ACTCH.4	Axis or spindle with logic number ·4·.
V.[2].A.ACTCH.1	Axis with index ·1· in the channel ·2·.
V.SP.ACTCH.2	Spindle with index ·2· in the system.
V.[2].SP.ACTCH.1	Spindle with index ·1· in the channel ·2·.

Remarks.

The variable returns the value of execution or preparation as follows. If the spindle belongs to the channel requesting the variable, it returns the preparation value; if the axis or spindle belongs to a different channel, the variable returns the execution value and interrupts block preparation.

Special returned values.

This variable returns one of the following values.

Value.	Meaning.
0	The axis or spindle is not in any channel.
1 - 4	Channel number.

6.4 Spindle swapping.

Initially, each channel has some spindles assigned to it as set by the machine parameters. While executing a program, the CNC can change channel spindles or just change the configuration of a channel by modifying the position of its spindles or eliminating some of them.

Not all the spindles can change channels or modify their position in the channel. Parameter AXISEXCH sets whether the spindle has permission to change channels and if it does, whether the change is temporary or permanent; in other words, whether the change is maintained after an M02, M30, a reset or after restarting the CNC. See *"2.1.3 Allowing to change channel axes and spindles."* on page 38.

The original configuration (the one defined in the machine parameters) of a channel where permanent changes have been made may be restored either by validating the machine parameters or by undoing the changes, for example using a part-program. It must be borne in mind that validating the machine parameters restores the configuration of all the channels.



The CNC will also restores the machine parameter settings if a checksum error occurs when powering up the CNC.

Commands for modifying the spindle configuration via program.

The following instructions are used to modify the configuration of a channel.

Instruction.	Meaning.
#SET SP	Set a new spindle configuration.
#CALL SP	Add a spindle to the configuration of the channel.
#FREE SP	Remove a spindle from the configuration of the channel.
#RENAME SP	Rename the spindles of a channel.

Changing the configuration of the spindles of a channel involves changing the master spindle of the channel. See *"7.1 The master spindle of the channel."* on page 112.

6.4.1 Set a new spindle configuration.

This instruction #SET SP may be used to define a new spindle configuration in the channel. This instruction is the same as programming #FREE SP of all the spindles and then #CALL SP of the new spindles.

When defining a new configuration, the CNC behaves as follows.

- The spindles existing in the channel and that were not programmed in the instruction are free. The CNC removes them from the configuration of the channel and does not assign them to another channel.
- The CNC adds the new spindles to the configuration as if they are free. If any spindle is in another channel, the channel requesting the spindle waits until the axis is set free.

When defining a new configuration, the order used to define the spindles in the instruction does not matter. The position of the spindles in the channel does not matter, either.

Programming.

When programming this instruction, you must define the new spindle configuration of the channel.

Programming format.

The programming format is the following; the parameters to be set appear between curly brackets and the optional parameters between angle brackets.

```
#SET SP [{sn}<,{sn}>]
```

{sn} Name of the spindle that is part of the new configuration.

```
#SET SP [S]
    (Configuring one spindle)
#SET SP [S1,S2]
    (Configuring two spindles)
```

6.4.2 Add a spindle to the configuration.

The instruction #CALL SP may be used to add one or several spindles to the channel. The CNC adds the new spindles to the configuration as if they are free. If any spindle is in another channel, the channel requesting the spindle waits until the axis is set free.

The position of the spindles in the channel is not relevant.

Programming.

When programming this instruction, you must define the axes to be added to the channel and the position where they will be placed in the channel. Optionally, one or several offsets may be applied to the defined axes.

Programming format.

The programming format is the following; the parameters to be set appear between curly brackets and the optional parameters between angle brackets.

```
#CALL SP [{sn}<,{sn}>]
```

{sn} Spindle name.

```
#CALL SP [S]
    (It adds spindle S to the configuration.)
#CALL SP [S1,S2]
    (It adds spindles S1 and S2 to the configuration.)
```

6.4.3 Remove a spindle from the configuration.

The instruction #FREE SP may be used to remove one or several spindles from the channel. The CNC removes them from the configuration of the channel and does not assign them to another channel.

Programming.

When programming this instruction, you must define the spindles to be removed from the configuration. The ALL command may be used to remove all the spindles of the channel.

Programming format.

The programming format is the following; the parameters to be set appear between curly brackets and the optional parameters between angle brackets.

```
#FREE AX ALL
#FREE AX [{sn}<,{sn}>]
```

{sn} Spindle name.

ALL Command to remove all the spindles from the channel

```
#FREE SP [S]
    (It removes the spindle S from the configuration)
#FREE SP [S1,S4]
    (It removes spindles S1 and S4 from the configuration)
#FREE SP ALL
    (It removes all the spindles from the configuration)
```

6.4.4 Rename the spindles of a channel.

The instruction #RENAME SP may be used to rename one or several spindles of the channel. For each programmed channel pair, the first spindle takes the name of the second one; if the second spindle is present in the configuration, it takes the name of the first spindle.

The change of the name of the spindles only remains during the execution of the program; the original names are restored at the beginning of the next program.

Programming.

When programming this instruction, you must define one or several pairs of spindles.

Programming format.

The programming format is the following; the parameters to be set appear between curly brackets and the optional parameters between angle brackets.

```
#RENAME SP [{sn1},{sn2}] <[{sn1},{sn2}]>
```

{sn1} Spindle whose name is to be changed.

{sn2} New spindle name.

```
#RENAME SP [S,S1]
    (Rename spindle S as S1.)
#RENAME SP [S1,S2][S3,S]
    (Rename spindle S1 as S2 and spindle S3 as S.)
```

Considerations.

Parameter RENAMECANCEL indicates whether the CNC keeps or cancel the names of the axes and spindles (#RENAME) after executing M02 or M30, after a reset or at the beginning of a new part-program in the same channel.

After turning the CNC off and on, the axes and spindles always keep their new name, except after a checksum error or after validating the machine parameters that involve restoring the original configuration of channels, axes or spindles. In either case, the axes and the spindles will recover their original names.

When a channel releases (frees) a spindle (instruction #SET or #FREE), the axis always recovers its original name.

Even if the #RENAME is kept (parameter RENAMECANCEL), the CNC cancels it if the channel recovers a spindle with the same name after a reset or after a new program starts. This occurs if the #RENAME uses the name of a spindle whose channel change permission is temporary or no_exchange (parameter AXISEXCH), that is not in the channel at the time.

6.4.5 Cancel the renaming of the spindles.

The #RENAME SP OFF instruction may be used to change the names of the indicated spindles regardless of what parameter RENAMECANCEL indicates.

Programming.

When programming this instruction, define the renamed spindles to be canceled; if no spindle is defined, it cancels the name change of all the spindles of the channel.

Programming format.

The programming format is the following; the parameters to be set appear between curly brackets.

#RENAME SP OFF [{Sn}, {Sn}, ...]

{Sn} Renamed spindle.

```
#RENAME SP OFF [S3]
    (Cancel the renaming of spindle S3).
#RENAME SP OFF
    (Cancel the renaming of all the spindles).
```

6.4.6 Variables related to the spindle configuration of the channel.

(V.)[ch].G.NSPDL

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.
Report variable (to be used from the scripts).

Number of spindles of the channel.

Syntax.

·ch· Channel number.

V.[2].G.NSPDL	Channel ·2·.

(V.)[ch].G.MASTERSP

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Logic number of the master spindle of the channel.

Syntax.

·ch· Channel number.

V.[2].G.MASTERSP	Channel ·2·.

(V.)[ch].A.ACTCH.xn
(V.)[ch].A.ACTCH.sn
(V.)[ch].SP.ACTCH.sn

Variable that can only be read from the program, PLC and interface.
Variable valid for rotary and linear axes as well as for spindles.
The variable returns the value of execution or preparation depending on the axis or spindle.

Current channel of the axis or of the spindle.

Syntax.

·ch· Channel number.

·xn· Name, logic number or index of the axis.

·sn· Name, logic number or index of the spindle.

V.A.ACTCH.Z	Z axis.
V.A.ACTCH.S	Spindle S.
V.SP.ACTCH.S	Spindle S.
V.SP.ACTCH	Master spindle.
V.A.ACTCH.4	Axis or spindle with logic number ·4·.
V.[2].A.ACTCH.1	Axis with index ·1· in the channel ·2·.
V.SP.ACTCH.2	Spindle with index ·2· in the system.
V.[2].SP.ACTCH.1	Spindle with index ·1· in the channel ·2·.

Remarks.

The variable returns the value of execution or preparation as follows. If the spindle belongs to the channel requesting the variable, it returns the preparation value; if the axis or spindle belongs to a different channel, the variable returns the execution value and interrupts block preparation.

Special returned values.

This variable returns one of the following values.

Value.	Meaning.
0	The axis or spindle is not in any channel.
1 - 4	Channel number.

6.5 Communication and synchronization.

Each channel may execute its own part-program independently and in parallel with the rest of the channels, but it can also communicate with them to share information or synchronize at particular points. The communication between channels takes place on the basis of a number of marks managed by the part-programs of each channel. These marks establish whether the channel is waiting to be synchronized or it may be synchronized, etc.

There are two different ways to synchronize, each offers a different solution. The synchronization marks of both methods are independent from each other; the marks of each method neither affect nor are affected by the marks of the other one.

Synchronization of channels interrupting the execution of all the channels involved.	**Synchronization of channels without interrupting the execution of all the channels involved.**
#MEET	#SIGNAL - #WAIT - #CLEAR
The CNC interrupts the execution in all the channels in order to synchronize.	The CNC does not interrupt the execution in all the channels in order to synchronize.
The marks are initialized after executing an M02 or an M30, after a reset or on power-up.	The marks are not initialized after executing an M02 or an M30, after a reset or on power-up.

Synchronization of channels interrupting the execution of all the channels involved. #MEET instruction.

The #MEET instruction provides the easiest way to synchronize. This instruction activates a mark in its own channel and sets the channel waiting until the same mark is active in the indicated channels. This method interrupts the execution in all the channels involved in the synchronization.

The set of marks being used are initialized after executing an M02 or an M30, after a reset or on power-up.

Synchronization of channels without interrupting the execution of all the channels involved. Instructions #SIGNAL - #WAIT - #CLEAR.

The instructions #SIGNAL, #WAIT and #CLEAR are a method somewhat more complicated than the previous one, but more versatile. This instruction #SIGNAL activates the marks in its own channel and #WAIT sets the channel waiting until the indicated mark is active in the indicated channel. This method does not interrupt the execution in all the channels in order to synchronize.

The set of marks being used are maintained after executing an M02 or an M30, after a reset or on power-up.

Other ways to synchronize channels.

The common arithmetic parameters can also be used to communicate and synchronize channels. By writing a certain value from a channel and later reading it from another channel, it is possible to set the condition to follow up on the execution of a program.

Accessing the variables of a channel from another channel can also be used as a way to communicate.

Swapping axes between channels also makes it possible to synchronize processes, because a channel cannot grab an axis until it has been released by another one.

6.5.1 Synchronization of channels interrupting the execution of all the channels involved.

The synchronization is programmed with the #MEET instruction. This instruction activates a mark in its own channel, interrupts the execution of the program and sets the channel waiting until the same mark is active in the indicated channels. All the channels interrupt the execution of their programs in order to synchronize.

Each channel has 100 marks numbered from 1 to 100; no order needs to be followed when using these marks. This instruction may be used to synchronize all the channels at the same time.

Programming.

When programming this instruction, you must define the number of the mark and the channels used for synchronization. There is no need to include the number of its own channel in each instruction because the mark is activated when executing the #MEET instruction; however, we recommend to program it to make the program easier to understand.

Programming format.

The programming format is the following; the parameters to be set appear between curly brackets and the optional parameters between angle brackets.

```
#MEET [{mark},{ch}<,{ch}>]
```

{mark} Synchronism mark.

{ch} Channel where the mark must be activated.

```
#MEET [1,3]
    (The CNC synchronizes the channel where the instruction is executed with channel ·3· using mark ·1·.)
#MEET [8,2,3]
    (The CNC synchronizes the channel where the instruction is executed with channels ·2· and ·3· using mark ·8·)
```

How is the synchronization carried out.

This instruction activates a mark in its own channel, interrupts the execution of the program and sets the channel waiting until the same mark is active in the indicated channels. When the last channel involved activates its mark, all the channels delete the mark used and resume the execution of their programs. All the channels interrupt the execution of their programs in order to synchronize.

In the following example, channels ·1·, ·2· and 3 wait for mark ·5· to be active in order to synchronize the execution.

Channel ·1·.	Channel ·2·.	Channel ·3·.
%PRG_1 … … #MEET [5,1,2,3] (Execution interrupted.)	%PRG_2 … #MEET [5,1,2,3] (Execution interrupted.)	%PRG_3 … … … … #MEET [5,1,2,3] (Execution interrupted.) (Synchronize the execution.)

6.5.2 Synchronization of channels without interrupting the execution of all the channels involved.

The instruction #WAIT interrupts the execution of the program and sets the channel waiting until the programmed mark is active in the indicated channels. The #SIGNAL instruction activates the indicated mark in its own channel without interrupting the execution of the program. The channels only interrupt the execution of their programs in order to synchronize if necessary.

Each channel has 100 marks numbered from 1 to 100; no order needs to be followed when using these marks. This instruction may be used to synchronize all the channels at the same time.

Programming.

When programming the instruction #WAIT, you must define the number of the mark and the channels used for synchronization. For the instruction #SIGNAL, only the number of the mark to be activated must be defined. For the #CLEAR, only the number of the mark to be deleted must be defined; if no mark is defined, the channel will delete all of them.

Programming format. #WAIT instruction.

The programming format is the following; the parameters to be set appear between curly brackets and the optional parameters between angle brackets.

```
#WAIT [{mark},{ch}<,{ch}>]
```

{mark} Synchronism mark.

{ch} Channel where the mark must be activated.

```
#WAIT [1,3]
    (The CNC synchronizes the channel where the instruction is executed with channel ·3· using mark ·1·.)
#WAIT [8,2,3]
    (The CNC synchronizes the channel where the instruction is executed with channels ·2· and ·3· using mark ·8·)
```

Programming format. #SIGNAL instruction.

The programming format is the following; the parameters to be set appear between curly brackets and the optional parameters between angle brackets.

```
#SIGNAL [{mark}<,{mark}>]
```

{mark} Synchronism mark.

```
#SIGNAL [1]
    (The CNC activates mark ·1· in the channel where the instruction is executed.)
#SIGNAL [3,6]
    (The CNC activates marks ·3· and ·6· in the channel where the instruction is executed.)
```

Programming format. #CLEAR instruction.

The programming format is the following; the parameters to be set appear between curly brackets and the optional parameters between angle brackets.

```
#CLEAR [{mark}<,{mark}>]
```

{mark} Synchronism mark.

```
#CLEAR [1]
    (The CNC deletes all the marks in the channel where the instruction is executed.)
#CLEAR [3,6]
    (The CNC deletes marks ·3· and ·6· in the channel where the instruction is executed.)
```

How is the synchronization carried out.

The #SIGNAL instruction activates the indicated mark in its own channel without interrupting the execution of the program. Depending on the status of the marks when executing the #WAIT instruction, the CNC behaves as follows.

- If when executing the #WAIT instruction, the programmed mark is not active, the channel interrupts the execution of the part-program and sets the channel waiting until the mark is active in the indicated channels. When the #SIGNAL instruction activates the mark , the channel will resume the execution of the program.
- If when executing the #WAIT instruction, the programmed mark is already active, the channel does not interrupt the execution of the part-program. In this case, the channels will not get synchronized.

The channel does not delete the marks used. Use the #CLEAR instruction to delete the marks.

6.5.3 Variables related to channel synchronization.

(V.)[ch].G.MEETST[mk]

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Status of the MEET type mark [mk] in the [ch] channel.

Syntax.

·ch· Channel number.

·mk· Number of the synchronization mark.

V.[2].G.MEETST[4]	Channel ·2·. Mark ·4·.

(V.)[ch].G.WAITST[mk]

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Status of the WAIT type mark [mk] in the [ch] channel.

Syntax.

·ch· Channel number.

·mk· Number of the synchronization mark.

V.[2].G.WAITST[4]	Channel ·2·. Mark ·4·.

(V.)[ch].G.MEETCH[nch]

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

MEET type mark originated in channel [nch] and expected by channel [ch].

Syntax.

·ch· Channel number.

·nch· Channel that originates the synchronization mark.

V.[2].G.MEETCH[4]	Channel ·2·. Channel ·4·.

(V.)[ch].G.WAITCH[nch]

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

WAIT type mark originated in channel [nch] and expected by channel [ch].

Syntax.

·ch· Channel number.

·nch· Channel that originates the synchronization mark.

V.[2].G.WAITCH[4]	Channel ·2·. Channel ·4·.

6.

PROGRAMMING IN A SYSTEM WITH CHANNELS.
Communication and synchronization.

FAGOR

CNC 8060
CNC 8065

(REF: 1402)

MULTI-SPINDLE SYSTEM. SPINDLE CONTROL.

7

The CNC can have up to four spindles distributed between the various channels of the system. A channel may have one, several or no spindles associated with it. Via part-program or in MDI/MDA mode, it is possible to indicate which spindle the commands are directed to; when not indicated, the commands are directed to the master spindle of the channel.

Each channel can only control its spindles; it is not possible to start up or stop the spindles of another channel directly. Indirectly, the CNC can control the spindles of another channel using the instruction #EXBLK.

Multi-spindle system.

A multi-spindle channel is one that has two or more spindles. Via part-program or MDI, it is possible to indicate which spindle the commands are directed to; when not indicated, the commands are directed to the master spindle of the channel.

All the spindles of the channel may be running at the same time. Also, each of them may be in a different mode; turn in different directions, be in positioning mode, etc.

Master spindle of the channel.

The master spindle is the main spindle of the channel. In general, whenever a channel has a single spindle, it will be its master spindle. When a channel has several spindles, the CNC will choose the master spindle according to the established criteria.

7.1 The master spindle of the channel.

The master spindle is the main spindle of the channel. It is the spindle that receives the commands when no specific spindle is mentioned. In general, whenever a channel has a single spindle, it will be its master spindle.

CNC criterion for selecting the master spindle after executing M02, M30 or after an emergency or a reset or restarting the CNC.

The selection of the master spindle in the channel depends on machine parameter MASTERSPDL. This parameter indicates whether the channel keeps the current master spindle or recovers its original master spindle after executing M02, M30 or after an emergency or reset or restarting the CNC.

MASTERSPDL	Meaning.
Temporary.	The channel recovers its original master spindle if it is free; otherwise, it selects as master the firs spindle available in the original configuration.
Maintained.	The channel keeps the active master spindle.

When a channel does not keep its master spindle, on CNC power-up and after a reset, the channel assumes as master spindle the first spindle defined by the machine parameters of the channel (original master). If this spindle is parked or "handed out" to another channel, the channel assumes as master spindle the next spindle defined by the machine parameters and so on. If the channel does not have spindles of the original configuration (the one defined by the machine parameters) because they are parked or "handed out", it assumes as master spindle the first one of the current configuration that is not parked.

Exchanging spindles between channels.

When spindles are being exchanged between channels, the behavior of this parameter also depends on parameter AXISEXCH that sets whether the channel change of a spindle is temporary or permanent. If the current master spindle of the channel is a spindle "loaned" by another channel and its permission to change channels is temporary (AXISEXCH = Temporary), the spindle returns to its original channel.

Which one is the master spindle after executing M30?

When executing an M30, it follows the same criteria, but considering that after executing this function, the temporary spindle swaps are not undone; they are undone at the beginning of the next program. This means that the original master might not be available after executing an M30, but it will be at the beginning of the next program. In this situation, after an M30, the channel will temporarily assume a master spindle that will change at the beginning of the next program.

Which one is the master spindle after modifying the configuration of the channel?.

If no master spindle is indicated, after parking or swapping spindles, it assumes one according to the following criteria. In general, whenever a channel has a single spindle, it will be its master spindle.

- If the whole system only has one spindle, it will be the master spindle of the current channel.
- If a spindle is added to a channel that does not have one, it will be the master spindle.
- If a channel releases its master spindle and it has only one spindle left, this one will be its new master spindle.
- If a channel having two spindles but no master spindle releases one of them, the remaining one will be its master spindle.
- At first, in a channel with several spindles, the master spindle will be the one configured by machine parameters.

- If two or more spindles remain in a channel and none of the previous rules can be applied, it applies the following criteria.

  If any of the spindles is the original master, it is assumed as master spindle. If it is parked, it selects the next spindle from the original configuration (those defined by the machine parameters) and so on.

  If no spindles from the original configuration are available in the channel, it assumes as master spindle the first one of its current configuration. If it is parked, it selects the next spindle and so on.

Which one is the master spindle after parking or unparking spindles?

The same treatment is applied as when modifying the configuration of the channel.

7.1.1 Manual selection of a master spindle

Selecting a new master spindle

Whenever a channel has a single spindle, it will be its master spindle. When a channel has several spindles and after swapping or parking spindles, the CNC will choose the master spindle according to the criteria described earlier. However, the user may select a different master spindle in MDI/MDA mode or via part-program using the #MASTER instruction.

Programming format.

The programming format is the following; the parameters to be set appear between curly brackets.

#MASTER {sn}

{sn} Spindle name.

```
#MASTER S
#MASTER S2
```

Canceling the master spindle

The master spindle may be selected at any time. If the master spindle changes channels, the channel will select a new master spindle according to the criterion described earlier.

On power-up, after executing an M02 or M30, and after an emergency or reset, the CNC acts as defined by the OEM (parameter MASTERSPDL).

7.2 Spindle speed.

The speeds of all the spindles of the channel may be programmed in the same block. It is not possible to program the speed of a spindle that is not in the channel.

The programmed speed stays active until another value is programmed. On power-up, after executing an M02 or M30, and after an emergency or reset, the spindles assume a ·0· speed.

Programming format.

The spindle speed is represented by the spindle name followed by the desired speed. For spindles like S1, S2, etc. the "=" sign must be included between the name and the speed; the "=" is not necessary for the spindle S.

The programming format is the following; the parameters to be set appear between curly brackets.

```
S..S9={vel}
```

S..S9 Spindle name.

{vel} Spindle speed.

```
S1000
S1=334
S150 S2=350
```

The speed may be programmed in rpm or in m/min (feet/min), depending on whether G97 or G96 is active. The default units are rpm.

Spindle start and stop.

Defining a speed does not imply starting the spindle. The startup is defined using the following auxiliary functions. See *"M03/M04/M05. Spindle start and stop."* on page 123.

Function.	Meaning.
M03	Starts the spindle clockwise.
M04	Starts the spindle counteclockwise.
M05	Stops the spindle.

Speed ranges (gears).

Each spindle may have up to 4 different ranges (gears). Each gear means a speed range for the CNC work in. The programmed speed must be within the active gear; otherwise, a gear change will be required. The CNC does not admit speeds higher than the one defined for the last gear.

The gear change may be automatic or manual. When the change is manual, the gear is selected with auxiliary functions M41 through M44. When the change is automatic, the CNC itself generates these functions according to the programmed speed. See *"M41-M44. Gear change."* on page 125.

7.3 G96/G97. Turning speed or constant surface speed.

The following functions are oriented to lathe type machines. In order for Constant Surface Speed mode to be available, the machine manufacturer must have set one of the axis -face axis- (parameter FACEAXIS) usually axis perpendicular to the shaft of the part. Milling type machines usually work at constant turning speed.

The functions related to spindle speed programming may be used to select either Constant Surface Speed mode or Constant turning speed mode. Constant Surface Speed is only available at the master spindle of the channel.

Function.	Meaning.
G96	Constant surface speed.
G97	Constant turning speed.

At constant surface speed, the CNC changes the spindle speed as the perpendicular axis moves in order to maintain the cutting speed constant between the tool and the part, thus optimizing the machining conditions. With constant turning speed, the speed stays constant regardless of the movement of the axes.

G96. Constant surface speed.

The G96 function only affects the master spindle of the channel. After executing G96, the CNC interprets that the spindle speeds programmed for the master spindle of the channel are in meters/minute (feet/minute). This work mode is activated when programming a new speed while G96 is active.

This function may be programmed anywhere in the program and it doesn't have to go alone in the block. It is recommended to program the speed in the same block as the G96 function. The spindle gear (range) must be selected in the same block or in a previous one.

Option ·1·.	Option ·2·.
G96 S110 M41	M41 G96 S110

When working at constant surface speed, it is recommended to limit by program the maximum turning speed that the spindle can reach. See *"G192. Turning speed limit."* on page 121.

G97. Constant turning speed.

The G97 function affects all the spindles of the channel. After executing G97, the CNC interprets that the programmed spindle speeds are in rpm.

This function may be programmed anywhere in the program and it doesn't have to go alone in the block. It is recommended to program the speed in the same block as the G97 function; if not programmed, the CNC assumes as programmed speed the one the spindle is currently turning at. The gear may be selected at any time.

Option ·1·.	Option ·2·.	Option ·3·.
G97 S1540 M43	M43 G97 S1540	G97 S1540 M43

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

Functions G96 and G97 are modal and incompatible with each other. On power-up, after executing an M02 or M30, and after an emergency or reset, the CNC assumes function G97.

7.3.1 Variables related to spindle speed.

PROGRAMMING THE SPEED.

(V.)[ch].A.SREAL.sn
(V.)[ch].SP.SREAL.sn

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Real spindle speed.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.SREAL.S	Spindle S.
V.SP.SREAL.S	Spindle S.
V.SP.SREAL	Master spindle.
V.A.SREAL.5	Spindle with logic number ·5·.
V.SP.SREAL.2	Spindle with index ·2· in the system.
V.[2].SP.SREAL.1	Spindle with index ·1· in the channel ·2·.

Remarks.

This variable takes into account the accelerations and decelerations of the machine. It returns a value of ·0· when the spindle is stopped, it returns the value in rpm when the spindle is in G96/G97 and in degrees/minute when the spindle is in M19.

SPINDLE SPEED IN G97.

(V.)[ch].A.SPEED.sn
(V.)[ch].SP.SPEED.sn

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Active spindle speed in G97.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.SPEED.S	Spindle S.
V.SP.SPEED.S	Spindle S.
V.SP.SPEED	Master spindle.
V.A.SPEED.5	Spindle with logic number ·5·.
V.SP.SPEED.2	Spindle with index ·2· in the system.
V.[2].SP.SPEED.1	Spindle with index ·1· in the channel ·2·.

Remarks.

The speed may be set by program or by PLC; the one set by PLC has the highest priority.

Speed set.	V.PLC.S.sn	V.A.PRGS.sn	V.A.SPEED.sn
By program; S5000. By PLC; none.	0	5000	5000
By program; S5000. By PLC; S9000.	9000	5000	9000
By program; S5000. By PLC; S3000.	3000	5000	3000
By program; S5000. By PLC; none. By MDI; S8000.	0	8000	8000
By program; S5000. By PLC; S9000. By MDI; S8000.	9000	8000	9000
By program; S5000. By PLC; S3000. By MDI; S8000.	3000	8000	3000

(V.)[ch].PLC.S.sn

Variable that can be read and written from the PLC and read from the program and from the interface. The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Speed active by PLC for G97.

The speed programmed by PLC prevails over the one programmed by program or MDI. To cancel the speed by PLC, set the variable to ·0·; the CNC applies the speed active by program.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.PLC.S.S2	Spindle S2.
V.PLC.S.5	Spindle with logic number ·5·.
V.[2].PLC.S.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.PRGS.sn
(V.)[ch].SP.PRGS.sn

Variable that can only be read from the program, PLC and interface. The variable returns the value of block preparation.

Channel [ch]. Speed active by program for G97.

Being G97 active, programming a new speed in MDI mode updates the value of this variable.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.PRGS.S	Spindle S.
V.SP.PRGS.S	Spindle S.
V.SP.PRGS	Master spindle.
V.A.PRGS.5	Spindle with logic number ·5·.
V.SP.PRGS.2	Spindle with index ·2· in the system.
V.[2].SP.PRGS.1	Spindle with index ·1· in the channel ·2·.

SPINDLE SPEED IN G96 (CSS).

(V.)[ch].A.CSS.sn
(V.)[ch].SP.CSS.sn

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Active spindle speed in G96.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.CSS.S	Spindle S.
V.SP.CSS.S	Spindle S.
V.SP.CSS	Master spindle.
V.A.CSS.5	Spindle with logic number ·5·.
V.SP.CSS.2	Spindle with index ·2· in the system.
V.[2].SP.CSS.1	Spindle with index ·1· in the channel ·2·.

Remarks.

The speed may be set by program or by PLC; the one set by PLC has the highest priority.

Speed set.	V.PLC.CSS.sn	V.A.PRGCSS.sn	V.A.CSS.sn
By program; S150. By PLC; none.	0	150	150
By program; S150. By PLC; S250.	250	150	250
By program; S150. By PLC; S100.	100	150	100
By program; S150. By PLC; none. By MDI; S300.	0	300	300
By program; S150. By PLC; S250. By MDI; S200.	250	200	250
By program; S150. By PLC; S100. By MDI; S200.	100	200	100

(V.)[ch].PLC.CSS.sn

Variable that can be read and written from the PLC and read from the program and from the interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Speed active by PLC for G96.

The speed programmed by PLC prevails over the one programmed by program or MDI. To cancel the speed by PLC, set the variable to ·0·; the CNC applies the speed active by program.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.PLC.CSS.S2	Spindle S2.
V.PLC.CSS.5	Spindle with logic number ·5·.
V.[2].PLC.CSS.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.PRGCSS.sn
(V.)[ch].SP.PRGCSS.sn

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Speed active by program for G96.

Being G96 active, programming a new speed in MDI mode updates the value of this variable.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.PRGCSS.S	Spindle S.
V.SP.PRGCSS.S	Spindle S.
V.SP.PRGCSS	Master spindle.
V.A.PRGCSS.5	Spindle with logic number ·5·.
V.SP.PRGCSS.2	Spindle with index ·2· in the system.
V.[2].SP.PRGCSS.1	Spindle with index ·1· in the channel ·2·.

7.4 G192. Turning speed limit.

Function G192 limits the spindle turning speed in both work modes; G96 and G97. This function is especially useful when working at constant cutting speed while machining large parts or when doing spindle maintenance work.

If function G192 is not programmed, the turning speed in each gear is limited by its machine parameter G00FEED.

Programming the turning speed limit.

The speed limit for all the spindles of the channel may be programmed in the same block. This function may be programmed while the spindle is running; in this case, the CNC will limit the speed to the new programmed value.

The turning speed limit is set by programming function G192 and then the maximum speed for each spindle. The spindle speed is represented by the spindle name followed by the desired speed. For spindles like S1, S2, etc. the "=" sign must be included between the name and the speed; the "=" is not necessary for the spindle S.

Programming format.

The programming format is the following; the parameters to be set appear between curly brackets.

```
G192 S..S9={vel}
```

{vel} Maximum spindle speed. The speed is given in rpm.

```
G192 S1000
G192 S1=334
G192 S150 S2=350
```

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

Function G192 is modal. Function G192 is canceled on power-up or after an emergency. The behavior of function G192 after executing an M02 or M30 and after a reset depends on the setting of machine parameter SPDLSTOP.

SPDLSTOP	Behavior of function G192
Yes	Functions M02, M30 and reset cancel function G192.
No	Functions M02, M30 and reset do not affect the spindle. The CNC maintains function G192.

7.4.1 Variables associated with the turning speed limit.

(V.)[ch].A.SLIMIT.sn
(V.)[ch].SP.SLIMIT.sn

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Active spindle speed limit.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.SLIMIT.S	Spindle S.
V.SP.SLIMIT.S	Spindle S.
V.SP.SLIMIT	Master spindle.
V.A.SLIMIT.5	Spindle with logic number ·5·.
V.SP.SLIMIT.2	Spindle with index ·2· in the system.
V.[2].SP.SLIMIT.1	Spindle with index ·1· in the channel ·2·.

Remarks.

The maximum speed may be set by program or by PLC; the one set by PLC has the highest priority.

(V.)[ch].PLC.SL.sn

Variable that can be read and written from the PLC and read from the program and from the interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Spindle speed limit active by PLC.

The speed programmed by PLC prevails over the one programmed by program or MDI. To cancel the speed by PLC, set the variable to ·0·; the CNC applies the speed active by program.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.PLC.SL.S2	Spindle S2.
V.PLC.SL.5	Spindle with logic number ·5·.
V.[2].PLC.SL.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.PRGSL.sn
(V.)[ch].SP.PRGSL.sn

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Spindle speed limit active by program.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.PRGSL.S	Spindle S.
V.SP.PRGSL.S	Spindle S.
V.SP.PRGSL	Master spindle.
V.A.PRGSL.5	Spindle with logic number ·5·.
V.SP.PRGSL.2	Spindle with index ·2· in the system.
V.[2].SP.PRGSL.1	Spindle with index ·1· in the channel ·2·.

7.5 M03/M04/M05. Spindle start and stop.

A speed must be set in order to start up a spindle. The following auxiliary functions must be used to start and stop the spindle.

Function.	Meaning.
M03	Start the spindle clockwise.
M04	Start the spindle counterclockwise.
M05	Stop the spindle.

These functions are modal and incompatible with each other and with function M19.

M03/M04. Start the spindle clockwise/counterclockwise.

Function M03 starts the spindle clockwise and function M04 counterclockwise. These functions should be set in the "M functions" table so they are executed at the end of the block where it is programmed.

These functions may be defined together with the programmed speed or in a separate block. If several spindles are programmed in a single block, functions M3 and M4 apply to all of them. If the function is programmed in a block that does not mention any spindle, the function will only be applied to the master spindle of the channel.

```
S1000 M3
    (The spindle "S" starts clockwise at 1000 rpm)
S1=500 M4
    (The spindle "S1" starts counterclockwise at 500 rpm)
S1000 S2=456 M3
    (Spindle S turning at 1000 rpm and S2 at 456 rpm, both clockwise)
M4
    (The master spindle starts counterclockwise)
```

Starting several spindles in different directions.

To associate these M functions to a particular spindle, define the name of the spindle next the each M function and separated by a period. The programming format is:

M3.Sn
M4.Sn

·sn· Name of the spindle (S, S1··S9).

```
M3.S S1000 S2=456 M4.S2
    (The spindle S turns clockwise at 1000 rpm)
    (The spindle S2 turns counterclockwise at 456 rpm)
```

M05. Stop the spindle.

Function M05 stops the spindle. This function should be set in the "M" function table so it is executed at the end of the block where it is programmed.

These functions may be defined together with the programmed speed or in a separate block. If the function is programmed in a block that does not mention any spindle, the function will only be applied to the master spindle of the channel.

To associate this M function to a particular spindle, define the name of the spindle next the M function and separated by a period. The programming format is:

M5.Sn

·sn· Name of the spindle (S, S1··S9).

```
S1000 S2=456 M5
    (Stops the master spindle)
M5.S M5.S2 S1=1000 M3.S1
    (Stops the spindles S and S2)
    (Spindle S1 turns clockwise)
```

Pre-set turning direction for a tool.

It is possible to set a preset turning direction for each tool. The value is defined in tool table.

When assigning a turning direction in the table, the CNC will verify, during execution, that the turning direction in the table is the same as the one programmed (M03/M04). If the two directions are not the same, the CNC will display the corresponding error message. The CNC verifies this every time an M03, M04 or M06.is programmed.

Knowing which is the preset turning direction.

The turning direction preset for each tool may be consulted in the tool table; the one for the active tool can also be consulted with the following variable.

(V.)G.SPDLTURDIR
This variable returns the preset turning speed of the active tool. ·0· value if there is no preset turning direction, ·1· if it is M03, and ·2· if it is M04.

Canceling the preset turning direction temporarily.

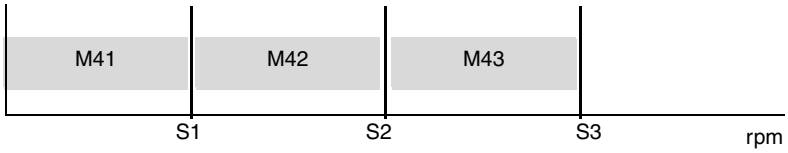
The preset turning speed of the active tool may be temporarily canceled from the part-program. This is done by setting the variable V.G.SPDLTURDIR = 0.

When changing a tool, this variable will take the corresponding value according to what has been set in the tool table.

7.6 M41-M44. Gear change.

Each spindle may have up to 4 different ranges (gears). Each gear means a speed range for the CNC work in. The programmed speed must be within the active gear; otherwise, a gear change will be required.

The gear change may be automatic or manual. When the change is manual, the gear is selected with auxiliary functions M41 (gear 1) through M44 (gear 4). When the change is automatic, the CNC itself generates these functions according to the programmed speed.



The graphic shows a spindle with three speed ranges (gears). The first gear goes from 0 to S1 rpm; the second one from S1 to S2; the third one from S2 to S3.

The configuration of the speed ranges or gears (automatic or manual change, maximum speed for each gear, etc.) is defined by the machine manufacturer. See *"How to know the gear configuration of a spindle."* on page 126.

Manual gear change.

When the change is manual, the gear is selected with auxiliary functions M41 through M44.

Function.	Meaning.
M41	Selects gear ·1·.
M42	Selects gear ·2·.
M43	Selects gear ·3·.
M44	Selects gear ·4·.

These functions may be defined together with the programmed spindles or in a separate block. If several spindles are programmed in a single block, functions M41 to M44 apply to all of them. If the functions are programmed in a block that does not mention any spindle, the functions will only be applied to the master spindle of the channel.

```
S1000 M41
    (Gear ·1· for spindle S)
S1=500 M42
    (Gear ·2· for spindle S1)
S1000 S2=456 M41
    (Gear ·1· for spindle S and S2)
M44
    (Gear ·4· for master spindle)
```

Starting several spindles in different directions.

To associate these M functions to a particular spindle, define the name of the spindle next the each M function and separated by a period. The programming format is:

M41.Sn
M42.Sn
M43.Sn
M44.Sn

·sn· Name of the spindle (S, S1··S9).

```
M41.S M42.S3
    (Gear ·1· for spindle S)
    (Gear ·2· for spindle S3)
```

Influence of the reset, turning the CNC off and of the M30.

Gears (speed ranges) are modal. On power-up, the CNC assumes the gear defined by the machine manufacturer. After executing an M02 or M30 and after an emergency or a reset, the CNC maintains the active gear.

Knowing which is the active gear.

The window for the M functions in the jog and automatic modes shows which is the active gear; if none is shown, it means that gear ·1· is the active one. The active gear may be consulted using the following variable.

(V.)[ch].G.MS[nb]
Status of the requested "M" function. Each function has a bit that indicates whether the relevant function is active (bit=1) or not (bit=0).

Gear change on Sercos spindles.

When using Sercos spindles, functions M41-M44 also involve changing the drive's velocity set.

How to know the gear configuration of a spindle.

Both the type of gear change (automatic or manual) and the maximum speed for each gear are defined by the machine manufacturer. The configuration may be checked directly in the machine parameter table or using the following variables.

(V.)[ch].MPA.AUTOGEAR.sn
(V.)[ch].SP.AUTOGEAR.sn
Channel [ch]. Automatic gear change on spindle sn.
This variable returns a ·1· if the gear changer is automatic and a ·0· if it is manual.

(V.)[ch].MPA.NPARSETS.xn
(V.)[ch].MPA.NPARSETS.sn
(V.)[ch].SP.NPARSETS.sn
Channel [ch]. Number of parameter sets available at the spindle sn.

(V.)[ch].MPA.DEFAULTSET.xn
(V.)[ch].MPA.DEFAULTSET.sn
(V.)[ch].SP.DEFAULTSET.sn
Channel [ch]. Default parameter set on spindle sn on power-up.

(V.)[ch].MPA.G00FEED[set].xn
(V.)[ch].MPA.G00FEED[set].sn
(V.)[ch].SP.G00FEED[set].sn
Channel [ch]. Maximum speed for each gear of spindle sn.

7.7 M19. Spindle orientation.

This work mode is only available on machines having a rotary encoder installed on the spindle.

Spindle orientation serves to position the spindle at a particular angle. The spindle orientation is defined with function M19. After executing function M19, the spindle no longer works in speed mode and it switches to positioning mode.

Programming a spindle orientation.

The positioning is defined by programming function M19 and then the position of each spindle. The position is represented by the spindle name followed by the desired angle. For spindles like S1, S2, etc. the "=" sign must be included between the name and the position value; the "=" is not necessary for the spindle S. The programming format is the following; the parameters to be set appear between curly brackets.

```
M19 Sn={pos}
```

·sn· Name of the spindle (S, S1··S9).

·pos· Spindle angular position.

```
M19 S0
    (Positioning of spindle S at 0º)
M19 S2=120.78
    (Positioning of spindle S2 at 120.78º)
```

If several spindles are programmed in a single block, function M19 applies to all of them. If the function is programmed in a block that does not mention any spindle, the CNC orients the master spindle at 0º.

```
M19 S1=10 S2=34
    (Positioning of spindle S1 at 10º and S2 at 34º)
M19
    (Positioning of the master spindle at 0º)
```

To orient the spindle to the ·0· position, it may also be programmed by defining, next to the M19, the spindle to be oriented. If no spindle is defined, the CNC assumes that the master spindle is the one to be oriented.

```
M19 S1
    (Positioning of spindle S1 at 0º)
M19 S1=0
    (Positioning of spindle S1 at 0º)
```

Every spindle positioning move requires an M19 and the positioning angle. Even if function M19 is active, if an "S" value is defined without an M19, the CNC assumes it as the new turning speed for the next time the spindle is turned on in speed mode using functions M03/M04.

This angular position is programmed in degrees and it is always assumed in absolute coordinates, thus not being affected by functions G90/G91. To do the positioning, the CNC calculates the module (between 0 and 360º) of the programmed value.

Properties of the function and Influence of the reset, turning the CNC off and of the M30 function.

M19 is modal and incompatible with M03, M04 and M05. On power-up, after executing an M02 or M30, and after an emergency or reset, the CNC sets the spindle in speed mode with function M05.

How is positioning carried out.

When executing function M19, the CNC behaves as follows.

1. The CNC stops the spindle (if it was turning).
2. The spindle no longer works in speed mode and it switches to positioning mode.
3. If it is the first time the M19 is executed, the CNC homes the spindle (home search).
4. It positions the spindle at 0º or at the angular position defined by the "S" code (if it has been programmed). To do that, it will calculate the module (between 0 and 360º) of the programmed value and the spindle will reach that position.

```
N10 G97 S2500 M03
    (The spindle turns at 2500 rpm)
N20 M19 S50
    (Spindle in positioning mode. The spindle orients at 50º)
N30 M19 S150
(Positioning at 150º)
N40 S1000
    (New spindle speed. The spindle stays in positioning mode)
N50 M19 S-100
    (Positioning at -100º)
N60 M03
    (Spindle controlled in speed. The spindle turns at 1000 RPM)
N70 M30
```

It executes the M19 for the first time.

When executing the M19 function for the first time, it homes the spindle. The M19 functions programmed afterwards only orient the spindle. To home the spindle again, use function G74.

7.7.1 Variables related to spindle orientation.

The following are used when the spindle is working in closed loop, it behaves like an axis. A theoretical coordinate is the position that the spindle must occupy at all times, a real coordinate is the one it actually occupies and the difference between these two is called "following error".

SPINDLE POSITION IN M19.

(V.)[ch].A.PPOS.sn
(V.)[ch].SP.PPOS.sn

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Programmed spindle position.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.PPOS.S	Spindle S.
V.SP.PPOS.S	Spindle S.
V.SP.PPOS	Master spindle.
V.A.PPOS.5	Spindle with logic number ·5·.
V.SP.PPOS.2	Spindle with index ·2· in the system.
V.[2].SP.PPOS.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.POS.sn
(V.)[ch].SP.POS.sn

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Real spindle position.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.POS.S	Spindle S.
V.SP.POS.S	Spindle S.
V.SP.POS	Master spindle.
V.A.POS.5	Spindle with logic number ·5·.
V.SP.POS.2	Spindle with index ·2· in the system.
V.[2].SP.POS.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.TPOS.sn
(V.)[ch].SP.TPOS.sn

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Theoretical spindle position.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.TPOS.S	Spindle S.
V.SP.TPOS.S	Spindle S.
V.SP.TPOS	Master spindle.

V.A.TPOS.5	Spindle with logic number ·5·.
V.SP.TPOS.2	Spindle with index ·2· in the system.
V.[2].SP.TPOS.1	Spindle with index ·1· in the channel ·2·.

SPINDLE SPEED IN M19.

(V.)[ch].A.SPOS.sn
(V.)[ch].SP.SPOS.sn

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Active spindle speed in M19 .

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.SPOS.S	Spindle S.
V.SP.SPOS.S	Spindle S.
V.SP.SPOS	Master spindle.
V.A.SPOS.5	Spindle with logic number ·5·.
V.SP.SPOS.2	Spindle with index ·2· in the system.
V.[2].SP.SPOS.1	Spindle with index ·1· in the channel ·2·.

Remarks.

The speed may be set by program or by PLC; the one set by PLC has the highest priority.

Speed set.	V.PLC.SPOS.sn	V.A.PRGSPOS.sn	V.A.SPOS.sn
By program; S.POS=180. By PLC; none.	0	180	180
By program; S.POS=180. By PLC; S.POS=250.	250	180	250
By program; S.POS=180. By PLC; S.POS=90.	90	180	90
By program; S.POS=180. By PLC; none. By MDI; S.POS=200.	0	200	200
By program; S.POS=180. By PLC; S.POS=250. By MDI; S.POS=200.	250	200	250
By program; S.POS=180. By PLC; S.POS=100. By MDI; S.POS=200.	100	200	100

(V.)[ch].PLC.SPOS.sn

Variable that can be read and written from the PLC and read from the program and from the interface.
The variable returns the execution value; reading it interrupts block preparation.

Channel [ch]. Speed active by PLC for M19.

The speed programmed by PLC prevails over the one programmed by program or MDI. To cancel the speed by PLC, set the variable to ·0·; the CNC applies the speed active by program.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.PLC.SPOS.S2	Spindle S2.
V.PLC.SPOS.5	Spindle with logic number ·5·.
V.[2].PLC.SPOS.1	Spindle with index ·1· in the channel ·2·.

(V.)[ch].A.PRGSPOS.sn
(V.)[ch].SP.PRGSPOS.sn

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.

Channel [ch]. Speed active by program for M19.

Syntax.

·ch· Channel number.

·sn· Name, logic number or index of the spindle.

V.A.PRGSPOS.S	Spindle S.
V.SP.PRGSPOS.S	Spindle S.
V.SP.PRGSPOS	Master spindle.
V.A.PRGSPOS.5	Spindle with logic number ·5·.
V.SP.PRGSPOS.2	Spindle with index ·2· in the system.
V.[2].SP.PRGSPOS.1	Spindle with index ·1· in the channel ·2·.

7.

MULTI-SPINDLE SYSTEM. SPINDLE CONTROL.

M19. Spindle orientation.

FAGOR

CNC 8060
CNC 8065

(REF: 1402)





FAGOR

CNC 8060
CNC 8065

(REF: 1402)