





(REF: 0809)

# **OPERATING MANUAL**

(Soft: V04.0x) (Ref: 0809)



#### **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:

- · 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.

If any of them is disabled, the CNC shows a warning message and it must be enabled to guarantee a safe working environment.

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

		tration of conformity [8070 CNC]	
		tration of conformity [15" LCD passive monitor]	
		on history	
	,	y conditions	
		anty terms	
		rial returning terms	
		ed documentation	
CHAPTER 1	GENE	ERAL CONCEPTS	
JIAI ILII I			
	1.1	CNC configuration. Hardware	
	1.2 1.2.1	Turning the CNC on and off	
	1.2.1	Emergency shutdown with battery  Work modes and software protection at the CNC	
	1.3	Description of the keys	
	1.4.1	Keys associated with the information on the screen	
	1.4.1		
	1.4.3	Description of the operator panel	
	1.5	Directory structure	
	1.5.1	MTB (Machine Tool Builder) directory	
	1.5.2	USERS directory	
		<u> </u>	
CHAPTER 2	HOW	TO OPERATE THE CNC	
	2.1	Screen description	17
	2.1.1	Detailed description of the CNC status bar	18
	2.2	Operating modes	
	2.2.1	How to access to the operating modes	
	2.2.2	Description of the various operating modes	
	2.3	Dialog boxes	
	2.4	Windows for warnings and errors	
	2.5	File selection window	
	2.6	Calculator	
	2.6.1 2.7	Defining expressions.	
	2.7	Keyboard shortcuts	32
CHAPTER 3	AUTO	DMATIC MODE	
	3.1	Displaying the automatic mode	36
	3.1.1	Description of the general status bar	37
	3.1.2	Channel synchronization window	
	3.1.3	Icon description (vertical softkeys)	
	3.2	Data screen	
	3.3	Data screen (softkeys)	
	3.3.1	First block	
	3.3.2	Stop condition	
	3.3.3 3.3.4	Block search	
	3.3.4	Active subroutines	_
	3.3.5	Find text  Program selection and execution	
	3.4.1		
	3.4.1	Program execution	
	3.4.2	Program execution  Executing program blocks separately	
	3.4.4	Program execution (retrace)	
	3.4.5	Tool inspection.	
	3.4.6	Block search	
CHAPTER 4	MANU	UAL (JOG) MODE	
	4.1	Appearance of the Manual (JOG) mode	60
	4.1.1	Description of a typical screen of this work mode	61

4.1.2



**CNC 8070** 

	4.2	Operations with the axes	64
	4.2.1	Home search	64
	4.2.2	Jogging the axes	65
	4.2.3	Jogging the axes with handwheels	
	4.2.4	Moving an axis to a particular position (coordinate)	69
	4.2.5	Coordinate preset	70
	4.3	Spindle control	
	4.4	Tool selection and tool change	
	4.5	Definition of cutting conditions	
	4.6	Automatic loading of zero offsets or fixture offset tables	74
CHAPTER 5	MANU	AL (JOG) MODE. TOOL CALIBRATION	
	5.1	Manual calibration. Calibration without a probe	77
	5.2	Semi-automatic calibration. Calibration with a probe	81
	5.3	Automatic calibration with a probe and a canned cycle	84
	5.3.1	Mill or lathe model ("trihedron" geometrical configuration)	84
	5.3.2	Lathe model ("plane" geometrical configuration)	87
CHAPTER 6	MANU	AL (JOG) MODE. PART CENTERING (MILL MODEL)	
	6.1	How to define the data	91
	6.2	Data programming	
	6.3	Basic operation	95
CHAPTER 7	EDITIN	NG-SIMULATION MODE	
	7.1	Appearance of the editing - simulation	98
	7.1.1	General screen description	
	7.1.2	Window description	
	7.1.3	Icon description (vertical softkeys)	
	7.2	Program editing and simulation	
	7.2.1	Program editing	
	7.2.2	Editing a program in the 8055 CNC language	
	7.2.3	Program simulation	
	7.2.4	Simulation errors	108
	7.3	Editing window	109
	7.3.1	Contextual programming assistance	112
	7.3.2	Help for programming subroutines	113
	7.3.3	Import DXF files	114
	7.3.4	Syntax errors when editing	115
	7.4	Editing window (softkeys)	116
	7.4.1	Open program	116
	7.4.2	File	117
	7.4.3	Undo	118
	7.4.4	Operations with blocks	
	7.4.5	Find/replace	
	7.4.6	Customizing	
	7.4.7	Profile editor	
	7.4.8	Canned cycles	
	7.4.9	Incline planes.	
	7.4.10	TEACH-IN	
	7.5	Graphics window	
	7.6	Program window	
	7.7	Program window (softkeys)	
	7.7.1	First block	
	7.7.2	Stop condition	
	7.7.3 7.7.4	Active subroutines.	
		Find text	
	7.8 7.8.1	Statistics window  Time estimates	
	7.8.1 7.9	Profile editor	
	7.9 7.9.1	How to use the profile editor.	
	7.9.1 7.9.2	Profile definition	
	7.9.2 7.9.3	Define any profile	
	7.9.3 7.9.4	• •	
	7.9.4 7.9.5	Define a circular profile  Define a rectangular profile	
	7.9.5 7.9.6	Modify a profile and insert corners	
	7.9.6 7.9.7	Profile editor. Example 1 (milling).	
	7.9.7 7.9.8	Profile editor. Example 1 (milling).	
	7.9.6	Profile editor. Example 3 (milling).	
	7.9.10		



**CNC 8070** 

CHAPTER 8	GRAP	HIC ENVIRONMENT (MILL MODEL)	
	8.1	Type of graphics	151
	8.2	Zoom	
	8.3	Dimensions	153
	8.4	Point of view	154
	8.5	Measurement	
	8.6	Clear screen	
	8.7	Colors	
	8.8	Options	
	8.9 8.10	Real coordinatesSimulation speed	
	0.10	Circulation speed	
CHAPTER 9	GRAP	HIC ENVIRONMENT (LATHE MODEL)	
	9.1	Type of graphics	163
	9.2	Zoom	
	9.3	Dimensions	
	9.4	Measurement	
	9.5	Clear screen	
	9.6	Colors	
	9.7 9.8	Options  Real coordinates	
	9.9	Simulation speed	
	0.0		
CHAPTER 10	MDI M	ODE	
	10.1	Appearance of the MDI mode	
	10.1.1	Window description	175
	10.2	Standard MDI window	
	10.2.1	Block editing and execution	
	10.3	Full MDI screen	_
	10.3.1	Block execution	
	10.3.2	Save the blocks as a program	180
CHAPTER 11	USER	TABLES	
	11.1	Appearance of the table mode	182
	11.1.1	Icon description (vertical softkeys)	
	11.2	Zero offset tables	184
	11.3	Fixture table	185
	11.4	Arithmetic parameter tables	
	11.5	Operations with tables	
	11.5.1	Data editing	
	11.5.2 11.5.3	Save and recall tables	
CHAPTER 12		AND MAGAZINE TABLE	
	12.1	Showing tables and common operations	
	12.1.1 12.1.2	Table selection	
	12.1.2	Save and load the tables	
	12.1.4	Printing the tables	
	12.1.4	Tool table	
	12.2.1	Description of the icons of the vertical softkey menu	
	12.2.2	The tool list	
	12.2.3	Description of the tool data	
	12.3	Operations with the tool table	
	12.3.1	Editing the tool table	
	12.4	Active-tools table	211
	12.4.1	Description of the icons of the vertical softkey menu	
	12.4.2	Changing the tool of the spindle	
	12.5	Table for the status of the tool change process	
	12.6	Magazine table	
	12.6.1	Description of the icons of the vertical softkey menu	
	12.6.2	List of magazine positions	
	12.6.3	Magazine information	
	12.7 12.7.1	Operations with the magazine table	
	12.7.1	Load / unload a tool to / from the tool changer arm	
		and a a too to / from the tool endinger and the first	



**CNC 8070** 

CHAPTER 13	UTILITIES MODE	
	13.1 Appearance of the utilities mode	226
	13.1.1 Screen description	227
	13.1.2 Window description	
	13.1.3 Vertical softkey menu (icons)	
	13.2 Utilities (Softkeys)	
	13.2.1 Sorted by	
	13.2.3 Search in files	
	13.2.4 Select all	
	13.2.5 Invert selection	
	13.2.6 New folder	
	13.2.7 Protection passwords	234
	13.2.8 Data safety backup. Backup - Restore	236
CHAPTER 14	PLC	
	14.1 Appearance of the PLC mode	238
	14.1.1 Screen description	239
	14.1.2 Icon description (vertical softkeys)	
	14.2 "Programs" service	
	14.2.1 PLC project (softkeys)	
	14.2.2 Files of the PLC project PLC (Softkey)	
	14.3 Program editing	
	14.4 Editing in C language or mnemonic language (softkeys)	
	14.4.2 File	
	14.4.3 Undo	
	14.4.4 Operations with blocks	
	14.4.5 Find/Replace	
	14.4.6 Customizing	
	14.5 Editing in contact (ladder) language (softkeys).	250
	14.5.1 Analyze	250
	14.5.2 File	250
	14.5.3 Editing	251
	14.5.4 View	
	14.5.5 Marks	_
	14.5.6 Find	
	14.5.7 Customizing	
	14.6 Program monitoring	
	14.6.1 Monitoring in C language or mnemonic language (softkeys)	
	14.7 "Commands" service	
	14.7.1 Options of the "Commands" service (softkeys)	
	14.8 "Outputs" service	
	14.8.1 Options of the "Outputs" service	
	14.9 "Logic analyzer" service	264
	14.9.1 Editing logic analyzer data	265
	14.9.2 Save, load and reset the analyzer configuration	
	14.9.3 Execute and analyze trace	
	14.9.4 Customize the appearance of the logic analyzer	
	14.10 "Monitoring" service	
	14.10.1 Description of resource tables	
	14.10.2 Definition of the table resources	
	14.10.3 Options of the "Monitoring" service (softkeys)	
	14.11. Options of the "Cross reference" service (softkeys)	
	14.11.1 Options of the Gloss reference service (softkeys)	
	14.12.1 Options of the "Statistics" service (softkeys)	
	14.13 "Messages" service	
	14.13.1 Options of the "Messages" service (softkeys)	
	14.13.2 Editing the message and error table	
	14.13.3 Displaying PLC messages	
	14.13.4 Displaying PLC errors	
	14.13.5 Grouping the additional information text files in a single file	283
	14.13.6 Save, load and print a message and error table	284
CHAPTER 15	MACHINE PARAMETERS	
	15.1 Appearance of the machine parameter tables	286
	15.1.1 Icon description (vertical softkeys)	
	. , ,	

FAGOR =

**CNC 8070** 

	15.2	Parameter table description	
	15.2.1	"M" function setting table	290
	15.2.2	Compensation table	291
	15.2.3	OEM parameters	292
	15.3	Operations with tables	
	15.3.1	Data editing and validation	
	15.3.2	Save and recall tables	
	15.3.3	Find text	
	15.3.4	Importing and exporting compensation tables	
	10.0.1	importing and experting compensation tables.	
CHAPTER 16	SETUP	ASSISTANCE	
	16.1	Oscilloscope	200
	16.1.1	Interface description	
	16.1.1	Configuration screen	
	16.1.2	Configure and execute the oscilloscope function	
	16.1.4	Machine parameter editing	
	_		
	16.2	The Bode diagram	
	16.2.1	Interface description	
	16.2.2	Machine parameter editing	
	16.2.3	Configuration screen	
	16.3	The circularity (roundness) test	
	16.3.1	Interface description	
	16.3.2	Configuring and executing the circularity (roundness) test	
	16.3.3	Configure the graphic environment	
	16.3.4	Define and execute the movement subroutine	
	16.3.5	Data capture for the graphic	
	16.3.6	Adjustment of the machine parameters involved	
	16.3.7	Validate the changes and save the configuration used	
	16.3.8	Machine parameters that may be modified	331
CHAPTER 17	17.1		334
CHAPTER 17		Appearance of the DDSSetup mode	
CHAPTER 17	17.1	Appearance of the DDSSetup mode	335
CHAPTER 17	17.1 17.1.1	Appearance of the DDSSetup mode	335
CHAPTER 17	17.1 17.1.1 17.1.2	Appearance of the DDSSetup mode	335 336 337
CHAPTER 17	17.1 17.1.1 17.1.2 17.2	Appearance of the DDSSetup mode	335 336 337 338
CHAPTER 17	17.1 17.1.1 17.1.2 17.2 17.3 17.4	Appearance of the DDSSetup mode	
CHAPTER 17	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5	Appearance of the DDSSetup mode	
CHAPTER 17	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6	Appearance of the DDSSetup mode	
CHAPTER 17	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6	Appearance of the DDSSetup mode	
CHAPTER 17	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7	Appearance of the DDSSetup mode	
CHAPTER 17	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343
CHAPTER 17	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9 17.10	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343
CHAPTER 17	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343
	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9 17.10	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343 344 344
	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9 17.10	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343 344 346
	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9 17.10	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343 344 346
	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9 17.10 DIAGN	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343 344 346
	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9 17.10 DIAGN  18.1 18.1.1	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343 344 346
	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9 17.10 DIAGN  18.1 18.1.1 18.1.2	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343 344 346 346
	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9 17.10 DIAGN  18.1 18.1.1 18.1.2 18.2 18.2.1	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343 344 346 346
	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9 17.10 DIAGN 18.1 18.1.1 18.1.2 18.2 18.2.1 18.2.2	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343 344 346 346 348 349 350 351 351 352 353
	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9 17.10 DIAGN 18.1 18.1.1 18.1.2 18.2 18.2.1 18.2.2 18.2.3	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343 344 346 346 348 349 350 351 351 352 353
	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9 17.10 DIAGN 18.1 18.1.1 18.1.2 18.2 18.2.1 18.2.2 18.2.3 18.3	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343 344 346 346 350 351 351 352 353 354 355
	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9 17.10 DIAGN 18.1 18.1.1 18.1.2 18.2 18.2.1 18.2.2 18.2.3 18.3 18.4 18.5	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343 344 346 346 348 349 350 351 351 352 353 354 355 356
	17.1 17.1.1 17.1.2 17.2 17.3 17.4 17.5 17.6 17.7 17.8 17.9 17.10 DIAGN 18.1 18.1.1 18.1.2 18.2 18.2.1 18.2.2 18.2.3 18.3 18.4	Appearance of the DDSSetup mode	335 336 337 338 339 340 341 342 343 344 346 346 348 350 351 351 352 353 354 355 356 356



**CNC 8070** 

# **ABOUT THE PRODUCT**

# **BASIC CHARACTERISTICS.**

Basic characteristics.	
PC-based open system.	Windows XP
Number of axes.	maximum 28.
Number of spindles.	maximum 4.
Number of tool magazines.	maximum 4.
Number of execution channels.	maximum 4.
Number of handwheels.	maximum 12.
Type of servo system. Type of digital servo system.	Analog / Digital Sercos
Communications.	(PC104) RS232 / Ethernet (ICU) RS485 / Ethernet (MCU) RS485 / Ethernet
Integrated PLC.	
PLC execution time.	< 1ms/K
Digital inputs / Digital outputs.	1024 / 1024
Marks / Registers.	8192 / 1024
Timers / Counters.	256 / 256
Symbols.	Unlimited
Block processing time.	< 1 ms

Remote modules.	
Communication with the remote modules.	CANopen / CANfagor
Digital inputs per module (CANopen / CANfagor).	16 or 32 / 16
Digital outputs per module (CANopen / CANfagor).	24 or 48 / 16
Analog inputs per module (CANopen / CANfagor).	4 / 8
Analog outputs per module (CANopen / CANfagor).	4 / 4
Inputs for the temperature sensors PT100 (CANopen).	2
Feedback inputs (CANfagor).	4
	Differential TTL / Sinusoidal

Customizing.	
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	



**CNC 8070** 

# **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.

	-OL- model	-M- model	-T- model
Open system. Access to the administrator mode.	Option	Option	Option
Number of execution channels	1 to 4	1 to 4	1 to 4
Number of axes	3 to 28	3 to 28	3 to 28
Number of spindles	1 to 4	1 to 4	1 to 4
Number of tool magazines	1 to 4	1 to 4	1 to 4
COCOM version	Option	Option	Option
Dual-purpose machines (M-T)	Option	Option	Option
Non-Fagor digital drive	Option	Option	Option
Tool radius compensation	Option	Standard	Standard
"C" axis	Option	Standard	Option
RTCP transformation	Not available	Option	Option
High speed machining (HSC).	Option	Option	Option
Probing canned cycles	Not available	Option	Option
Drilling ISO cycles for the GP model. (G80, G81, G82, G83).	Option		
Tandem axes	Option	Option	Option
Synchronism and cams	Option	Option	Option
Tangential control	Option	Option	Option



**CNC 8070** 

# DECLARATION OF CONFORMITY [8070 CNC]

# The manufacturer:

Fagor Automation S. Coop.

Barrio de San Andrés  $N^{\circ}$  19, C.P. 20500, Mondragón -Guipúzcoa- (Spain).

#### We declare:

We declare under our exclusive responsibility the conformity of the product:

# Numerical Control Fagor 8070 CNC

Referred to by this declaration with following directives:

# Low-voltage regulations.

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

# Regulation on electromagnetic compatibility.

EN 61000-6-4	Generic regulation on emissions in industrial environments.	
EN 55011	Radiated. Class A, Group 1.	
(*) EN 55011	Conducted. Class A, Group 1.	
(*) EN 61000-3-2	Current harmonics.	
(*) EN 61000-3-3	Flickers and Voltage fluctuations.	
EN 61000-6-2	Generic regulation on immunity in industrial environments.	
EN 61000-4-2	Electrostatic discharges.	

EN 61000-4-3 Radiofrequency radiated electromagnetic fields. EN 61000-4-4 Bursts and fast transients.

(\*) EN 61000-4-5 High Voltage conducted pulses (Surges).

EN 61000-6-4 Conducted disturbance induced by radio frequency fields.

EN 61000-4-8 Magnetic fields to Mains frequency.
EN 61000-4-11 Voltage fluctuations and Outages.

ENV 50204 Fields generated by digital radio-telephones.

(\*) Only for the 8070 model with central unit PCI.

As instructed by the European Community Directives: 73/23/CEE modified by 93/68/EEC on Low Voltage and 89/336/CEE modified by 92/31/EEC and 93/68/EEC on Electromagnetic Compatibility and their updates.

In Mondragón, March 20th 2007.

agor Automation a. Ocop. inter-

Pcio.: Julen Busturia

FAGOR

**CNC 8070** 

# DECLARATION OF CONFORMITY [15" LCD PASSIVE MONITOR]

#### The manufacturer:

Fagor Automation S. Coop.

Barrio de San Andrés Nº 19, C.P. 20500, Mondragón -Guipúzcoa- (Spain).

#### We declare:

We declare under our exclusive responsibility the conformity of the product:

# Numerical Control Fagor Passive monitor LCD-15

Referred to by this declaration with following directives:

# Low-voltage regulations.

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

#### Regulation on electromagnetic compatibility.

EN 61000-6-4	$\label{eq:Generic regulation on emissions in industrial environments.}$
EN 55011	Radiated. Class A, Group 1.

EN 61000-6-2 Generic regulation on immunity in industrial environments.

EN 61000-4-2 Electrostatic discharges.

EN 61000-4-3 Radiofrequency radiated electromagnetic fields.

EN 61000-4-4 Bursts and fast transients.

EN 61000-4-6 Conducted disturbance induced by radio frequency fields.

EN 61000-4-8 Magnetic fields to Mains frequency. EN 61000-4-11 Voltage fluctuations and Outages.

ENV 50204 Fields generated by digital radio-telephones.

As instructed by the European Community Directives: 73/23/CEE modified by 93/68/EEC on Low Voltage and 89/336/CEE modified by 92/31/EEC and 93/68/EEC on Electromagnetic Compatibility and their updates.

In Mondragón, March 20th 2007.

Fcio.: Julen Busturia

FAGOR

**CNC 8070** 

# **VERSION HISTORY**

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

Ref. 0201 Software V01.00

First version.

Ref. 0212 Software V01.10

Set the numbering of the digital I/O.	Machine parameters:     NDIMOD, DIMODADRR, DIMOD, NDOMOD,     DOMODADRR, DOMOD.
Probe management from the digital inputs; it is not possible to manage remote feedback inputs ("Counter" module).	<ul> <li>Machine parameters: PRBDI1, PRBDI2, PRBPULSE1, PRBPULSE2.</li> </ul>
Tabletop probe configuration.	<ul> <li>Machine parameters: PROBE, PRB1MIN, PRB1MAX, PRB2MIN, PRB2MAX, PRB3MIN, PRB3MAX.</li> </ul>
Define the repositioning feedrate after tool inspection.	Machine parameter: REPOSFEED.
New treatment of the JOG keys. Different keys to select the axis and the direction.	Machine parameter: JOGKEYDEF.
Kinematics for rotary tables (TYPE9 to TYPE12).	Kinematics TYPE9 through TYPE12.
Know the dimensions of the kinematics on an axis.	Variable (V.)A.HEADOF.xn
Keyboard simulation from the PLC.	Variable: (V.)G.KEY
Park and unpark Sercos axes from the PLC.	PLC command: PARK, UNPARK.
Jog mode. Tool calibration with or without probe.	
Jog mode. Automatic loading of zero offsets table.	
Jog mode. Programming of feedrate "F" and spindle speed "S".	
MDI mode. Block syntax check.	
Utilities mode. Define protection passwords.	
Block search. Define the first block.	
Improved tool table.	
Axis selection/deselection to move it with a handwheel.	
Simulate the theoretical path.	
Confirm the execution of a program pressing the [START] key in a mode other than automatic.	
General scaling factor.	New instruction, #SCALE.
Probe selection.	New instruction, #SELECT PROBE.
Probing canned cycles.	New instruction, #PROBE.
Programming of warnings.	New instruction, #WARNING.
Block repetition.	New instruction, #RPT.
Know the active general scaling factor.	Variable: (V.)G.SCALE
Knowing which is the active probe.	Variable: (V.)G.ACTIVPROBE
Improved programming of high speed machining.	#HSC instruction.
Improved programming of axis swapping.	<ul> <li>Instructions #SET, #CALL, #FREE, #RENAME.</li> </ul>
The number of macros in a program is now limited to 50.	Macros.

Ref. 0501 Software V02.01

Windows XP operating system.	
Emergency shutdown with battery (Central unit PC104).	
Multi-channel system, up to 4 channels. Swapping of axes and spindles, communication and synchronization between channels, common arithmetic parameters, access variables by channel, etc.	
Multi-spindle system, up to 4 spindles.	
Tool management with up to 4 magazines.	
Parameter matching between the CNC and the Sercos drive.	
Sercos control in velocity.	
New kinematics table-spindle (TYPE13 to TYPE16).	Kinematics TYPE13 through TYPE16.
New kinematics for C axis (TYPE41 to TYPE43).	Kinematics TYPE41 through TYPE43.
New languages (Basque and Portuguese).	Machine parameter: LANGUAGE.
Placing the vertical softkeys on the left or on the right.	Machine parameter: VMENU.



**CNC 8070** 

Tandem axes.	Machine parameters:
randem axes.	TANDEM, TMASTERAXIS, TSLAVEAXI TORQDIST, PRELOAD, PRELFITI, TPROGAI TINTTIME, TCOMPLIM.
Gantry axis. Maximum difference allowed between the following errors of both axes before issuing a warning.	Machine parameter: WARNCOUPE.
Apply cross compensation to either theoretical or real coordinates.	Machine parameter: TYPCROSS.
Apply leadscrew compensation to either theoretical or real coordinates.	Machine parameter: TYPLSCRW.
Tool radius compensation mode (G136/G137) by default	Machine parameter: IRCOMP.
Defining the type of reference pulse.	Machine parameter: REFPULSE.
Memory sharing between applications.	Machine parameter: PLCDATASIZE.
OEM generic machine parameters.	Machine parameter: MTBPAR.
Reading Sercos variables from the CNC.	Machine parameter: DRIVEVAR.
Electronic-cam editor.	Machine parameter: CAM.
Backlash peak compensation.	Machine parameters:     BAKANOUT, BAKTIME, ACTBAKAN.
New behavior for rotary axes.	Machine parameters:     AXISMODE, UNIDIR, SHORTESTWAY.
Sercos transmission at 8 MHz and 16 MHz.	Machine parameter: SERBRATE.
Define the anticipation time for the axes to be considered to be in position.	Machine parameters: ANTIME.      DI Constitute ADVIANDOS
DLO. The TMODED ATION was a ball the control of the side of the	PLC marks: ADVINPOS.  PLC marks: TMORERATION
PLC. The TMOPERATION may take the values 13 and 14.	PLC marks: TMOPERATION.     PLC marks: MMCW/DC
PLC. Detect when the operating system locks up.  PLC. Disable the cross compensation tables.	PLC marks: MMCWDG.     PLC marks: DISCROSS.
PLC. Disable the cross compensation tables.  PLC. Correct the parallelism on Gantry axes.	PLC marks: DISCHOSS.     PLC marks: DIFFCOMP.
PLC. Correct the parallelism on Gantry axes. PLC. Execute CNC blocks.	PLC marks: DIFFCOMP.     PLC command: CNCEX.
1 EG. EXCOURTE ON O DIOUNG.	PLC command. CNCEX.     PLC marks: FREE.
PLC. Reading arithmetic parameters and OEM parameters with CNCRD returns the value multiplied by 10000 (reading in float mode).	PLC command: CNCRD.
PLC. Define external symbols.	PLC command: PDEF.
The RESETIN mark is not necessary to park/unpark axes or spindles from the PLC.	PLC marks: RESETIN, PARK, UNPARK.
The "(V.).TM.MZWAIT" variable is not necessary in the subroutine associated	Subroutine associated with M6.
with M06.  Apply filters to eliminate the resonance of the spindle when it works as C axis	<ul><li>Variable: (V.).TM.MZWAIT</li><li>Frequency filters. "C" axis. Rigid tapping.</li></ul>
<ul> <li>The tool variables will be read asynchronously when the tool is neither the active one nor in the magazine.</li> <li>The tool variables will be written asynchronously whether the tool is the active one or not.</li> <li>The variables referred to local arithmetic parameters of the active levels</li> </ul>	
will be read and written asynchronously.	V
Know the software version.	Variable: (V.)G.SOFTWARE
Variables to be set via PLC.	<ul> <li>Variables:         <ul> <li>(V.)A.PLCFFGAIN.xn</li> <li>(V.)A.PLCACFGAIN.xi</li> </ul> </li> <li>(V.)A.PLCPROGAIN.xn</li> </ul>
Variables for adjusting the position.	Variables: (V.)A.POSINC.xn (V.)A.TPOSINC.xn (V.)A.PREVPOSINC.xn
Fine adjustment variables.	Variables: (V.)A.FEED.xn (V.)A.GCEL.xn (V.)A.JERK.xn (V.)A.JERK.xn (V.)A.TJERK.xn
Obtain information about the feedback inputs.	Variables: (V.)A.COUNTER.xn (V.)A.ASINUS.xn (V.)A.BSINUS.xn
Spindle parking and unparking.	PLC marks: PARK, UNPARK.     Instructions #PARK, #UNPARK.
Tool radius compensation.	
Behavior of the beginning and end of tool radius compensation when not programming a movement.	
Changing the type of radius compensation while machining.      Changing the type of radius compensation while machining.      Changing the type of radius compensation while machining.	
Via program, loading a tool in a specific magazine position.	a New instruction WMOALL
Programming of modal subroutines.	New instruction, #MCALL.
Executing a block in a channel.	New instruction, #EXBLK.      NP command
Programming the number of repetitions in the block.	NR command.
Direct resolution of 2D and 3D pockets without requiring a softkey.	
Simulating a canned cycle of the editor separately.  Importing DXF files from the program editor or from the profile editor.	
Importing programs of the 8055/8055i CNC from the program editor.	
Use a softkey to select the repositioning of the spindle after tool inspection.	
Backup-restore utility.	
Improved profile editor.	



CNC 8070

Assistance in the program editor. Contextual programming assistance.

- When programming "#", it shows the list of instructions.
- When programming "\$", it shows the list of instructions.
- When programming "V.", it shows the list of variables.

Specific password for the machine parameters for kinematics.

Save the CAN configuration for testing it when starting up the system.

The diagnosis mode shows detailed information on the Sercos connection (Type and version of the drive and motor connected to it).

It is possible to print all the information on the configuration from any section of the diagnosis mode.

It is possible to simulate a cycle separately from the cycle editor.

Setup assistance. Oscilloscope, Bode diagram, circularity test.

Ref. 0504 Software V02.03

New values of machine parameter SERPOWSE for the "Sercos II" board.	<ul> <li>Machine parameters: SERPOWSE</li> </ul>
Independent-axis programming commands.	<ul> <li>PLC command: MOVE, FOLLOW.</li> </ul>
Electronic-cams programming commands.	PLC command: CAM.
New signals that may be consulted and changed for the independent interpolator (electronic cam and independent axis)	
The simulated axes are ignored regarding the validation code.	
When unifying parameters, G00FEED and MAXVOLT are not sent out to the drive.	
Electronic cam programming (real coordinates).	New instruction, #CAM.
Synchronization of independent axis (real coordinates).	New instruction, #FOLLOW.
Movement of the independent axis.	New instruction, #MOVE.
DDSSETUP mode.	
G31. Temporary polar origin shift to the center of interpolation.	G31 function.
G112. Change the drive's parameter set.	G112 function.

Ref. 0509 Software V03.00

la dia a suita	
Incline axis.	Marking a server of a NIMODE
CAN bus type selection.	Machine parameter: CANMODE.
Permit using the G95 function in jog mode.	Machine parameter: FPRMAN.
Lathe model. Select graphics configuration.	Machine parameter: GRAPHTYPE.
Lathe model. Select axis configuration.	Machine parameter: GEOCONFIG.
Select the set of parameters for synchronization.	Machine parameter: SYNCSET.
"C" axis maintained.	Machine parameter: PERCAX.
Improved definition of kinematics for the C axis.	
Magazine-less system.	
Ground tools for a turret magazine. The TMOPERATION register may take the values 3, 4, 9 and 10.	PLC register: TMOPERATION.
Commands CNCRD and CNCWR. The channel number and the indexes may be defined in the variables using an integer, a register or a symbol.	PLC commands: CNCRD and CNCWR.
Variable to read the accumulated PLC offset.	Variable: (V.)[ch].A.ACTPLCOF.xn
Variable to obtain a linear estimation of the following error.	Variable: (V.)[ch].A.FLWEST.xn
Variables to read the instant value of feed-forward or AC-forward.	<ul> <li>Variables: (V.)[ch].A.ACTFFW.xn (V.)[ch].A.ACTACF.xn</li> </ul>
Variable to know the line number of the file being executed.	Variable: (V.)[ch].G.LINEN
Variable to know what kind of cycle is active.	Variable: (V.)[ch].G.CYCLETYPEON
Variable to know the tool orientation.	Variable: (V.)[ch].G.TOOLDIR
Variable to know whether the HSC mode is active or not.	Variable: (V.)[ch].G.HSC
Variable to know the theoretical feedrate on 3D path.	Variable: (V.)[ch].G.F3D
Variable to know the number of the warning being displayed.	Variable: (V.)[ch].G.CNCWARNING
The variable (V.)G.CNCERR is now per channel.	Variable: (V.)G.CNCERR
Select the type of loop, open or closed, for the spindle.	New instruction, #SERVO.
Spindle synchronization.	New instruction, #SYNC.
Spindle synchronization.	New instruction, #TSYNC.
Spindle synchronization.	New instruction, #UNSYNC.
Select milling cycles at a lathe model.	New instruction, #MILLCY.
Select turning cycles at a milling model.	New instruction, #LATHECY.
Define a kinematics when activating the C axis.	#CYL instruction.
Define a kinematics when activating the C axis.	#FACE instruction.
Improved coordinate transformation (#CS/#ACS).	Instructions #CS, #ACS.
Keep the part zero when deactivating the transformation.	
Working with 45° spindles. Select between the two choices.	
Keep the rotation of the plane axes with MODE 6.	
G33. New parameter (Q1) to define the entry angle.	G33 function.
G63. Tool inspection is possible during rigid tapping.	G63 function.



**CNC 8070** 

G112. G112 is no longer admitted for the spindle.	<ul> <li>G112 function.</li> </ul>	
New criteria when assuming a new master spindle in the channel.		
Improved tool table.		

Ref. 0601 Software V03.01

Machine parameters: CANLENGTH.
<ul> <li>Machine parameters: LINKCANCEL.</li> </ul>
Machine parameters: COMPCANCEL.
Machine parameters:     FBACKSRC, FBACKDIFF.
<ul> <li>PLC marks: FBACKSEL(axis), ACTBACK(axis).</li> </ul>
PLC marks: SYNCRONP.
PLC marks: SERCOSRDY.
<ul> <li>Variables: (V.)TM.TOOLCH1[mz] (V.)TM.TOOLCH2[mz]</li> </ul>
#EXEC instruction.
#EXBLK instruction.

Ref. 0606 Software V03.10

Feedrate. Maximum machining feedrate.	Machine parameter: MAXFEED.
Feedrate. Default machining feedrate when none has been programmed.	Machine parameter: DEFAULTFEED.
The user keys may be configured as jog keys.	Machine parameter: USERKEYDEF.
Disabling a keyboard or jog panel integrated into the CAN bus.	PLC mark: PANELOFF.
Handwheel with push-button. Selecting an axis sequentially for jogging it with	PLC mark: NEXTMPGAXIS.
the handwheel.	•
CNCEX command. Aborting CNCEX commands launched from the PLC.	PLC mark: PLCABORT.
CANopen protocol.	Machine parameter: CANMODE.
Block search. Functions M, H, F, S are not sent out to the PLC.	Machine parameter: FUNPLC.
Tapping. Changing the override while threading.	Machine parameters:
	THREADOVR, OVRFILTER.
Dead axis. Handling the blending of (transition between) blocks.	PLC mark: DEAD(axis)
Kinematics. Integrating OEM kinematics through machine parameters.	
Kinematics. More axes can now get involved in a kinematics (from 5 to 8).	
Kinematics. Type 41/42. Angular offset for the rotary axis.	<ul> <li>Kinematics TYPE41 and TYPE42.</li> </ul>
Kinematics. Type 41/42. Tool misalignment with the C axis.	<ul> <li>Kinematics TYPE41 and TYPE42.</li> </ul>
Kinematics. Type 43. Angular offset for the rotary axis.	Kinematics TYPE43.
OFM machine parameters	

- Range of parameters that can be written from the part-program, from the PLC or from the interface.
- Range of parameters affected by the change of units.
- Each parameter may have a different describing comment associated with it.

New FAGOR low passing filters.

Home search. New home searching method for spindles with home switch. The spindle goes through the home switch twice.

Configuring 2 axes with the same feedback input and analog output.

The CNC displays the warnings generated at the drive.

M function table. Each M function may have a different describing comment associated with it.

General handwheel. The CNC may have several general handwheels.

 $General\,hand wheel.\,A\,general\,hand wheel\,can\,move\,several\,axes\,at\,the\,same$ time.

Improvements in the looks of some softkeys of the editor.

Improvements in the looks of some softkeys of the graphics window.

Editing mode. Programming help files for OEM and global subroutines.



**CNC 8070** 

Editing mode. Help file with the list of available subroutines.	
Editing mode. Improved contextual assistance.	
Editing mode. New softkey for deactivating the contextual assistance.	
Editing mode. Improvements in the looks of the softkeys.	
The automatic mode offers a softkey for selecting the program that is being edited.	
The automatic and jog modes show the status of the _FEEDHOL mark.	PLC mark: _FEEDHOL.
The automatic and jog modes show the status of the _INHIBIT mark of the axes and spindle.	PLC mark: INHIBIT.
Automatic mode. It shows information on all the spindles.	
Jog mode. It shows information on all the spindles.	
"Retrace" function.	
Tangential control.	
Tool table. New softkey for initializing the positions; T1 in position 1, T2 in position 2, etc.	
Tool table. New softkeys for copying and pasting the data of a tool offset.	
The CNC checks whether the programmed turning direction (M3/M4) matches the one preset in the tool table.	
Generating the warranty registration report.	
Hiding the window for errors and warnings.	
M02/M30. There is no need to program M02 or M30 to end a part program.	Functions M02/M30.
Canceling the preset turning direction of a tool.	Variables: (V.)G.SPDLTURDIR
Change the maximum feedrate allowed in the channel from the PLC.	Variables: (V.)[ch].PLC.PLCG00FEED
Show the status of the emergency relay.	Variables: (V.)G.ERELAYST
HSC. New FAST mode.	#HSC instruction.
"C" axis. The #CYL instruction requires programming the radius.	#CYL instruction.
M function table. New field to define whether they are sent out to the PLC or not during block search.	
Improved block search.	
Tool calibration.	
Manual calibration. When calibration is done, pressing [START] assumes the new values.	
Semi-automatic calibration. Calibration of lathe tools.	
<ul> <li>Semi-automatic calibration. When calibration is done, pressing [START] assumes the new values.</li> </ul>	
<ul> <li>Automatic calibration. When calibration is done, the CNC assumes the new values.</li> </ul>	

Ref. 0608 Software V03.11

Simulator Possibility to use the dongle (hardware key) in a network.	
Line graphics. Improved resizing of the graphics on the screen.	
"Retrace" function. Several improvements to the retrace function.	
HSC. New command CORNER.	#HSC instruction.
The default value of some machine parameters is different for the CNC and for the simulator installed on a PC.	
G33. The override limitation is maintained while returning to the beginning of the thread.	G33 function.
RTCP. Home search is now possible on the axes that are not involved in RTCP.	
Abort the execution of the program and resume it somewhere else.	<ul> <li>New instruction, #ABORT.</li> </ul>

Ref. 0610 Software V03.12

In parameter matching, the CNC sends parameters it sends REFSHIFT and FBMIXTIME.	
Time constant for combined feedback.	<ul> <li>Machine parameters: FBMIXTIME.</li> </ul>
Machine parameters: FBMIXTIME.	
When the axes are position-Sercos; during parameter matching, the CNC sends the value of parameter REFSHIFT to the drive so it takes it into account; this way the CNC coordinate and that of the drive will be the same.	

# Software V03.13

Sign criteria for tool offsets (dimensions) and tool wear.	<ul> <li>Machine parameters: TOOLOFSG.</li> </ul>



**CNC 8070** 

Define the tool wear with incremental or absolute values.	<ul> <li>Variables: (V.)TM.TOOLCH1[mz] (V.)TM.TOOLCH2[mz].</li> </ul>
Variables V.TM.TOOLCH1[mz] / V.TM.TOOLCH2[mz] may be written from the PLC.	
MDI mode. Cancel the block being executed while keeping the machining conditions.	

Ref. 0704 / Ref. 0706 Software V03.14

MCU and ICU central unit.	
battery powered RAM. Connecting handwheels to the central unit. local I/O. Local feedback inputs. Loca probes.	
The handwheels may be connected to the central unit.	<ul> <li>Machine parameters: COUNTERTYPE, COUNTERID.</li> </ul>
Local feedback inputs.	Machine parameters:     COUNTERTYPE, COUNTERID.
Management of local I/O.	<ul> <li>Machine parameters: NLOCOUT, EXPSCHK.</li> </ul>
Number of non-volatile PLC registers.	Machine parameter: BKUPREG.
Number of non-volatile PLC counters.	Machine parameter: BKUPCOUN.
Number of common non-volatile arithmetic parameters.	Machine parameter: BKUPCUP.
Configuring local probes.	<ul> <li>Machine parameters: PROBETYPE, PRBID.</li> </ul>
Spindle home search.	Machine parameter: REFINI.
Define whether the spindle is homed automatically with the first movement or not.	
The application may be restarted while turning the CNC off.	
The task window may be accessed by clicking on the OEM icon (top left of the status bar).	
The channels may be accessed by clicking on the icons of the status bar).	
The pages of an operating mode may be accessed by clicking on the mode name (top right of the status bar).	
The turning speed limitation (G192) is also applied when the spindle is working at constant turning speed (G97)	G192 function.

Ref. 0707 Software V03.15

In parameter matching, the CNC sends parameter ABSOFF when using absolute feedback.	
On rotary axes or spindles working in velocity-Sercos, the CNC calculates the module of the coordinate. In parameter matching, drive parameter PP76(7) is set to 0.	
Know the type of hardware.	Variable: (V.)G.HARDTYPE
Theoretical tool feedrate along the path.	Variable: (V.)[ch].G.PATHFEED
Managing an analog axis through the analog output and the second feedback of a Sercos drive.	
Every time the diagnosis mode is accessed, the CNC creates the files SystemInfo.txt and SercosInfo.txt.	
PLC errors may have an additional data file associated with them, same as PLC messages.	
User tables. The zero offset table shows the spindles that may be activated	
as C axis.	
Zero offsets for the C axis.	
The CNC shows a warning when a channel is expecting a tool that is being used in another channel.	

Ref. 0709 Software V03.16

Machine parameters: KINID



Tandem spindles.

Machine parameters: KINID

enclosure.

**CNC 8070** 

Diagnosis mode. Monitoring of the temperature of the CPU, board and

The CNC uses the combined feedback to calculate the velocity command, but

it uses the direct feedback to calculate the compensations, circularity test, etc.

The CNC does not assume any kinematics on power-up.

Ref. 0712 Software V03.17

On rotary axes with module and spindle working in velocity-Sercos mode, with a whole gear ratio and with drive parameter PP76(7)=1, the parameter matching does not redefine parameter PP76(7)=1; the CNC shows a warning so the user recalculates the value of drive parameter PP4 (GC6 command).

In a tandem system, the master axis or spindle must have external feedback and the slave internal feedback.

C axis maintained after executing M02, M30 or after an emergency or reset.

• Machine parameter: PERCAX.

Ref. 0801 Software V03.20

The CNC has a different MTB folder for each type of software installed; MTB_T	
for lathe, MTB_M for mill and MTB_MC for motion control.  Configure the PT100 inputs.	Machine parameters: NPT100, PT100.
1	Machine parameters: NFT100, F1100.      Machine parameters: MAXDIFF.
Coordinate compensation on gantry axes.  Status of the local probes.	Variables: (V.)G.PRBST1 (V.)G.PRBST2.
Feedback alarms.	Machine parameter: FBACKAL.
By default, the feedback alarms must be activated.	PLC mark: REFPOIN(axis).
When a feedback alarm occurs on the analog axes, the REFPOIN mark	- I LO Mark. HELL ON (axis).
is set to (=0).	
Set change.	
<ul> <li>For the CNC to assume the new parameter set, it must wait for the PLC to receive the confirmation of one of the marks GEAR1 to GEAR4.</li> </ul>	
<ul> <li>The gear change concludes when the PLC receives the confirmation signal AUXEND.</li> </ul>	
<ul> <li>Sercos spindle. The set change only affects the drive when it implies a change of gear ratio.</li> </ul>	
The CNC lets change the gear of the slave axis or spindle of a tandem.	
Coordinate latching with the help of a probe or a digital input.	Variables:     (V)[ab] A LATCH vp. (V)[ab] A LATCH vp.
	(V.)[ch].A.LATCH.xn (V.)[ch].A.LATCH.xn • PLC command: TOUCHPROBE
	PLC command: TOUCHPROBE     PLC marks:
	PROBE1ACTIVE, PROBE2ACTIVE,
	LATCH1ACTIVE(axis), LATCH2ACTIVE(axis),
	LATCH1DONE(axis), LATCH2DONE(axis).
PLC. The PLC informs that there is an OEM password.	PLC mark: PSWSET.
PLC. The PLC informs that, for the spindle, the parameter set selected at the CNC does not match that of the PLC.	PLC mark: GEAROK.
PLC. Initiate the CNC shut-down sequence.	PLC mark: CNCOFF.
PLC. Dynamic distribution of the machining operations between channels.	<ul> <li>PLC marks: DINDISTC1, DINDISTC2, DINDISTC3 DINDISTC4.</li> </ul>
PLC. Electronic cam programming (theoretical coordinates).	PLC command: TCAM.
PLC. The PLC program can have several mnemonic files (extension "plc").	
PLC. There is no need to use the partition character "\" in the PLC program to divide a logic expression in two lines.	
PLC. When defining each PLC error, it is possible to select whether it opens the emergency relay or not.	
PLC. Grouping the additional information text files in a single file.	
PLC. Contact (ladder) editor.	
Axis synchronization. Managing a rotary axis as an infinite axis making it possible to increase the feedback count of the axis indefinitely (wihout limits) regardless of the value of the module.	Variables: (V.)[ch].A.ACCUDIST.xn
Errors and warnings.	
From the errors and warnings, it is possible to access the errors solving (troubleshooting) manual.	
Errors between 10000 and 20000 are reserved for the OEM so he can create his own warning or error texts in different languages.	
New kinematics for spindles (TYPE17 to TYPE24).	Kinematics TYPE17 through TYPE24.
Show a warning and interrupt program execution.	New instruction, #WARNINGSTOP.
Electronic cam programming (theoretical coordinates).	New instruction, #TCAM.
Dynamic distribution of the machining operations between channels.	New instruction, #DINDIST.
The CNC can park the main axes.	
The axes may be programmed using the "?" wild card that refers to the axis position in the channel.	Wild card "?".
Functions G130 (percentage of acceleration) and G132 (percentage of jerk) may be applied to the spindles.	Functions G130 and G132.
Profile editor. Axes coordinated with auto-scale and name of the axes.	
Profile editor. Zoom and movement of the graphics area via keyboard.	
Profile editor. At the lathe model, the orientation of the axes is defined by parameter GRAPHTYPE.	Machine parameter: GRAPHTYPE.
Edisimu mode. Incline plane programming assistance.	
Edisimu mode. To simulate the program, when pressing the "START" softkey, the CNC assumes the real configuration of the spindles of the channel and the configuration of the machine parameters. The starting coordinates for	



**CNC 8070** 

Edisimu mode. New window for consulting the status of the subroutines, canned cycles, block repetition and loops.
Edisimu mode. The "START" softkey saves the program being edited.
Automatic mode. New functions and instructions that cancel the retrace function.
Automatic mode. New window for consulting the status of the subroutines, canned cycles, block repetition and loops.
Automatic mode. The [START] key saves the program being edited.
Diagnosis mode. Generate the Fagor file for error diagnosis.
Tool table. When selecting an incremental wear, it is possible to define the maximum increment possible; by default 0.5 mm (0.019685 inch).
Machine parameters tables. Import and export compensation tables.
Within a work mode, select the different pages in reverse order using the [SHIFT] key.
Setup assistance. Bode.
Interface related variables.

Ref. 0809 Software V04.00

Unicode.	
Mew language (Chinese).	Machine parameter: LANGUAGE.
When the CNC is installed as a simulator at a PC, the drive may be a simulated type or a Sercos type.	Machine parameter: DRIVETYPE.
In the machine parameter table, an icon indicates which parameters are involved in parameter matching.	
Handwheels. There can now be up to 12 handwheels.	Machine parameter: NMPG.
The CNC applies module compensation throughout the entire revolution of the axis.	Machine parameter: MODCOMP.
HSC. Eliminate the first resonance frequency of the machine when generating the velocity command.	Machine parameter: FREQRES.
Home search moving the axis to the reference point.	<ul> <li>Machine parameter: POSINREF.</li> </ul>
Delay estimate at the drive.	<ul> <li>Machine parameter: AXDELAY.</li> </ul>
Transfer inhibit for the independent axes.	<ul> <li>Machine parameter: XFITOIND.</li> </ul>
	<ul> <li>PLC mark: _XFERINH and _XFERINH(axis).</li> </ul>
Status of the axis position loop.	<ul> <li>PLC mark: LOPEN(axis).</li> </ul>
PLC. Detect overtemperature at the CNC.	<ul> <li>PLC mark: OVRTEMP.</li> </ul>
PLC. There can now be up to 1024 PLC messages.	PLC resources: MSG.
PLC. There can now be up to 1024 PLC errors.	PLC resources: ERR.
On a turret type tool magazine, the TMOPERATION mark can take the value of 15.	PLC mark: TMOPERATION.
Handwheels. Inhibit the handwheels of the system.	<ul> <li>PLC mark: INHIBITMPG1/INHIBITMPG12.</li> </ul>
Cancel spindle synchronization after executing M02, M30 or after an error or	<ul> <li>Machine parameter: SYNCCANCEL.</li> </ul>
a reset.	<ul> <li>Instructions #SYNC and #TSYNC.</li> </ul>
Positioning a turret magazine whether there is a tool in the indicated position or not.	#ROTATEMZ instructions.
A channel maintains its master spindle after executing M02, M30 or after an emergency or a reset or restarting the CNC.	<ul><li>Machine parameter: MASTERSPDL.</li><li>#MASTERinstruction.</li></ul>
Force the change of gears and/or of the parameter set of a Sercos drive	PLC mark: SERPLCAC.
	<ul> <li>Variable: (V.)A.SETGE.xn</li> </ul>
Set a machine coordinate.	PLC mark: REFPOIN(axis).
	G174 function.
There can now be up to 99 zero offsets.	G159 function.
There can now be up to 100 synchronization marks.	<ul> <li>Instructions #MEET, #WAIT and #SIGNAL.</li> </ul>
Select a turret position.	#ROTATEMZ instructions.
Axis synchronization. Managing a rotary axis as an infinite axis making it possible to increase the feedback count of the axis indefinitely (wihout limits) regardless of the value of the module.	Variables: (V.)[ch].A.PREVACCUDIST.xn
Variables. The variable (V.)[ch].E.PROGSELECT can be written via part-program, PLC and interface. This variable can only be written with the value of ·0·	Variables: (V.)[ch].E.PROGSELECT
Variables. The following variables are valid for the spindle.	Variables: (V.)[ch].A.MEAS.sn (V.)[ch].A.ATIPMEAS.sn (V.)[ch].A.MEASOF.sn (V.)[ch].A.MEASOK.sn (V.)[ch].A.MEASIN.sn



**CNC 8070** 

(REF: 0809)

• Programming in Polar coordinates.

Programming in incremental coordinates.

Best zoom, display part zero and auto-zoom from the keyboard.

Improved softkey menu.

Jog mode. New softkey to turn the CNC off.

Jog mode. In handwheel mode, next to each axis, the CNC indicates whether that axis has an individual handwheel associated with it or not.

Jog mode. The screen shows the tool dimensions.

Automatic mode. The screen shows the tool dimensions.

Handwheels. The general handwheels can move axes with an associated individual handwheel.

Handwheels. Number of pulses sent by the handwheel since the system was started up.

Variables: (V.)G.HANDP[hw]

# Feed handwheel.

Diagnosis mode. View the error and warning history issued by the CNC.

#### Edisimu mode and PLC mode.

- · New hotkey to redo an operation.
- The editor shows the line number.
- The option "Find/replace" permits selecting the search direction, up or down. New softkey to look for the text without replacing it.
- The editor adjusts the long blocks to the size of the window dividing the block into several lines.
- The editor offers hotkeys [CTRL]+[+] and [CTRL]+[-] to increase or decrease the size of the editor font. If the CNC has a mouse with a wheel, the [CTRL] key combined with this wheel can also be used to increase and decrease the size of the text font.
- In large files (more than 200 kB), the editor cancels the syntax coloring.
- In large files (more than 200 kB), the editor does not save the program when changing blocks; the editor saves the program when the user has not modified the program for about 5 seconds.

#### Edicimu mode

- Comments having an asterisk (\*) and programmed at the beginning of the block allow to group blocks. Blocks programmed between these comments will be grouped and may be expanded or shrunk the same way as the cycles or profiles.
- Having the "Hide cycles/profiles" option active, when the cursor moves over a hidden element, it expands automatically; when the cursor moves out of the element, it shrinks again.
- The editor offers the [ALT]+[-] hotkey to expand y hide cycles, profiles and
  grouped blocks. If the CNC has a mouse, click on the symbol located to
  the right of the cycle, profile or group of blocks to expand them and hide
  them.
- In large files (more than 200 kB), the editor does not hide the canned cycles or the profiles.

PLC mode. New softkeys to sort the files that make up the PLC project.



**CNC 8070** 

# 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 Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

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 Before manipulating the connectors (inputs/outputs, feedback, etc.)

connected to AC power.

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

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. In order to avoid electrical discharges, always work under 90% of

relative humidity (non-condensing) and 45 °C (113 °F).

Do not work in explosive environments. In order to avoid risks or damages, do no work in explosive

environments.



**CNC 8070** 

#### 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.

machine.

Avoid disturbances coming from the 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

Ambient conditions. The storage temperature must be between +5 °C and +45 °C (41 °F

and 113 °F).

The storage temperature must be between -25  $^{\circ}$ C and 70  $^{\circ}$ C (-13  $^{\circ}$ F

and 158 °F).

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.



**CNC 8070** 

# **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.



**CNC 8070** 

# **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.

FAGOR has exclusive competence in deciding whether the repair enters within the term defined as the warranty period.

#### **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.



**CNC 8070** 

#### **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.



**CNC 8070** 

# **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 8070

# **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 75° 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.



**CNC 8070** 

# **RELATED DOCUMENTATION**

Here is a list of the manuals available for your CNC, all of them included in the CD-Rom that comes with the product. Some of these manuals are also available in paper upon request.



Manual available in electronic format, included in the CD-Rom.



Manual available in paper.

# -OEM- manuals

Manuals directed to the machine builder or person in charge of the installation and start-up. The -OEM-manuals come in Spanish and English.

Manual	Description	
Hardware configuration (·M/T· model)	This manual describes the hardware configuration and the technical data of each element.	
Installation manual (·M/T· model)	This manual describes how to install and set the CNC up.	
Hardware history (·M/T· model)	This manual describes the hardware configuration and the technical data of elements that are no longer sold.	

## -USER- manuals

Manuals directed to the end user or CNC operator. The -USER- manuals come in several languages.

Manual	Description	
Operation manual (·M/T· model)	This manual describes how to operate the CNC.	
Programming manual (-M/T- model)	This manual describes how to program the CNC.	
Probing (·M· model)	This manual describes how to program probing movements and probing canned cycles. Milling model.	<b>©</b>
Probing (·T· model)	This manual describes how to program probing movements and probing canned cycles. Lathe model.	
Machining canned cycles (·M· model)	This manual describes how to program the machining canned cycles. Milling model.	
Machining canned cycles (·T· model)	This manual describes how to program the machining canned cycles. Lathe model.	
Quick reference (·M/T· model)	Summary guide of the PLC programming language.	<b>(</b>
Programming examples (·M· model)	Manual with programming examples for the mill model.	<b>©</b>
Programming examples (·T· model)	Manual with programming examples for the lathe model.	<b>(a)</b>



**CNC 8070** 

# -OEM / USER- manuals

Other manuals, directed to the machine manufacturer and to the end user.

Manual	Description	
New features	It is an optional manual that describes the new features and modifications that have been implemented in the CNC since the previous version and are not included in the manuals.	
Error solution	This manual offers a description of some error messages that may appear on the CNC indicating the probable causes that originate them and how to solve them.	
Execution channels	This manual describes how to configure and work in a multi-channel system.	
Monographic subjects	This manual describes how to configure and work with some CNC features.	<b>©</b>



**CNC 8070** 

# **GENERAL CONCEPTS**

1

This manual describes how to operate with the CNC using the keyboard, monitor and the operator panel. It also describes the various CNC operating modes as well as how to operate with each of them.

# 1.1 CNC configuration. Hardware

# Monitor, keyboard and operator panel

The CNC assembly comprises the monitor, keyboard and the operator panel.

- The monitor shows the CNC's status and, in general, the information about the whole system.
- The keyboard is used to communicate with the CNC; the operator may request information on commands or change the CNC status using new instructions.
- The operator panel is used to govern the various elements of the machine, including the movement of the axes and the control of the spindle, etc.

# **Modules**

Depending on the CNC model, it may be connected in different ways. For example, it would be possible to connect devices such as:

- · Floppy disk drive.
- CD-ROM unit.
- · Printer.
- PC-compatible keyboard.
- · Mouse with PS-2 connector.
- Devices with USB interface.
- Ethernet connection.
- Etc.

For further information, see the hardware manual.



**CNC 8070** 

# 1.

GENERAL CONCEPTS

CNC configuration. Hardware



**CNC 8070** 

(REF: 0809)

# **Network connection (Ethernet)**

The CNC may be connected to a computer network through specific connectors. The CNC must be configured as any node of the network as if it were a regular PC.

The following actions are possible when having a CNC configured as a node within the computer network:

- Access from any PC to the part-program directory of the CNC.
- Access from the CNC to any PC, to execute, simulate or edit programs. The program to be executed needs not be in the local disk.
- Copy programs and tables from the CNC to a PC and vice versa.
- Edit, modify, delete, rename, etc. the programs stored at the CNC.
- Perform a telediagnosis of the CNC.

The way the CNC is turned on and off depends on the disk installed (read-only disk or not) and on how it has been set by the machine manufacturer. However, the most common way to do it is as follows.

# Turning the CNC on

After powering up the unit, the operating system (Windows XP) will start up first. Then, and depending on how the manufacturer has set it, the CNC8070 application will either start up automatically or will have to be initiated by clicking on the icon shown on the screen.

While starting up, it will display the initial standard CNC screen or the initial screen created by the machine manufacturer for that purpose. Once the CNC is running, it will show the screen for the work mode (automatic or jog) selected by the machine manufacturer.

## Messages and options during the start-up.

When powering the CNC up, it informs on the contingencies that may be interesting. Depending on what the CNC checks on power-up, the CNC shows, if necessary, the option to "Restore backup copy and continue". This option shows the available backup copies and, once it is selected, the CNC will rename the current MTB folder and will restore the one in the selected backup copy. If the data bases are not valid, the CNC updates them.

When restoring a backup copy or the data bases while the CNC is in user mode, the CNC shows a message indicating that the change is temporary. These two situations occur because in both cases, the CNC modifies files that are write-protected in user mode. To make these change permanent, start the CNC up in setup mode.

Keeping the [END] key pressed while starting the CNC up cancels that process and the CNC shows the options to resume the start-up, cancel the start-up or restore a backup copy.

## Peculiarities of a write-protected (read-only) disk.

The CNC application will only start up when the unit is in one of the following work modes; the CNC application does not start up in administrator mode.

· Setup mode.

This mode must only be used to update the CNC software and to set up the machine. The access to this mode is protected with the password "machine parameters", defined in the utilities mode. On power-up, the CNC shows a warning indicating that the disk is unprotected.

• User mode.

It is the usual work mode for the user, once the setup is completed. The manufacturer must deliver this unit to the user set up to start up in this mode. The access to this mode is not protected with the password.

FAGOR

**CNC 8070** 

# FAGOR =

**CNC 8070** 

(REF: 0809)

## Turning the CNC off.

To turn the CNC off, press the key combination [ALT]+[F4]. The CNC must not be turned off if there is any program in execution.

After closing the CNC application and depending on how the manufacturer has set it, the unit will turn off automatically or it will be required to select *Shut down the system* option of the *Start* menu. Once the application is closed, the screen will show a message indicating to the operator that the unit may then be turned off.



The unit must be turned off with the on/off switch after having closed the application using the key combination mentioned earlier. Turning the unit off incorrectly may cause the loss of information about:

- · Active offsets (zero offsets, part offset, etc.).
- · Coordinates.
- Parts counter.
- · Active axis sets.
- Information about the next tool.

If on power-up, it displays the error " 12 - Checksum error in CNC data", it means that the CNC has been turned incorrectly (due to a power failure, etc.) and consequently that information has been lost: When this error message is displayed, home (reference) the axes again and activate the offsets (part zero included) and the sets of axes.

# Make a backup copy at a CNC with a disk that is not write-protected (not readonly).

If when turning the CNC off or resetting it, it detects a configuration change, the CNC will show the option to make a backup copy of the new configuration (MTB folder). The backup copies are saved in the "\Backup" folder in a compressed format.

## Make a backup copy at a CNC with a disk that is write-protected (read-only).

· Setup mode.

If the CNC is started up in setup mode, it offers the option to save a backup copy. Once setup is completed, it is recommeded to make a backup copy of the configuration; if there is none, the CNC will make one automatically.

Every time the CNC is turned off, it asks if the setup is completed. If setup is not completed. it will continue in setup mode the next time the system is restarted. If setup is completed, the CNC makes a backup copy if necessary and it will work in user mode the next time the system is restarted. The CNC makes a backup copy when there are changes in the machine parameters, in the PLC program or in the OEM subroutines.

· User mode.

If the CNC is started up in user mode, it does not offer the option to save a backup copy. The manufacturer must have saved the backup copy of the configuration during the setup. The backup copies are saved in the "\Backup" folder in a compressed format.

# 1.2.1 Emergency shutdown with battery

The central unit is powered by an external DC power supply (24 V DC). Optionally, an external battery may be connected to ensure the detection of power supply voltage drops and that the unit is turned off properly.

When a power supply failure occurs (drop at the 24 V DC) and there is a battery connected to the central unit, the latter responds as follows:

- If the supply is interrupted for less than 2 seconds.
  - The screens shows the corresponding warning and the system recovers fine. CAN errors may occur due to the lack of 24 V DC at the remote modules.
- If the supply is interrupted for more than 2 seconds.

After the 2 seconds, the screen shows the corresponding error and it initiates the automatic turn-off sequence. First, it stops the machine if it is in execution. It closes the CNC8070 application, then the whole system and finally turns the battery off.



GENERAL CONCEPTS

Turning the CNC on and off



**CNC 8070** 

# 1.3 Work modes and software protection at the CNC.

Fagor delivers the unit with a protected compact flash type disk that is write-protected except for the folders or files that must be unprotected for the normal operation of the CNC. The changes made to protected folders or files will be operative until the unit is turned off and back on, the CNC will then restore the initial configuration. The changes made to unprotected areas of the disk will remain.

The disk has been pre-configured with three access modes, each offers a different protection level. The unit shows the active work mode with an icon on the task bar of the operating system, next to the clock.

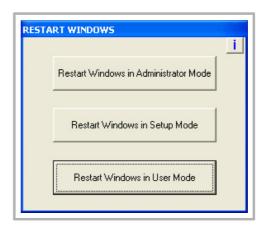
Icon.	Work mode.
<b>(</b>	Administrator mode.
<b>F</b>	Setup mode.
P	User mode.



These work modes will not be available when upgrading the software from a version where the unit has a disk that is not write-protected (not read-only). Since it is not write-protected, the changes made to it will remain.

# Changing the work mode and that of the protected folders.

Pressing the icon that indicates the active work mode, the unit shows the next window to switch from one work mode to another. This process implies restarting the unit and if the manufacturer has set it up this way, it will also be necessary to enter the corresponding password.



# Protecting or unprotecting the folders.

When selecting the -i- at the top right hand side, the unit shows the list of folders and files that are unprotected at the time. In administrator mode and the list of folders being visible, pressing [CTRL]+[ALT]+[TAB]+[SHIFT] it is possible to protect or unprotect any folder or file of the system that is not key for the proper operation of the CNC. Press the keystroke sequence.





**CNC 8070** 

GENERAL CONCEPTS

## Administrator mode.

The access to the administrator mode is enabled with the validation code. If you don't have this software option, you will not be able to access the administrator mode and, therefore, you will not be able to install third-party software.



This mode must only be used to install non-Fagor software or to change the system configuration. There is not protection level in administrator mode, the whole disk is unprotected. The CNC8070 does not start up in this mode.

The access to this mode is protected with the password "administrator mode", defined in the utilities mode.



The unit shows the following image on the desk, with red background, indicating the active work mode and warning that it is not a safe mode.

When starting the unit up in this work mode, it will request the access password.

# Setup mode.



This mode must only be used to update the CNC software and to set up the machine. The setup mode has an intermediate protection level where everything that may be changed while setting the machine up is unprotected.

The access to this mode is protected with the password "machine parameters", defined in the utilities mode.



The unit shows the following image on the desk, with yellow background, indicating the active work mode and warning that it is not a safe mode.

## User mode.



It is the usual work mode for the user, once the setup is completed. It has the maximum protection level where only the folders and files that may be changed during the normal operation of the machine are unprotected. Part-programs must be saved in the "..\USERS" folder; the CNC considers the files saved in other folders as temporary files and will be deleted when the CNC is turned off.

The access to this mode is not protected with the password.



**CNC 8070** 

# 1.4.1 Keys associated with the information on the screen

# Function keys.

Functions keys may be used to select the different options shown on the screen.

Keys.	Meaning.
F1 F2	Horizontal softkeys.
	It selects the options of the horizontal menu.
F8 F12	Vertical softkeys.
	It selects the options of the vertical icon menu.

# Browsing keys.

Keys.	Meaning.
	Screen change.  Depending on how the OEM has set this key, it will be possible to perform one of the following operations.
	<ul> <li>Access the various screens of the active work mode sequentially, pressing [SHIFT] at the same time inverts the sequence. In the PLC mode, it switches between the various active services.</li> </ul>
	<ul> <li>Sequentially access the different channels.</li> </ul>
	<ul> <li>The horizontal softkey menu shows the available screens and the vertical menu shows the available channels.</li> </ul>
	Window change.  It is used to switch between the different windows of the screen.
1	Previous menu.  On the horizontal softkey menu, it lets you go up from the softkey sub-menu to the previous level from where that menu was accessed.

# Help key.

	Keys.	Meaning.
ſ	HELP	Help.
		It accesses the CNC's system help.

**1.** 

**GENERAL CONCEPTS**Description of the keys



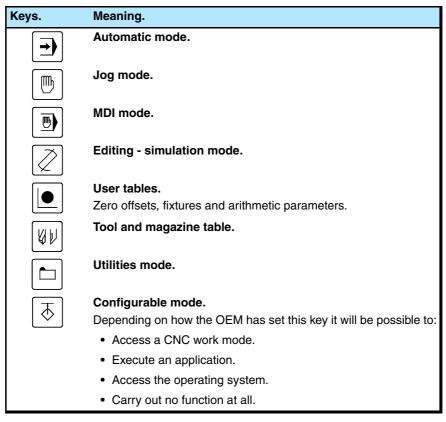
**CNC 8070** 

# 1.4.2 Keyboard layout

Depending on the utility of the different keys, the CNC keyboard may be considered to be laid out as follows.

## Work modes.

To access the different CNC work modes directly.



After accessing the work mode, these keys may be used to access the various screens of the active work mode sequentially, pressing [SHIFT] at the same time inverts the sequence.

# Numeric keypad.

It includes the necessary keys to enter numeric data and define the arithmetic operations as well as the following special keys:

Keys.	Meaning.
	Movement arrow keys.  They move the cursor one position to the left, right, up or down.
	Previous or next page.  They show the previous or next page when the program (part-program or PLC program) has more than one information page.
Home End	Home and end keys. It moves the cursor the beginning or end of the line.

1.

GENERAL CONCEPTS

Description of the keys



**CNC 8070** 

# FAGOR

**CNC 8070** 

(REF: 0809)

# Alphanumeric keyboard

It includes the necessary keys to edit programs as well as the following special keys:

Keys.	Meaning.
	Shift key for capital letters.
SHIFT	By keeping it pressed and in combination with a another key, it accesses a second set of characters for that other key shown at the top of that key. When combined with a letter, it writes it in capital letters.  When combined with arrow keys, it selects the text the cursor slides
	on.
ALT GR	Alternate characters.  By keeping it pressed in combination with another key, it accesses the third set of character of the that key, shown at the bottom right of that key.
	ASCII characters.
ALT	If a number is keyed in while keeping this key pressed, it will write the character corresponding to the ASCII code keyed in.
CAPS	Capital letters.  It toggles between uppercase and lowercase letters. It only affects alphabetical characters.
<b>(</b>	Delete character (back space). It deletes the character to left of the cursor.
SUP	<b>Deleted character.</b> It deletes the character to the right of the cursor or the selected text.
INS	Insert or overwrite.  It toggles between insert and overwrite modes. When inserting, the new text is added to the previous one, whereas when overwriting, the new text replaces the previous one.
	Tab.
-	It moves the cursor to the next field of the active menu.
ESC	Escape key It cancels the current operation without assuming the changes.
ENTER 🔶	Command validation. It is used to validate the data of the current operation and the program blocks edited in editing - simulation mode.
RECALL +	Data recall.  Having the TEACH-IN option active, it enters the axis position value into the block being edited.  When selecting a profile or conversational canned cycle in the part-program, it accesses either the profile editor or canned cycle editor accordingly.

# 1.4.3 Description of the operator panel

Depending on the utility of the different elements, the operator panel may be considered to be laid out as follows:

# JOG keys

They are used to jog the axes. There are two types of JOG panels:

There are two keys for each axis. One to jog the axis in the positive direction and another one to move it in the negative direction.

Keys.	Meaning.
X+ 7+	Jogging the axis in the positive direction.  The movement is carried out at the jog feedrate.
X- 7-	Jogging the axis in the negative direction.  The movement is carried out at the jog feedrate.
	Rapid jogging of the axis.  When combined with another JOG key, it moves the axis at rapid jog feedrate.

There is a key for each axis and two keys for moving direction, common to all the axes. Both keys (axis and direction) must be pressed to jog the axis.

Keys.	Meaning.
X 7	Axis selection.
+ -	Selection of the moving direction.
	Rapid jogging of the axis.  When combined with another JOG key, it moves the axis at rapid jog feedrate.

# **Feed selectors**

The operator panel has two selectors:

Keys.	Meaning.
©	Movement type selector.
100 10 1 100 JOG	Select the type of jog movement, which may be either by handwheels, incremental jog or continuous jog.
- M]	In handwheel mode, it selects the multiplying factor for the handwheel pulses (x1, x10 o x100).
	In incremental mode, it selects the incremental value of the axis movements.
	Feedrate override % selector.
70 80 90 100 110 60 0 130 40 130 40 150 150 160 180 170 180 170 180 180 180 170 180	To select the override percentage to be applied to the programmed and jog feedrates, between 0% and 200%.

1.

GENERAL CONCEPTS

Description of the keys



**CNC 8070** 

# Spindle control.

It is used to govern the spindle. It consists of the following keys.

Keys.	Meaning.
	Start the spindle clockwise.
	To stop the spindle.
	Start the spindle counterclockwise.
- +	To vary the spindle speed percentually.
Ö	Spindle orientation.

# Execution keys.

Keys.	Meaning.
	Cycle start key.
	It executes the program selected in the execution mode regardless of the active work mode (except for MDI mode). If the program has been interrupted, it resumes its execution.
	If MDI mode is active, it executes the block just edited.
	Cycle stop key.  It interrupts the execution of the program or the MDI block.
	RESET key.
RESET	It initializes the system setting the initial conditions as defined by machine parameters.
	Single-block execution mode.
	It selects the single block execution mode. When this mode is active, the execution of the program is interrupted at the end of each block.
ZERO	Home search.
	It is used to reference (home) one or several axes at the same time using a subroutine.

# External devices.

The functions of these keys are defined by the machine manufacturer and they allow controlling the various devices of the machine (coolant, chip remover, etc.).



**CNC 8070** 

The necessary files for the CNC are located in the directory C:\CNC8070 and its relevant subdirectories.

## BACKUP Folder for backup copies

This folder contains the backup copies of the MTB folder, in compressed files.

## **CONFIGURATION Directory of MTB folders.**

This folder contains the different MTB folders of the system; MTB\_T for lathe, MTB\_M for mill and MTB\_MC for motion control. On system startup and depending on the validation code, the system will move the corresponding CONFIGURATION folder to the CNC8070 and will rename it MTB.

## DIAGNOSIS Folder for diagnosis data

This folder contains relevant information for proper error diagnosis, including the reportfagor.zip file. The \Capturas folder keeps the point captures.

## FAGOR Version directory

This directory contains the software corresponding to the CNC version installed. Software updates are carried out in this directory and they do not affect the contents of directories MTB and USERS.



Do not change the contents of this directory. Only authorized personnel from Fagor Automation may modify the contents of this directory.

Fagor Automation shall not be held responsible of the performance of this CNC if the contents of this directory have been changed.

## MTB OEM directory

This directory is especially directed at machine manufacturers. This directory contains the modifications made by the machine manufacturer at the CNC like, for example, the PLC program, machine parameters, custom settings, new screens, integrating external applications, etc.

No MTB folder appears when installing the software for the first time. On system startup and depending on the validation code, the system will move the corresponding CONFIGURATION folder to the CNC8070 and will rename it MTB.

When changing the validation code, the system will move the MTB folder to the CONFIGURATION folder and will give it its old name; then, it will move the corresponding CONFIGURATION folder to CNC8070 and will rename it MTB.

If the OEM has created the MTB folder manually, for example by copying it from a backup, the system will not make any changes on power-up nor when modifying the validation code.

## TMP Temporary files

The directory contents are erased every time the CNC is turned on. This directory is used by the CNC to save the temporary files generated while operating.

## USERS User directory

It is especially directed at users. The purpose of this directory is to provide the user with a memory space for storing part-programs, profiles, etc. as they are generated.

## UNINST Uninstall directory

This directory contains the necessary files to uninstall the software of the 8070 CNC. To uninstall, double-click on the *fimain.exe* file and follow the instructions displayed on the screen.

1.

GENERAL CONCEPTS
Directory structure



**CNC 8070** 

# 1.5.1 MTB (Machine Tool Builder) directory

This directory is especially directed at machine manufacturers. This directory contains the modifications made by the machine manufacturer at the CNC like, for example, the PLC program, machine parameters, custom settings, new screens, integrating external applications, etc.

## DATA

This directory contains:

- The databases for machine parameters, tables, etc. and the safety backup (in ASCII) of those tables.
- (\*.dat) files related to the machining canned cycles (cycle editor).
- The copies made for storing data after turning the CNC off (coordinates, zero offsets, etc.).
- Each ..\LANG\lagnuage folder contains the file cncError.txt that contains the OEM
  messages and errors in different languages. If an error text is not in the folder of
  the language active at the CNC, it looks for it in ..\LANG\ENGLISH\cncError.txt;
  if it is not there, the CNC will issue the relevant error message.

#### **DRIVE**

This directory contains the information regarding the DDSSETUP.

## **KINEMATIC**

This directory contains the information regarding the OEM kinematics.

#### **MMC**

This directory contains the CNC custom setting made by the machine builder:

- The directory "...\MMC\CONFIG", the configuration files (ini) and the files that may be modified using the screen customizing tool (Fguim.exe).
- In the directory "...\MMC\IMAGES", the machine builder may include all the files of the application regarding bitmaps, videos, icons, etc.

#### **PLC**

This directory keeps the information regarding the PLC integrated by the machine builder:

- The directory "...\PLC\LANG" contains the PLC messages and error messages in the different languages
- The directory "...\PLC\PROJECT" contains the files that make up the PLC project and the object file.
- The directory "...\PLC\WATCH" contains the sets saved from monitoring and the logic analyzer.

## **RELEASE**

When the machine builder integrates his/her own application into the CNC, the components that have been created (\*.OCX files) will have to be located in this directory.

## **SUB**

When the machine builder integrates his/her own subroutines (tool change, home search, etc.), they will have to located in this directory.

The "\HELP\language" folder contains the help files associated with OEM subroutines and the pcall.txt file that contains the list of OEM subroutines. If these files are not in the folder of the language active at the CNC, the editor will not provide them as help.

# **TUNING**

This directory stores the configurations saved by the user in the setup assistance.



**CNC 8070** 

#### 1.5.2 **USERS** directory

This directory is especially directed at users. The purpose of this directory is to provide the user with a memory space for storing part-programs, profiles, etc. as they are generated. We recommend to save these programs in the folders created for this purpose in order to make it easier and faster to find them and make safety backup copies.

In a system with a write-protected disk, the programs created by the user must be saved in this folder; the only one that is not protected. Any program saved in a protected folder will be temporary and will be deleted when the CNC is turned off.

**HELP** The "\HELP\language" folder contains the help files associated with user-defined

global subroutines and the pcall.txt file that contains the list of user subroutines.

**POCKET** This directory saves the profiles that have been created using the profile editor and are related to the pockets of the conversational canned cycles.

**PRG** This directory saves the part-programs created by the user who may create new subdirectories and store the programs in a more orderly fashion.

> The folder "...\PRG\_8055\_TO\_8070" stores all the programs translated from the 8055 format into the 8070 language . The converted (translated) program is saved with the same name but with the extension m55 (milling program) or t55 (lathe

program).

**PROFILE** This directory saves the profiles that have been created using the profile editor and

are related to conversational canned cycles.

**REPORTS** This directory saves the BMP files generated when printing a graphic image to a file.

It also saves the reports (prn format) generated in the Diagnosis mode.

**CNC 8070** 

**FAGOF** 

GENERAL CONCEPTS
Directory structure



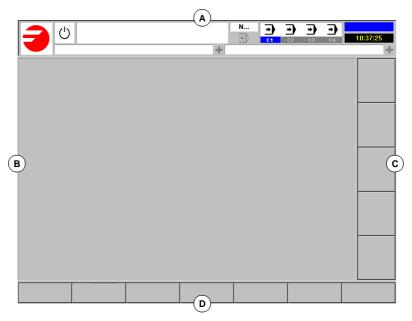
**CNC 8070** 

# HOW TO OPERATE THE CNC

2

# 2.1 Screen description

The CNC screens shows the following information:



## A. General CNC-status bar.

It shows the information on the program being executed, the active operating mode and the PLC messages. See "Detailed description of the CNC status bar" on page 18.

**B.** Screen for the active work mode.

The information shown in this area depends on the active work mode (automatic, jog, etc.). The information shown in each work mode is described in the relevant chapter.

C. Vertical softkey menu (icons).

The menu options change depending on the active work mode. The different options are selected using softkeys F8 through F12.

This menu may be placed either on the left or on the right of the screen.

D. Horizontal softkey menu.

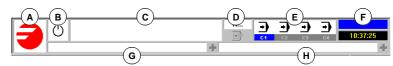
The menu options change depending on the active work mode. The different options are selected using softkeys F1 through F7.



**CNC 8070** 

# 2.1.1 Detailed description of the CNC status bar

The top of the screen shows the following information:



A. Icon (customizable) identifying the manufacturer.

Mouse-clicking on the icon gives access to the task window (same as pressing the keystroke sequence [CTRL]+[A]).

**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.



Programmed stopped.



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.

The background color will be different depending on the status of the program.

- **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. Click on the icons to access the desired channel.



Execution mode.



Jog mode.



MDI mode.

The channel synchronization window may be expanded using the [ALT]+[S] keys. 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.

An LED of various colors 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.

White No synchronization mark expected.

Green Synchronization mark expected. The mark is set to 1 in the channel

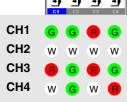
that originates it.

Red Synchronization mark expected. The mark is set to 0 in the channel

that originates it.



**CNC 8070** 



Channel 1 (CH1) is expecting synchronism marks from the rest of the channels. The marks of channels 2 and 4 are set to 1. The mark of channel 3 is set to zero.

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-).

**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.

G. Active CNC message.

For each channel, it 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.

H. 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.



**CNC 8070** 

Operating modes

# 2.2 Operating modes

# 2.2.1 How to access to the operating modes

The CNC operating modes may be accessed from the keyboard or from the task window that is displayed with the key combination [CTRL]+[A] or by clicking on the OEM icon (top left of the status bar).

Each operating mode may consist of several screens or pages. Use the operating mode accessing key to switch between the various screens (pages) from the change key (if the OEM has configured it that way) or by clicking at the top right of the status bar (name of the operating mode).

When pressing one of these keys, the CNC will show the next screen and when pressing them together with the [SHIFT] key, it will show the previous screen. The screen selection is rotary in such a way that when pressing this key on the last screen, it shows the first one.

# Accessing the operating modes from the keyboard

The following operating modes may be accessed from the keyboard.

**→** 

Automatic mode.



Jog mode.



MDI mode.



Editing - simulation mode.



User tables.



Tool and magazine table.



Utilities mode.

Also, the keyboard has a special key that may be configured by the machine manufacturer to access their specific operating modes.



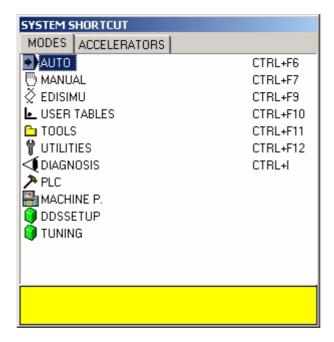
- · Machine parameters.
- PLC.
- Diagnosis mode.
- · Setup assistance.



**CNC 8070** 

# Task window

All CNC operating modes may be accessed from the task window. To open the task window, use the key combination [CTRL]+[A] or click on the OEM icon (top left of the status bar). Press [ESC] to close the window without making a selection.



## Moving around in the task window

Besides the mouse, the following keys may be used.

[♠] [♣] Moves the cursor through the window.

[CTRL]+[TAB] Changes the panel.

[ENTER] It accesses the selected mode.

2.

HOW TO OPERATE THE CNC
Operating modes



**CNC 8070** 

Operating modes

# 2.2.2 Description of the various operating modes

#### **Automatic mode**

It is used to execute part-programs in automatic or single-block mode.

While executing a part-program, any other operating mode may also be accessed (except "MANUAL" and "MDI") without interrupting the execution of the program. This way, a program may be edited while another one is running (background editing). The running program cannot be edited.

## Manual (jog) mode

It is used to control the machine movements manually using the keys of the operator panel, perform a home search, set the part zero, etc.

## **Editing - simulation mode**

It is used for editing and simulating part-programs even while running another part-program.

## **MDI** mode

It is used for editing and executing blocks in ISO code or in high-level language.

#### **User tables**

It is used for manipulating the CNC tables regarding part-programs (zero offsets, fixtures and arithmetic parameters).

## Tool and magazine table

It is used to edit CNC tables regarding the tools and the tool magazine.

#### **Utilities mode**

It is used to handle CNC part-programs (copy, delete, rename, etc.). A password may also be set to access the various operating modes.

Likewise, it offers the possibility to do a backup or a restore of the desired part of the disk.

#### PLC

It is used to operate with the PLC (edit a program, monitor, change the status of its variables, etc.).

# Machine parameter table

It is used for setting the machine parameters to adapt the CNC to the machine.

## Tuning mode. Setup assistance

For faster and easier machine setup. This mode offers the axis adjustment guide, the circle test, the oscilloscope and the Bode graphs.



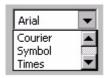
**CNC 8070** 

#### **Dialog boxes** 2.3

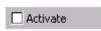
The dialog boxes consist of the following elements. All the actions may be carried out with the mouse or via keyboard.



 Selection panels. It selects among the different option groups within the same dialog box.

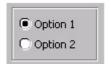


· Drop list. It selects an option from a list. Clicking on the right icon, the list expands.



· Selection box.

It activates an option.



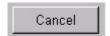
· Option selection.

It selects an option from a group. Selecting one cancels the previous one.



· Color pallet.

Select a color.



· Selection buttons.

They access a group of options or close the dialog box.

After making the changes, the dialog boxes are closed using one of the buttons that let you accept or reject the changes made.

# Using the keyboard

The data of the dialog boxes are defined via keyboard as follows.

- Using the [CTRL]+[TAB] combination, it is possible to select different panels of the dialog box.
- The [TAB] key moves the cursor through the elements of the box.
- The [SPACE] key activates the selection boxes, selects an options and selects a color from the color pallet.
- The [ENTER] key presses the button that is selected.
- The [ESC] key closes the dialog box without accepting the changes.
- On the drop lists, the desired option may be selected using the keys on the keyboard.
- The arrow keys [♠][♠][♠][♠] move the cursor over the color pallet and over the options of a list.

Dialog boxes

HOW TO OPERATE THE CNC



**CNC 8070** 

# 2.4 Windows for warnings and errors

When an error occurs, the CNC will display a window describing the cause of the error. There are three types of errors. The top of the window shows the category and it will have a different color depending on the type of error it shows.

WARNING Green window.
ERROR Red window.
FATAL ERROR Purple window.

When several warnings and/or errors occur simultaneously, they are displayed from the one with the highest priority to the one with the lowest priority. A down arrow indicates that there are errors or warnings with lower priority and an up arrow indicates that there are some with higher priority. The order of priority appears next to the arrows. The operator can toggle the different active errors or messages using the [ $\clubsuit$ ] keys. Pressing the [ESC] key closes the windows for errors and messages one by one.

The warning and error windows may be displayed or hidden with the keystroke combination [ALT]+[W] or clicking on the windows and on the status bar. When these windows are hidden, the status bar will show an indicator with the active error number. If several errors are active, it will highlight the "+" sign next to the number.

## Help for the warnings and errors.



Some warning and errors, besides their description, can show additional information on their causes and solutions. If the warning or error offers this option, a help icon will appear to the right of its number. This additional information may be displayed by pressing the [HELP] key or clicking on the help icon. To close the window for warnings and errors, press [ESC].

## WARNING

The system warning mesages are just warnings, they do not interrupt the execution of the part-program and may be eliminated by pressing the [ESC] key.

The warnings programmed with the instruction #WARNINGSTOP interrupt the execution of the program at the point where this instruction has been programmed. In this type of warnings, It's up to the user to either resume the execution at this point, [START] key, or abort the program, [RESET] key.

## **ERROR**

It may be a syntax error in a program, PLC generated errors, etc. They are errors that must be corrected.

They interrupt the execution of the program. Although the window displaying them may be closed by pressing [ESC], it does not mean that the error status has been taken care of. To do that, press [RESET]. After pressing [RESET] the CNC assumes the initial conditions set by the machine manufacturer with the machine parameters.

While the error window is active, no other action will be possible but removing it (it is not possible to change operating modes).

Refer to the help associated with the error or the error solving manual for a detailed description of the cause of the error and its possible solutions.

## **FATAL ERROR**

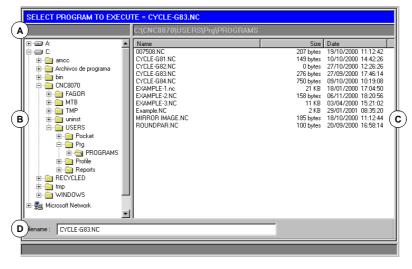
They are errors that force the operator to turn the CNC off. If the error persists, contact the Service Department at Fagor Automation.



**CNC 8070** 

The file selection window is common to all operating modes. This window is displayed when, from an operating mode, the operator selects the option to open, save or import a file, open or load a table, etc.

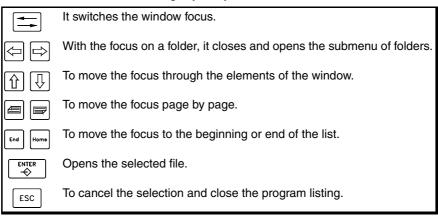
From this window, it is possible to either select an existing file or create a new one. A new file may be created only when it is a valid action. Depending on the operating mode it is accessed from, the list will only show the proper files.



- A. Type of file to be selected.
- B. Folders accessible from the CNC, as well as how they are structured. It shows all the elements accessible from the CNC, such as the hard disk, floppy disk drive, CD-ROM unit. etc.
- C. Files contained in the selected folder.
- **D.** Area to define the name of the new files. A file can also be selected when knowing its name.

# Move around in the window

Besides the mouse, the following keys may be used.



2.

HOW TO OPERATE THE CNC
File selection window



CNC 8070

# Sei

To make searching easier, the file list may be sorted according to different criteria. See "Sorting the list of files" on page 26.

A file may be selected from the list:

To select a file from the list

- 1. Select the folder that contains the program.
- 2. Select the file from the list or write its name in the bottom window.
- 3. Press [ENTER] to accept the selection.

Pressing [ESC] cancels the operation at any time and closes the window.

#### How to search an element from the list

An element (folder or file) may be selected from the list by moving the cursor to the desired element or using an alphanumeric keyboard. Pressing a key will select the first element from the list starting with that letter or number. Pressing it again will select the second one and so on.

The files can also be selected using the "Find file" softkey.

# Creating a new file

To create a new file:

- 1. Select the destination folder for the file.
- 2. Write the file name at the bottom window.
- 3. Press [ENTER] to confirm the action.

Pressing [ESC] cancels the operation at any time and closes the window.

# Sorting the list of files

The file list may be ordered alphabetically, by size or by date.

- When using a mouse, select the column header with the pointer. Every time the same header is selected, it toggles the sorting criteria from ascending to descending and vice versa.
- From the softkey menu. Every time the same softkey is selected, the CNC toggles the sorting criteria from ascending to descending and vice versa.

## File search

It is used to look for files whose names contain a particular character string. When selecting this option, the CNC shows a dialog box requesting the text to be found.

The programs are searched one by one. It may be searched using either the softkey menu or the following keys.



To search for the next program (in descending order).



To search for the previous program (in ascending order).



To end the search and close the dialog box.

When defining a search, the softkey menu shows the "Next" and "Previous" options.

- Search next, it looks for the next program (in descending order).
- Search previous, it looks for the previous program (in ascending order).

Depending on how the search is carried out, the focus goes to the last file found, whose name also appears at the top of the window. To end the search, press [ESC].



**CNC 8070** 

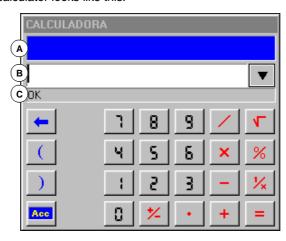
The calculator may be accessed from any task window or directly using the key combination [CTRL]+[K]. Press [ESC] to close the calculator.

The calculator may be accessed from any operating mode. When accessing from an editable element, the result displayed by the calculator may be inserted. An editable element is any element that may have a focus or cursor, such as the program editor, tables, editable data, etc.

To insert the result, press the [INS] key. It closes the calculator and inserts the result in the position that the cursor had when accessing the calculator.

## The calculator looks like this

The calculator looks like this.



**A.** Results window. It shows the result of the expression after accepting it with the [ENTER] key.

This value may be recovered with the "Acc" button so it can be used in later calculations.

**B.** Editing window. It shows the expression being defined. The expression may consist of one or several operations that may be defined directly from the keyboard or with the softkey menu options.

It saves the list of the last operations.

**C.** Explorer window. It shows, at all times, the result from evaluating the expression being defined. When selecting a portion of the expression at the edit window, it will show the result from evaluating that portion.

The result from evaluating the expression may be:

OK: ######## Numeric.
OK: 1.#INF Infinite.

Error: <> Syntax error.

Wrong expression Wrong expression.

## **Operations history**

The expressions already accepted become part of the history and may be displayed using the relevant keys  $[\, \bigstar \,] [\, \maltese \,]$ . After selecting an expression from the window, press the [ENTER] key to recover it. The [ESC] key closes the history window.

HOW TO OPERATE THE CNC
Calculator



**CNC 8070** 

# Softkey menu

The softkey menu shows all the available operations. The expressions may be edited using either the softkey menu or the mouse with the calculator keys.

# Work units

The result of the operations may be given in various units. Use the following icons to change the units. It highlights the units currently selected.



Hexadecimal (\$), decimal (D) or binary (B) units.



Units in degrees or radians.



**CNC 8070** 

# 2.6.1 Defining expressions

An expression may consist of one or more operations. Each one of them may be defined by any valid combination of variables, constants, functions and operations.

Press [ENTER] To accept the expression entered and calculate the value.

## **Quick parenthesis insertion**

To place a portion of an expression between parenthesis, select the portion and press one of the parenthesis keys "(" or ")".

If while a portion of the expression is selected, an operation key is pressed, the selection will appear between parenthesis and it will be preceded by the operation just defined.

Example: Having the expression "1+1/X+1/(X+1)+1". When selecting the "1/(X+1)+1" portion and pressing the "SIN" option, the expression will become "1+1/X+Sin(1/(X+1)+1)".

## Variables for the calculator

Up to 26 variables (from "A" to "Z", except " $\tilde{N}$ ",) may be used to store numeric values and use them later.

Assigning a value to a variable: A = 34.234

Reference to a variable: Sin(A/2)

A single expression may contain both assignment and reference operations. Use the ":" character as separator.

A=34.234:Sin(A/2) is the same as Sin((A=34.234)/2)



The values of the calculator variables "A" - "Z" are independent from the values of local parameters "A" - "Z" (also called P0 through P25).

## **Constants**

The following constants are available and they may be accessed from the softkey menu.

PI	Value of pi (3.14159)
MM -> INCHES	mm/inch conversion factor.
INCHES ->MM	inch/mm conversion factor.

## Various operations

In the following examples, the "x" and "y" values indicate any valid combination of constants, variables or expressions.

## Binary operations.

AND	Binary AND	1001 AND 1010 = 1000
OR	Binary OR	1001 OR 1010 = 1011
XOR	Exclusive OR	1001 XOR 1010 = 0011
NOT	Inverse	NOT 101 = 1···1010



**CNC 8070** 

## Arithmetic operations.

x + y	Add	
<i>x</i> - <i>y</i>	Subtract	
<i>x</i> * <i>y</i>	Multiply	
x/y	Divide	
<i>x</i> % <i>y</i>	Percentage	10%50 = 5
<i>x</i> ^ <i>y</i>	Power	2^3 = 2 * 2 * 2 = 8
<i>x</i> !	Factor	3! = 3 * 2 * 1 = 6

## Trigonometric operations.

SIN(x)	Sine	SIN 30 = 0.5
COS(x)	Cosine	COS 60 = 0.5
TAN(x)	Tangent	TAN 45 = 1
INV SIN(x)	Arc-sine	InvSin 0.5 = 30
INV COS(x)	Arc-cosine	InvCos 0.5 = 60
INV TAN(x)	Arc-tangent	InvTan 1 = 45

#### Conversion functions.

MM -> INCHES	It converts from millimeters to inches.
INCHES ->MM	It converts from inches to millimeters.
ABS(x)	Absolute value.
INT(x)	Integer.

## **Extended functions**

SQRT x	Square root	SQRT 16
LN x	Neperian log	LN 100
LOG x	Decimal log.	LOG 100
e^ <i>x</i>	"e" function	InvLn 3
10 ^ x	Decimal exponent	InvLog 50
INTEGRAL x	It calculates the integral	N=100:A=1:B=5:Integral(x+2)
ZERO x	Function zero	N=100:E=1e-10:A=5:Zero(x^2)

## **INTEGRAL** function

It returns as result, the integral defined by the function between the limits defined by the "A" and "B" variables.

The precision of the result depends on the "N" variable, that indicates the number of intervals used to divide the function in order to calculate the integral. If "N" is less than or equal to zero, it will be ignored and the number of intervals will be 100. If "N" is greater than 500000, the intervals will be 500000.

## **ZERO** function

It returns as result, the value that makes the function to be zero. Since there may be several values satisfying this condition, the result will depend on the initial value of the exploration defined with the "A" variable.

The precision of the result depends on the "N" variable that indicates the number of iterations of the resolution algorithm. If "N" is less than or equal to zero, it will be ignored and the number of iterations will be 100. If "N" is greater than 500000, the intervals will be 500000.



**CNC 8070** 

The "E" variable defines the error allowed so the result of the function with the calculated value is smaller than "E". This variable is useful when not knowing what value to assign to "N". In that case, it is recommended to assign to "N" a very large value and to "E" the error allowed.

2.

HOW TO OPERATE THE CNC
Calculator

FAGOR

CNC 8070

#### **Keyboard shortcuts** 2.7

# Operations at the interface

To perform different operations with the CNC.

[CTRL] + [W] Minimize/Maximize the CNC.

[CTRL] + [J] Show / hide the virtual operator panel.

[CTRL] + [M] Show / hide the PLC message list.

[CTRL] + [O] Show / hide the CNC message list.

Show / hide the window for errors and warnings. [ALT] + [W]

[ALT] + [F4] Turn the CNC off.

# Work modes

To select the CNC operating modes.

[CTRL] + [A] To show the task window.

[CTRL] + [F6] Automatic mode.

[CTRL] + [F7] Jog mode.

> [CTRL] + [F8] MDI mode.

[CTRL] + [F9] Editing - simulation mode.

[CTRL] + [F10] User tables.

[CTRL] + [F11] Tool and magazine table.

Utilities mode.

Calculator. [CTRL] + [K]

# **CNC** keys

[CTRL] + [F12]

Same as pressing the following CNC keys

[CTRL] + [F1] Previous menu. Ð

[CTRL] + [F2] Switch window.

[CTRL] + [F3] Switch screens. RECALL

Same as pressing the following keys of the operator panel.

[RECALL] key.

[CTRL]+[S] [START] key.

[CTRL] + [F5]

[CTRL]+[P] [STOP] key.

[CTRL]+[R] [RESET] key.

[CTRL] + [B] "Single block" execution key.  $\blacksquare$ 



**CNC 8070** 

Keyboard shortcuts available at the part-program and PLC program editor.

[CTRL]+[C] Copy the selected text.

[CTRL]+[X] Cut the selected text.

[CTRL]+[V] Paste the selected text.

[CTRL]+[Z] Undo the last change.

[CTRL]+[Y] Redo the selected text.

[CTRL]+[G] Save the program/Recover the original program.

[CTRL]+[+] Zoom in.

[CTRL]+[–] Zoom out.

[ALT]+[-] Hide or expand a cycle.

[CTRL]+[HOME] Move the cursor to the beginning of the program.

[CTRL]+[END] Move the program to the end of the program.

[CTRL]+[TAB] Toggle between the editor and the error window.

2.

HOW TO OPERATE THE CNC
Keyboard shortcuts



**CNC 8070** 



**CNC 8070** 

# **AUTOMATIC MODE**

3

# **Automatic mode selection**



The automatic mode may be selected from any other operating mode by pressing its relevant key.

This work mode may be accessed from any channel and even from several channels at the same time. A different program may be executed in each channel.

From the automatic mode, and even while a program is in execution, it is possible to access another work mode except the JOG mode and the MDI mode.

# Operations in automatic mode

The following operations are possible in this work mode:

- Select and execute a part-program.
  - The execution may be carried out in -automatic- mode or in -single block- mode.
- Set the execution conditions (first and last blocks) before executing the program.
- · Searching a block.

Recover the history of a program up to a particular block, with the option to change the active F, S, M and H functions to then resume program execution from that block.

- Display various data related to axes position.
  - (Command, position referred to part zero or to home, following error, etc.).
- Display a graphic representation of the program being executed.
- Perform a tool inspection while executing the program.
- Changing technological functions F, S, M, H.

While the program is interrupted, it is possible to executed M and H functions and modify the F and S functions.



**CNC 8070** 

# 3.1 Displaying the automatic mode

When this operating mode is active, it will be indicated at the top right side of the general status bar.

This mode may be displayed with several screens. The current screen and the total number of available screens are shown at the upper right hand side of the general status bar.

Switching between the different screens is done by pressing the automatic mode accessing key. The screen selection is rotary in such a way that when pressing this key on the last screen, it shows the first one of the operating mode.

# Standard configuration

The standard configuration of this work mode, described in this manual corresponds to the one supplied by Fagor. In some cases, the manual shows a sample screen, instead of that of the CNC in order to make it easier to understand.

These screens may be customized by the machine manufacturer changing its contents as well as its appearance. They may also remove certain screens or add some of their own.

# **Description of the screens**

As mentioned earlier, the automatic mode may consist of one or several of the following screens (later sections of this chapter describe each one in further detail).

#### Data screen.

They show execution related data: axis position, "M" and "G" function history, active tool and tool offset, spindle speed and axis feedrate.

When this screens shows a window with the program selected for execution, it will be possible to select the execution starting and ending conditions as well as to recover the program history up to a particular block.

#### Graphic screen.

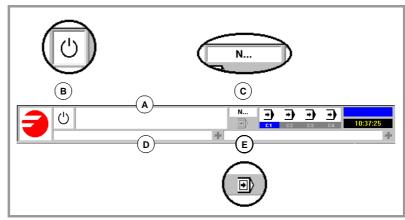
It shows a graphic representation of the program during execution. It also allows taking measurements on the graphics.



**CNC 8070** 

#### 3.1.1 Description of the general status bar

The general status bar shows the following information associated with the automatic mode:



A. Program selected in the active channel for execution.

The background color of the window will be different depending on the program

**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.



Program waiting to be executed.

Background color: White.



Program in execution.

Background color: Green.



Program interrupted.

Background color: Dark green.



Program in error.

Background color: Red.

- C. Number of the block in execution.
- D. Active CNC message.

For each channel, it 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].

The list of messages shows, next to each message, the channel where it is active.

E. Icon that indicates that the "Single-block" execution mode is active. This execution mode is selected from the operator panel.

**AUTOMATIC MODE** Displaying the automatic mode



**CNC 8070** 

3.

AUTOMATIC MODE Displaying the automatic mode

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	No synchronization mark expected.
Green	Synchronization mark expected. The mark is set to $\cdot 1 \cdot$ in the channel that originates it.
Red	Synchronization mark expected. The mark is set to $\cdot 0 \cdot$ in the channel that originates it.



(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-).

The previous graph shows for example:

- Channel 1 (CH1) is expecting synchronism marks from the rest of the channels. The marks of channels 2 and 4 are set to ·1·. The mark of channel 3 is set to ·0·.
- Channel 2 (CH2) is not expecting any synchronism mark.



**CNC 8070** 

# 3.1.3 Icon description (vertical softkeys)

The icon menu offers all the icons associated with this operating mode regardless of the active screen.

The icons are activated with their associated softkey (F8 to F12).

#### List of icons



# Program selection.

It selects a program for execution.



# Begin tool inspection.

(This icon is only available when program execution is interrupted).

To begin tool inspection in order to check its condition.

While tool inspection is active, the axes may be jogged and the spindle may be started or stopped.



#### **End tool inspection**

(This icon is only available when tool inspection is active).

It ends tool inspection.



# Changing technological functions.

(This icon will only be available when the program is interrupted).

This icon may be used to change the machining conditions executing any M, F, H, S function. The changes made remain when resuming the execution of the program.



# Selecting the program that is being edited.

It selects directly the program being edited.



**CNC 8070** 

# 3.2 Data screen

A typical screen of the automatic mode can show the following information:



- **A.** Program window. It shows data on the program selected for execution and lets selecting the initial and final execution blocks. It shows the following data.
  - Name of the program or global subroutine, called upon from the program, that is being executed.
  - Blocks of the program or subroutine being executed. During execution, the cursor shows the block being executed.

When accessing the automatic mode and after finishing the execution of the program, it shows the portion of the main program even if there are local subroutines defined in the program.

- Line of the program where the cursor is.
- **B.** Information related to the position of the axes. In the screen configuration supplied by Fagor, the data displayed will be different on each screen. Displaying the following information is the most typical.
  - The programmed coordinate, in other words, the target position.
  - The current position of the axes referred to part zero or machine reference zero (home) being this position referred to the tip of the tool or to its base.
  - The following error (axis lag).

If the name of the axis appears in red, it means that the PLC is inhibiting the movement of the axes (INHIBIT mark active).

- C. Active "M" functions.
- **D.** Active "G" functions and high-level commands.
- **E.** Tool related information. The information the CNC shows in this area depends on the model, lathe or mill.

At a mill model, the screen shows the number of the active tool "T", the active tool offset "D" for that tool and the number of the next tool "NxTool". The CNC also shows the radius and length of the active tool.

At a lathe model, the screen shows the number of the active tool "T", the active tool offset "D" for that tool. The screen also shows the tool offsets on each axis.

F. Information related to the spindle speed "S".

"Sreal" Actual spindle speed.
"Sprog" Programmed speed.

"S%" Percentage of speed override being applied on to the programmed value.

Depending on the active screen, it will also show the spindle position "SPOS" and its following error "SFWE".

The screen only shows the data on one spindle. If there are several spindles in the channel, the data on the next spindle may be displayed by pressing the "S" key.



UTOMATIC MODE

Data screen



**CNC 8070** 

If the text "Sreal" appears in red, it means that the PLC is inhibiting the movement of the spindle (INHIBIT mark active). The status of the mark is not displayed if the screen does not show this text.

**G.** Information related to the feedrate "F" of the axes:

"Freal" Actual feedrate of the axes.

"Fprog" Programmed feedrate.

"F%" Percentage of feedrate override being applied on to the

programmed value.

If the text "Freal" appears in red, it means that the PLC is inhibiting the movement of the axes (FEEDHOL mark active). The status of the mark is not displayed if the screen does not show this text.

- H. Execution (cycle) time "CyTime" of the program and number of parts made "Partc".
- I. CNC messages.

# Softkey menus

When selecting the program screen on the softkey menus, the options associated with that window will be displayed.

The options available in the softkey menu are the following. See "3.3 Data screen (softkeys)" on page 42.

"First block"

To set the first block of the execution.

• "Stop condition"

To select the block where the execution of the program or subroutine will be interrupted.

· "Block search"

To recover the program history up to a particular block and resume execution from that block on.

• "EXBLK"

It may be used to execute program blocks separately.

• "Active subroutines"

It may be used to display information related to the execution status of the subroutines, canned cycles, repetition blocks and loops.

"Find text"

It may be used to search for a text or placing the cursor on a particular line of the program.

3.

AUTOMATIC MODE

Data screen



**CNC 8070** 

# 3.3.1 First block

It sets the block currently selected with the cursor as the first block to begin execution. When not setting the starting block, the program will begin executing from the first block.

The last block may be selected using the cursor or the "Find text" or "Go to line" options of the softkey menu.

# Keys to move the cursor Moves the cursor line by line. To move the cursor page by page. End Home Moves the cursor to the first block or to the end of the program.

The first block being set remains active until it is canceled (using the [ESC] key) or until the program is executed; in that case the first block of the program will be assumed as the first block.





**CNC 8070** 

# To set the block where the execution of the program or subroutine will be interrupted. After executing that block, the execution may be resumed with the [START] key or canceled with the [RESET] key.

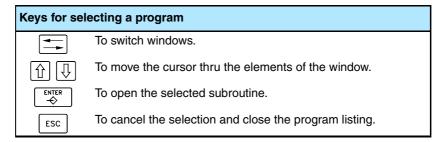
If the last block is not set, the program execution will end executing one of the end-of-program functions "M02" or "M30".

J

When selecting this option, the softkey menu of the CNC shows the options available for selecting the stop condition. To return to the main menu, press the key for the previous menu.

#### **Subroutine selection**

To select the stop condition in a global subroutine which has been called upon from the program. When selecting this option, the CNC shows a list of programs available for selecting the desired subroutine.



To select a subroutine from the list.

- 1. Select the folder containing the subroutine.
- 2. Select the program containing the subroutine or write its name in the bottom window.
- **3.** Press [ENTER] to open the program or [ESC] to cancel the selection and close the program listing.

The program window will show the contents of the selected subroutine.

## Last block

It sets as execution interrupting block the one currently selected with the cursor. If the last block is not set, the program execution will end executing one of the end-of-program functions "M02" or "M30".

The last block may be selected using the cursor or the "Find text" or "Go to line" options of the softkey menu.

Keys to move the cursor	
<u>(1)</u> [J]	Moves the cursor line by line.
	To move the cursor page by page.
End	Moves the cursor to the first block or to the end of the program.

The set last block remains active until canceled (with [ESC] key) or until the program is executed; in that case, the execution will end in one of functions "M02" or "M30".

# Number of times

It sets as stop condition, that the block selected as the last block has been executed a specific number of times.

When selecting this option, the CNC requests the number of times that the block must be executed before ending the execution of the program. After entering the number of times, press [ENTER] to validate the value or [ESC] to cancel it.

3.

AUTOMATIC MODE
Data screen (softkeys)



**CNC 8070** 

# 3.3.3 Block search

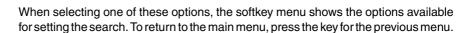
Ð

With this option, it is possible, in a program or subroutine, to set the last block in order to recover the program history and resume program execution from that point on with the same conditions as if it were executed from the beginning.

When recovering the program history, the CNC reads it up to the indicated block activating the "G" functions it reads along the way. Likewise, it sets the feedrate and spindle speed conditions of the program and calculates the position where the axes should be. The M functions are sent depending on how the machine is configured; they may be sent while reading the program or at the end of the program.

The CNC has two search modes, automatic and manual.

- The automatic search may be used to restore the program history up to the block where the execution was canceled or up to a block near it, set by the operator.
   The CNC remembers the block where interruption was canceled, thus not being necessary to set the stop block.
- The manual search may be used to restore the program history up to a particular block set by the operator.



#### Manual block search.

#### Subroutine selection.

To select the stop condition in a global subroutine which has been called upon from the program. When selecting this option, the CNC shows a list of programs available for selecting the desired subroutine.

To select a subroutine from the list.

- 1. Select the folder containing the subroutine.
- 2. Select the program containing the subroutine or write its name in the bottom window.
- **3.** Press [ENTER] to open the program or [ESC] to cancel the selection and close the program listing.

The program window will show the contents of the selected subroutine.

# Stop block.

It sets as the last block for program history recovery the one currently selected with the cursor. If the program is executed next, it resumes at this block.

The last block may be selected using the cursor or the "Find text" option of the softkey menu.

## Number of times.

It sets as stop condition, that the block selected as the stop block has been executed a specific number of times.

When selecting this option, the CNC requests the number of times that the block must be executed before ending the history recovery. After entering the number of times, press [ENTER] to validate the value or [ESC] to cancel it.

If a block containing a multiple machining cycle in ISO code (G160, G161, G162, G163, G164, G165) or of the cycle editor (L MULTIPLE.CYC) has been selected as a stop block, the block search will end just before the beginning of the nth modal cycle repeated in the multiple machining.

# Interruption point.

When pressing this softkey, the CNC finds the block where the execution was interrupted and it selects it as stop block. This block may be changed with the "Stop block" softkey.

3.

AUTOMATIC MODE

Data screen (softkeys)



**CNC 8070** 

It establishes the block currently selected with the cursor as the first block of the block search. If the first block is not defined, the block search starts at the first block of the program.

The first block may be selected using the cursor or the "Find text" option of the softkey.

The first block may be selected using the cursor or the "Find text" option of the softkey menu.

# Automatic block search.

#### Number of blocks.

With this option, it is possible to set the block search so it ends n blocks before the block where the execution was interrupted. If the number of blocks is  $\cdot 0 \cdot$  (zero), the block search in the block where the execution was interrupted.

## Simplified for CAM.

This option is oriented to CAM programs. With this option, not all the motion blocks are simulated; the CNC removes from the simulation the blocks between the last block that changes the motion conditions and the n-th block before the interruption block.

#### Interruption point.

When pressing this softkey, the CNC finds the block where block search will end. The last block will be the one where the execution was interrupted; the "block number " softkey may be used to set the block search to end n blocks before the one where the execution was interrupted.

#### First block.

It establishes the block currently selected with the cursor as the first block of the block search. If the first block is not defined, the block search starts at the first block of the program.

The first block may be selected using the cursor or the "Find text" option of the softkey menu.

3.

AUTOMATIC MODE

Data screen (softkeys)



**CNC 8070** 

This option may be used to toggle between the display of program blocks and the display of information related to the status of the subroutines, canned cycles, repetition blocks and loops.

Having this option active and the program execution interrupted, the user can use the cursor to select an information line and press [ENTER] to skip to the corresponding program block.

# Description of the information displayed.

The information is laid out in a table with four columns and having the following meaning.

Column.	Meaning.
S	Nesting level of the subroutine.
Р	Nesting level of the local parameters used in the subroutine.
Sub	Name of the program, subroutine or canned cycle.
Ор	Type of block being simulated. The loops are shown with a progress bar and a text indicating the loop it is in.

The window shows the following information.

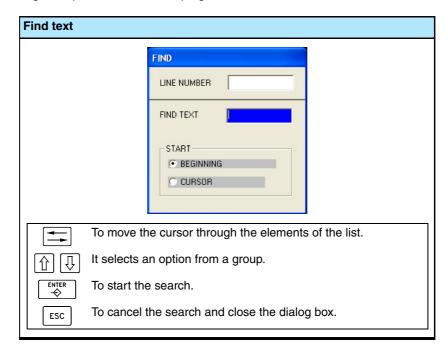
Type of block.	Information displayed.
	Name of the subroutine
L	Name of the subroutine
LL	Name of the subroutine
#CALL	Name of the subroutine
#PCALL	Name of the subroutine
#MCALL	Name of the subroutine
G180 through G189	Name of subroutine associated with the G function.
M function.	Name of the subroutine associated with the M function.
T function.	Name of the subroutine associated with the T function.
Canned cycle.	G function associated with the canned cycle.
#EXEC	Name of the subroutine
#EXBLK	
\$RPT	Number of current loop and total number of loops.
\$FOR	Number of current loop and total number of loops.
\$WHILE	Number of current loop.
\$DO	Number of current loop.
\$IF	Text "IF".
\$SWITCH	Text "SWITCH".
\$WHILE \$DO \$IF	Number of current loop.  Number of current loop.  Text "IF".



**CNC 8070** 

# **3.3.5** Find text

When selecting this option, the CNC shows a dialog box with two areas so the user can go to a particular line of the program or look for a text.



#### Go to line

This option may be used to position the cursor on a particular line of the program.

In this area of the dialog box, the CNC requests the line number to go to. Key in the desired number and press [ENTER], the cursor will then go to that line.

#### Find text

This option may be used to find text or a character string throughout the program.

In this area of the dialog box, the CNC requests the text to look for. It is also possible to select whether the search must start at the beginning of the program or at cursor position.

To start the search, press [ENTER] and the cursor will position on the text found. Pressing [ENTER] again, the CNC will look for the next match and so on.

To end the search, press [ESC]. The cursor will position on the block containing the text searched.



AUTOMATIC MODE

Data screen (softkeys)



**CNC 8070** 

**AUTOMATIC MODE** 

Program selection and execution

To select a program, press one of the following softkeys.



This softkey shows the list of available programs. See "2.5 File selection window" on page 25.



This softkey selects directly the program being edited in the same channel.

Once a program has been selected, its name appears on the general status bar. For each channel, it shows the name of the program selected in that channel.



**CNC 8070** 

**AUTOMATIC MODE** 

# 3.4.2 Program execution

The name of the program selected in the channel for execution appears on the general status bar. See "3.1.1 Description of the general status bar" on page 37.

If not indicated otherwise, the program execution will begin from the first block of the program to the execution of one of the end-of-program functions "M02" or "M30". As an option, it is possible to define the first and last blocks of the execution. See "3.3 Data screen (softkeys)" on page 42.

## Start executing

To start the execution of the program, press [START] on the Operator Panel.



When pressing [START], the CNC saves the program being edited, even if the programs being edited and executed are different.

The program may be executed in –single block– or –automatic– mode; the mode may be selected even while executing the program. When –single block– is active, the screen will display the relevant symbol on the general status bar.

If the –single block– mode is active, program execution will be interrupted at the end of each block; the [START] key must be pressed again to execute the next block. If the –automatic– mode is active, the program will be executed all the way to the end or up to the block selected as end of execution.

#### Interrupting the execution

The [STOP] key interrupts the execution of the program. Press [START] again to resume execution from where it was interrupted.

The execution may be interrupted at any time, except when threading. In that case, it will be interrupted at the end of the threading operation.

#### Stop the execution

The [RESET] key cancels the execution of the program and resets the CNC to its initial conditions.



The program selected for execution may be executed in any operating mode by pressing [START] on the Operator Panel. The CNC will request confirmation before starting to execute the program.



**CNC 8070** 

From the automatic mode, it is possible to execute blocks of a program separately; i.e. it is possible to select a block of the program and execute only that block. Blocks executed like this change the history of the M and G functions.

Press the "EXBLK" softkey of the horizontal menu to enable this function. Being this option active, every time the [START] key is pressed, it only executes the block selected in the active program. Once that block is executed, another block may be executed by selecting it with the cursor and pressing [START] again and so on. The blocks may be selected with the  $[\clubsuit]$   $[\clubsuit]$  keys.

AUTOMATIC MODE
Program selection and execution



**CNC 8070** 

**AUTOMATIC MODE** 

# 3.4.4 Program execution (retrace)

The retrace function stops the execution of the program and starts executing backwards the path traveled so far at the current block plus the last n blocks executed. The number of blocks to retrace has been preset by the OEM; a typical value being 75 blocks.

When the retrace function is canceled, the CNC resumes the normal execution of the program. During the retrace function, the program history is not updated; the CNC maintains the history of the point where the retrace function was activated.

#### Turn the retrace function on.

This feature is handled by the PLC. Usually, this feature is turned on and off using an external push button or key configured for that purpose.

The retrace function may be interrupted with the [STOP] key. Pressing [START] while the retrace function is interrupted resumes the execution in retrace mode. The retrace function can also be executed block by block (single block mode). The single block mode may be activated at any time, even when the retrace function is active.

The retrace function may be activated during an interpolation, in the middle of a block and also at the end of the block, whether the execution was interrupted by M0 or by the single block mode.

The retrace function cannot be activated while executing G33, G63, G100 or G04 type blocks. The CNC first finishes the execution of these blocks and then activates the execution in retrace mode. In the case of G33, G63 and G100, the retrace function is canceled; with function G04, the execution in retrace mode continues.

#### When is the retrace function canceled.

The CNC channel cancels the retrace function in the following cases.

- The user cancels the retrace function.
  - Since this feature is handled from the PLC, the user may cancel it from the operator panel if the manufacturer has put a button or key for that.
- The CNC channel has executed all the blocks stored for the retrace function.
- The CNC channel has reached the beginning of the program.
- The CNC channel has reached a block that cannot be executed with the retrace function.

The retrace function is also canceled after a reset or an M30.

# Resume the execution after the retrace function has been canceled.

How does the CNC resume the execution, once the retrace function has ended, depends on the setting of the PLC. If once the retrace function has ended, the CNC does not restore the normal execution of the program, it will show a warning indicating that the retrace function must be deactivated (e.g. from the outside operator panel if the manufacturer has provided a button or key to that effect). The same occurs when trying to execute a new program after executing the retrace function.

# Influence of certain functions while executing in retrace mode.

## Machining conditions.

Being retrace active, the movements are carried out backwards keeping the feedrate and the spindle used to execute them originally.



**CNC 8070** 

Activating, canceling and changing Constant Surface Speed (G96 G97) cancel the retrace function. However, if constant surface speed is on, it respects the change of spindle speed depending on the coordinate of the diameter axis.

#### Tool and offset.

Blocks with T and D programming (next tool and offset) are ignored during the retrace function. Blocks with D programming (change of active tool offset) cancel the retrace function.

#### ·M· functions.

The behavior of the retrace function when executing M functions depends on how the machine manufacturer has set it up. When the CNC finds an M function, it can either ignore it and keep executing blocks in retrace or cancel the "retrace" function. In any case, the following M functions behave as follows.

- Functions M00 and M01 are always executed; they are sent to the PLC and [CYCLE START] must be pressed to resume execution in retrace.
- Functions M03 and M04 are always ignored; the CNC does not start the spindle nor does it change its turning direction.
- While the spindle is running, function M05 cancels the "retrace" function; the CNC does not stop the spindle. If the spindle was stopped, this function is ignored.
- Functions M19, M41, M42, M43 y M44 cancel the retrace function.

# Functions that cancel the retrace function.

Definition.	Functions and instructions.	
Gear change.	M41 M42 M43 M44	
Spindle orientation.	M19	
Effective spindle stop. If the spindle was stopped, this function does not cancel the retrace function.	M5	
Change of work plane.	G17 G18 G19 G20	
Changing zero offset and fixture offsets, etc. Functions G53-G59, G159 only cancel the retrace function if they cancel an active zero offset.	G92 G158 G53-G59 G159	
Probing.	G100 G101 G102 #SELECT PROBE.	
Activate, cancel or modify constant surface speed.	G96 G97	
"C" axis. The instruction #CAX OFF only cancels the retrace function if it cancels an active #CAX.	#CAX ON #CAS OFF #FACE ON #FACE OFF #CYL ON #CYL OFF	
Tapping.	G33 G63.	
Changes in feed-forward or in AC-forward.	G134 G135	
Home search.	G74	
Changing the parameter set.	G112	
Changing the software limits.	G198 G199	
Synchronizing.	#SYNC	
High speed machining.	#HSC ON #HSC OFF	
Incline axis. Freezing (suspending) an incline axis does not cancel the retrace function.	#ANGAX ON #ANGAX OFF	
Slaving axes.	#LINK #UNLINK	
Axis parking.	#PARK #UNPARK	



**CNC 8070** 

Definition.	Functions and instructions.
Tangential control.	#TANGCTRL ON #TANGCTRL OFF #TANGCTRL SUSP #TANGCTRL RESUME
Axis swapping.	#SET AX #CALL AX #PUT AX #RENAME AX
Changing tool length due to tool change.	#TOOL AX #TLC OFF
Programming in the machine coordinate system.	#MCS ON #MCS OFF
Coordinate transformation. Instructions #ACS DEF, #ACS ACT, #CS DEF and #CS ACT do not cancel the retrace function.	#ACS ON #ACS OFF #ACS NEW #CS ON #CS OFF #CS NEW #RTCP OFF
Kinematics change.	#KIN ID
Independent axis.	#MOVE #MOVE ABS #MOVE ADS #MOVE INF
Electronic cam.	#CAM ON #CAM OFF #FOLLOW ON #FOLLOW OFF



**CNC 8070** 

# 3.4.5 Tool inspection.



This option is only available when the running program has been interrupted. Activating tool inspection makes the following operations possible:

- Jogging the axes using the JOG keys of the operator panel or handwheels.
- Stop and start the spindle using the keys on the operator panel.
- Access the CNC tables (tools, tool offsets, etc.) and modify their data.
- Changing the machining conditions executing any M, F, H, S function through the MHSF softkey.

Once tool inspection is completed and before resuming the execution of the program, the spindle must be started and the axes repositioned to the point where tool inspection began. Once the axes are repositioned, press [START] to resume program execution.

#### Rigid tapping and tool inspection mode

When interrupting the rigid tapping and accessing the tool inspection mode, it is possible to jog the axes (only in jog mode) that are involved in threading. When moving the axis, the interpolated spindle will also move; the spindle used to make the thread. If rigid tapping involves several axes, when moving one of them, all the other ones involved in the thread will also move.

This allows moving the axis into or out of the thread as often as desired until pressing the repositioning softkey. The axes move at the programmed F except when an axis or spindle exceeds its maximum feedrate allowed (parameter MAXMANFEED), in which case, the feedrate will be limited to that value.

The spindle jogging keys are disabled during tool inspection. It is only possible to get out of the thread by jogging one of the axes involved in rigid tapping. Functions M3, M4, M5 and M9 cannot be programmed at the spindle; they are ignored.

While repositioning, when selecting one of the axes of the thread on the softkey menu, it will move all the axes and spindle involved in the thread.

## Reposition the axes and the spindle.



To end tool inspection and reposition the spindle and the axes at the point where tool inspection began, press the relevant icon. The axes and the spindle may be repositioned simultaneously.

After pressing this icon, the CNC will show a list of the axes that are out of position. If the spindle was stopped during tool inspection, next to the list of axes, it will also show the status that the spindle had before the inspection.

# Reposition the axes.

The axes may be repositioned one by one or all at the same time. To reposition them, select the axes with the relevant softkey and press [START]. The axes reposition at the feedrate defined by the OEM.

The axes may be repositioned using the JOG keys or handwheels. On the other hand, repositioning may be interrupted (using the [STOP] key) to select other axes. Once one axis has reached its position, it will no longer be available.



**CNC 8070** 

The spindle turning direction may be restored together with the axes or separately. To do this, it shows, next to the axes to be repositioned, the previous status of the spindle (M3, M4 or M19). To restore the turning direction, select the softkey and press [START].

# Changing the machining conditions.

When entering into tool inspection mode and also after repositioning the axes, it is possible to change the machining conditions using the "MHSF" softkey. Pressing this softkey allows editing the feedrate and speed values, as well as activating M and H functions. Press [START] to assume the new values. The CNC keeps the new values when resuming the execution.

Use the [TAB] key to move through the various data. Press [ESC] or the "MHSF" softkey to return to the standard screen of the automatic mode.

AUTOMATIC MODE
Program selection and execution



**CNC 8070** 

# 3.4.6 Block search.

With block search, it is possible to recover the program history up to a particular block in such way that if program execution is resumed at that block, it will do so with the same conditions as if it were executed from the beginning.

When recovering the program history, the CNC reads it up to the indicated block activating the "G" functions it reads along the way. Likewise, it sets the feedrate and spindle speed conditions of the program and calculates the position where the axes should be. The M functions are sent depending on how the machine is configured; they may be sent while reading the program or at the end of the program.

#### Block search and canned cycles.

If a canned cycle is not selected as stop block, the block search only simulates the T, F, S changes and the movement to the last point. The surface milling, grooving and profile machining cycle, instead of simulating the movement to the last point, simulate a movement to the point defined by the safety Z and the reference corner or profile entry point.

If a block containing a multiple machining cycle has been defined as a stop block, it is possible to define the number of times that the machining operation is repeated. The block search will end just before the beginning of the n-th modal cycle repeated in the multiple machining operation.

# Types of block search.

The CNC has two search modes, automatic and manual. See "3.3.3 Block search" on page 44.

#### Automatic block search.

The automatic search may be used to restore the program history up to the block where the execution was canceled or up to a block near it, set by the operator. The CNC remembers the block where interruption was canceled, thus not being necessary to set the stop block.

#### Manual block search.

The manual search may be used to restore the program history up to a particular block set by the operator. The last block may be in a subroutine.

In this search, it is possible to set as an ending condition to repeat the stop block a particular number of times, for example multiple machining cycles, loops, etc.

# Executing the block search.

To recover the program history, proceed as follows:

- 1. Selecting the type of search: automatic or manual.
- 2. Selecting the stop block.

In the automatic block search, there is no need to select the stop block; by default, the CNC runs the search up to the block where the program was interrupted. However, the CNC allows setting as stop block a point near the interruption point. The "number of blocks" softkey determines how many blocks before the interruption point the search ends. Pressing the "interruption point" softkey, the cursor is placed on the block where the search will be interrupted.

The manual block search requires defining the point where the search will end. The stop block may be in a subroutine. When selecting as stop block a block with repetition, multiple machining, etc. it is possible to define the number of times that this block must be repeated in order to end the search.



**CNC 8070** 

- 3. Selecting the starting block for the search. If the first block is not selected, the block search starts at the beginning of the program.
- 4. Press the [CYCLE START] key to start the block search.

When recovering the program history, the CNC reads it up to the indicated block activating the "G" functions it reads along the way. Likewise, it sets the feedrate and spindle speed conditions of the program and calculates the position where the axes should be. The M functions are sent depending on how the machine is configured; they may be sent while reading the program or at the end of the program.

- 5. Depending on how the treatment of functions M, H, F, S is configured, it may be necessary to decide which ones are sent out to the PLC.
- 6. Reposition the axes to the point to resume execution.
- 7. The machining conditions may be modified using the "MHSF" softkey.
- 8. Press [START] to execute the program.

# Treatment of functions M, H, F, S.

Whether the functions M, H, F, S will be sent to the PLC or not during the block search depends on the setting of machine parameter FUNPLC. For the M function, it must also be borne in mind the how they have been set in the M function table.

The subroutine associated with the M functions is executed when the M is sent out to the PLC.

### Machine parameters. M function table.

The M function table has an MPLC field to define whether the function is to be sent out to the PLC or not. All M functions set in the table will be sent out to the PLC or not depending on the setting of this field; the rest of M functions will be sent or not depending on the setting of machine parameter FUNPLC.

#### Machine parameter FUNPLC = Yes. The functions are sent out to the PLC.

In this case, the functions are sent out to the PLC during block search as they are being read. Once block search ends and after repositioning the axes, the CNC shows the "MHSF" softkey for executing any function of this type, thus modifying the machining conditions.

# Machine parameter FUNPLC = No. The functions are not sent out to the PLC.

In this case, the functions are not sent out to the PLC during block search. After the search, the CNC screen shows the history of those functions so the user can enable them in the desired order.

• Mandatory "M" functions. List of M functions active up to the block reached and that must be executed in order to resume execution. These functions have a special meaning for the PLC.

This window will only show one of the functions are M03/M04/M05/M19 on one hand, M08/M09 on the other hand and M41/M42/M43/M44. The rest of the M functions such as M00, M01, M02, M6, M30 are not shown because they are not modal.

- Other M functions. List of "M" functions active up to the block reached. These functions do not have a special meaning for the PLC and need not be executed. These functions may be executed in any order, in groups or one by one, repeated,
- H functions. List of "H" functions active up to the block reached. These functions may be executed in any order, in groups or one by one, repeated, etc.
- "F" and "S" functions. The programmed "F" and "S" values may be modified. The changes stay valid until they are modified by the program being executed.

Use the [TAB] key to move through the various windows. The [♠] [♠] keys may be used to move the cursor through the M and H functions of a window, the [ENTER] key selects them or deselects them and the [START] key executes them Press the "MHSF" softkey to return to the standard screen of the automatic mode.

The CNC shows the M and H functions sent out to the PLC in green and the ones selected to be sent to PLC in red.



**AUTOMATIC MODE** 



**CNC 8070** 

# Reposition the axes.



To reposition the axes, press the corresponding icon and the CNC will show the axes that are out of position. The axes may be repositioned individually or several at the same time in one of the following ways:

- Manually. Jog the axes with the handwheels or with the JOG keys. The movement is limited by the repositioning end point and the corresponding software limit.
- Automatically. Select the axes with the relevant softkey and press [START].
   Repositioning may be interrupted (using the [STOP] key) to select other axes.

When an axis reaches the repositioning end point, it is no longer available; however, this axis may be moved with the handwheel or the JOG keys under the same conditions as before. Once all the axes have been repositioned, none of them may be moved.

# Changing the machining conditions.

After repositioning the axes and before resuming the execution, it is possible to change the machining conditions using the "MHSF" softkey. Pressing this softkey allows editing the feedrate and speed values, as well as activating M and H functions. Press [START] to assume the new values. The CNC keeps the new values when resuming the execution.

Use the [TAB] key to move through the various data. Press [ESC] or the "MHSF" softkey to return to the standard screen of the automatic mode.



**CNC 8070** 

# MANUAL (JOG) MODE

4

# Jog (manual) mode selection.



The manual (JOG) mode may be accessed from any other operating mode, except when a program is in execution, by pressing its relevant key.

In the same channel, the jog mode cannot be accessed while a program is running or a block is being executed in MDI. The jog mode may be accessed in the rest of the channels.

If the execution of the program has been interrupted, the CNC will cancel the execution of the program and will go into jog mode. A program is interrupted when the [STOP] has been pressed or is waiting in single-block mode.

When accessing the jog mode from the automatic mode, the CNC keeps the machining conditions selected in the last mode.

# Operations in this work mode.

The following operations are possible in this work mode:

- Display various data related to axes position.
   (Position referred to part zero or to home, following error, etc.).
- Home the axes (Machine reference zero search).
- Jog the axes using the keys on the operator panel or electronic handwheels.
- Move an axis to a position by previously selecting the target point.
- · Preset a coordinate.
- Start and stop the spindle manually using the keys of the operator panel.
- · Change tools.
- Activate up to 16 external devices using the keys of the upper side of the operator panel. The external devices associated with each key must be defined by the machine manufacturer.
- Calibrate a tool in jog mode (without probe), in semi-automatic mode (when using a table-top probe) or using the tool calibration cycle (also when using a table-top probe).
- · Part centering (mill model).
- Turn the CNC off from the softkey menu.



**CNC 8070** 

This mode may be displayed with several screens. The current screen and the total number of available screens are shown at the upper right hand side of the general status bar.

Switching between the different screens is done by pressing the jog mode accessing key. The screen selection is rotary in such a way that when pressing this key on the last screen, it shows the first one of the operating mode.

# Standard configuration.

The standard configuration of this work mode, described in this manual corresponds to the one supplied by Fagor. In some cases, the manual shows a sample screen, instead of that of the CNC in order to make it easier to understand.

These screens may be customized by the machine manufacturer changing its contents as well as its appearance. They may also remove certain screens or add some of their own.



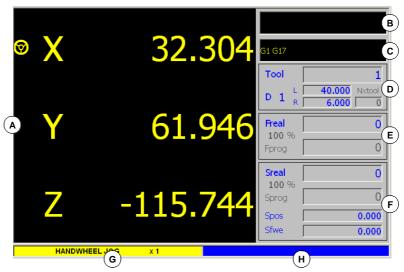
MANUAL (JOG) MODE Appearance of the Manual (JOG) mode



**CNC 8070** 

#### Description of a typical screen of this work mode 4.1.1

A typical screen of the jog mode can show the following information:



A. Information related to the position of the axes. In the screen configuration supplied by Fagor, the data displayed will be different on each screen.

The most common thing is to show the current position of the axes referred to part zero in large characters, and the amount of following error (axis lag) in small characters. Another more detailed option shows the coordinates of the tool tip and those of the tool base, being both referred to part zero and machine zero.



In handwheel mode, this symbol next to an axis indicates that the axis has an individual handwheel associated with it.

If the name of the axis appears in red, it means that the PLC is inhibiting the movement of the axes (INHIBIT mark active).

- B. Active "M" functions.
- C. Active "G" functions and high-level commands.
- D. Tool related information. The information the CNC shows in this area depends on the model, lathe or mill.

At a mill model, the screen shows the number of the active tool "T", the active tool offset "D" for that tool and the number of the next tool "NxTool". The CNC also shows the radius and length of the active tool.

At a lathe model, the screen shows the number of the active tool "T", the active tool offset "D" for that tool. The screen also shows the tool offsets on each axis.

E. Information related to the spindle speed "S".

"Sreal" Actual spindle speed.

"Sprog" Programmed speed.

"S%" Percentage of speed override being applied on to the programmed

Depending on the active screen, it will also show the spindle position "SPOS" and its following error "SFWE".

The screen only shows the data on one spindle. If there are several spindles in the channel, the data on the next spindle may be displayed by pressing the "S" key twice (the first time to program a turning speed).

If the text "Sreal" appears in red, it means that the PLC is inhibiting the movement of the spindle (INHIBIT mark active). The status of the mark is not displayed if the screen does not show this text.





**CNC 8070** 

F. Information related to the feedrate "F" of the axes:

"Freal" Actual feedrate of the axes.

"Fprog" Programmed feedrate.

"F%" Percentage of feedrate override being applied on to the

programmed value.

If the text "Freal" appears in red, it means that the PLC is inhibiting the movement of the axes (FEEDHOL mark active). The status of the mark is not displayed if the screen does not show this text.

**G.** Moving mode selected at the jog selector switch on the operator panel.

H. Reserved.



**CNC 8070** 

#### 4.1.2 Softkey menus.

# Horizontal softkey menu.



#### Turning the CNC off.

This softkey initiates the CNC shut-down sequence, it is the same as pressing the key combination [ALT]+[F4].

# Vertical softkey menu.



#### Display units (millimeters/inches).

It is used to change the units for displaying the position (coordinates) of the linear axes and the feedrate data. Toggling these units does not affect the rotary axes which will always be displayed in degrees.

The icon highlights the units currently selected (millimeters or inches).

It must be borne in mind that the units can only be changed on the data display. A program assumes the units defined with the active function "G70" or "G71", or, when not programmed, the units set by the machine manufacturer.



# Display units (millimeters/inches).

It is used to change the units for displaying the position (coordinates) of the linear axes and the feedrate data. Toggling these units does not affect the rotary axes which will always be displayed in degrees.

The icon highlights the units currently selected (millimeters or inches).

It must be borne in mind that the units can only be changed on the data display. A program assumes the units defined with the active function "G70" or "G71", or, when not programmed, the units set by the machine manufacturer.



#### Load zero offset or fixture offset tables.

It may be used to save the active zero offset in the zero offset or fixture offset table.



# Tool calibration (mill model).

It accesses the tool calibration screens. See chapter "5 Manual (jog) mode. Tool calibration".



## Tool calibration (lathe model).

It accesses the tool calibration screens. See chapter "5 Manual (jog) mode. Tool calibration".



# Part centering (mill model).

It accesses the part centering screen.



MANUAL (JOG) MODE

**FAGOF** 

**CNC 8070** 

MANUAL (JOG) MODE
Operations with the axes

# 4.2 Operations with the axes

#### 4.2.1 Home search

It is the operation used to synchronize the system. This operation must be carried out when the CNC loses the position of the origin point (e.g. by turning the machine off).

When "searching home", the axes move to the machine reference point and the CNC assumes the coordinate values assigned to that point by the machine manufacturer, referred to machine zero. When using distance-coded reference marks or absolute feedback, the axes will only move the distance necessary to verify their position.

The axes may be homed manually or automatically.

- Manual homing is done one axis at a time, from the operator panel.
- Automatic homing is done with a subroutine defined by the machine manufacturer.

# Manual home search

It is done one axis at a time. Home search of an axis is carried out as follows:

- XYZ
- **1.** Select the axis to be homed using the alphanumeric keyboard. The CNC will highlight that axis to indicate that it is selected.

To select the numbered axes (e.g. "X1"), select any axis and then move the selection until positioning on the desired one. The focus moves with the  $[\mbox{$\mbox{$$$}\mbox{$$}\mbox{$$}}]$  keys.



- 2. Press the homing key [ZERO]. The CNC will display the "1" symbol in the numeric area indicating that a home search will take place.
- ESC
- 3. Press [START] to go ahead with the home search or [ESC] to cancel the operation.



When searching home in this mode, the CNC cancels the zero offset, the fixture offset and the measuring offset. It assumes the machine reference zero point (home) as the new part zero.

# **Automatic home search**

This homing method is only available if the machine manufacturer has previously defined a homing subroutine.

Home search using a subroutine is carried out as follows:



1. Press the homing key [ZERO]. The CNC will display a dialog box requesting confirmation to execute the home search.



2. Press [START] to go ahead with the home search or [ESC] to cancel the operation.



**CNC 8070** 

# 4.2.2 Jogging the axes

The axes may be jog using the JOG keyboard on the operator panel. Two types of movements may be made.

- Continuous jog, when the axis moves while acting on the keyboard.
- Incremental jog, when the axis moves a specific distance every time the operator acts on the keyboard.

The type of jog is selected with the jog selector switch on the operator panel.



Continuous jog



Incremental jog



Handwheels

# JOG keypad.

There are two types of jog keyboards depending on the behavior of the keys.



Z+





The keypad has two keys for each axis. One to jog the axis in the positive direction and another one to move it in the negative direction.



To move a single axis, press the axis key and the one for its jogging direction.









The keypad has a key for each axis and two keys for moving direction, common to all the axes.



To jog an axis requires activating both the axis key and the moving direction. There are two options, depending on how the jog keyboard has been configured.

- The axis will move while both keys are pressed, the axis key and the direction key.
- When pressing the axis key, the key remains active. The axis will move while the direction key is kept pressed. To de-select the axis, press [ESC] or [STOP].

# User keys as jog keys

The CNC offers the OEM the possibility to enable the user keys as jog keys. The user keys defined this way behave like the jog keys.

#### Feedrates.



The movement is carried out at the feedrate defined by the OEM. The feedrate may be varied between 0% and 200% using the feedrate override switch on the operator panel.



MANUAL (JOG) MODE

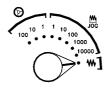
Operations with the axes



**CNC 8070** 

# **Continuous JOG**

In continuous jog, the axes keep moving while the jog keyboard is acted upon. Continuous jog allows moving several axes at the same time.



- 1. Turn the jog selector switch of the operator panel to the continuous jog position on the dial.
- 2. Jog the desired axis using the JOG panel (keypad). The axis will move while acting upon the JOG panel.

If while moving, a second axis is selected, the new one will move at the same time and under the same conditions.

In continuous jog, the axes keep moving while the jog keyboard is acted upon. Continuous jog allows moving several axes at the same time.

If while the axes are moving, the rapid key is pressed, the axes will move at the rapid rate set by the machine manufacturer. This feedrate will be applied while that key is kept pressed and, when released, the axes will recover their previous feedrate. This rapid rate may be varied between 0% and 200% with the feedrate override switch on the operator panel.

# Incremental JOG

In incremental jog, the axis moves a specific distance every time the key is pressed. In incremental jog, the axes may be jogged simultaneously.

#### Proceed as follows:

 Turn the jog selector switch of the operator panel to one of the incremental jog positions. Each position will move the axis a fixed distance; the typical values are the following.

Position.	Movement for each key push.
1	0.001 mm or 0.0001 inches
10	0.010 mm or 0.0010 inches.
100	0.100 mm or 0.0100 inches.
1000	1.000 mm or 0.1000 inches.
10000	10.000 mm or 1.0000 inches.

2. Jog the desired axis using the JOG panel (keypad). Every time the JOG panel is acted upon, the axis will move the distance indicated on the dial of the jog selector switch.

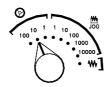
If while moving, a second axis is selected, the new one will move at the same time and under the same conditions.



**CNC 8070** 

#### 4.2.3 Jogging the axes with handwheels

Electronic handwheels may be used to move the axes. Depending on the type of handwheel, The CNC may have general handwheels to move any axis or individual handwheels that will only move their associated axes.



To move the axes with the handwheels, turn the jog selector switch of the operator panel to one of the handwheel positions. Every position indicates the multiplying factor applied to the handwheel pulses; the typical values are the following.

Position.	Movement per revolution of the handwheel.
1	0.100 mm or 0.0100 inches.
10	1.000 mm or 0.1000 inches.
100	10.000 mm or 1.0000 inches.

Once the desired resolution has been selected and depending on the type of handwheel being used, general or individual, proceed as follows.

#### General handwheel

The CNC may have several general handwheels. The general handwheel is not associated with any axis in particular, it may be used to move any axis of the machine even if it has an individual handwheel associated with it.

- If there are several axes selected in handwheel mode, the general handwheel will move all of them.
- If an axis has been selected which as an individual handwheel selected with it, this axis may be moved with the general handwheel, with the individual one or with both at the same time. When using both handwheels simultaneously, the CNC will add or subtract the pulses provided by both handwheels depending on which direction they are turned.
- If the CNC has several general handwheels, any of them can move the axes selected in handwheel mode. When using several handwheels simultaneously, each axis involved will be applied the sum of the increments of all the handwheels.

These are the steps to follow for moving one or several axes with the general handwheel.

- 1. Select the axis or axes to be jogged. The CNC will highlight the selected axes. When selecting an axis or quitting the handwheel mode using the movement selector, the previous one is automatically deselected.
- 2. Once the axis has been selected, the CNC will move it as the handwheel is turned depending on the setting of the selector switch and on the turning direction of the handwheel.

The feedrate depends on how fast the handwheel is turned.

# Selecting the axes to be jogged

There are two ways to select the axes.

- 1. On the JOG keyboard, press one of the keys for the axis to be jogged. Selecting an axis de-selects the previous one. To select several axes, press one of the keys of each key at the same time.
  - An axis needs not belong to the active channel in order to be selected. An axis from one channel may be set in handwheel mode from another channel, if the channel of the axis is also in jog mode.
- 2. When using a handwheel with a push-button, the push-button may be used to select, sequentially, the axes to be jogged.
  - Pushing the button selects the first one of the axes being displayed. If an axis has already been selected, it de-selects it and selects the next one. If it was the last one, it selects the first one again.



MANUAL (JOG) MODE



**CNC 8070** 

MANUAL (JOG) MODE
Operations with the axes

Only the axes being displayed in the active channel may be selected, regardless of the channel they belong to. The axes of another channel cannot be selected if they are not being displayed.

An axis is de-selected when quitting the handwheel mode using the movement selector and after a reset. If an axis has been set in handwheel mode from the PLC, it can only be deactivated from the PLC; a reset does not deactivate it.

#### Selecting an axis from the automatic mode

When having only one channel, if while in automatic mode, you set the switch in handwheel mode and select an axis, when going to jog mode, it maintains the selected axis.

#### Individual handwheel

The CNC can have several individual handwheels, where each of them is associated with a particular axis. The CNC moves each axis as its relevant handwheel is turned depending on the setting of the selector switch and on the turning direction of the handwheel.



In handwheel mode, this symbol next to an axis indicates that the axis has an individual handwheel associated with it.

When moving several axes simultaneously using handwheels, all the axes having their own handwheel plus the ones that may be selected with the general handwheel may be involved. When moving several axes at the same time, the feedrate of each axis depends on how fast its associated handwheel is turned.



It may happen that depending on the turning speed and the selector position, the CNC be demanded a faster feedrate than the maximum allowed. In that case, the CNC will move the axis the indicated distance but at the maximum feedrate allowed.

# Feed handwheel.

Usually, when machining a part for the first time, the feedrate is controlled by the switch on the operator panel. The "feed handwheel" allows using one of the handwheels of the machine to control that feedrate depending on how fast the handwheel is turned.



This feature must be managed by the PLC. Usually, this feature is turned on and off using an external push button or key configured for that purpose.



**CNC 8070** 

# 4.2.4 Moving an axis to a particular position (coordinate)

When moving the axes to a specific position, they must do so one by one following these steps:







**1.** Select the axis to be homed using the alphanumeric keyboard. The CNC will highlight that axis to indicate that it is selected.

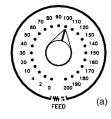
To select the numbered axes (e.g. "X1"), select any axis and then move the selection until positioning on the desired one. The focus moves with the  $[\, lacktriangle\,] \label{eq:constraint}$  keys.

- 2. Enter the coordinate of the target point.
- 3. Press [START] to execute the movement or [ESC] to cancel the operation.

# Feedrate behavior

The moving feedrate depends on whether G00 or G01 is active.

- If G00 is active, the movement is carried out at the rapid rate defined by the machine manufacturer.
- If G01 is active, the movement is carried out at the active feedrate. If no feedrate
  is active, the movement is executed at the feedrate defined by the machine
  manufacturer.



This feedrate may be varied between 0% and 200% using the feedrate override switch<sup>(a)</sup> on the operator panel. The percentage will be applied on to all the movements carried out in G00 and in G01.



MANUAL (JOG) MODE
Operations with the axes



**CNC 8070** 

# 4.2.5 Coordinate preset

Coordinates must be preset axis by axis following these steps:







- **1.** Select the axis to be homed using the alphanumeric keyboard. The CNC will highlight that axis to indicate that it is selected.
  - To select the numbered axes (e.g. "X1"), select any axis and then move the selection until positioning on the desired one. The focus moves with the  $[\mbox{$\mbox{$$$}\mbox{$$$}}][\mbox{$\mbox{$$$$}\mbox{$$$}}]$  keys.
- 2. Key in the desired preset value.
- **3.** Press [ENTER] to preset the entered value or [ESC] to cancel the operation.

The preset may be canceled by homing the axes one by one or by means of function "G53".



MANUAL (JOG) MODE
Operations with the axes



**CNC 8070** 

#### Displaying the data of several spindles.

The screen only shows the data on one spindle. If there are several spindles in the channel, the data on the next spindle may be displayed by pressing the "S" key. The first push is to program the turning speed, the second one shows the data on the second spindle and so on.

#### Spindle control

The spindle may be controlled manually using the following keys of the operator panel. The keys always refer to the master spindle of the active channel.

The spindle speed should be set (in the MDI mode) before selecting the turning direction, thus avoiding a sudden start of the spindle when setting an "S" because the turning direction was active.

#### Starts the spindle clockwise. Same as function M03.

Starts the spindle clockwise at the active speed. Displays function M03 in the history of the current machining conditions.

#### Start the spindle counteclockwise. Same as function M04.

Starts the spindle counterclockwise at the active speed. Displays function M04 in the history of the current machining conditions.

#### To stop the spindle. Same as function M05.

To stop the spindle. Displays function M05 in the history of the current machining conditions.

#### Spindle speed override.

It may be used to vary the spindle turning speed, a typical value being a variation between 50% and 120%. Every time one of these keys is pressed, the speed varies 5%

These values may be different depending on how the machine manufacturer has defined them.

#### Spindle orientation. Same as function M19.

It orients the spindle.

Displays function M19 in the history of the current machining conditions.

#### Customizing the keys

The machine manufacturer sets the minimum and maximum percentages between which the speed may be varied as well as the percentage of variation every time one of these keys is pressed.

They are set using the following machine parameters.

MINOVR Minimum override.

MAXOVR Maximum override.

STEPOVR Override increment (step).

4.

MANUAL (JOG) MODE
Spindle control



**CNC 8070** 

# 4.4 Tool selection and tool change

The tool located in the spindle may be changed in manual mode. Proceed as follows.

- **1.** Press [T] at the alphanumeric keyboard. The CNC will highlight the current tool indicating that it is selected.
- **2.** Key in the number of the tool to be placed in the spindle.
- ${\bf 3.}\,$  Press [START] to execute the tool change or [ESC] to cancel the operation.



MANUAL (JOG) MODE
Tool selection and tool change



**CNC 8070** 

# 4.5 Definition of cutting conditions

In jog mode, it is possible to set the turning speed "S" and the feedrate "F".

- The feedrate will correspond to the active feedrate.
- The speed entered will correspond to the master spindle of the active channel.

#### Proceed as follows.

- 1. Press the [S] key on the alphanumeric keypad to select a spindle speed or the [F] key to select a feedrate. The CNC will highlight the relevant data indicating that it is selected.
- **2.** Enter the desired spindle speed or feedrate.
- 3. Press [START] to assume the entered value or [ESC] to cancel the operation.



MANUAL (JOG) MODE
Definition of cutting conditions



CNC 8070

# 4.6 Automatic loading of zero offsets or fixture offset tables



It may be used to save the active zero offset in the zero offset or fixture offset table. This operation is carried out using the "Load table" icon.

Proceed as follows:

- 1. Press the icon for "Load table". The CNC shows the list of available zero offsets and fixture offsets.
- 2. Select the zero offset or fixture offset where you wish to save the active offset.
- 3. Press [ENTER] to enter the offset in the table.

The [ESC] key cancels the operation at any time.



**CNC 8070** 

# MANUAL (JOG) MODE. TOOL CALIBRATION

Tool calibration is available in the jog mode. The softkey to access tool calibration will be different depending on the software installed (lathe model or mill model). To quit the calibration mode and return to jog mode, press the [ESC] key.



Icon to access tool calibration in a mill model.



Icon to access tool calibration in a lathe model.

The CNC offers in both models the possibility to calibrate lathe tools and milling tools. The CNC will show the necessary data and will update the help graphics according to the selected tool.

#### Types of calibration

There are several ways to calibrate a tool. Some ways are only available when using a table-top probe.





Only manual calibration is possible when not using a table-top probe. All types of calibration are available when using a table-top probe. The different calibration methods may be selected from the vertical softkey menu.

The active kinematics are taken into account and do not prevent tool calibration in this mode. Manual or semi-automatic calibration will not be possible if a coordinate ( #CS or #ACS) transformation is active or when either the RTCP or TLC function is active.

#### Manual calibration.

It is done without the table-top probe. A reference part is required to calibrate the tool. All the movements are carried out manually.

#### Semi-automatic calibration.

Available when using a table-top probe. The positioning movements are carried out manually and the CNC executes the probing movements.

#### Automatic calibration.

Available when using a table-top probe. The CNC executes all the movements using the calibration canned cycle #PROBE.



**CNC 8070** 

#### **Probe selection**

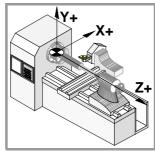
Two probes may be configured at the CNC. The probe active at the time is used for calibration. The active probe may be changed via part-program or MDI using the instruction #SELECT PROBE.

#SELECT PROBE [1]
Selects the first probe.

#SELECT PROBE [2]
Selects the second probe.

# Geometrical configuration of the axes: "plane" or "trihedron".

At the lathe model, the geometrical configuration of the axes may be either of the "plane" or "trihedron" type depending on the availability of a third main axis, usually the  $\cdot Y \cdot$  axis. The different calibration modes adapt to the current configuration showing the necessary data for each one of them.

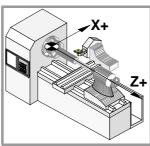


# Geometrical configuration of "trihedron" type axes.

It is the typical configuration of a milling machine or of a lathe that has a third main axis ( $\cdot Y \cdot$  axis).

There are three axes forming a Cartesian XYZ type trihedron like on a milling machine. There may be more axes besides those forming the trihedron.

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).



#### Geometrical configuration of "plane" type axes.

It is the typical configuration of a lathe.

There are two axes forming the usual work plane. There may be more axes, but they cannot be part of the trihedron; they must be auxiliary, rotary, etc.

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

The work plane is always G18; the plane cannot be changed via part-program.

#### 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) and Z (second) 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 #TOOL AX instruction or the G20 function.



**CNC 8070** 

#### Manual calibration. Calibration without a probe 5.1

In this mode, only the active tool can be calibrated and it may be a milling tool or a lathe tool. The CNC will show the necessary data and will update the help graphics according to the selected tool.



- A. Machine data. Position of the axes, tool and active tool offset, real spindle speed and real feedrate of the axes.
- B. Area to define the calibration data.
- C. Drawing indicating that tool calibration is allowed. If the drawing is not displayed, some of the data is missing.

#### **Tool calibration**

Since there is no probe, a reference part is required to calibrate the tool. The calibration consists in moving the tool manually until it touches the part and then validating the calibration on each axis. After validating them, the new values are saved in the tool table.

#### Selecting a tool

The tool and the active tool offset may be changed from the calibration mode. After defining the new tool or tool offset in the cycle data, press [CYCLE START] and the CNC will execute the tool change.

Bear in mind that if the defined tool is the active tool, when pressing [START] the CNC assumes the values that the offset has at the time.

#### Calibration in a mill model

On milling tools, the tool length is calibrated and the wear value is set to zero. The radius and radius wear may be set manually.

To calculate the length, it takes into account the coordinate of the longitudinal axis of the tool in the active plane (G17, G18, G19, G20), the tool orientation on the (#TOOL AX) axis and the coordinate of the reference part. For the calibration, it assumes that the coordinate of the reference part refers to the longitudinal axis of

For the lathe tools, it calibrates the tool offsets on each axis. The offset wears are set to zero.



MANUAL (JOG) MODE. TOOL CALIBRATION



**CNC 8070** 

#### Calibration in a lathe model

For the lathe tools, it calibrates the tool offsets on each axis. When validating the calibration in one of the offsets, the wear of that offset is reset to zero.

There are two options for the milling tools and may be selected with the following icons.



• Tool length calibration (only with "trihedron" type axis configuration).

This option may be used to update the length value and resets the wear value to zero. It also updates the tool table data.



· Tool offset calibration.

This option may be used to update the value of the offsets on each axis. The offset wears are set to zero.

#### Validating the calibration.

They are validated from the vertical softkey menu. Once the tool has been calibrated, when pressing [START] the CNC assumes the new values of the offset.



It validates the calibration of a milling tool.



It validates the calibration of a lathe tool. It shows an icon for each offset.

When on a lathe the axis have a "trihedron" type configuration, the calibration on the axis perpendicular to the work plane is done using the horizontal softkey menu.

#### **Definition of data**

To define the data, place the focus on the relevant data, key in the desired value and press [ENTER].

#### For a lathe tool.

The nomenclature of the axes depends on the geometrical configuration of the "plane" or "trihedron" axes. For a "plane" configuration, the names of the axes assume the DIN standard for lathes; the X axis as the abscissa axis and the Y axis as the ordinate axis.

Data	Meaning
<b>Z</b> p <b>X</b> p	Dimensions of the reference part being used in the calibration. These coordinates are referred to the main axes of the tool.
Т	Tool to be calibrated.
D	Tool offset to be calibrated.
Off Y Off X	Tool offsets on each axis.
Lw	Offset wear on each axis.

When a lathe has a third axis perpendicular to the work plane ("trihedron" geometrical configuration), the CNC will also show its data and calibration will be possible on that axis. The data of the third axis may be hidden or shown using the horizontal softkey menu.



**CNC 8070** 

#### For a milling tool.

Data	Meaning
Zp	Coordinate of the reference part being used in the calibration. This coordinate is referred to the longitudinal axis of the tool.
Т	Tool to be calibrated.
D	Tool offset to be calibrated.
L	Tool length.
Lw	Length wear.
R	Tool radius.
Rw	Radius wear.

#### **Tool calibration steps**

To calibrate the length, follow these steps.

- **1.** Define the dimensions of the reference part being used in the calibration.
- 2. Select the tool and the offset to be calibrated. After the selection, the CNC shows the dimensions defined in the tool table for that offset.

To calibrate a tool, it must be the active tool. When selecting a tool and pressing [ENTER], the CNC only shows the data for that tool. Press [CYCLE START] for the CNC to make the tool change so it becomes the active tool. See "Selecting a tool" on page 81.

- 3. Calibrate the tool. Approach the tool manually until touching the part and then validate the calibration using the softkey menu.
  - After validating the calibration, it updates the values and initializes the wear value to zero. Them, the new values are saved in the tool table.
- 4. Press [START] for the CNC to assume the new values of the offset.

To calibrate another tool, repeat steps 2 and 3.

#### Considerations for the offsets and their wear.

It must be borne in mind that the offset of a tool on an axis is the distance between the base of the tool and its tip (nose). This means that when calculating the offset of a milling tool on an axis that includes the radius dimension, that radius is included in the offset. The same is true for the tool length.

When calibrating the offsets of a milling tool, the length value is deleted but not the radius value.

#### Sign criteria for the offsets and their wear.

The sign criterion for the offsets and their wear is established by machine parameter TOOLOFSG.

TOOLOFSG	Meaning.
Negative.	Tool calibration returns a negative offset. The offset wear must be entered with a positive value.
Positive.	Tool calibration returns a positive offset. The offset wear must be entered with a negative value.

#### Entering wear values incrementally or absolutely.

In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute. See "Select the type of wear values to enter, incremental or absolute." on page 207.





**CNC 8070** 

5\_

Manual calibration. Calibration without a probe

MANUAL (JOG) MODE. TOOL CALIBRATION

FAGOR

CNC 8070

(REF: 0809)

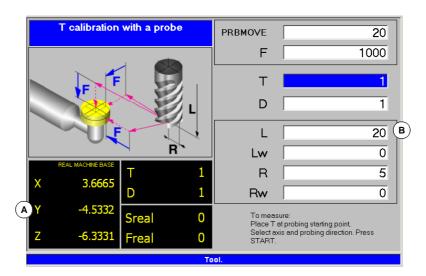
Using incremental wear, the value entered by the user will be added (or subtracted if it is negative) to the absolute value of the wear. After pressing [ENTER] to accept the new value, the wear field will show the resulting absolute value.

Initial wear	Incremental wear	Total wear
1	0.2	1.2
1	-0.2	0.8
-1	0.2	-0.8
-1	-0.2	-1.2

#### 5.2 Semi-automatic calibration. Calibration with a probe

This option is only available when using a tabletop probe installed on the machine.

On a milling model, it may be used to calibrate the length and radius of the milling tools and the offsets of the lathe tools. On a lathe model, it may be used to calibrate the offsets of any tool.



- A. Machine data. Position of the axes, tool and active tool offset, real spindle speed and real feedrate of the axes.
- B. Area to define the calibration data.

The tool must be in the spindle. After the calibration, the wear is reset to zero.

When changing the tool data, the tool table data is updated after calibration.

#### **Tool calibration**

The calibration consists in manually approaching the tool to the probe and then command the CNC to execute the probing movement. The CNC will move the tool on the selected axis until touching the probe. After touching the probe, it concludes the calibration on that axis and updates the values.

#### Selecting a tool

The tool and the active tool offset may be changed from the calibration mode. After defining the new tool or tool offset in the cycle data, press [CYCLE START] and the CNC will execute the tool change.

Bear in mind that in this calibration mode, the [CYCLE START] key has two functions. If a new tool has been selected, it executes the tool change. If the selected tool is the active one, pressing [CYCLE START] initiates the calibration.

#### Calibration in a mill model

On milling tools, it calibrates the radius and length of the tool. After calibrating one of the two dimensions, its wear value is reset to zero.

For the lathe tools, it calibrates the tool offsets on each axis. The offset wears are set to zero.

#### Calibration in a lathe model

For the lathe tools, it calibrates the tool offsets on each axis. When validating the calibration in one of the offsets, the wear of that offset is reset to zero.

MANUAL (JOG) MODE. TOOL CALIBRATION



**CNC 8070** 

# Validating the calibration

Use the horizontal softkey menu to select the axis and the moving direction for the calibration. Once selected and after placing the tool in the spindle, press [CYCLE START] to start the calibration. The tool will move in the indicated direction until touching the probe and it will then conclude the calibration updating the tool data with the measured values.

Once the tool has been calibrated, the CNC shows a message proposing to press [START] so the CNC assumes the new values of the offset. When pressing [START] while this message is displayed, the CNC assumes the new values of the offset; if the message is not displayed, pressing [START] executes the probing movement again.

Once a movement has been selected, the window will show a help drawing indicating the type of calibration to be done, length or radius.

#### **Definition of data**

To define the data, place the focus on the relevant data, key in the desired value and press [ENTER].

Data	Meaning
PRBMOVE	Maximum probing distance. If the CNC does not receive the probe signal before reaching moving this probing distance, it stops the axes.
F	Probing feedrate.
Т	Tool to be calibrated.
D	Tool offset to be calibrated.
L	Tool length.
Lw	Length wear.
R	Tool radius.
Rw	Radius wear.

#### Tool calibration steps

To calibrate the tool, follow these steps:

- 1. Define the probing distance and feedrate.
  - If the feedrate is not defined, the probing movement will be made at the feedrate set by the OEM.
- 2. Select the tool and the offset to be calibrated. After the selection, the CNC shows the dimensions defined in the tool table for that offset.
  - To calibrate a tool, it must be the active tool. When selecting a tool and pressing [ENTER], the CNC only shows the data for that tool. Press [CYCLE START] for the CNC to make the tool change so it becomes the active tool. See "Tool calibration" on page 81.
- **3.** Manually approach the tool to the probe until it is placed on the path that will be used for probing.
  - To calibrate the radius with a cylindrical probe, the path must coincide with the probe's center point; if not, the radius will be calculated wrong.
- **4.** Calibrate the tool. Select the axis and the probing direction on the softkey menu and press [START].
  - The probe moves in parallel to the axis and in the selected direction until touching the probe. It updates the measured value and resets the wear value to zero. The data is stored in the tool table.
- **5.** Press [START] again for the CNC to assume the new values of the offset. For the new values to be assumed, press [START] while the bottom message is displayed; otherwise, it executes the probing movement again.



**CNC 8070** 

#### Considerations for the offsets and their wear.

It must be borne in mind that the offset of a tool on an axis is the distance between the base of the tool and its tip (nose). This means that when calculating the offset of a milling tool on an axis that includes the radius dimension, that radius is included in the offset. The same is true for the tool length.

When calibrating the offsets of a milling tool, the length value is deleted but not the radius value.

#### Sign criteria for the offsets and their wear.

The sign criterion for the offsets and their wear is established by machine parameter TOOLOFSG.

TOOLOFSG	Meaning.
Negative.	Tool calibration returns a negative offset. The offset wear must be entered with a positive value.
Positive.	Tool calibration returns a positive offset. The offset wear must be entered with a negative value.

#### Entering wear values incrementally or absolutely.

In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute. See "Select the type of wear values to enter, incremental or absolute." on page 207.

Using incremental wear, the value entered by the user will be added (or subtracted if it is negative) to the absolute value of the wear. After pressing [ENTER] to accept the new value, the wear field will show the resulting absolute value.

Initial wear	Incremental wear	Total wear
1	0.2	1.2
1	-0.2	0.8
-1	0.2	-0.8
-1	-0.2	-1.2

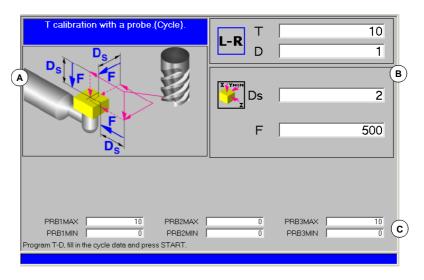


**CNC 8070** 

## 5.3 Automatic calibration with a probe and a canned cycle

#### 5.3.1 Mill or lathe model ("trihedron" geometrical configuration)

This option is only available when using a tabletop probe installed on the machine. This mode may be used to calibrate both milling and lather tools. The CNC will show the necessary data and will update the help graphics according to the selected tool.



- **A.** Machine data is optionally shown. Position of the axes, tool and active tool offset, real spindle speed and real feedrate of the axes.
- B. Area to define the calibration data.
- C. Probe position.

#### **Tool calibration**

The calibration is done using a probing canned cycle. The CNC moves the tool until touching the probe and validates the calibration on each axis. The tool may be calibrated on both axes of the plane or on the three axes of the trihedron.

The calibration begins when pressing the [CYCLE START] key. When the CNC finishes the calibration on the selected axes, it updates the tool table with the measured values. Also, the CNC assumes the new values.

XYZ FSTD The horizontal softkey menu offers the option to show the machine data such as position of the axes, feedrate, etc. This data is shown on the help graphics of the window.

#### Selecting a tool

In this calibration mode, the cycle itself changes the tool and the tool offset. There is no need to previously place the tool in the spindle.

Bear in mind that pressing the [CYCLE START] key starts the calibration cycle.

#### Select another position for the probe

The calibration uses the probe position defined in the machine parameters. Optionally, another position may be defined for the probe which will only be valid for the defined calibration. The new position does not affect the values defined in the machine parameters.



The data for defining the new probe position is accessed using the vertical softkey menu.



**CNC 8070** 

#### **Tool calibration**

There are two options for the milling tools and may be selected with the following icons.





· Calibrate the offsets and resets the wears to zero.

• Calibrate the length and radius and measure the wears.

For the lathe tools, it calibrates the tool offsets on each axis. The offset wears are set to zero.

#### **Definition of data**

To define the data, place the focus on the relevant data, key in the desired value and press [ENTER]. To change icons, place the focus on it and press [SPACE].

#### To calibrate the length, radius and wears of a milling tool.

The data shown depends on the calibration option selected with the horizontal softkey menu. This menu may be used to select the length and/or radius calibration and whether to calculate their wear or not. If the wears are not calculated, they are reset to zero after the calibration.

Data	Meaning
Т	Tool to be calibrated.
D	Tool offset to be calibrated.
Ds	Safety distance.
F	Probing feedrate.  If not defined, the movements are carried out at the default feedrate, set by the machine manufacturer.
N	Number of cutters of the tool. If defined with a $\cdot 0 \cdot$ value, the CNC knows the location of a cutter and it will only make the movement once. The spindle turning speed must be $\cdot 0 \cdot$ . If defined with a value other than $\cdot 0 \cdot$ , all cutters will be calibrated. The
	CNC makes an initial movement to locate a cutter; then, stops the spindle and makes a precise measurement of each cutter. It is necessary to define the spindle speed and the Dm distance.
Dm	Distance the edge of the tool separates from the center of the probe to position the next cutter.
S	Spindle speed.
Y_x	Probe side to be touched.
STOP	Behavior when exceeding the maximum wear permitted; reject the tool or change it with another one from the same family.
Lw	Maximum length wear allowed.
Rw	Maximum radius wear allowed.
PRB1MAX  PRB2MIN	Probe position.  The values defined here are only taken into account during the calibration cycle; they do not modify the machine parameter values.





**CNC 8070** 

#### To calibrate the offsets of a milling or lathe tool.

Data	Meaning
Т	Tool to be calibrated.
D	Tool offset to be calibrated.
Ds	Safety distance.
F	Probing feedrate.  If not defined, the movements are carried out at the default feedrate, set by the machine manufacturer.
PRB1MAX  PRB2MIN	Probe position.  The values defined here are only taken into account during the calibration cycle; they do not modify the machine parameter values.
X YMIN Z	This icon sets the number of axes to calibrate on.

#### **Tool calibration steps**

To calibrate the length, follow these steps.

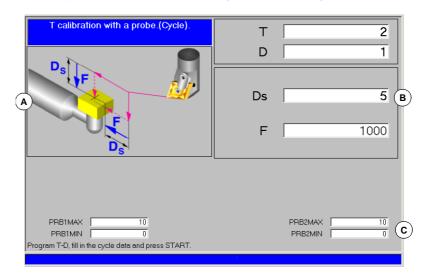
- 1. Select the tool and the offset to be calibrated. It is not necessary to insert the tool into the spindle; the CNC carries out this operation if necessary.
- **2.** Define the data defining the calibration. To calibrate a milling tool, use the horizontal softkey menu to select the desired operation.
- **3.** Press the [CYCLE START] key to start the calibration. The CNC calibrates the tool making all the necessary movements; there is no need to manually approach the tool. If necessary, the CNC makes the tool change.
- 4. After the calibration It updates the tool table data.



**CNC 8070** 

#### Lathe model ("plane" geometrical configuration) 5.3.2

This option is only available when using a tabletop probe installed on the machine. This mode may be used to calibrate both milling and lather tools. The CNC will show the necessary data and will update the help graphics according to the selected tool.



- A. Machine data is optionally shown. Position of the axes, tool and active tool offset, real spindle speed and real feedrate of the axes.
- B. Area to define the calibration data.
- C. Probe position.

#### **Tool calibration**

The calibration is done using a probing canned cycle. The CNC moves the tool until touching the probe and validates the calibration on each axis. The tool is calibrated on the two axes of the plane.

The calibration begins when pressing the [CYCLE START] key. When the CNC finishes the calibration on the selected axes, it updates the dimensions and the wears. Them, the new values are saved in the tool table.

XYZ FSTD

The horizontal softkey menu offers the option to show the machine data such as position of the axes, feedrate, etc. This data is shown on the help graphics of the window.

#### Selecting a tool

In this calibration mode, the cycle itself changes the tool and the tool offset. There is no need to previously place the tool in the spindle.

Bear in mind that pressing the [CYCLE START] key starts the calibration cycle.

#### Select another position for the probe

The calibration uses the probe position defined in the machine parameters. Optionally, another position may be defined for the probe which will only be valid for the defined calibration. The new position does not affect the values defined in the machine parameters.



The data for defining the new probe position is accessed using the vertical softkey menu.

#### **Tool calibration**

For milling and lathe tools, it calibrates the tool offsets on each axis. The offset wears are set to zero.





**CNC 8070** 

#### **Definition of data**

To define the data, place the focus on the relevant data, key in the desired value and press [ENTER]. To change icons, place the focus on it and press [SPACE].

Data	Meaning
T	Tool to be calibrated.
D	Tool offset to be calibrated.
Ds	Safety distance.
F	Probing feedrate.  If not defined, the movements are carried out at the default feedrate, set by the machine manufacturer.
PRB1MAX  PRB2MIN	Probe position.  The values defined here are only taken into account during the calibration cycle; they do not modify the machine parameter values.

#### **Tool calibration steps**

To calibrate the length, follow these steps.

- 1. Select the tool and the offset to be calibrated. It is not necessary to insert the tool into the spindle; the CNC carries out this operation if necessary.
- 2. Define the data defining the calibration.
- **3.** Press the [CYCLE START] key to start the calibration. The CNC calibrates the tool making all the necessary movements; there is no need to manually approach the tool. If necessary, the CNC makes the tool change.
- 4. After the calibration It updates the tool table data.



**CNC 8070** 

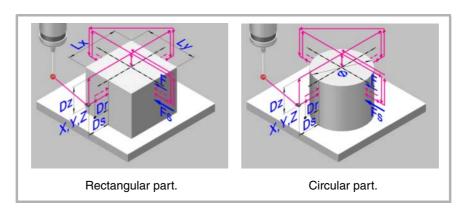
# MANUAL (JOG) MODE. PART CENTERING (MILL MODEL)

Part centering is available in the jog mode. This option is only available at the mill model. To quit the part centering mode and return to jog mode, press the [ESC] key.



Icon to access the part centering mode.

This mode may be used to calculate the center of a rectangular or circular part of known dimensions as well as, in rectangular parts, the inclination of the part with respect to the abscissa axis. The type of part to be centered is selected with the parameters of the cycle.



#### Requirements prior to the calibration.

Before executing the cycle, the probe must be placed near the part and in front of the probing point, as centered as possible and in the Z coordinate where probing will take place.

To execute this cycle, the probe must be properly calibrated.

#### Requirements prior to the calibration.

To center the part, follow these steps.

- 1. Select the probe and the offset. If not selected, the probe to be used must be defined in the parameters of the cycle.
- 2. Enter the data of the cycle.
- **3.** Press the [START] key to start the cycle. The CNC makes the necessary moves to center the part.
- **4.** When done with the cycle, the CNC updates the corresponding parameters.



**CNC 8070** 

#### Data returned by the cycle after the measurement.

Once the cycle has ended, the CNC returns the real values obtained in the measurement in the following arithmetic parameters:

P296 Inclination angle between the part and the abscissa axis (rectangular

part).

P297 Part surface coordinate.

P298 Part center coordinate along the abscissa axis.

P299 Part center coordinate along the ordinate axis.

Optionally, in this cycle, it is possible to preset the coordinates to select a new part zero and, in rectangular parts, rotate the coordinate axes to align the axes with the part.



**CNC 8070** 

#### 6.1 How to define the data.

To enter or modify a data, it must be selected; i.e. it must have the editing focus on it. The parameters of the cycles may be selected with the  $[\clubsuit] [\clubsuit] [\clubsuit] [\clubsuit]$  keys or with the direct access keys. The first data of each group may also be selected by pressing the page-up and page-down keys.

The direct access keys correspond to the name of the parameters; [F] for feedrates, [T] for tools, etc. Every time the same key is pressed, it selects the next data of the same type.

#### Manual data entry.

To modify a numerical data, key in the desired value or press [DEL] to leave the data undefined. In either case, press [ENTER] for the cycle to assume the new value.

Press the [SPACE] key to change the status of this icon.

#### Leaving some data undefined.

Some data may be left undefined (empty checkbox). In this case, the cycle behaves as follows.

- If the cycle position is not defined, it is executed at the current position the axes when calling the cycle.
- If the tool number is not defined, it will be executed with the tool that is active at the time of execution.

#### Defining data using arithmetic parameters..

Numerical data may be defined using global arithmetic parameters (P100-P9999) or common ones (P10000-P19999). In this case, when executing the cycle, these data will assume the value that the parameter has at the time.

When using global parameters, bear in mind that some cycles modify the value of these parameters at the end of the execution. Refer to each cycle to see which parameters it modifies.

#### Teach-in mode for data entry.

The Teach-in mode is always active and the bottom of the screen shows a window with the axes of the channel. The Teach-in mode may be used to jog the axes and assign their current position to the data that define the position of the cycle. The axes may be jogged using the jog keypad, the handwheels or via MDI.

To assign a value to a data, select it with the cursor (focus on it) and press the [RECALL] key. The data is taken from the channel where the part centering mode is active.

- The X axis related data takes the coordinate of the first axis of the channel.
- The Y axis related data takes the coordinate of the second axis of the channel.
- The Z axis related data takes the coordinate of the third axis of the channel.



MANUAL (JOG) MODE. PART CENTERING (MILL MODEL)



**CNC 8070** 

Data programming.

# 6.2 Data programming.

#### Probe data.

#### ·Tp· Number of the tool that identifies the probe.

Number of the tool used to define the probe in the tool table. Optional parameter, if not defined or set to 0, use the probe that is in the spindle when executing the cycle.

#### ·Dp· Number of the tool offset that identifies the probe.

Offset associated with the probe, used to execute the cycle.

#### Probing movement.

#### ·X Y Z· Probe position to execute the cycle.

Xm Probe position along the abscissa axisYm Probe position along the ordinate axis.

Zm Probe position along the axis perpendicular to the plane.

The part centering cycle is executed in the active plane.

#### ·icon· Geometry of the part to be centered.

This parameter indicates the type of part to be centered.



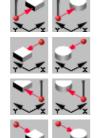
Rectangular part centering.



Circular part centering.

#### ·icon· Axis and direction of the first probing movement.

This parameter sets the axis on which the fist probing movement will take place.



The probe moves in the positive direction of the  $\boldsymbol{X}$  axis.

The probe moves in the negative direction of the X axis.

The probe moves in the positive direction of the Y axis.

The probe moves in the negative direction of the Y axis.

#### ·icon· Surface coordinate measuring.

This parameter indicates whether the cycle must also measure the position of the top surface of the part.



The cycle does not measure the surface coordinate.

The cycle measures the surface coordinate.



**CNC 8070** 

#### ·icon· Preset the part zero after ending the cycle.

This parameter indicates whether the part zero is to be preset or not, if so, the point taken as reference. This point may be preset with any value using parameters .Px Pv Pz.



Do not preset the part zero.



Preset the part zero at the center of the part.





Preset the part zero at one of the corners (the cycle shows an icon for each corner).

#### ·icon· Pattern rotation.

For rectangular parts, this parameter indicates whether a coordinate rotation is to be applied or not with the measured angle.



Do not rotate the coordinates (pattern).



Rotate the coordinates (pattern).

#### ·Lx Ly Ø· Part dimensions.

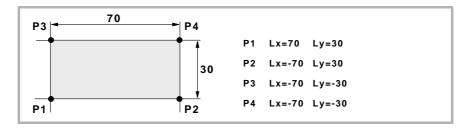
These parameters establish the dimensions of the rectangular or circular part.

Rectangular part. Part length along the abscissa axis. Lx

Ly Rectangular part. Part length along the ordinate axis.

Ø Circular part. Part diameter.

For a rectangular part, parameters ·Lx· and ·Ly· indicate the length of the pocket on each axis. The sign indicates tool orientation.



#### ·Ds· Safety distance.

Optional parameter; if not defined, it assumes the distance between the part and the probe position when calling the cycle.

Distance with respect to the point to measure, to which the probe approaches before making the probing movement.

#### ·Dr· Withdrawal distance after initial probing.

This parameter sets the distance the probe withdrawals after the first probing movement. Once it withdraws this distance, the CNC makes a second probing movement.

#### ·Dz· Safety distance along Z.

Distance for the probe to go up for its movements over the part.

#### ·Fs· Feedrate for the first probing movement.

Optional parameter; if not defined, the cycle assumes the value of machine parameter PROBEFEED of the axis.

This parameter sets the feedrate for the first probing movement. Then, the CNC will repeat the probing movement at feedrate ·F·.



Data programming.

**FAGOR** 

**CNC 8070** 

Data programming.

# ·F· Probing feedrate.

Optional parameter; if not defined, the cycle assumes 10% of the value of machine parameter PROBEFEED of the axis.

This parameter sets the feedrate for the second probing movement.

#### ·icon· Feedrate for approach movements.

This parameter sets the type of feedrate for the movements to the approach points.

The movements are carried out in rapid.

**W** 

The movements are carried out at work feedrate.

#### Coordinate preset value.

#### ⋅Px Py Pz⋅ Coordinate preset value on each axis.

Px Preset value on the abscissa axis. Py Preset value on the ordinate axis.

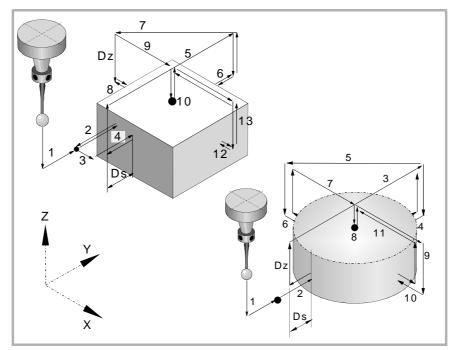
Pz Preset value on the axis perpendicular to the plane.

These parameters are only valid when presetting the part zero and any this point may be assigned any value.



**CNC 8070** 

## 6.3 Basic operation.



1. Approach movement (if any of parameters ·XYZ· has been defined).

Movement of the probe at the selected feedrate from the cycle calling point to the first approach point defined by parameters ·XYZ·.

This approach movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.

#### 2. Probing of the first point

The cycle makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.

First probing movement. Probe movement along the selected axis at feedrate ·Fsuntil the probe signal is received.

Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in  $\cdot Dr \cdot$ .

Second probe. Probing movement at feedrate  $\cdot F \cdot$  until the probe signal is received.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the starting point.

- If it is a rectangular part, approach movement for calculating the inclination.Movement parallel to the probed side to touch a different point of the same side.
- **4.** If it is a rectangular part, probing movement to calculate the inclination of the part. Probing movement at feedrate ·F· until the probe signal is received. The probe returns to the safety position located at a ·Ds· distance from the part.
- 5. Approach movement to the second probing point.

Movement of the probe from the first approach point to the second located in front of the first one.

The perpendicular axis moves the  $\cdot Dz \cdot$  distance up in rapid (G00). Then the probe moves over the part, at the selected feedrate, to the next approach point. The probe moves a  $\cdot Dz \cdot$  distance back down at feedrate  $\cdot Fs \cdot$ 

For the movement, the cycle takes into consideration the dimensions of the part and, if it is rectangular, its inclination angle. If in this last movement, the probe touches the part, the CNC shows an error at the cycle parameter value.

6. Probing of the second point.

It is done like the previous one. The probe returns to the safety position located at a ·Ds· distance from the part.



Basic operation.

FAGOR

**CNC 8070** 



Basic operation.



**CNC 8070** 

(REF: 0809)

- 7. Approach movement to the third probing point.
  - Movement of the probe from the second approach point to the third.
- 8. Probing of the third point.
  - It is done like the previous one.
- 9. Approach movement to the top surface (only if it has been programmed).
  - When selecting to measure the surface, the probe moves the ·Dz· distance up in rapid (G00) and the probe moves to the center of the part.
- 10. Measuring the top surface (only if it has been programmed).

The probe makes a first probing move, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.

First probing movement. Probing movement, at feedrate ·Fs·, until touching the part surface.

Withdrawal movement. Return of the probe in rapid (G00) the distance indicated

Second probe. Probing movement at feedrate ·F· until the probe signal is received.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the safety position.

**11.**Approach movement to the fourth probing point.

Movement of the probe from the third approach point to the fourth.

**12.**Probing of the fourth point.

It is done like the previous one.

13. Probe positioning at the calculated center.

This movement is carried out in two stages. It first moves along the perpendicular axis in rapid (G00) and then in the plane.

# **EDITING-SIMULATION MODE**

7

#### Selecting the editing and simulation mode



The editing-simulation mode may be accessed from any other operating mode, even while a program is in execution, by pressing the relevant key.

This work mode may be accessed from any channel and even from several channels at the same time. A different program may be edited and simulated in each channel.

#### What can you do in this work mode?

The following operations are possible in this work mode:

- Edit, modify or view the contents of a part-program.
  - A program may be edited in CNC programming language (ISO or high level) with the assistance of a conversational editor for machining cycles and positioning assisted by a profile editor or using the TEACH-IN feature.
- Simulate the execution of a program displaying a graphic representation of the program being simulated.
- Estimate the total program execution time and the machining time for each tool.

#### Program editor, Unicode format.

The editor of the CNC admits Unicode characters. When saving a program, the editor respects its original format, ANSI or Unicode, except when editing a Unicode character, in which case the CNC will always save the program in Unicode format.



Los programas en formato Unicode no son compatibles con versiones anteriores a la versión V4.0. Un programa en formato Unicode no podrá ser editado ni ejecutado en una versión anterior a la V4.0.

A third-party text editor (like Windows notepad) may be used to convert Unicode format programs into ANSI format; but special characters that have no ANSI equivalent will be lost in the process.



**CNC 8070** 

**FAGOR** 

**CNC 8070** 

(REF: 0809)

When this operating mode is active, it will be indicated at the top right side of the general status bar.

This mode may be displayed with several screens. The current screen and the total number of available screens are shown at the upper right hand side of the general status bar.

Switching between the different screens is done by pressing the editing-simulation mode accessing key. The screen selection is rotary in such a way that when pressing this key on the last screen, it shows the first one of the operating mode.

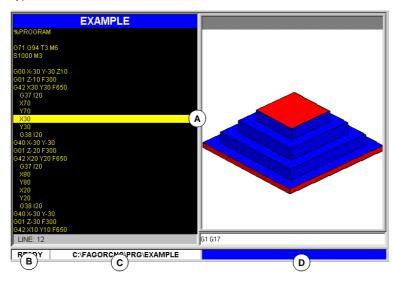
#### Standard configuration

The standard configuration of this work mode, described in this manual corresponds to the one supplied by Fagor. In some cases, the manual shows a sample screen, instead of that of the CNC in order to make it easier to understand.

These screens may be customized by the machine manufacturer changing its contents as well as its appearance. They may also remove certain screens or add some of their own.

#### 7.1.1 **General screen description**

On a typical screen for this work mode, the information is laid out as follows.



A. Windows of the editing-simulation mode.

It shows the operating windows of the selected screen. Every screen may consist of one or more windows. See "7.1.2 Window description" on page 100.

B. Status of the program selected in this work mode or channel number when using

In any case, the background color will be different depending on the status of the program being simulated.

Ready Background color: White. In simulation Background color: Green. Interrupted Background color: Dark green.

In error Background color: Red.

C. Program name and location.

The background color will be different depending on the status of the program.

D. CNC messages.





**CNC 8070** 

As mentioned earlier, each screen may consist of one or many of the following windows (later sections of this chapter describe each one of them in greater detail):

- Edit window: For editing new programs or modify the existing ones. Editing is
  possible using a profile editor, a conversational canned cycle editor or using the
  TEACH-IN feature.
- Graphic window: This window shows a graphic representation of the program during the simulation. It also allows taking measurements on the graphics.
- Program window: For selecting the starting and ending conditions of the simulation.
- Statistics window: For estimating the machining time for each tool and the total program execution time.



When the screen consists of several windows, the softkey menu will show the options of the active window. To switch windows and access the desired softkey menu, press the relevant key.

EDITING-SIMULATION MODE Appearance of the editing - simulation



**CNC 8070** 

#### 7.1.3 Icon description (vertical softkeys)

The icons are activated with their associated softkey (F8 to F12).



The icon menu always shows all the icons related to this operating mode regardless of which window is active. The group of options shown on this menu may be expanded with the softkey associated with the "+" icon that offers a different set of options.

#### List of icons



#### START (simulation)

(This icon will not be available while simulating a program).

It starts the program simulation or resumes it if it was interrupted.

During simulation, the graphic window will show the graphic representation of the program being simulated.



#### STOP (simulation)

(This icon will only be available when simulating a program).

It interrupts program simulation.

Simulation will resume by pressing the START icon.



#### **RESET** (simulation)

It cancels program simulation.

If an error occurs during simulation, it eliminates the error status and returns the simulation mode to its initial conditions.



#### Channel change for the editing and simulation mode

(This icon will only be available when using channels).

It serves to change the channel being displayed for editing and simulation. It does not affect the active channel at the CNC.



#### "Continuous" or "single block" mode

It selects the "single block" or "continuous" mode; either one may be selected even while executing a program.

When "single block" mode is active (the icon will appear pressed), program simulation will be interrupted at the end of each block. When the "automatic" mode is active, the simulation will take place until the end of the program or up to the block selected as end of simulation.



#### Check program syntax

It analyzes the program looking for syntax errors. The syntax check is not available for programs written in 8055 CNC language.

If there are no errors, the bottom of the screen will display a message indicating that the program is correct. If there are syntax errors, they will be shown at the bottom of the editing window.



CNC 8070

**FAGOR** 



#### **Time estimates**

(This icon is only available when the statistics window is displayed).

It offers an estimate of the total execution time at 100% of the programmed feedrate. For this time estimate, the CNC analyzes:

- The machining and positioning time for each tool used in the program.
- The number of "M" functions that are executed.
- The number of tool changes performed.

The result will appear at the statistics window.



#### Simulation setup

It shows the simulation setup options.

- Activate and cancel tool radius compensation.
- · Conditional stop.
- Activate and deactivate the software travel limits.
- · Block skip condition.
- Cancel the channel synchronization in the simulation.



**CNC 8070** 

# EDITING-SIMULATION MODE Program editing and simulation

## 7.2 Program editing and simulation

#### 7.2.1 Program editing

The process of editing or modifying a program is the following:

- 1. Open the program at the editor or create a new program.
- 2. Use the cursor to select the block where the new blocks will be inserted. Edit the program blocks using the editing modes available.
- Save the program so it can be modified or executed later on. This operation may be performed automatically as the program is being edited or if the editor has been personalized.

#### Available editing modes

Any of the various editing modes available may be used while editing a program. The available editing modes are:

#### **CNC language**

It is edited block by block and each one may be written in ISO language or in High level language.

When editing high level commands and depending on the type of command, the editor offers a list of available commands. See "7.3.1 Contextual programming assistance." on page 112.

#### 8055 CNC language

Los programas pieza se pueden editar tanto en el lenguaje propio del 8070 como en el lenguaje del 8055. La programación en lenguaje del CNC 8055 se habilita en el editor de programas pieza, desde la softkey "Personalizar" del menú horizontal. Within this option, activate the softkey of the 8055 editor. See "7.2.2 Editing a program in the 8055 CNC language" on page 104.

#### Conversational editor for canned cycles

To define machining and positioning canned cycles quickly and easily. This mode offers the following advantages:

- There is no need to know the canned cycle parameters.
- The CNC only lets entering the data being shown, thus avoiding any data entry errors when defining the cycles.
- · The programmer is assisted at all times with help messages.

When done editing the cycle, the CNC generates the necessary blocks and it will add them to the program inserting them after the block indicated by the cursor.

#### **Profile editor**

To edit new profiles quickly and easily. The editor shows a graphic representation of the profile being defined.

After defining the profile data, the CNC generates the necessary blocks and it will add them to the program inserting them after the block indicated by the cursor.

#### Incline planes

Help to the user for programming incline planes using the instructions #CS and #ACS. Using the "Insert" softkey, the block corresponding to the programmed instruction is inserted after the block where the cursor is.

#### **TEACH-IN**

It is basically the same as editing in CNC language, except when it comes to programming coordinates. This option displays the coordinates of each axis and lets enter them directly into the block indicated by the cursor.



**CNC 8070** 

#### 7.2.2 Editing a program in the 8055 CNC language

Los programas pieza se pueden editar tanto en el lenguaje propio del 8070 como en el lenguaje del 8055. La programación en lenguaje del CNC 8055 se habilita en el editor de programas pieza, desde la softkey "Personalizar" del menú horizontal. Dentro de esta opción, activar la softkey del editor 8055. Con esta opción desactivada, el CNC siempre trabajará con el lenguaje del 8070.

Aunque esta opción esté activa, también se puede editar programas en el lenguaje del 8070. El CNC asume uno u otro lenguaje en función del nombre asignado al programa pieza. Once a programming language has been selected, it does not admit instructions of the other language; i. e. Both languages cannot be cobined in the same part-program.

Con esta opción activa, en el CNC se pueden editar y ejecutar programas pieza escritos en el lenguaje del 8055. El CNC no permite ejecutar instrucciones del CNC 8055 desde el modo MDI.

Regardless of the selected programming language, the editor will be that of the 8070 CNC.

#### Editing a program in the 8055 CNC language

Having set the editor to accept the syntax of the 8055 CNC, the CNC assumes that this language will be used in the following cases. In the rest of the cases, the CNC interprets that the 8055 CNC language will not be used and assumes the syntax of the 8070 CNC.

- When the program name has the extension *pit* (lathe program) or *pim* (mill program). These are the native extensions of the 8055 CNC.
- Cuando el nombre del programa sea un valor numérico de no más de 6 dígitos (999999), que es como se denomina a los programas en el CNC 8055. En estos casos el CNC le añade automáticamente la extensión pim (en fresa) o pit (en torno).

Once the program has been opened, edit it using the 8055 CNC language; the CNC will not admit instructions of the 8070 CNC.

#### **Program editing limitations**

It is not possible to program calls to subroutines that are contained in other programs.

El análisis sintáctico no está disponible para programas escritos en el lenguaje del 8055. La softkey para realizar el análisis sintáctico del programa estará deshabilitada.

# Executing and simulating a program edited in the 8055 CNC language

Para ejecutar o simular un programa escrito en el lenguaje del CNC 8055 (aquellos con extensiones *pim* ó *pit*), el CNC lo traduce al lenguaje del CNC 8070. El CNC siempre simula y ejecuta el programa traducido, que además es el que se muestra en la pantalla durante la ejecución.

If when selecting the program, a file with extension *pit* or *pim* is selected, the CNC opens the converted file and displays it on the screen.

If any error occurs during execution, the CNC will display the block that caused it. The block will be in 8070 language, but it will be easily identified due to the translation format.



**CNC 8070** 

#### The conversion of the part-program

The CNC keeps both programs; the one written in 8055 format (the one edited) and its equivalent in 8070 format (the converted one). The converted (translated) program is saved in the following folder with the same name but with the extension m55 (milling program) or t55 (lathe program).

```
C:\Cnc8070\Users\Prg\PRG_8055_TO_8070
```

The program is only converted one; the first time that is simulated or the first time that the program is selected in automatic mode.

Si se modifica el programa editado en lenguaje del CNC 8055, el CNC vuelve a traducirlo al formato del CNC 8070. Si se modifica el programa traducido al lenguaje del CNC 8070, no se actualiza el programa editado en lenguaje del CNC 8055.

If the syntax of the converted block is considerably different from the original, the CNC adds the original block as comment so as to identify it easily.

8055 language.	8070 language.	
G XYZ	G0 X Y0 Z0 (G XYZ)	
T1.2 M6	T1 D1 M06 (T1.2 M6)	
G01 G05 G90 F1000	G01 G05 G90 F1000	
X10 Y23 Z33	X10 Y23 Z33	
G75 X100	G100 X100 (G75 X100)	
M30	М30	

When the cannot translate a block because it has a function without equivalent at the 8070, it will show the message "function without translation". The syntax analyzer will show this message when the index of a machine parameter is indicated parametrically.



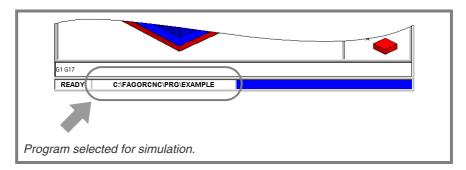
**EDITING-SIMULATION MODE** 



**CNC 8070** 

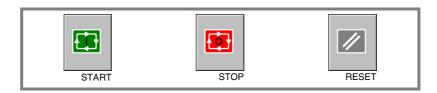
#### 7.2.3 Program simulation

The graphic window shows the program selected at the editing window and its name appears at the bottom center of the screen.



The program simulation procedure is the following:

- 1. Choose the type of graphic representation, its dimensions and the point of view. This data may also be modified during the simulation of the program.
- 2. Activate the desired simulation options using the icon menu.
- **3.** Press the START icon to start the simulation. Every time the START icon is pressed, the editor saves the program being edited in the channel, even when the "save always" option is not active.
- **4.** The simulation may be interrupted with the STOP icon or canceled with the RESET icon.



The simulation of the program starts at the first block of the program and ends after executing one of the end-of-program functions "M02" or "M30". As an option, it is possible to define the first and last blocks of the simulation. See "7.6 Program window" on page 128.

To simulate the program, the CNC assumes the real configuration of the spindles of the channel and the configuration of the machine parameters.

#### "Single block" mode of simulation.

If the –single block– mode is active, program execution will interrupted at the end of each block; the [START] icon must be pressed again to continue.



The –single block– mode is selected from the icon menu and may be selected before or while simulating the program.

#### Simulation options

The available simulation options are accessed from the icon menu. Pressing the icon displays a window that shows the following options.



#### Tool radius compensation

It activates or cancels tool radius compensation to simulate the program.

Tool radius compensation being off, the simulation will ignore the programmed tool radius compensation.



**CNC 8070** 



# Conditional stop during simulation

It simulates the external "conditional stop" switch.

While active, the program simulation will be interrupted at the blocks having a "conditional stop" function "M01". The simulation will resume when pressing the START icon.



#### **Software limits**

It activates or deactivates the software limits for program simulation.

With the limits activated, if the software limits are reached during simulation, a simulation error message will be issued. If not active, the software limits will be ignored during simulation.



# **Block jump**

It simulates the external "block skip" switch.

While active, the blocks having the block-skip character "/" will be ignored (not simulated).



# Cancel channel synchronization

There is one icon for each channel. It cancels the channel synchronization wait periods during simulation.

When active, the wait period will end immediately and it will resume the execution of the program.



**CNC 8070** 

**FAGOR** 

# 7.2.4 Simulation errors

When an error occurs, the CNC will display a window describing the cause of the error. These errors are displayed in the middle of the screen, regardless of which window is active. Pressing the [ESC] key closes the windows one by one. Use the  $[\mbox{$\frac{1}{2}$}]$  keys to see the various windows without closing them.

There are two types of errors. The top of the window shows the category and it will have a different color depending on the type of error it shows.

#### WARNING

It is just a warning, it does not interrupt the simulation of the program.

The warnings programmed with the instruction #WARNINGSTOP interrupt the execution of the program where the instruction is located. In this case, It's up to the user to either resume the simulation at this point, [START], or abort the program, [RESET].

# **ERROR**

The errors do not interrupt the simulation of the program.

Although the window displaying them may be closed by pressing [ESC], it does not mean that the error status has been taken care of; to do that, press the [RESET] icon. The program can be neither edited nor simulated while the error stays state active.



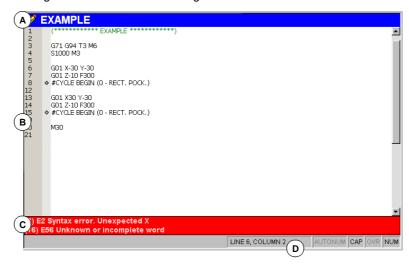
**CNC 8070** 

# 7.3 Editing window

It may be used to edit, modify or see the contents of a part-program and check the program for syntax errors.

# Description of the editing window

The editing window shows the following information:



#### A. Title bar

Name of the program selected for editing. It also indicates whether it is a readonly program or not (if read-only, while simulating or executing).

An "\*" next to the program means that the program has been modified since last saved (only if automatic program saving is off).

## B. Edit area.

Line number and area for editing the program.

- C. Editing errors (if any) and programming assistance. If the text is not display in full, place the focus in this area and move the text using the [♠][♠][♣][♣] keys.
  - This area shows, on a red background, the errors that occur while editing the program or the errors found after running a syntax check of the program.
  - This area shows, on a blue background, the contextual assistance offered by the editor when programming commands in high level language.

## D. Status bar.

CAP:

Information about cursor position and the status of the editor options such as:

AUTONUM: Automatic block numbering. When active, the CNC automatically numbers the new blocks being generated.

Capital letters. When active, the text is always written in capital

letters.

OVR: Overwrite text. It toggles between overwriting and inserting text.

When active, it overwrites the existing text.

NUM: Numeric keypad active.



EDITING-SIMULATION MODE
Editing window



**CNC 8070** 

# Softkey menu of the editing window

When selecting the editing window, the horizontal softkey menu shows the options for this window. The set of options offered by this menu may be changed using the "+" softkey which offers a new set of options.

The options available in the softkey menu are the following. See "7.4 Editing window (softkeys)" on page 116.

"Open program" Open a program at the editor.

"File" To restore, save, save with another name or print a

program. It can also be used to import the contents of another program, of a DXF file or PIM and PIT files.

"Undo" To "undo" the last modifications made.

"Operations with blocks" To copy, cut and paste information of a block or set of

blocks within the program. It can also copy a block or set

of blocks as an independent program.

"Find/replace" To search for a line or text in the program being edited.

It can also replace a text with another text.

"Customize" To customize the appearance and the properties of the

editing window.

"Profile editor"

To access the profile editor and define a new profile or

modify an existing one.

"Cycle editor"

To access the conversational editor for machining and

positioning canned cycles.

"Incline planes" It may be used to access the help for programming

incline planes.

"TEACH-IN" To turn on or off the TEACH-IN mode.

# Hotkeys of the editor.

Shortcut	Function.
[CTRL]+[C]	Copy the selected text.
[CTRL]+[X]	Cut the selected text.
[CTRL]+[V]	Paste the selected text.
[CTRL]+[Z]	Undo the last change.
[CTRL]+[Y]	Redo the selected text.
[CTRL]+[G]	Save the program/Recover the original program.
[CTRL]+[+]	Zoom in.
[CTRL]+[-]	Zoom out.
[ALT]+[-]	Hide or expand a cycle.
[CTRL]+[HOME]	Move the cursor to the beginning of the program.
[CTRL]+[END]	Move the program to the end of the program.
[CTRL]+[TAB]	Toggle between the editor and the error window.



**CNC 8070** 

# Functionalities of the editor.

# Group lines at the editor.

Comments having an asterisk (\*) and programmed at the beginning of the block allow to group blocks. Blocks programmed between these comments will be grouped and may be expanded or shrunk the same way as the cycles or profiles.

# Expand y hide cycles, profiles and grouped blocks.

The editor offers the [ALT]+[-] hotkey to expand y hide cycles, profiles and grouped blocks. If the CNC has a mouse, click on the symbol located to the right of the cycle, profile or group of blocks to expand them and hide them.

Having the "Hide cycles/profiles" option is active, the editor only shows the name of the canned cycle or of the profile. Having this option active, when the cursor moves over a hidden element, it expands automatically; when the cursor moves out of the element, it shrinks again.

#### Zoom at the editor.

The editor has the following hotkeys to increase or decrease the size of the editor font. If the CNC has a mouse with a wheel, the [CTRL] key combined with this wheel can also be used to increase and decrease the size of the text font.

[CTRL]+[+] Zoom in. [CTRL]+[-] Zoom out.

#### Multi-line blocks.



The editor adjusts the long blocks to the size of the window dividing the block into several lines. On the right side of each cut line, the editor shows a symbol to indicate that the block continues in the next line.



**CNC 8070** 

Editing window

# 7.3.1 Contextual programming assistance.

The contextual assistance is shown when editing commands in high level language.

- Keying in "V" shows the list of CNC variables.
- Keying in "#" shows the list of CNC instructions.
- Keying in "\$" shows the list of flow control instructions of the CNC.

In all of them, when keying the following letter of the variable or instruction name, the cursor of the list goes to the first command that starts with that letter. The  $[\, \, \, \, \, ][\, \, \, \, \, ]$  keys may be used to move the cursor through the list of commands and the  $[\, \, \, \, ][\, \, \, \, \, ]$  key to enter the selected command into the block that is being edited.

#### Entering a help element in the block being edited.

When an element has been selected from the list and [ENTER] is pressed, the editor inserts in the cursor position the element selected in the drop menu.

- If the statement does not have parameters, the editor inserts the whole statement.
- If the statement has parameters, but the text written by the user does not have any parameters, the editor inserts only the fixed part of the statement. If the user has written a parameter, the editor does not insert anything.
- If it is not an axis variable and/or array variable, the editor inserts the whole variable.
- If it is an axis variable and/or array variable, but the user has not written it, the editor inserts only the fixed part of the variable. If it is an axis variable and/or array variable and the user has written it, the editor does not insert anything.

After inserting an element, if it is necessary, the bottom of the screen maintains the contextual assistance for that element to complete the editing of the block. Pressing [ENTER] again, the contextual assistance disappears from the bottom of the screen.

#### Activating the contextual assistance.

The contextual assistance is activated with the "PROG HELP" softkey of the screen editing menu. The softkey appears pressed when contextual assistance is enabled.

Contextual assistance is only available when using the 8070 CNC language.



**CNC 8070** 

# 7.3.2 Help for programming subroutines.

# Subroutine help files.

Help files may be associated with each OEM subroutine (G180 through G189) and each global subroutine called upon using #MCALL or #PCALL and they will be displayed while editing. Each subroutine may have two help files; a text file (txt) and an image file (bmp).

The help window is displayed while editing, after a blank space or tab following a G180-G189 or following the name of the subroutine. The help window is only informative, it cannot be accessed with the cursor nor browse through it. When the help file is displayed, its text may be inserted into the part-program using the [INS] key. The help window closes with [ESC], deleting the key word or going on to another line of the program.

The help window is only available at the editor of the 8070, the "Editor 8055" softkey being deactivated. This help is always active, regardless of the editing help softkey, "Prog. help".

# List of available subroutines.

The editor allows having in a text file (txt) a list that is displayed while editing the part-program, every time a #PCALL or an #MCALL instruction is edited.

The editor shows the list of subroutines while editing, after a blank space or tab following a G180-G189 or following a #PCALL or #MCALL instruction. This list works the same way as the lists of variables, it is possible to use the arrow keys to scroll through the various elements. Pressing [ENTER], the editor inserts the selected line in the current position of the cursor. The list of subroutines disappears with [ESC], deleting the key word or going on to another line of the program.

This help is always active, regardless of the editing help softkey, "Prog. help".



For further information on how to define the help on subroutines, refer to the programming manual.



**CNC 8070** 

Editing window

# 7.3.3 Import DXF files

The DXF format is standard for exchanging graphic files. Importing this type of files makes it possible to generate the part-program directly from a drawing. The DXF file may consist of points, lines and arcs. It can also consist of polylines, but they must be previously uncombined.

The program editor and the profile editor can import DXF files. When selecting this option, the editor shows a list of the programs that may be imported. Select the desired program from the list and press [ENTER].

After selecting the file, define how the various layers of the DXF file are to be converted into ISO code. Once this data has been set, press the "Convert" softkey to import the file into the part-program.

# **DXF** file configuration

When importing a DXF file, it shows the configuration window to define how the different layers of the drawing must be treated. The configuration window shows three areas clearly and distinctly.

#### Layer description area

The DXF files may consist of layers and each one has different heights of the drawing. All the layers together make up the whole drawing.

When importing the file, you can decide which layers to include in the part-program. By default, all the layers are included. To exclude a layer, select it and press the "Disable layer" softkey.

For each layer, it indicates its priority and offset (height) on the perpendicular axis.

## Layer priority and offset

It shows the data of the layer selected by the cursor. For each layer, one must define its priority and offset (height) on the perpendicular axis.

- The priority defines the order in which the layers will be executed; i.e. the order in which they will be included in the part-program. Those with priority ·1· will be executed first and so on.
- The offset (height) on the perpendicular axis permits executing each layer in the desired Z coordinate (or that of the relevant perpendicular axis).

## Work plane

The work plane must be defined before importing the file in the part-program. The plane is defined by selecting the abscissa axis, the ordinate axis and the perpendicular axis.

When importing it in the profile editor, the plane will be the one selected at the editor and only the perpendicular axis can be selected.



**CNC 8070** 

# 7.3.4 Syntax errors when editing

The syntax errors occurred while editing or after running a syntax check will be displayed at the bottom of editing window.

To toggle the cursor between the editor and the error listing, press the key combination [CTRL]+[TAB].

# Errors while editing

While editing, each block is analyzed when entered. If a syntax error is detected in the block, the error window will display the following information:

- · Position of the error in the block.
- · Error number and explanatory text.

# Errors after a syntax check



The syntax check is executed from the icon menu. The syntax check is not available for programs written in 8055 CNC language.

The syntax check checks all the blocks of the program. If a syntax error is detected, the error window will show the following information.

- · Location and name of the program being checked.
- Line number and position of the error within the block.
- · Explanation of the error.

Moving the cursor through the errors of the window, the editor will highlight the block containing the selected error. Use the  $[\, \, \bullet \,] [\, \, \bullet \,]$  keys to move the cursor. Press [ENTER] to select the block containing the error or press [ESC] to close the error window.

If the text is not display in full, place the focus in this area and move the text using the [+][+][+][+][+] keys.



EDITING-SIMULATION MODE
Editing window



**CNC 8070** 

# 7.4 Editing window (softkeys)

# 7.4.1 Open program

To select a program for the editing - simulation mode. This program may be a new one or an existing one. A different program may be edited and executed in each channel.

When selecting this option, the CNC shows a list of the available programs. See "2.5 File selection window" on page 25.

To select a program from the list:

- 1. Select the folder that contains the program. If it is a new program, it will be saved in this folder.
- 2. Select the program from the list or write its name in the bottom window. To edit a new program, write the name of the program in the bottom window.
- **3.** Press [ENTER] to accept the selection and open the program or [ESC] to cancel it and close the program listing.



**CNC 8070** 

# 7.4.2 File

It is used to restore, save, import or print a program.

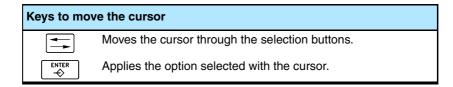


When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

#### File "Restore original"

This option is only available when the "auto save" option is active. See "7.4.6 Customizing" on page 121.

It is used to restore the original program without the changes made since the last time it was opened. When selecting this option, the CNC requests confirmation of the command:



#### File "Save"

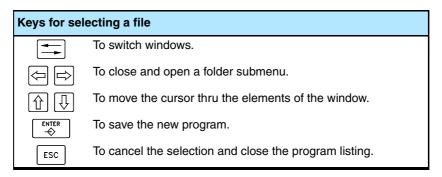
This option is only available when the "auto save" option is not active. See "7.4.6 Customizing" on page 121.

To save the program being edited.

#### File "Save as"

It is used for saving a program that is being edited with a different name. After saving the program, one may continue editing the new program

When selecting this option, the CNC shows a list with all the programs already saved.



To save a program with another name:

- 1. Select the destination folder.
- Write the program name in the bottom window. To replace an existing program, select it from the list
- Press [ENTER] to save the program or [ESC] to return to the editor without saving the program.

Once the program has been saved, the top of the editing window will show the name of the new program.

# File "Include program"

It may be used to import the content of a part-program into the one being edited. Any program that may be accessed from the CNC may be imported, even the program currently in execution.

When selecting this option, the CNC shows a list of the programs that may be imported into the one being edited.



**CNC 8070** 

# Keys for selecting a program



To switch windows.



To move the cursor thru the elements of the window.



To save the new program.



To cancel the selection and close the program listing.

To import a program from the list:

- 1. Select the desired program from the list or write its name in the bottom window.
- 2. Press [ENTER] to import the program or [ESC] to cancel the selection and close the program listing.

The selected program is added to the one being edited after the block indicated by the cursor.



The program may be selected from the list as follows:

- By moving the cursor to the desired element.
- Using the alphanumeric keyboard. Pressing a key will select the first element from the list starting with that letter or number. Pressing it again will select the second one and so on.

Programs may also be selected using the "File search" option of the softkey menu.

#### File "Print"

This softkey may be used to print the program in the pre-determined printer.

#### File "Import"

It permits importing DXF, PIM and PIT files into the program being edited. See "7.3.3 Import DXF files" on page 114.

- The DXF format is standard for exchanging graphic files. Importing this type of files makes it possible to generate the part-program directly from a drawing. The files must consist of points, lines and arcs.
- Los archivos PIM y PIT son los programas pieza utilizados por el CNC 8055. Al importar este tipo de fichero, su lenguaje de programación se adecua al utilizado por el CNC 8070.

When selecting this option, the CNC shows a list of the programs that may be imported into the one being edited. Select the desired program from the list and press [ENTER].

# 7.4.3 Undo

This softkey may be used to "undo" the last modifications made. The modifications are undone one by one starting from the most recent one. The CNC offers the following keyboard shortcuts to undo and redo the operations.

[CTRL]+[Z]

Undo the last change.

[CTRL]+[Y]

Redo the selected text.



**CNC 8070** 

# 7.4.4 Operations with blocks

It is used to copy, cut and paste the information of a block or set of blocks and export this information as an independent program.

This option is only available when there is a text selected in the program or on the clipboard. To select a text in the program, keep the [SHIFT] key pressed while moving the cursor.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

# Operations with blocks "Copy"

It copies the selected text onto the clipboard.

#### Operations with blocks "Cut"

Copies the selected text onto the clipboard and deletes it from the program.

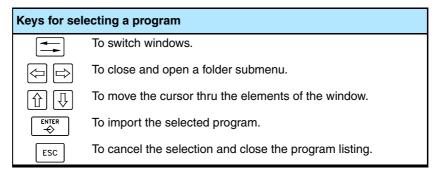
# Operations with blocks "Paste"

Pastes the contents of the clipboard into the program.

# Operations with blocks "Copy to program"

Saves the selected texts as an independent program.

When selecting this option, the CNC shows a list of the available programs.



To save the text as a program:

- 1. Select the destination folder.
- 2. Write the program name in the bottom window. To replace an existing program, select it from the list.
- **3.** Press [ENTER] to save the program or [ESC] to cancel the selection and close the program listing.



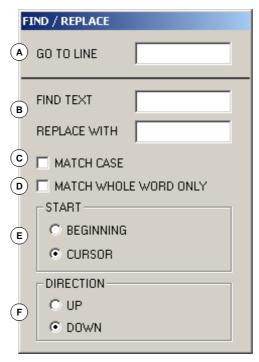
**CNC 8070** 

# 7.4.5 Find/replace

It is used for searching for a line or a text in a program. It can also replace a text with another text.

When selecting this option, the CNC shows a dialog box requesting the line number or the text to look for. When defining a text search, certain options may also be defining that allow:

- A. Go to a line of the program.
- **B.** Replacing the text being searched with another in the program.
- C. Ignore uppercase and lowercase.
- **D.** Consider the text to find as a whole word.
- **E.** Select whether the search starts at the beginning of the program or at the cursor position.
- F. Direction of the search.



After defining the search options, press [ENTER] to start the search or [ESC] to cancel it. The text found in the program will be highlighted and the softkey menu will show the following options:

- "Replace" option, to replace the highlighted text.
- "Replace all" option, to replace the text throughout the whole program.
- "Find next" option, to skip this text and keep on searching.
- "Find previous" option, to look for the text without replacing it.

To end the search, press [ESC].



**CNC 8070** 

# 7.4.6 Customizing

To customize the appearance and the properties of the editing window.

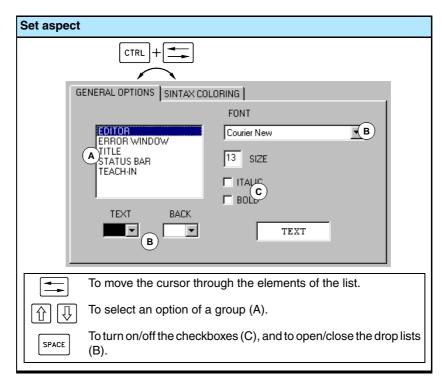


When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

#### Customizing "Appearance" (looks)

It is used for personalizing the look of the different elements of the editing window and the colors of the elements of the program. In large programs (more than 200 kB), the editor cancels the syntax coloring.

When selecting this option, the CNC shows the dialog box with two panels. The first one has the elements of the editing window that may be customized and the second one those of the program. The panel is selected by the key combination [CTRL]+[TAB].



After defining the new look, use the cursor to select one of the buttons here below to accept or ignore the changes made and press [ENTER]. The dialog box may also be closed directly without accepting the changes by pressing [ESC].

# Screen editor "Save always"

This softkey is used to activate the automatic program saving feature. When this option is active, the CNC will automatically save the program every time the cursor changes blocks. In large programs (more than 200 kB), the CNC saves the program when the user has not modified the program for about 5 seconds.

The softkey appears pressed when this option is enabled. If this option is not active, the program is saved from the softkey menu. See "7.4.2 File" on page 117.

## Screen editor "Autonum."

This softkey may be used to activate the block auto-numbering feature. The softkey appears pressed when this option is enabled.

While autonumbering is active, the CNC will insert the block number automatically every time a new block is generated.





**CNC 8070** 

**FAGOR** 

**CNC 8070** 

(REF: 0809)

#### Screen editor "Autonum. parameters"

It is used for selecting the starting block number and the sequencing step for the following blocks when autonumbering the blocks.

When selecting this option, the CNC shows a dialog box requesting both data. After defining these values, press [ENTER] to accept them or [ESC] to ignore them.

#### Customizing "TEACH-IN parameters"

It may be used to set the behavior of each axis in TEACH-IN mode. The behavior of an axis is defined by the properties "visible" and "selected".

"Visible" The axis is displayed in the TEACH-IN window.

"Selected" The axis is included in the blocks that are edited by pressing

the [RECALL] key.

Depending on these properties, one of the following behaviors may be assigned to the axes:

· Selected and visible:

The axis is displayed in the TEACH-IN window and it is included in the blocks being edited by pressing the [RECALL] key.

· Not selected and visible:

The axis is displayed in the TEACH-IN window, but it is not included in the blocks being edited by pressing the [RECALL] key.

· Not selected and not visible:

The axis is neither displayed in the TEACH-IN window nor included in the blocks being edited by pressing the [RECALL] key.

After defining the behavior of the axes, press [ENTER] to accept the selection or [ESC] to reject it.

#### Screen editor "Hide cycles/profiles"

This softkey may be used to hide the contents of the conversational canned cycles and of the profiles. Being this option active, the editor only shows the name of the canned cycle or profile; otherwise, it shows the whole contents. The softkey appears pressed when this option is enabled. In large programs (more than 200 kB), the editor does not hide the canned cycles or the profiles.

Having this option active, when the cursor moves over a hidden element, it expands automatically; when the cursor moves out of the element, it shrinks again.

# Screen editor "Prog. help"

This softkey activates the contextual assistance for programming commands in high level language. The softkey appears pressed when contextual assistance is enabled.

Contextual assistance is only available when using the 8070 CNC language. See "7.3.1 Contextual programming assistance." on page 112.

#### Screen editor "8055 Editor"

This softkey activates editing in the 8055 CNC language. See "7.2.2 Editing a program in the 8055 CNC language" on page 104.

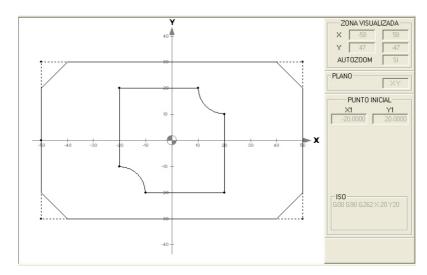
Having set the editor this way, the CNC assumes that this language will be used in the following cases.

- When the program name has the extension *pit* (lathe program) or *pim* (mill program). These are the native extensions of the 8055 CNC.
- When the program name is a number with no more than 6 digits (999999) like the program names at the 8055 CNC.

# 7.4.7 Profile editor

It is used for accessing the profile editor.

When selecting this option, the softkey menu shows the profile editor options. To return to editing the program, press "End".



The profile editor may also be accessed by selecting a profile in the program and pressing [RECALL].

The section on "7.9 Profile editor" of this chapter offers a more detailed description on how to use the profile editor.



EDITING-SIMULATION MODE
Editing window (softkeys)



**CNC 8070** 

#### 7.4.8 **Canned cycles**

It is used for accessing the canned cycle editor.



When selecting this option, the softkey menu shows the available canned cycles. To return to the main menu, press the key for the previous menu.

When selecting a canned cycle, the CNC displays the window for defining that canned cycle. The cycle defining window may also be accessed by selecting a canned cycle in the program and pressing [RECALL].

The cycle being edited may also be simulated in the window for defining the canned cycle.



Canned cycles have a specific manual both for milling and turning. Refer to the documentation included in the CD-Rom that comes with the product for further detail.

**EDITING-SIMULATION MODE** Editing window (softkeys)



**CNC 8070** 

# 7.4.9 Incline planes.



This softkey may be used to access the help screen for programming incline planes. When selecting this option, the softkey menu shows the different ways to program incline planes. To return to the main menu, press the key for the previous menu.

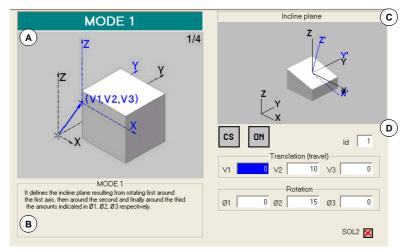
The screen for defining incline planes may also be accessed by selecting one in the program and pressing [RECALL]. Incline planes programmed directly in ISO code using the instructions #CS and #ACS can also be restored using the [RECALL] key. This way, it is possible to check the programmed parameters and the resulting incline plane.

Once the incline plane has been defined, press [ESC] to quit the editor. The softkey menu will show the "insert incline plane" to insert the defined incline plane into the program. The block for the defined inclined plane is inserted after the block where the cursor is.



For further information on how to program incline planes, instructions #CS and #ACS, refer to the programming manual.

# Description of the help screen for incline planes.



- **A.** Help graphics. Sequence of drawings showing each step for defining the incline plane. When the focus is on a programmable parameter, the sequence will stop and show the relevant explanatory drawing.
- **B.** Short explanation on how to program the selected incline plane.
- **C.** Drawing of the resulting incline plane. The drawing is updated (refreshed) as the user programs the different parameters that affect the geometry of the incline plane. The drawing shows the following elements.
  - The incline plane in white.
  - The reference system of the incline plane, X' Y' Z', in blue.
  - When there is a translation vector (V1, V2, V3), the drawing will show a
    reference system at the lower left side of the drawing. The distance between
    this reference system and the cube is not proportional; the graphics is merely
    approximate.

When the drawing is selected with the focus, the cube may be rotated using the  $[\!\![ \bullet ]\!\!] \!\![ \bullet ]\!\!] \!\![ \bullet ]\!\!] \!\![ \bullet ]\!\!]$  keys or using the wheel of the mouse.

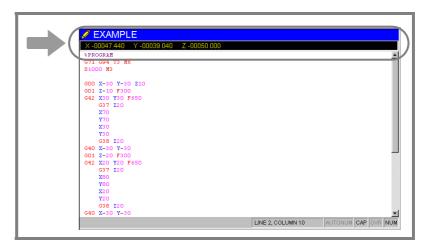
**D.** Parameters to define the incline plane.



**CNC 8070** 

It is used to activate or deactivate the TEACH-IN mode; in this mode, the axes may be moved manually and their position may be assigned to a block.

When active, the top of the editing window shows the position of the axes defined as "visible" for the TEACH-IN mode. See "7.4.6 Customizing" on page 121.



When TEACH-IN mode is active, it is possible to keep editing the coordinates of the axes directly from the keyboard or they may be assigned the current position of the machine axes. Both editing methods may be used indistinctly, even while defining a block.

To define the coordinates of one or several axes using TEACH-IN:

- Move the axes to the desired position using the JOG keys, the handwheels or the MDI mode.
- 2. In the part-program, edit the name of the axes whose position is to be defined or not select any axis if you wish to define the position of all of them.
- 3. Press the [RECALL] key.

If an axis of the channel has been edited, the CNC assigns the current position of that axis as the program coordinate. The axis is displayed in the TEACH-IN window.

If only the block number or an empty line has been edited, it inserts a block with the position of all the axes of the channel defined as "selected" for the TEACH-IN mode. See "7.4.6 Customizing" on page 121.

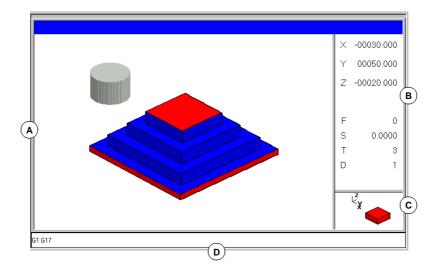
If a character has been edited other than the axis name or the block number, it does not insert anything and the cursor stays in the same place.



**CNC 8070** 

# 7.5 Graphics window

It is used to show a graphic representation of the program being simulated and take measurements over the drawing. The CNC has different types of graphics. See chapter "8 Graphic environment (mill model)". See chapter "9 Graphic environment (lathe model)".





EDITING-SIMULATION MODE
Graphics window

FAGOR

**CNC 8070** 

Program window

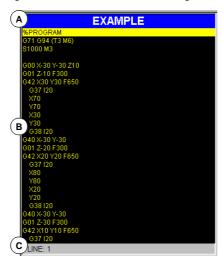
# 7.6 Program window

It shows the contents of the program selected for simulation and allows selecting the first and last blocks of the simulation. When not selected, the simulation will begin at the first block of the block and will end after executing one of the end-of-program functions "M02" or "M30".

While simulating, the window cursor will show the block being simulated.

# Description of the program window

The program window shows the following information:



#### A. Title bar.

Name of the program selected for simulation.

B. Program blocks selected for simulation.

While simulating, the cursor will indicate the block being simulated. The "active subroutines" option being active, the window displays information related to the execution of subroutines, canned cycles, repetition blocks and loops.

C. Program line.

Line of the program where the cursor is.

# Softkey menus of the program window

When selecting the program window, the horizontal softkey menu will show the options related to this window.

The options available in the softkey menu are the following. See "7.7 Program window (softkeys)" on page 129.

"First block"

To set the first block of the simulation.

"Stop condition"

To select the block where the simulation of the program or subroutine will be interrupted.

• "Active subroutines"

It may be used to display information related to the simulation status of the subroutines, canned cycles, repetition blocks and loops.

"Find text"

It may be used to search for a text or placing the cursor on a particular line of the program.

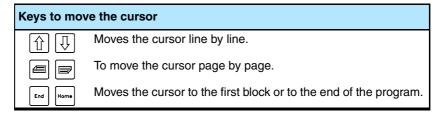


**CNC 8070** 

# 7.7.1 First block

It sets as the first simulation block, the block where the cursor is. When not setting the first block, the simulation will begin at the first block of the program.

The last block may be selected using the cursor or the "Find text" or "Go to line" options of the softkey menu.



The first block defined, remains active until canceled (with [ESC]) or until the program is simulated; in that case, the first block of the program will become the first block of the simulation.



**CNC 8070** 

**FAGOR** 

# 7.7.2 Stop condition





It is used to establish, in the program or in a subroutine, the block where the program simulation will be interrupted. After executing that block, the simulation may be resumed with the [START] icon or canceled with the [RESET] icon.

7

START RESET

If no last block is established, the simulation of the program will end after executing one of the end-of-program functions "M02" or "M30".



When selecting this option, the softkey menu of the CNC shows the options available for selecting the stop condition. To return to the main menu, press the key for the previous menu.

#### Subroutine selection

To select the stop condition in a global subroutine which has been called upon from the program. When selecting this option, the CNC shows a list of programs available for selecting the desired subroutine.

# Keys for selecting a program To move the cursor thru the elements of the window. To open the selected subroutine. ESC To cancel the selection and close the program listing.

To select a subroutine from the list.

- Select from the list the program containing the subroutine or write its name in the bottom window.
- **2.** Press [ENTER] to open the program or [ESC] to cancel the selection and close the program listing.

The program window will show the contents of the selected subroutine.

#### Last block

It sets the cursor position as the block where simulation is interrupted. If no last block is established, the simulation of the program will end after executing one of the end-of-program functions "M02" or "M30".

The last block may be selected using the cursor or the "Find text" or "Go to line" options of the softkey menu.

Keys to move the cursor	
①贝	Moves the cursor line by line.
	To move the cursor page by page.
End	Moves the cursor to the first block or to the end of the program.

The established last block remains active until it is canceled (with [ESC]) or a program is simulated; in that case, the execution of one of the functions "M02" or "M30" is set as the stop condition.

#### **Number of times**

It sets as stop condition, that the block selected as the last block has been executed a specific number of times.

When selecting this option, the CNC requests the number of times the block must be executed before finishing. After entering the number of times, press [ENTER] to validate the value or [ESC] to cancel it.



**CNC 8070** 

#### 7.7.3 **Active subroutines.**

This option may be used to toggle between the display of program blocks and the display of information related to the status of the subroutines, canned cycles, repetition blocks and loops.

Having this option active and the program simulation interrupted, the user can use the cursor to select an information line and press [ENTER] to skip to the corresponding program block.

# Description of the information displayed.

The information is laid out in a table with four columns and having the following meaning.

Column.	Meaning.
S	Nesting level of the subroutine.
Р	Nesting level of the local parameters used in the subroutine.
Sub	Name of the program, subroutine or canned cycle.
Ор	Type of block being simulated. The loops are shown with a progress bar and a text indicating the loop it is in.

The window shows the following information.

Type of block.	Information displayed.
L	Name of the subroutine
LL	Name of the subroutine
#CALL	Name of the subroutine
#PCALL	Name of the subroutine
#MCALL	Name of the subroutine
G180 through G189	Name of subroutine associated with the G function.
M function.	Name of the subroutine associated with the M function.
T function.	Name of the subroutine associated with the T function.
Canned cycle.	G function associated with the canned cycle.
#EXEC	Name of the subroutine
#EXBLK	
\$RPT	Number of current loop and total number of loops.
\$FOR	Number of current loop and total number of loops.
\$WHILE	Number of current loop.
\$DO	Number of current loop.
\$IF	Text "IF".
\$SWITCH	Text "SWITCH".

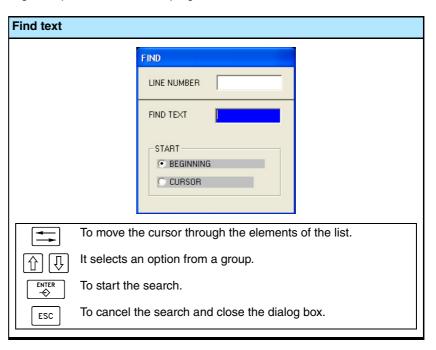


**EDITING-SIMULATION MODE** 



**CNC 8070** 

When selecting this option, the CNC shows a dialog box with two areas so the user can go to a particular line of the program or look for a text.



# Go to line

This option may be used to position the cursor on a particular line of the program.

In this area of the dialog box, the CNC requests the line number to go to. Key in the desired number and press [ENTER], the cursor will then go to that line.

#### Find text

This option may be used to find text or a character string throughout the program.

In this area of the dialog box, the CNC requests the text to look for. It is also possible to select whether the search must start at the beginning of the program or at cursor position.

To start the search, press [ENTER] and the cursor will position on the text found. Pressing [ENTER] again, the CNC will look for the next match and so on.

To end the search, press [ESC]. The cursor will position on the block containing the text searched.





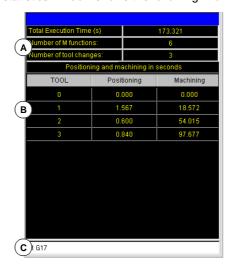
**CNC 8070** 

# 7.8 Statistics window

It shows an estimate of the total execution time of the program and of the machining time for each tool.

# Description of the statistics window

This statistics window shows the following information:



#### A. General information.

It shows a time estimate of program execution at 100% of the programmed feedrate, the number of "M" functions executed and the number of tool changes made.

B. Machining time for each tool.

It shows a list of the tools used in the program indicating the machining time for each tool and the total positioning time.

C. "G" functions active during simulation.

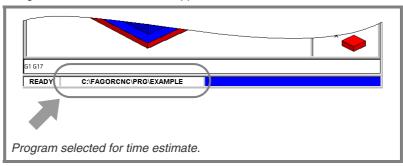


**CNC 8070** 

Statistics window

# 7.8.1 Time estimates

The statistics window shows an estimate of the execution of the program selected in the editing window, and whose name appears at the bottom center of the screen.



The process to estimate time is the following:

1. Use the program window to select the first and last block for the execution time estimate.

If not selected, the execution time estimate will be done from the first block of the program to the execution of one of the end-of-program functions "M02" or "M30".



- 2. Activate the desired simulation options using the icon menu.
- 3. Start the estimate calculation by pressing its TIMES icon.

# Simulation options

These options may also be turned on or off while simulating the program. When they are active, its corresponding icon of the icon menu will appear pressed.

The simulation options available are:



# "Single block" mode

It selects the "single block" mode so the execution time calculation is interrupted at the end of each block.



# **Block jump**

It simulates the external "block skip" switch.

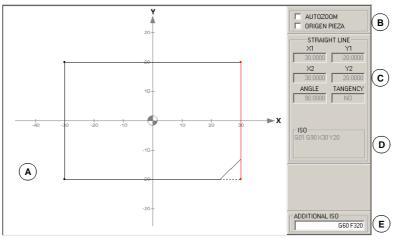
When active, the CNC will NOT calculate the execution time of the blocks containing the block-skip character "/".



**CNC 8070** 

#### **Profile editor** 7.9

The profile editor looks like this.



A. Graphic area. Graphic representation of the profile being drawn, axes coordinated with autoscale and name of the axes that make up the plane. The name of the axis indicates the positive direction of the axis. At the lathe model, the direction and position of the axes are defined by machine parameter GRAPHTYPE.

The user can move the graphics using the arrow keys, enlarge or reduce the displayed area using the [+] or [-] key, apply the best zoom with the [=] key, keep the part zero visible with the [/] key and activate the autozoom with the [\*] key. These options will not be available when a menu for editing data or selecting items is active at the editor.

- B. Status of the autozoom and part zero options, regarding the display of the profile at the editor.
- C. Data entry area.
- D. Translation (conversion) of the selected profile or part of it into ISO code
- E. Area used to enter the values of the corners or the ISO coded text to be added to the element.

# Softkey menu.

The options that may be selected from the softkey menu make it possible to edit profiles, modify edited profiles, select the zoom, the work plane, undo the last change and end the editing session. While editing or modifying the profile, the softkey menu offers the option to undo the last operation. Likewise, it offers the option to save the profile at any time.

Softkey.	Meaning.
Edit	To select the type of profile to be edited. Also to enlarge a profile, import a profile saved in DXF format. See "7.9.2 Profile definition" on page 138.
Modify	To modify, insert or delete elements of a profile. See "7.9.6 Modify a profile and insert corners" on page 143.
Displayed area	To modify the zoom of the graphics area.
Plane	To select the axes of the plane and their direction.
End	It ends the profile editing session. Before exiting the profile editor, it will offer the option to save or not the edited profile.
Undo	To "undo" the last operation made.
Save and continue	To save the profile and continue editing. Using this key does not require that the profile be completed.



**EDITING-SIMULATION MODE** Profile editor

**FAGOR** 

**CNC 8070** 

# Displayed area

The softkey menu of this screen may be used to modify the zoom of the graphics area. When accessing the "display area" menu, the softkey menu shows the following options:

Softkey.	Meaning.
Zoom + Zoom -	To enlarge or reduce the display area.  The user can enlarge or reduce the displayed area using the +] or [-] key and move the graphics with the arrow keys.
Optimum area	It selects the best display area, i.e. it places the profile in the center of the graphics area and selects the best zoom possible to show the whole profile.  The user may apply the best area with the [=] key.
Part zero	It keeps the part zero visible at all times.  The user may change the status of this option with the [/] key.
Autozoom	When applying Autozoom, every time a new section is inserted which goes beyond the screen, the profile will automatically be centered and zoomed in or out to show the whole profile. This way, it will show the whole profile again. The user may change the status of this option with the [*] key.
Validate	It may be used to confirm the changes made.

# Define the work plane.

The softkey menu of this screen may be used to modify the axes of the plane and their directions. When accessing the plane menu, the softkey menu shows the following options:

Softkey.	Meaning.
Abscissa axis	It may be used to select the abscissa axis.
Abscissa direction	It may be used to select the direction of the abscissa axis.
Ordinate axis	It may be used to select the ordinate axis.
Ordinate direction	It may be used to select the direction of the ordinate axis.
Validate	It may be used to confirm the changes made.

# End the session at the editor.

The "End" softkey ends the profile editing session. Before exiting the profile editor, it will offer the option to save or not the edited profile.

Softkey.	Meaning.
Save profile	It inserts the profile in the program and exits the profile editor.
Do not save profile	It does not insert the profile in the program and it exits the profile editor.
Continue	It does not insert the profile in the program and it goes on editing profile.

If the profile has been resolved, the CNC will insert it in the part program that is being edited. If the profile cannot be resolved due to lack of data, the CNC will issue the relevant message.



**CNC 8070** 

# 7.9.1 How to use the profile editor.

Several profiles may be edited without having to exit the profile editor. To edit a profile, proceed as follows:

- 1. Define the work plane at the profile editor.
- 2. Select the type of profile to be edited, such as a circular or rectangular profile or any profile.
- 3. For a rectangular or circular profile, define its data and insert it. For any profile, first select the starting point of the profile. Once the first point has been selected, draw the profile, which will be made up of straight and curved sections. If it has corner rounding, chamfering or tangential entries and exits, use one of these methods:
  - Treat them as individual sections when having enough information to define them.
  - Ignore them while defining the profile and, once it has been defined, select the corners that have those characteristics and insert them.
- 4. End the profile editing session by inserting them into the program. The portion of ISO-code program corresponding to the edited profile will be identified with the line "(#PROFILE)" or it will appear framed between the lines "(#PROFILE BEGIN)" and "(#PROFILE END)".

# **Data editing**

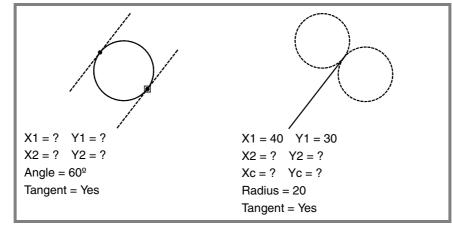
All data need not be defined; but it is recommended to define all the known data. To define the profile data, proceed as follows:

- 1. Press the softkey corresponding to the data to be defined.
- 2. Key in the desired value (which may be a numeric constant or an expression entered via the calculator). Use the [SPACE] key to change the value of a non-numerical data (tangency, direction, etc.). Press [CTRL]+[K] to access the calculator.
- Press [ENTER] to accept the defined value or [ESC] to reject it and return to the previous one. If the entered value is accepted, the CNC will select the next data.
- **4.** Once all the data has been defined, press the "Validate" softkey and the CNC will show the profile that has been defined.

#### Displaying incomplete sections

If there isn't enough data to show the defined section, the CNC will draw as much of it as it knows. The sections that are not fully defined will be shown with a dash line.

If there are more than one possibility, use the arrow keys to view the available options one by one except the ones that could generate tangency errors later on. To select the desired option, press [ENTER]. The sections with several possibilities will be shown in green whereas the rest of the sections will be shown in white.





**CNC 8070** 

# 7.9.2 Profile definition

With the softkey menu of this screen, it is possible to define any profile, a circular or a rectangular one. It can also be used to enlarge a profile already defined or import a profile saved in DXF format.

Softkey.	Meaning.
Profile	To edit any profile by defining the straight and curved sections that make it up. See "7.9.3 Define any profile." on page 139.
Circle	For quickly defining a circular profile. See "7.9.4 Define a circular profile." on page 141.
Rectangle	For quickly defining a rectangular profile. See "7.9.5 Define a rectangular profile." on page 142.
Expanding a profile	For adding a straight line or an arc in any position of a previously edited profile.
Import profile	For importing a previously saved profile. These profiles may also be imported in DXF format besides in ISO format. See "7.3.3 Import DXF files" on page 114.
Undo	To "undo" the last operation made.
Save and continue	To save the profile and continue editing. Using this key does not require that the profile be completed.





**CNC 8070** 

# 7.9.3 Define any profile.

For any element of the profile, the softkey menu may be used to define the data in Cartesian or Polar coordinates as well as in absolute or incremental coordinates.

Softkey.	Meaning.
Cartesian Polar	Softkey to select either Cartesian or Polar coordinates. The softkey is only available when allowed by the selected data.
ABS INC	Softkey to select either absolute or incremental coordinates. The softkey is only available when allowed by the selected data.
Polar origin	Softkey to define the Polar origin. The softkey is only available when Polar coordinates are active.
Direction	Direction of the profile; clockwise or counterclockwise.
Validate	Validate the profile defined.

# Define the starting point of the profile.

When selecting a new profile, the starting point must always be defined first. Once the profile's starting point has been defined, the softkey menu will show the necessary options to define the profile. To return to the main menu, press the key for the previous menu.

The starting point may be edited both in Cartesian and Polar coordinates, but always in absolute coordinates.

# Define a straight section of the profile.

The end point of the section may be edited both in Cartesian and Polar coordinates as well as in absolute or incremental coordinates. Being the incremental coordinates active, the editor will show the symbol  $\Delta$  next to the affected data.

#### Cartesian coordinates.

The data definition area shows the following data.

Data.	Information
X1, Y1	Coordinates of the starting point of the section on each axis of the active plane at the editor.
X2, Y2	Coordinates of the end point of the section on each axis of the active plane at the editor.
Angle	Angle of the profile with the abscissa axis.
Tangency	Tangency with the previous section of the profile.

## Polar coordinates.

The data definition area shows the following data.

Data.	Information
r1, θ1	Radius and Polar angle of the section's starting point.
r2, θ2	Radius and Polar angle of the section's end point.
Angle	Angle of the profile with the abscissa axis.
Tangency	Tangency with the previous section of the profile.



**CNC 8070** 

# Define a curved section (clockwise or counterclockwise) of the profile.

The end point and the center of the section may be edited both in Cartesian and Polar coordinates as well as in absolute or incremental coordinates. Both points may have different types of coordinates and, besides, when using Polar coordinates, both points may have a different Polar origin. Being the incremental coordinates active, the editor will show the symbol  $\Delta$  next to the affected data.

#### Cartesian coordinates.

The data definition area shows the following data.

Data.	Information
X1, Y1	Coordinates of the starting point of the section on each axis of the active plane at the editor.
X2, Y2	Coordinates of the end point of the section on each axis of the active plane at the editor.
Xc, Yc	Coordinates of the center of the section on each axis of the active plane at the editor.
Radius	Radius of the arc.
Tangency	Tangency with the previous section of the profile.

#### Polar coordinates.

The data definition area shows the following data.

Data.	Information
r1, θ1	Radius and Polar angle of the section's starting point.
r2, θ2	Radius and Polar angle of the section's end point.
rc, θc	Radius and Polar angle of the section's center point.
Radius	Radius of the arc.
Tangency	Tangency with the previous section of the profile.

# Start a new profile.

This softkey may be used to add a new profile to the current one, for example to define a pocket with islands.

# Modify the last element of the profile.

This softkey may be used to modify the last element added to the profile.

# Undo the last operation.

This softkey may be used to "undo" the last operation made.

# Save and continue.

To save the profile and continue editing. Using this key does not require that the profile be completed.



**CNC 8070** 

# 7.9.4 Define a circular profile.

The softkey menu may be used to define the data in Cartesian or Polar coordinates as well as in absolute or incremental coordinates.

Softkey.	Meaning.
Cartesian Polar	Softkey to select either Cartesian or Polar coordinates. The softkey is only available when allowed by the selected data.
ABS INC	Softkey to select either absolute or incremental coordinates. The softkey is only available when allowed by the selected data.
Polar origin	Softkey to define the Polar origin. The softkey is only available when Polar coordinates are active.
Direction	Direction of the profile; clockwise or counterclockwise.
Validate	Validate the profile defined.

The type of coordinates may be changed at any time and the editor will update the displayed values.

- The starting point and the center may be edited both in Cartesian and Polar coordinates, but both points must have the same type of coordinates. Therefore, a change of coordinate type affects both points. If programmed in Polar coordinates, the Polar origin will be the same for both.
- The starting point of the circle may only be edited both in absolute coordinates, whereas the center may be edited in both absolute and incremental coordinates.

# Define the data of the profile.

The data definition area shows all the data needed to define a profile. The data shown by the editor depends on the active coordinate type, Cartesian or Polar. Being the incremental coordinates active, the editor will show the symbol  $\Delta$  next to the affected data.

# Cartesian coordinates.

The data definition area shows the following data.

Data.	Information
X1, Y1	Coordinates of the starting point of the profile on each axis of the active plane at the editor.
Xc, Yc	Coordinates of the center of the profile on each axis of the active plane at the editor.
Radius	Radius of the arc.
Direction	Direction of the profile; clockwise or counterclockwise.

# Polar coordinates.

The data definition area shows the following data.

Data.	Information
r1, θ1	Radius and Polar angle of the profile's starting point.
rc, θc	Radius and Polar angle of the profile's center point.
Radius	Radius of the arc.
Direction	Direction of the profile; clockwise or counterclockwise.



**CNC 8070** 

# 7.9.5 Define a rectangular profile.

The softkey menu may be used to define the data in Cartesian or Polar coordinates.

Softkey.	Meaning.
Cartesian Polar	Softkey to select either Cartesian or Polar coordinates. The softkey is only available when allowed by the selected data.
Polar origin	Softkey to define the Polar origin. The softkey is only available when Polar coordinates are active.
Direction	Direction of the profile; clockwise or counterclockwise.
Validate	Validate the profile defined.

The type of coordinates may be changed at any time and the editor will update the displayed values.

- The starting point may be edited both in Cartesian and Polar coordinates.
- The starting point of the circle can only be edited in absolute coordinates.

# Define the data of the profile.

The data definition area shows all the data needed to define a profile. The data shown by the editor depends on the active coordinate type, Cartesian or Polar.

# Cartesian coordinates.

The data definition area shows the following data.

Data.	Information
X1, Y1	Coordinates of the starting point of the profile on each axis of the active plane at the editor.
XL, YL	Length of the profile on each axis of the active plane at the editor.
Angle	Angle of the profile with the abscissa axis.
Direction	Direction of the profile; clockwise or counterclockwise.

#### Polar coordinates.

The data definition area shows the following data.

Data.	Information
r1, θ1	Radius and Polar angle of the profile's starting point.
XL, YL	Length of the profile on each axis of the active plane at the editor.
Angle	Angle of the profile with the abscissa axis.
Direction	Direction of the profile; clockwise or counterclockwise.



**CNC 8070** 

## 7.9.6 Modify a profile and insert corners

To modify the defined profiles by either modifying or deleting the existing elements or inserting new elements. Likewise, it is possible to insert rounding, chamfers and tangential entries or exits.

When selecting this option, the softkey menu will show the necessary options to modify the profile.

Softkey.	Meaning.
Modify element	This softkey may be used to modify any data of a section of the profile.
Insert element	This softkey may be used to insert a new element in any position of the profile.
Delete element	This softkey may be used to delete an element of the selected profile.
Corners	This softkey may be used to include rounding, chamfers, tangential entries or exits in the defined profile.
Additional ISO	This softkey may be used to add an ISO coded line to a previously closed profile. Once the ISO coded line to be added has been entered, confirm the command by pressing the [ENTER] key.
Undo	To "undo" the last operation made.
Save and continue	To save the profile and continue editing. Using this key does not require that the profile be completed.

## **Modify element**

This softkey may be used to modify any data of a section of the profile. Once the desired element has been selected, one may modify the type of section (straight or arc) or its data. Once the element has been modified, press "Validate" to confirm the changes.

The CNC recalculates the new profile according to the data used to define that section and the next one (tangency, angle, etc.)

## Insert element

This softkey may be used to insert a new element in any position of the profile. After selecting the section after which the element is to be inserted, select the type of section (straight or arc) to be inserted, define its parameters and press the "Validate" softkey.

The CNC recalculates the new profile according to the data used to define that section and the next one (tangency, angle, etc.)

## **Delete element**

This softkey may be used to delete an element of the selected profile. Once the element to be deleted has been selected, confirm the command by pressing [ENTER]. The CNC recalculates the new profile.



**CNC 8070** 

Profile editor

## Corner definition

This softkey may be used to include rounding, chamfers, tangential entries or exits in the defined profile. When selecting this option, the softkey menu shows the type of corners that can be inserted.

Softkey.	Meaning.
Rounding	To insert a rounding at the profiles corners where it is possible.
Chamfer	To insert a chamfer at the profiles corners where it is possible.
Tangential entry	To add a tangential tool entry at the beginning of the profile.
Tangential exit	To add a tangential tool exit at the end of the profile.

Once the type of corner to be inserted has been selected, the CNC will highlight in red one of the corners of the profile. Use the softkey menu to select another element of the profile or select a corner of another profile.

Keys to sele	ect a corner
	To select a corner of the profile.
	To select a corner of another profile.

Select the profile corner to be modified and the CNC will request the value (radius or size) of the corner to be inserted:

- For a rounding, enter the rounding radius.
- For a chamfer, enter the size of the chamfer.
- For a tangential entry, enter the entry radius.
- For a tangential exit, enter the exit radius.

After defining the value of the corner, press [ENTER] to insert the corner in the profile. Then, one may select another corner or quit this mode by pressing [ESC].



**CNC 8070** 

# R10.00

## Profile definition without rounding, chamfers or tangential entries and exits.

Starting point	X = 80	Y =-20			
Straight	X = 80	Y =20			
Straight	X = 20	Y =20			
Straight	X = 20	Y =80			
Straight	X = 60	Y =80			
Counterclockwise arc	X = 100	Y =80	Center X = 80	Center Y = 80	Radius = 20
Straight	X = 140	Y =80			
Straight	X = 140	Y =20			
Straight	X = 80	Y =20			
Straight	X = 80	Y =-20			

## Definition of rounding, chamfers and tangential entries and exits

Select the "CORNERS" option and define:

Tangential entry	Select point "1"	Assign radius = 5	Press [ENTER]
Chamfer	Select point "2"	Assign size = 10	Press [ENTER]
Rounding	Select point "3"	Assign radius = 10	Press [ENTER]
Rounding	Select point "4"	Assign radius = 5	Press [ENTER]
Rounding	Select point "5"	Assign radius = 5	Press [ENTER]
Rounding	Select point "6"	Assign radius = 10	Press [ENTER]
Chamfer	Select point "7"	Assign size = 10	Press [ENTER]
Tangential exit	Select point "1"	Assign radius = 5	Press [ENTER]

Press [ESC] to quit the "CORNER" option.

## **End of editing**

Select the "END" option and save the profile. The CNC quits the profile editor and inserts the profile in the part-program.



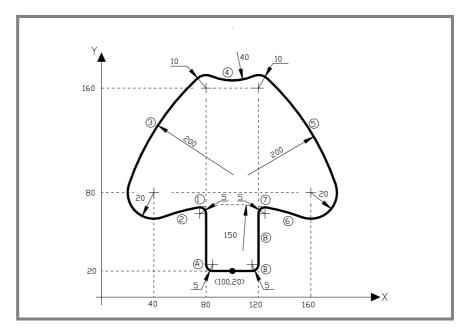
Profile editor

**EDITING-SIMULATION MODE** 



**CNC 8070** 

## Profile editor. Example 2 (milling). 7.9.8



## Definition of a profile without rounding.

Starting point	X = 100	Y =20
Straight	X = 80	Y =20
Straight	X = 80	Angle = 90
Counterclockwise arc (1)	Center X = 75	Radius = 5

Tangency = Yes Counterclockwise arc (2) Center X = 100 Radius = 150 Tangency = Yes

Clockwise arc Center X = 40Center Y = 80 Radius = 20 Tangency = Yes

The CNC displays the options for section 2. Select the right one.

The CNC displays the options for section 1. Select the right one.

Radius = 200 Clockwise arc Tangency = Yes

Center X = 80 Center Y = 160 Clockwise arc Radius = 10 Tangency = Yes

The CNC displays the options for section 3. Select the right one.

Counterclockwise arc (4) Radius = 40 Tangency = Yes

Clockwise arc Center X = 120 Center Y = 160 Radius = 10 Tangency = Yes

The CNC displays the options for section 4. Select the right one.

Clockwise arc (5) Radius = 200 Tangency = Yes

Clockwise arc Center X = 160 Center Y = 80 Radius = 20 Tangency = Yes

The CNC displays the options for section 5. Select the right one.

Counterclockwise arc (6) Center X = 100Radius = 150 Tangency = Yes

The CNC displays the options for section 6. Select the right one.

Counterclockwise arc (7) Center X = 125Radius = 5 Tangency = Yes

> The CNC displays the options for section 7. Select the right one. Y = 20

The CNC displays the options for section 8. Select the right one.

Straight X = 100

X = 120

Straight (8)

## Rounding definition "A" and "B".

Select the "CORNERS" option and define:

Rounding Select point "A" Assign radius = 5 Press [ENTER] Rounding Select point "B" Assign radius = 5 Press [ENTER]

Press [ESC] to quit the "CORNER" option.

## End of editing

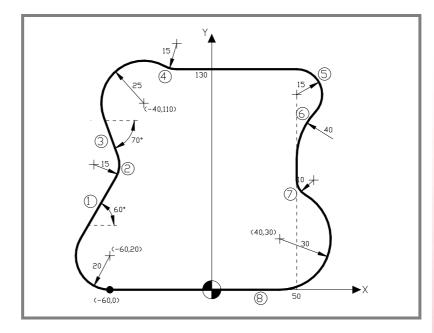
Select the "END" option and save the profile. The CNC quits the profile editor and inserts the profile in the part-program.

Tangency = Yes



**CNC 8070** 

## 7.9.9 Profile editor. Example 3 (milling).



## Profile definition.

X = -60Y =0 Starting point

Center X = -60Center Y = 20 Radius = 20 Clockwise arc

Angle = 60 Straight (1) Tangency = Yes

The CNC displays the options for section 1. Select the right one.

Tangency = Yes Radius = 15 Counterclockwise

Angle = -70Tangency = Yes Straight

Radius = 25 Center Y = 110 Clockwise arc Center X = -40Tangency = Yes

The CNC displays the options for section 3. Select the right one.

The CNC displays the options for section 2. Select the right one.

Counterclockwise (4) Radius = 15 Tangency = Yes

arc

Straight Y = 130Angle = 0Tangency = Yes

The CNC displays the options for section 4. Select the right one.

Center X = 50 Radius = 15 Tangency = Yes Clockwise arc

The CNC displays the options for section 5. Select the right one.

Radius = 40 Counterclockwise Tangency = Yes arc

> X = 50Angle = 270Tangency = Yes Straight

> > The CNC displays the options for section 6. Select the right one.

Radius = 10 Tangency = Yes Counterclockwise

Clockwise arc Center X = 40Center Y = 30 Radius = 30 Tangency = Yes

The CNC displays the options for section 7. Select the right one.

Straight Y = 0Tangency = Yes

The CNC displays the options for section 8. Select the right one.

## End of editing

Select the "END" option and save the profile. The CNC quits the profile editor and inserts the profile in the part-program.

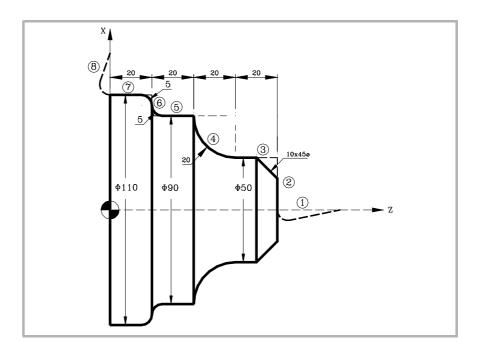
**EDITING-SIMULATION MODE** Profile editor



**CNC 8070** 

## 7.9.10 Profile editor. Example 4 (lathe)

EDITING-SIMULATION MODE



## Profile definition without rounding, chamfers or tangential entries and exits.

Starting point		Z = 100	X = 0			
Straight	(1)	Z = 80	X = 0			
Straight	(2)	Z = 80	X = 50			
Straight	(3)	Z = 60	X = 50			
Clockwise arc	(4)	Z = 40	X = 90	Zc = 60	Xc = 90	R = 20
Straight	(5)	Z = 20	X = 90			
Straight	(6)	Z = 20	X = 110			
Straight	(7)	Z = 0	X = 110			
Straight	(8)	Z = 0	X = 150			

## Definition of rounding, chamfers and tangential entries and exits

## Select the "CORNERS" option and define:

Tangential entry	Select the corner "1-2"	Assign radius = 5	Press [ENTER]
Chamfer	Select the corner "2-3"	Assign size = 10	Press [ENTER]
Rounding	Select the corner "5-6"	Assign radius = 5	Press [ENTER]
Rounding	Select the corner "6-7"	Assign radius = 5	Press [ENTER]
Tangential exit	Select the corner "7-8"	Assign radius = 5	Press [ENTER]

Press [ESC] to quit the "CORNER" option.

## **End of editing**

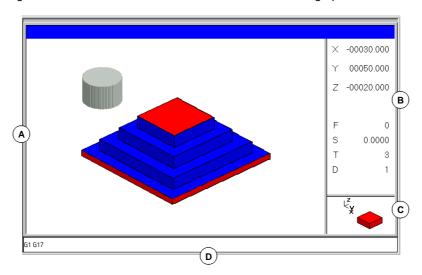
Select the "END" option and save the profile. The CNC quits the profile editor and inserts the profile in the part-program.



**CNC 8070** 

## GRAPHIC ENVIRONMENT (MILL MODEL)

The graphic environment displays a graphic representation of the program that is being executed or simulated and take measurements on the graphics.



## A. Display area.

Shows a graphic representation of the tool paths or of the part as the program is being executed or simulated.

## B. Dialog area.

Shows the position of the axes and the machining conditions active in the execution or simulation (feedrate, spindle speed, tool and tool offset).

When selecting specific options of the softkey menu, it shows the information associated with that option which may be defined by the user.

## C. Information area.

Shows the work plane appearing in the display area and an illustration representing the size of the graph and the portion of the graphic area selected with the zoom.

On 3D graphics, the illustration shows the point of view of the graph displayed and it may be changed by the operator.

Keys for selecting the point of view		
$\hat{1}$	Select a new point of view.	
ENTER -♦	Assumes the new point of view selected.	
ESC	Cancels the selected point point of view.	

**D.** This area only appears during simulation and shows the "G" functions active in the simulation.



**CNC 8070** 

## Softkey menus of the graphics window

When selecting the graphic window, the horizontal softkey menu shows the options associated with that window. The set of options offered by this menu may be changed using the "+" softkey which offers a new set of options.

The options available in the softkey menu are the following.

• "Type of graphics"

To select the type of graphics.

• "ZOOM"

To enlarge or reduce the whole graph shown or part of it.

• "Dimensions"

To define the size of the graphic representation.

• "Point of view"

To change the point of view of the graph and show it from another perspective.

"Measurement"

To measure the distance between two points.

"Clear screen"

It is used to clear the screen or delete the graphics displayed.

"Colors

To set the appearance of the graphics on display.

• "Options"

To set the appearance and some options of the graphic window.

• "+Real"

To display either the real or the theoretical tool path.

• "Speed"

To change the simulation speed.



**CNC 8070** 

## Type of graphics 8.1

The various types of graphics may be line (3D lines, XY, XZ, YZ and Combined) or solid (Sections and 3D solid). Line graphics show the tool path with lines of different colors and solid graphics show an image of the part.

The type of graphics selected will remain active until another type is selected or the graphic display is deactivated or the CNC is turned off. Likewise, when changing the type of graphics, the CNC will maintain the graphic conditions (zoom, graphic parameters, display area, etc.), defined for the last graphic.



When selecting this option, the softkey menu shows the types of graphics available. To return to the main menu, press the key for the previous menu.

## "3D lines" Graphics

It displays a three-dimensional graph of the tool paths.

## "Sections" Graphics

It displays a top view (XY plane) of the part; it shows the machining depths in different tones. It also displays the XZ and YZ sections for the areas shown by the indicators of the top view.

These indicators may be moved around using the  $[\begin{cal} \begin{cal} \beg$ different sections of the part. The CNC shows dynamically the new section being

The indicators may be moved at any time even while executing the program.

## "XY" - "XZ" - "YZ" graphics

It displays the tool paths in the XY, XZ or YZ plane.

## "Combined" graphics

It divides the display area in four quadrants and displays the tool path corresponding to each plane XY, XZ, YZ and to the 3D view.

## "Solid 3D" Graphics

It displays three-dimensional graphics of the machining of the part. Starting out with a 3D block which is "machined" as the program is executed or simulated.

Type of graphics

GRAPHIC ENVIRONMENT (MILL MODEL)



**CNC 8070** 

## 8.2 Zoom

The zoom option may be used to enlarge or reduce the whole graph shown or part of it. This option is not available in the "Combined" type of graphics.

After selecting the "Zoom" option, a zoom frame will appear over the graphics. This frame may be enlarged, reduced and moved around over the graphics already displayed in order to select a particular portion of it to zoom into or out of.

## 

The graphics at the lower right-hand side of the screen shows two figures. The one shown with lines only, indicates the dimensions of the display area and the one with colored sides indicates the portion selected with the zoom.



When selecting this option, the softkey menu shows the available zoom options. To return to the main menu, press the key for the previous menu.

## Zoom "Initial"

Restores the size of the display are set via program or using the "Dimensions" option.

## Zoom "Automatic"

The CNC uses the zoom that it considers best according to the movements programmed.

## Zoom "Previous"

It displays up to two zooms defined earlier. After the second one, it shows again the one defined last.

## Zoom "Limits"

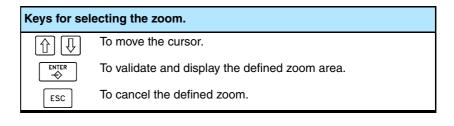
Only for the "Sections" type. In this graphics, zooming is done by moving the indicators that appear framing the graphic sections.

With this option, it is possible to select the axis whose indicator is to be moved.

The indicator may also be selected with the [+] and [-] keys of the numeric keyboard in a rotary fashion (Xmin, Xmax, Ymin, Ymax, Zmin, Zmax).

## Zoom "Edit"

It is used to manually edit the zoom values. It is edited in the dialog area of the graphic window that shows the dimensions of the zoom frame





**CNC 8070** 

## 8.3 **Dimensions**

It is used to define the size of the graphic representation by setting the maximum and minimum coordinates of the graphics on each axis.



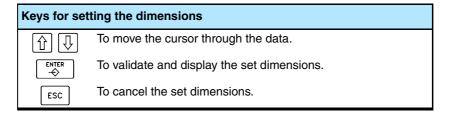
When selecting this option, the softkey menu shows the options available for setting the dimensions. To return to the main menu, press the key for the previous menu.

## **Dimensions "Automatic"**

The CNC sets the dimensions that it considers best according to the movements programmed.

## Dimensions "Edit"

It is used to manually edit the dimension values. It is edited in the graphic window that shows the dimensions of the graphics on each axis.







**CNC 8070** 

Point of view

## 8.4 Point of view

It is used to select the point of view on 3D graphics. This option is only available for the types of graphics "Combined", "3D lines", "Sections" and "Solid 3D".

The orientation of the graphics may be directly selected at the graphic window by orienting the XY plane and the Z axis. The XY plane may be rotated  $360^{\circ}$  and the Z axis  $90^{\circ}$ . The figure at the lower right-hand side of the screen shows the point of view currently selected.

## Keys for selecting the point of view ☐ ☐ ☐ ☐ To orient the XY plane. ☐ ☐ ☐ ☐ To orient the Z axis. ☐ ☐ ☐ To validate and display the selected point of view.

To cancel the selected point point of view.



When selecting this option, the softkey menu shows the options available for selecting the point of view. To return to the main menu, press the key for the previous menu.

## Point of view "Edit"

ESC

It is used to manually edit the orientation of the axes of the graphics. It is edited in the dialog area of the graphic window that shows the current orientation of the axes.

Keys for defining the point of view			
	To move the cursor through the data.		
ENTER 🔷	To validate and display the defined point of view.		
ESC	To cancel the defined point of view.		

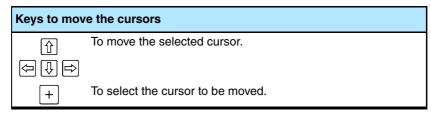


**CNC 8070** 

### 8.5 Measurement

To measure the distance between two points. This option is only available for the types of graphics "XY", "XZ", "YZ" and "Solid 3D".

When selecting this option, the section being measured will appear on the graphics with two cursors and a dashed line. The cursor currently selected will appear in red.



The dialog area will show the coordinates of both cursors, the distance between them on the straight line and the components of that distance on the axes of the active plane. The coordinates of the selected cursor will appear in red.



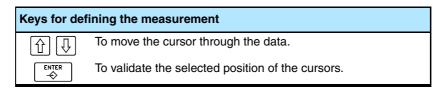
When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

## Measurement "Point1" & "Point2"

It is used to select the cursor to be moved (same as using the [+] key).

## Measurement "Edit"

It is used to manually edit the position of the cursors. It is edited in the dialog area of the graphic window that shows the position of both cursors.







**CNC 8070** 

If a solid graphic type is selected, the graphic representation will be reset and it will return to its initial state without machining.



**CNC 8070** 

## 8.7 Colors

It is used to change the colors used in the graphic representation.

In the line graphics, it is possible to choose the color for each type of tool path;
 the color for rapid movements, paths with tool compensation, etc.

The real coordinates are only available for the execution of the program. The real coordinate is the actual position of the tool which differs from the command coordinate in the amount of following error (axis lag).

• In solid graphics, only the color of each side of the solid may be selected.



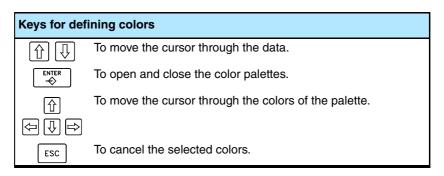
When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

## Colors "Apply"

It assumes the new colors and applies them to the blocks drawn next. If the new colors are not applied, the graphics are drawn with the old colors.

## Colors "Edit"

It is used to select the new colors for the graphics. They are selected in the dialog area of the graphic window that show the current colors.



8.

Colors

FAGOR

CNC 8070

## 8.8 Options

It is used to set the appearance and some functions of the graphic window. These options may be used at any time, even while executing a program.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

## Option "Activate"

This softkey may be used to activate the graphics. The softkey appears pressed when this option is enabled. The status of this softkey cannot be modified while executing or simulating a program.

When the graphic representation is deactivated and activated, the current graphic is erased; but the display conditions are kept active (type of graphics, zoom, graphic parameters and display area) that were active before that mode was deactivated.

## Option "Simple"

This softkey shows the single window for graphics. The softkey appears pressed when this option is enabled.

The single window hides the dialog and data areas on the right-hand side of the graphic window so the drawing occupies the whole graphic window.

## **Option "Hide Tool"**

This softkey is used to hide or show the tool while simulating in "3D solid" graphics. The softkey appears pressed when this option is enabled.

Lathe tools are never displayed regardless of the status of this softkey. When it is a lathe tool, this softkey may be activated to hide the tool, but it cannot be deactivated to show the tool.

## Option "Print"

This softkey may be used to print the graphics in the pre-determined printer or save it as a file (bmp format) at the CNC. When selecting the "File" option, it will be saved in the folder "C:\Cnc8070\Users\Reports\"; the file name may be selected using the "Print configuration" softkey.

When selecting this option, the CNC will show a dialog box requesting the print destination (printer or file). After selecting the destination, press [ENTER] to print it or [ESC] to cancel it.

## Option "Print setup"

It is used to set the printing configuration. When selecting this option, the CNC shows a dialog box where the following may be defined:

- The title of the graphics that will appear next to it in the print.
- The name of the file where the graphics will be stored when printing out to a file.

After filling out the data, press [ENTER] to accept them or [ESC] to cancel them.



**CNC 8070** 

It is used to draw the real tool path or the theoretical tool path. This option is only available in line type graphics.

This option is only available when executing the program; not when simulating it. When selecting this option (the softkey will appear pressed), the CNC draws the actual (real) tool path.

8.

GRAPHIC ENVIRONMENT (MILL MODEL)
Real coordinates

FAGOR

**CNC 8070** 

## Simulation speed 8.10

To change the simulation speed.

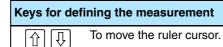


When selecting this option, the softkey menu shows the options to set the simulation speed. To return to the main menu, press the key for the previous menu.

This option is only available when simulating the program; not when exeucting it.

## Speed "Edit"

It is used to select the new simulation speed. It is selected using the graduated ruler that indicates the active simulation speed.



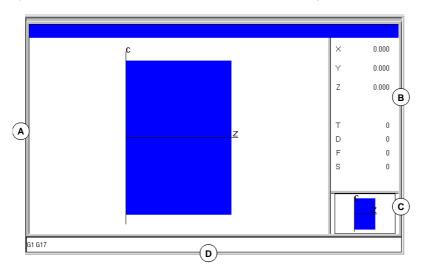
Simulation speed **GRAPHIC ENVIRONMENT (MILL MODEL)** 



**CNC 8070** 

## GRAPHIC ENVIRONMENT (LATHE MODEL)

The graphic environment displays a graphic representation of the program that is being executed or simulated and take measurements on the graphics.



## A. Display area.

Shows a graphic representation of the tool paths or of the part as the program is being executed or simulated.

## B. Dialog area.

Shows the position of the axes and the machining conditions active in the execution or simulation (feedrate, spindle speed, tool and tool offset).

When selecting specific options of the softkey menu, it shows the information associated with that option which may be defined by the user.

## C. Information area.

Shows the work plane appearing in the display area and an illustration representing the size of the graph and the portion of the graphic area selected with the zoom.

**D.** This area only appears during simulation and shows the "G" functions active in the simulation.



**CNC 8070** 

## Softkey menus of the graphics window

When selecting the graphic window, the horizontal softkey menu shows the options associated with that window. The set of options offered by this menu may be changed using the "+" softkey which offers a new set of options.

The options available in the softkey menu are the following.

• "Type of graphics"

To select the type of graphics.

• "ZOOM"

To enlarge or reduce the whole graph shown or part of it.

• "Dimensions"

To define the size of the graphic representation.

• "Measurement"

To measure the distance between two points.

• "Clear screen"

It is used to clear the screen or delete the graphics displayed.

• "Colors"

To set the appearance of the graphics on display.

• "Options"

To set the appearance and some options of the graphic window.

• "+Real"

To display either the real or the theoretical tool path.

• "Speed"

To change the simulation speed.



**CNC 8070** 

## Type of graphics 9.1

The various types of graphics may be grouped into line graphics (XZ, XC. ZC and combined) and solid graphics (Solid XZ, Solid XC, Solid ZC). Line graphics show the tool path with lines of different colors and solid graphics show an image of the part.

The type of graphics selected will remain active until another type is selected or the graphic display is deactivated or the CNC is turned off. Likewise, when changing the type of graphics, the CNC will maintain the graphic conditions (zoom, graphic parameters, display area, etc.), defined for the last graphic.



When selecting this option, the softkey menu shows the types of graphics available. To return to the main menu, press the key for the previous menu.

## "XZ" - "XC" - "ZC" graphics

It displays the tool paths in the XZ, XC or ZC plane.

## "Combined" graphics

It divides the display area in four quadrants and displays the tool path corresponding to each plane planos XZ, XC or ZC.

## "Solid 3D" Graphics

It displays a graphic of the machining of the part. Starting out with a solid block which is "machined" as the program is executed or simulated.



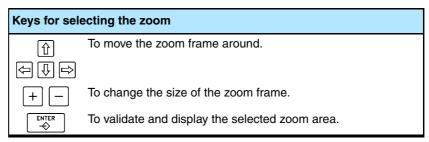


**CNC 8070** 

## 9.2 **Zoom**

The zoom option may be used to enlarge or reduce the whole graph shown or part of it. This option is not available in the "Combined" type of graphics.

After selecting the "Zoom" option, a zoom frame will appear over the graphics. This frame may be enlarged, reduced and moved around over the graphics already displayed in order to select a particular portion of it to zoom into or out of.



A window is displayed on the graphics at the lower right-hand side of the screen. This window indicates the graphic area selected with the zoom.



When selecting this option, the softkey menu shows the available zoom options. To return to the main menu, press the key for the previous menu.

## Zoom "Initial"

Restores the size of the display are set via program or using the "Dimensions" option.

## Zoom "Automatic"

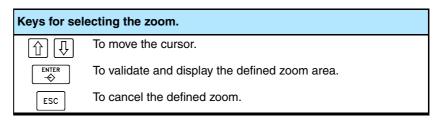
The CNC uses the zoom that it considers best according to the movements programmed.

## Zoom "Previous"

It displays up to two zooms defined earlier. After the second one, it shows again the one defined last.

## Zoom "Edit"

It is used to manually edit the zoom values. It is edited in the dialog area of the graphic window that shows the dimensions of the zoom frame





**CNC 8070** 

## 9.3 Dimensions

It is used to define the size of the graphic representation by setting the inside and outside diameters of the part and the maximum and minimum coordinates of the graphics on the longitudinal axis.



When selecting this option, the softkey menu shows the options available for setting the dimensions. To return to the main menu, press the key for the previous menu.

## **Dimensions "Automatic"**

The CNC sets the dimensions that it considers best according to the movements programmed.

## Dimensions "Edit"

It is used to manually edit the dimension values. It is edited in the dialog area of the graphic window that shows the dimensions of the graphics.

## 



.ATHE MODEL)

Dimensions



**CNC 8070** 

Measurement

## 9.4 Measurement

To measure the distance between two points. This option is not available in the "Combined" type of graphics.

When selecting the "Measuring" option, the section being measured will appear on the graphics with two cursors and a dashed line. The cursor currently selected will appear in red.

# Keys to move the cursors To move the selected cursor. □ □ □ To select the cursor to be moved.

The dialog area will show the coordinates of both cursors, the distance between them on the straight line and the components of that distance on the axes of the active plane. The coordinates of the selected cursor will appear in red.



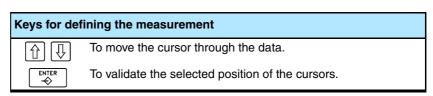
When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

## Measurement "Point1" & "Point2"

It is used to select the cursor to be moved (same as using the [+] key).

## Measurement "Edit"

It is used to manually edit the position of the cursors. It is edited in the dialog area of the graphic window that shows the position of both cursors.





**CNC 8070** 

## 9.5 Clear screen

It is used to clear the screen or delete the graphics displayed.

If a solid graphic type is selected, the graphic representation will be reset and it will return to its initial state without machining.

9.

Clear screen



**CNC 8070** 

## 9.6 Colors

It is used to change the colors used in the graphic representation.

- In the line graphics, it is possible to choose the color for each type of tool path; the color for rapid movements, paths with tool compensation, etc.
  - The real coordinates are only available for the execution of the program. The real coordinate is the actual position of the tool which differs from the command coordinate in the amount of following error (axis lag).
- In solid graphics, it is possible to choose the color of the solid, of the tool, etc.



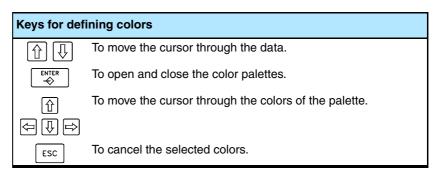
When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

## Colors "Apply"

It assumes the new colors and applies them to the blocks drawn next. If the new colors are not applied, the graphics are drawn with the old colors.

## Colors "Edit"

It is used to select the new colors for the graphics. They are selected in the dialog area of the graphic window that show the current colors.





**CNC 8070** 

## 9.7 **Options**

It is used to set the appearance and some functions of the graphic window. These options may be used at any time, even while executing a program.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

## Option "Activate"

This softkey may be used to activate the graphics. The softkey appears pressed when this option is enabled. The status of this softkey cannot be modified while executing or simulating a program.

When the graphic representation is deactivated and activated, the current graphic is erased; but the display conditions are kept active (type of graphics, zoom, graphic parameters and display area) that were active before that mode was deactivated.

## Option "Simple"

This softkey shows the single window for graphics. The softkey appears pressed when this option is enabled.

The single window hides the dialog and data areas on the right-hand side of the graphic window so the drawing occupies the whole graphic window.

## Option "Lines"

This softkey hides the solid part of the graphics and only shows the tool paths. The softkey appears pressed when this option is enabled.

## Option "Print"

This softkey may be used to print the graphics in the pre-determined printer or save it as a file (bmp format) at the CNC. When selecting the "File" option, it will be saved in the folder "C:\Cnc8070\Users\Reports\"; the file name may be selected using the "Print configuration" softkey.

When selecting this option, the CNC will show a dialog box requesting the print destination (printer or file). After selecting the destination, press [ENTER] to print it or [ESC] to cancel it.

## Option "Print setup"

It is used to set the printing configuration. When selecting this option, the CNC shows a dialog box where the following may be defined:

- The title of the graphics that will appear next to it in the print.
- The name of the file where the graphics will be stored when printing out to a file.

After filling out the data, press [ENTER] to accept them or [ESC] to cancel them.



**GRAPHIC ENVIRONMENT (LATHE MODEL)** 



**CNC 8070** 

GRAPHIC ENVIRONMENT (LATHE MODEL)

Real coordinates



**CNC 8070** 

## 9.9 Simulation speed

To change the simulation speed.

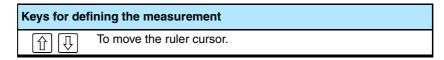


When selecting this option, the softkey menu shows the options to set the simulation speed. To return to the main menu, press the key for the previous menu.

This option is only available when simulating the program; not when exeucting it.

## Speed "Edit"

It is used to select the new simulation speed. It is selected using the graduated ruler that indicates the active simulation speed.





Simulation speed



**CNC 8070** 



**CNC 8070** 

## **MDI MODE**

10

## MDI mode selection



The MDI mode may be accessed from any other operating mode, except when a program is in execution (even it is interrupted) by pressing its relevant key.

This work mode may be accessed from any channel. The blocks are executed in the active channel.

The MDI mode lays over all the other work modes in such a way that when quitting the MDI mode by pressing [ESC], the CNC goes into the work mode from where the MDI mode was accessed.

## Operations in MDI mode

The following operations are possible in this work mode:

- · Edit and execute individual blocks.
- Save the set of blocks executed as an independent program.



**CNC 8070** 

## 10.1 Appearance of the MDI mode

When this operating mode is active, it will be indicated at the top right side of the general status bar.

This mode may be displayed with several screens. The current screen and the total number of available screens are shown at the upper right hand side of the general status bar.

Switching between the different screens is done by pressing the MDI mode accessing key. The screen selection is rotary in such a way that when pressing this key on the last screen, it shows the first one of the operating mode.

## Standard configuration

The standard configuration of this work mode, described in this manual corresponds to the one supplied by Fagor. In some cases, the manual shows a sample screen, instead of that of the CNC in order to make it easier to understand.

These screens may be customized by the machine manufacturer changing its contents as well as its appearance. They may also remove certain screens or add some of their own.



MDI MODE
Appearance of the MDI mode



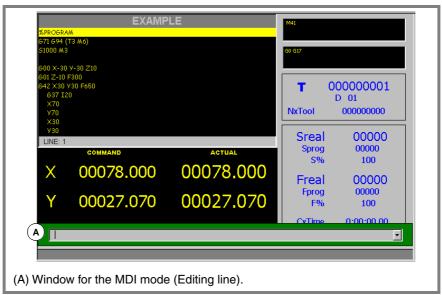
**CNC 8070** 

## 10.1.1 Window description

As mentioned earlier, the MDI mode may be displayed with two different windows (later sections of this chapter describe how to use them).

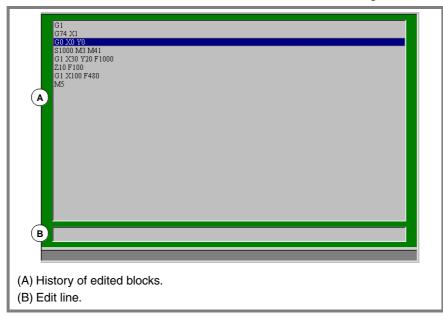
## Standard window

The standard window only displays one edit line for editing new blocks.



## Full screen

The full screen shows the blocks saved so far and an edit line for editing new blocks.



10.

Appearance of the MDI mode



**CNC 8070** 

MDI MODE Standard MDI window

## 10.2 Standard MDI window

This window only shows the edit line of the MDI mode.

It is possible to edit and execute new blocks or recover blocks saved in the block history, which may be modified before executing them.

## **Description**

The standard MDI mode window shows the following information:



A. Window for the MDI mode (Editing line).

Window where the blocks to be executed are edited. Blocks are edited one by one.



**CNC 8070** 

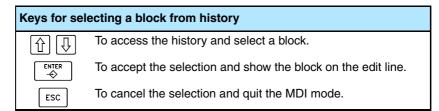
## 10.2.1 Block editing and execution

In this window, it is possible to edit and execute new blocks or the ones saved in the block history. Only the blocks previously executed will be saved in the block history.

While editing, it analyzes the syntax of the block being edited. When trying to execute, if the block is incorrect, it shows a warning message and it does not execute it.

## **Block editing**

The blocks may be edited directly on the edit line or may be recalled from the block history.



The recalled blocks may be modified before executing them as if they were new ones.

## **Block execution**

The block on the edit line is executed by pressing [START] at the operator panel. Once the block has been executed is saved in the block history. The block being either in execution or interrupted, the [ESC] key may be used to hide the MDI mode without canceling the execution.

The [STOP] key interrupts the execution of the block. Press [START] again to resume execution from where it was interrupted.



Being the execution interrupted, the CNC shows the "CANCEL" softkey that may be used to cancel the execution of the block while keeping the programmed machining conditions. This softkey cancels the execution of the block without doing a general reset of the CNC. Once the block execution has been canceled, it is added to the block history.

The [RESET] key cancels the execution of the block and resets the CNC to its initial conditions.

10.

MDI MODE Standard MDI window



**CNC 8070** 

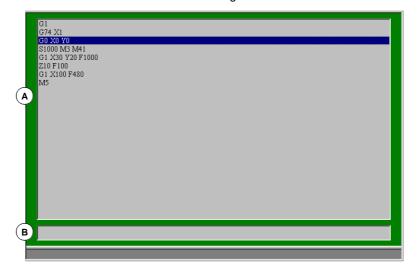
## 10.3 Full MDI screen

This window shows the blocks saved so far and the edit line of the MDI mode.

It is possible to edit and execute new blocks or recover blocks saved in the block history, which may be modified before executing them.

## **Description**

The full MDI mode screen shows the following information:



A. History of edited blocks.

The block history shows a list of the blocks edited in MDI mode. Every time a new block is edited, it is added to this history.

B. Edit line.

The blocks to be executed are edited on the edit line. Blocks are edited one by one.

## Softkey menu

When selecting the full screen, the horizontal softkeys show the options related to this window.

The options available in the softkeys menu are:

"New block"

It positions the cursor on the edit line.

"Modify

Recovers the block selected by the cursor and it displays it on the edit line.

This option is the same as pressing [ENTER].

• "Delete"

It deletes the block selected by the cursor.

"Cancel edit"

It cancels the editing of the block and it shows an empty edit line.

This option is only available when editing a block.

• "Delete all"

It deletes all the blocks from the block history.

• "Save"

It saves all the history blocks as an independent program.

QW = U



**CNC 8070** 

### 10.3.1 Block execution

In this window, it is possible to edit and execute new blocks or the ones saved in the block history.

### **Block editing**

The blocks may be edited directly on the edit line or may be recalled from the block history.

### 

The recalled blocks may be modified before executing them as if they were new ones.

The blocks edited or modified appearing on the edit line may be executed or saved in the block history.

### Saving a block

Press [ENTER] to save a block in the history without executing it.

If it is a new block, it will be added to the history whereas if it is modified block, it will replace the previous one in the history.

### **Block execution**

The block on the edit line is executed by pressing [START] at the operator panel. Once the block has been executed is saved in the block history. The block being either in execution or interrupted, the [ESC] key may be used to hide the MDI mode without canceling the execution.

The [STOP] key interrupts the execution of the block. Press [START] again to resume execution from where it was interrupted.



Being the execution interrupted, the CNC shows the "CANCEL" softkey that may be used to cancel the execution of the block while keeping the programmed machining conditions. This softkey cancels the execution of the block without doing a general reset of the CNC. Once the block execution has been canceled, it is added to the block history.

The [RESET] key cancels the execution of the block and resets the CNC to its initial conditions.

10.

MDI MODE Full MDI screen



**CNC 8070** 

### 10.3.2 Save the blocks as a program

To save the block history as an independent program, press the "Save" softkey and the CNC will show a list of the programs saved at the CNC.

### Keys for selecting a block from history To switch windows. To move the cursor thru the elements of the window. ENTER To save the new program. ESC To cancel the selection and close the program listing.

To save the program:

- **1.** Write the program name in the bottom window. To replace an existing program, select it from the list.
- 2. Press [ENTER] to save the program or [ESC] to return to the MDI mode without saving the program.



MDI MODE Full MDI screen



**CNC 8070** 

### **USER TABLES**

11

### User table selection



The CNC tables may be accessed from any other operating mode, even while a program is in execution, by pressing the key for these tables.

The user tables consist of the following tables:

- · Zero offset table.
- · Clamp tables (fixtures).
- Table of arithmetic parameters, global, local and common.

In order to activate a fixture zero or part zero offset, those values must be previously stored in the relevant CNC table.

### What can you do in this work mode?

The following operations are possible in this work mode:

- Edit and modify the tables for zero offsets and parameters.
- Save the content of a table.
- Recover the content of a table.
- Print the content of a table.
- Initialize a table.



**CNC 8070** 

### 11.1 Appearance of the table mode

This operating mode consists of several tables. The various tables may be selected using the horizontal softkeys.

Some tables are common to all the channels and others belong to each channel. In this case, by default, they show the ones of the active channel; but it is possible to access those of any other channel from the vertical softkey menu.

The zero offset tables and fixture offset tables are common to all the channels; however, in each channel they show the axes of that channel. When applying an offset in a channel, it is only applied to the axes that are part of the channel at the time.

The tables available are:

- · Zero offset table.
- · Fixture table.
- Table of global parameters. There is a table for each channel.
- Table of local parameters. There are seven tables for each channel, one table per level.
- Table of common parameters. The table is common to all the channels.

### Standard configuration

The standard configuration of this work mode, described in this manual corresponds to the one supplied by Fagor. In some cases, the manual shows a sample screen, instead of that of the CNC in order to make it easier to understand.



USEK I ABLES
Appearance of the table mode



**CNC 8070** 

### 11.1.1 Icon description (vertical softkeys)

The icons are activated with their associated softkey (F8 to F12).



The icon menu offers all the options for the selected table. The group of icons of this menu may be expanded with the softkey associated with the "+" icon that offers a new set of icons.

### List of icons.



### Display units (millimeters/inches)

To toggle the units for the position of the linear axes. Toggling these units does not affect the rotary axes which will always be displayed in degrees.

The icon highlights the units currently selected (millimeters or inches).

It must be borne in mind that the unit change is only valid for display and data entry. A program assumes the units defined with the active function "G70" or "G71", or, when not programmed, the units set by the machine manufacturer.



### Initialize table

Resets all the table data to "0". The CNC will request confirmation of the command.



### Find text

It is used to find a text or a value in the table.

When selecting this option, the CNC shows a dialog box requesting the text to be



### Accessing the tables of other channels

(This icon will only be available when using channels).

Some tables only show the data of the active channel. This icon gives access to the tables of the other channels.



### Select axes

It may be used to select the axes to be displayed in the tables.

When using several channels, only those axes assigned to the active channel may be accessed.



### Save tool

It saves the values of the table into a file. This file may be saved at the CNC, in a floppy disk or at another CNC (or PC) connected through Ethernet.



### Recall table

It restores the values of the table previously saved into a file.



### **Print table**

This softkey may be used to print the table in the pre-determined printer or save it as a file (prn format) at the CNC.



**FAGOR** 

**CNC 8070** 

### 11.2 Zero offset tables

This table contains the absolute zero offsets and the PLC offset of all the axes and spindles that may be activated as C axis. The zero offsets associated with the possible C axes are always visible, even when the C axis is not active.



Although the table is common to all the channels, when accessing from a particular channel, it only shows the axes and spindles of that channel. The offsets of other channels may be accessed from the vertical softkey menu.

The table shows the axes and spindles that are in the channel at the time; in other words, after swapping axes or spindles, the CNC updates the table.



- A. Machine axes and display units (millimeters, inches or degrees).
- B. Zero offsets.
- C. Zero offset value in each axes of the channel.

The end of this chapter describes how to edit this table.

### **PLC** offset

It cannot be directly defined in the table. The values are set from the PLC or from the part-program using high-level variables.

It is used, for example, to correct deviations originated by machine dilatations.

The CNC always adds the PLC offset to the selected zero offset.

### Absolute zero offsets

Besides setting the values directly in the table, they may also be set from the PLC or via part-program using high-level variables.

The zero offsets are used to place the part zero at different positions of the machine. When applying a zero offset, the CNC assumes as the new part zero the point set by the selected zero offset referred to the clamp zero (if active).

To apply an absolute zero offset, it must be activated via program using the relevant function.



USER TABLES
Zero offset tables

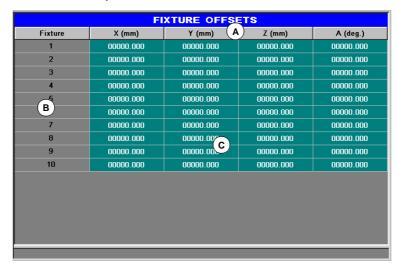


**CNC 8070** 

### 11.3 Fixture table

This table stores the clamp offsets for each axis. There are up to 10 clamp offsets.

Although the table is common to all the channels, when accessing from a channel, it only shows the axes of that channel. The offsets of other channels may be accessed from the vertical softkey menu.



- A. Machine axes and display units (millimeters, inches or degrees).
- B. Clamp offset.
- C. Clamp offset value in each axes of the channel.

The end of this chapter describes how to edit this table.

### **Fixture offset**

Besides setting the values directly in the table, they may also be set from the PLC or via part-program using high-level variables.

The clamp offsets are used to set the position of the clamping system of the machine. When applying a clamp offset, the CNC assumes as new clamp zero the point set by the selected offset referred to machine reference zero (home).

To apply a clamp offset it must be activated from the program using the relevant variable.



**USER TABLES**Fixture table



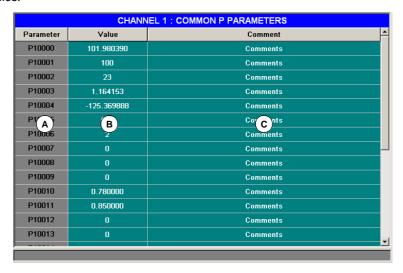
**CNC 8070** 

### 11.4 Arithmetic parameter tables

There are the following arithmetic parameter tables:

- · Common parameters.
- Global parameters. There is a table for each channel.
- Local parameters. There are seven tables for each channel, one table per nesting level (7 levels).

The CNC generates a new nesting level for local parameters every time parameters are assigned to a subroutine. The end of this chapter describes how to edit these tables.



- A. Parameter list.
- B. Parameter value.
- C. Parameter describing comment (only in the common-parameters table).

This field offers the possibility to associate a short description with the M function. This field is for information only; it is not used by the CNC.

The comments are saved in the file UCPComments.txt and it is possible to have one file per language. These files are saved in the folder "../MTB /data /Lang".

### **Arithmetic parameters**

The OEM defines the range of local and global parameters up to a maximum of 100 local parameters (P0-P99) and 9900 global parameters (P100-P9999).

When the local parameters are used in a subroutine calling block, they can also be referred to by the letters A-Z (except " $\tilde{N}$ ") in such a way that "A" is the same as P0 and "Z" is the same as P25. That is why the local parameter tables show the parameter number next to their associated letter.

The parameter values may be set directly in the table or from the PLC or via partprogram. In this case, the table values are updated after carrying out the operations indicated in the block being executed.

The parameter values may be displayed either in decimal notation (6475.873) or scientific (0.654E-3).



USER TABLES
Arithmetic parameter tables

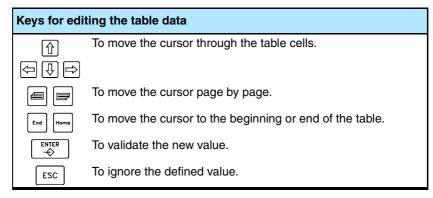


**CNC 8070** 

### 11.5 Operations with tables

### 11.5.1 Data editing

Select the desired table using the softkey menu:



To edit the table data, proceed as follows:

- 1. Use the cursor to select the cell whose value is to be changed.
- 2. Key in the new value.
- **3.** Press [ENTER] to accept the new value or [ESC] to ignore the new value and recover the previous one.

11.

**USER TABLES**Operations with tables



**CNC 8070** 

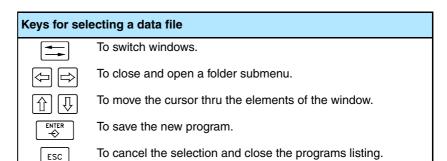
### 11.5.2 Save and recall tables

### Saving a table

It may be used to save the table data, in ASCII format, in a file.



After selecting the table whose data is to be saved, press the "Save" icon and the CNC shows a list with the tables that are already saved.



To save the table data, proceed as follows:

- 1. Select the destination folder.
- 2. Define the file name at the bottom window. To replace an existing file, select it from the list.
- 3. Press [ENTER] to save the file or [ESC] to return to the table without saving the file.

Depending on the table being saved, the CNC will assign one of the following extensions to the file:

Extension	Table type
*.UPO	Zero offset table.
*.UPF	Fixture table.
*.UPP	Parameter tables.

### Recall a table

It may be used to restore the table data, in ASCII format, from a file.



After selecting the table whose data is to be restored, press the "Load" icon and the CNC shows a list with the tables that are already saved.

To recover the table data, proceed as follows:

- 1. Select the folder containing the file.
- 2. Select the file from the list or write its name in the bottom window.
- 3. Press [ENTER] to recall the data from the file or [ESC] to return to the table without recalling the data.



USER TABLES
Operations with tables



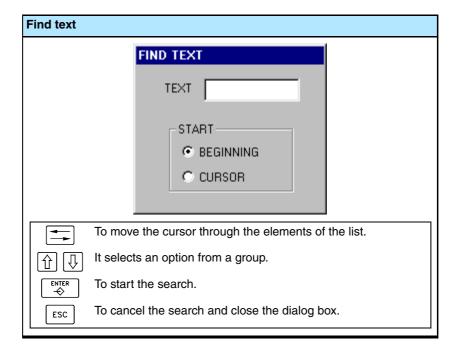
**CNC 8070** 

### 11.5.3 Find text

It is used to find text or a value in the table.



To perform the search, press the "Find" icon and the CNC will show a dialog box requesting the text or value to be searched. It is also possible to select whether the search must start at the beginning of the table or at the current cursor position.



11.

**USER TABLES**Operations with tables



CNC 8070



USER TABLES
Operations with tables



CNC 8070

### TOOL AND MAGAZINE TABLE

### Selecting the tool tables and magazine tables.



The CNC tables may be accessed from any other operating mode, even while a program is in execution, by pressing the key for these tables.

They may be divided into the following tables:

- · Tool table.
- · Active-tools table.
- Table for the status of the tool change process.
- · Tool magazine tables.

In order to load a tool in the magazine or in the spindle, that tool must have been previously defined in the corresponding table of the CNC.

### What can you do in this work mode?

The following operations are possible in this work mode:

- · Editing and modifying the tool table
- Display and manage the tool arrangement in the magazines and in the tool changing arms (if any).
- Display the information about the tool change process; the operation carried out when executing an M06, manager status, change status (in execution or at rest), magazine involved in the change (if the change is taking place) and whether the change process is or not in an error state.
- Load and unload a tool from the magazines through the spindle, using a maneuver.
- Saving the table contents.
- Recalling the table contents.
- · Printing the table contents.
- Initializing the tables.



**CNC 8070** 

### 12.1 Showing tables and common operations

### 12.1.1 Table selection

This operating mode consists of several tables. The various tables may be selected using the horizontal softkeys.

Softkey	Table
T	Tool table.
	This table defines the tools available and the data associated with each one of them. This table is common to all the magazines.
	Active-tools table.
	This table shows the tool that is active in each channel and the data associated with it.
PR	Change process.
	This table monitors the tool changes being executed in each channel.
Mach Mz	Tool magazine table (there is one table per magazine)
MZ MZ	For each magazine, it shows the tool distribution and the remaining life time of each tool (if tool life monitoring is active). If the magazine has a changer arm, it shows the tool located in it.
	The description of the magazine type can also be shown.
	The icon associated with this table depends on the software configuration (lathe or mill).

When selecting one of these tables, it is displayed on the screen and the vertical icon menu shows the icons associated with that table. Later sections of this chapter show a more detailed description of the icons and operations that may be carried out in each table.



Pressing the key for the previous menu deselects the active table. When no table has been selected, the vertical icon menu may be used to save and restore all the tables.

### Simultaneous operations for all the tables

If no table is currently selected, the vertical softkey menu will show the following softkeys to save or load all the tables. See "12.1.3 Save and load the tables" on page 194.



### Save the tables

Saves the data of all the tables.



### Recall the tables

It restores the data previously saved for all the tables.



**CNC 8070** 

# Showing tables and common operations

### 12.1.2 Search for a text in the tables

It is possible to search for a text or a value in the list of tools and magazine positions. The search is carried out from the vertical softkey menu.



This icon starts the search. Once the icon has been pressed, the CNC will display a dialog box to define the search criteria. The defined criteria is maintained until a new one is defined.



This icon is shown when a search criteria has been defined; it makes it possible to search for the next match using the current search criteria.

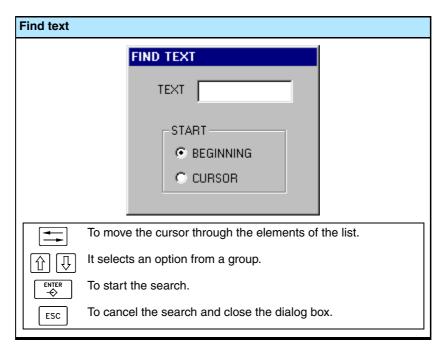
To start or resume the search, press [ENTER] and the focus will position on the first match found. Every time one of the icons is pressed, it offers the option to search for the next match or to define a new search.

The search may be canceled by pressing the [ESC] key.

### Define the search criteria.

Every time one of these icons is pressed, it shows a dialog box to define the search criteria. The following may be defined in this dialog box:

- · The text or value to search for.
- The beginning point of the search, namely either from the beginning of the table or from the cursor position.



12.

**TOOL AND MAGAZINE TABLE** 



**CNC 8070** 

### 12.1.3 Save and load the tables

From the vertical softkey menu, it is possible to make a backup copy of the table data (recommended). With these files, it is possible to recover the table data when needed.

These tables are saved in the an ASCII file. These files may be saved at the CNC, in a floppy disk or at another device (CNC, PC, etc.) connected through Ethernet. By default, they are saved in the folder "C:\CNC8070\ MTB\ DATA" or in the last folder selected by the user.

These tables are saved in the following files. The table monitoring the change processes is informative only; it is not copied anywhere.

Table	File
Tool table.	Tools.TPT
Active-tools table.	Toolsact.TPA
Tool magazine table (if there is only one magazine).	Magazine.TPM
Tool magazine table (if there are several magazines).	Magazine1.TPM
	• • •
	Magazine4.TPM

### Considerations for saving and loading tables.

Although each table may be saved separately, it is recommended to always have a copy of all the tables. Also, the following must be borne in mind when loading the tables:

- Loading the tool table initializes the magazine tables and the active-tools table.
   When changing the list of available tools, it may not be coherent with the tool distribution in the magazine or in the spindles. This is why after loading this table it is necessary to define (or load) the magazine tables and the active-tools table, if any, in that order.
- Loading the magazine table initializes the active-tools table. This is because when
  loading the magazine tables, the new tool arrangement may not be coherent with
  the active tools. This is why after loading this table, it is necessary to load the
  active-tools table.

### Save the tables

The tables may be saved one by one or all of them simultaneously. In either case, the action is carried out from the vertical softkey menu using one of the following icons.



Saving all the tables at the same time.

To save all the tables at the same time, no table must be selected. See *"12.1.1 Table selection"* on page 192.



Saving each table separately.

To save the tables one by one, select each table from the horizontal softkey menu.

After pressing the icon, the CNC will ask where to save the data files. Select the desired folder and press [ENTER]. The selection process may be canceled by pressing the [ESC] key.



**CNC 8070** 

### Recall the tables

It may be used to restore the data, in ASCII format, of the tables from the files. The tables may be loaded one by one or all of them simultaneously. In either case, the action is carried out from the vertical softkey menu using one of the following icons.



Loading all the tables at the same time.

To load all the tables at the same time, no table must be selected. See "12.1.1 Table selection" on page 192.



Loading each table separately.

To load the tables one by one, select each table from the horizontal softkey menu. To load the data of all the tables (recommended), follow a particular loading order to guarantee data consistency.

After pressing the icon, the CNC will ask the location of the data files. Select the desired folder and press [ENTER]. The selection process may be canceled by pressing the [ESC] key.

### Order (sequence) to follow when loading the different

As mentioned earlier, there are two ways to load the table data from the ASCII files; loading all of them at the same time or selecting each table and loading its data.

### Loading all the tables at the same time.

In this case, it is up to the CNC to set the order (sequence) used to load the data.

### Order (sequence) to follow when loading the tables one by one.

This way, each table is selected and its data loaded. In this case, the following sequence must be followed when loading the tables.

1. First load the tool table.

This table defines the system tools. Loading the tool table initializes the tables of the magazines and the active-tools table.

2. Second, load the tool magazine tables.

Since the tool list has changed, if the magazine tables are not loaded, it is not possible to assure that the new tools are the ones located in the magazine tool or in the active-tools table.

3. Finally, load the active-tools table.

When loading the tables of magazines, it is unsure that the current tool positions will be compatible with the active tools.

**FAGOF** 

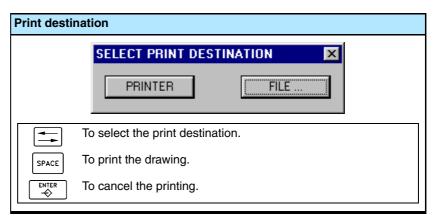
**CNC 8070** 

Some of the tables may be printed in a printer accessible from the CNC or as a file (PRN format). When the tables are saved as a file, it may be saved at the CNC, in a floppy disk or at any other device (CNC, PC, etc.) connected through Ethernet. By default, the files are saved in the folder "C:\CNC8070\ USERS\ Reports".

In either case, the action is carried out from the vertical softkey menu using the following icon.



This icon starts printing. When pressing this icon, the CNC will show a dialog box requesting the print destination for the table (printer or file).



After selecting the target, press [ENTER] to start printing. Press [ESC] to cancel the selection.



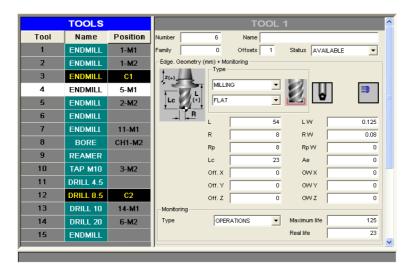
**CNC 8070** 

### 12.2 **Tool table**

This table defines the tools available and the data associated with each one of them. The tool list is common to the whole system, i.e. is common to all the available magazines. Once the tools have been defined, they may be distributed in the various magazines.

-

The table is divided in two panels. To switch panels, press the panel-change key.



The left panel shows the list of available tools. For each tool, it indicates the position and the magazine where it is located, whether it is a ground tool or the tool is active in a channel.

The right panel shows the data of the tool selected on the list. From the vertical softkey menu, it is possible to select the data shown on the right panel. See "12.2.3 Description of the tool data" on page 201.

### About the tools

Each tool is identified by its number, that is unique for the whole system; it cannot be repeated in different magazines nor in ground tools. This table indicates the position and the magazine where each tool is located, whether it is a ground tool or the tool is active in a channel.

### **Ground tools**

A ground tool is a tool that is not stored in any magazine and is loaded manually when requested. Ground tools are also defined in the tool table, but they are not associated with any magazine position.

Ground tool loading and unloading is global to the system; it is not associated with any particular magazine or channel.

TOOL AND MAGAZINE TABLE Tool table



**CNC 8070** 

### 12.2.1 Description of the icons of the vertical softkey menu

The options of the vertical softkey menu depend on which table is active.



The vertical softkey menu shows all the options for the selected table. The group of options shown on this menu may be expanded with the softkey associated with the "+" icon that offers a different set of options.

The options shown for the tool table are. Bear in mind that the table is divided into two panels. There are options that are valid for both panels and options that are only available on one of them.



### Display units (millimeters/inches)

It is used to change the units for data display. The change of units is only valid for display and data entry. For programming, the CNC assumes the units set by the active function G70 or G71.

The icon highlights the units currently selected (millimeters or inches).



### Find text

To search for a text on a tool list. See "12.1.2 Search for a text in the tables" on page 193.



### Add tool

(This icon is only available for the tool list).

It adds a new tool to the list.



### Delete tool

(This icon is only available for the tool list).

It removes a tool from the list. A tool cannot be deleted if it is in the tool magazine.



### Delete data

(This icon is only available for tool data).

It deletes the data where the cursor is. When deleting a data, it assumes its default value.



### Configuring the tool table

(This icon is only available for tool data).

It is used to configure the data shown in the tool table.



### Initializing the table

It initializes the tool table. The CNC will request confirmation of the command.

Initializing the tables eliminates all the tools from the list. It also initializes the active-tools table and the magazine tables because the available tools have been erased.



### Save table

It saves the table data in a file. See "12.1.3 Save and load the tables" on page 194.



### Recall table

It recalls the table data previously saved in a file. See "12.1.3 Save and load the tables" on page 194.

Bear in mind that loading the tool table initializes the magazine tables and the active-tools table.



CNC 8070

### Print table

This softkey may be used to print the table in the pre-determined printer or save it as a file (prn format) at the CNC. See "12.1.4 Printing the tables" on page 196.



### Copy offset data

It copies the data of the offset being displayed onto the clipboard. The data saved may be pasted to a new offset.



### Paste offset data

It pastes the offset data that is saved on the clipboard.

12.

TOOL AND MAGAZINE TABLE

Tool table



**CNC 8070** 

### 12.2.2 The tool list

The tool list appears on the left panel of the tool table. The list shows the available tools and their position. The CNC updates data of the list every time a tool change is carried out.

### **Tool number**

It is assigned automatically when the tool is added to the list. It may be modified by the user in the data window.

### **Tool name**

Name identifying the tool defined by the user in the data window. It may be edited directly on the list.

### **Tool position**

It indicates the position of the tool, in a magazine, in the spindle or in the claws of the tool changer arm.

C1-C4 It is in one of the spindles.

M1-M4 It is in one of the magazines. In this case, it also indicates its

magazine position.

CH1-CH2 It is in the holders of the tool changer arm.

If none of these positions is indicated, it means that it is a ground tool. Ground tools are not stored in the magazine and are loaded manually when requested.

### How to move around on the tool list

Keys to select a tool from the list		
	To scroll list up and down line by line.	
	To move the cursor page by page through the list.	
End Home	To move the cursor to the beginning or end of the list.	
ENTER	To access the data of the selected tool.	



**CNC 8070** 

### 12.2.3 Description of the tool data

The tool data appears on the right panel of the tool table. It shows the data of the tool selected on the list.

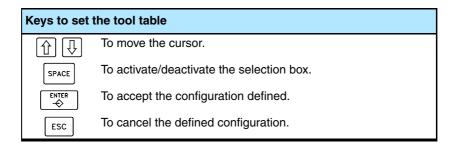
This data must be defined by the user. When using tool life monitoring, the CNC is in charge of updating the value of the actual (real) life.

### Configuring the tool data display

The data being shown may be configured from the vertical softkey menu.



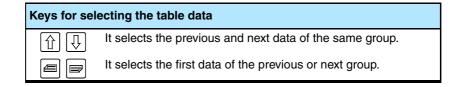
When pressing this icon, the CNC shows a dialog box for selecting the data to be shown. To hide a data, cancel the relevant check box.



### Description of the tool data

The tool data is grouped as follows:

- · Tool identification.
- · Tool geometry.
- · Tool monitoring.
- · Information for the magazine.
- · Custom.
- Comment.



### **Tool identification**

Information identifying the tool: number, name, family, number of tool offsets and status.

### **Tool number**

It is assigned automatically when the tool is added to the list. This number may be changed if it is not in the magazine, in the spindle or on the tool changer arm.

The tool number may be any integer between 1 and 99999999; by default, it is assigned the first available value on the list.

When entering an existing tool number, the CNC displays the data for that tool.



**CNC 8070** 

### 12

AZINE TABLE
Tool table

A tool family is a group of tools that share similar characteristics. This information is used when using an automatic tool changer so the CNC can replace the worn-out or rejected tool with a similar one.

Name identifying the tool. This data can also be defined on the tool list.

The tool name may be up to 32-characters long.

When requesting a new tool, the CNC checks whether it is worn out (real life greater than nominal life) or it has been rejected. If so, it selects the next tool in the table that belongs to the same family.

The family or a tool may be any integer between 0 and 99999999. The  $\cdot 0 \cdot$  family is the same as not having a family; i.e. the tools belonging to the  $\cdot 0 \cdot$  family cannot be replaced with another one.

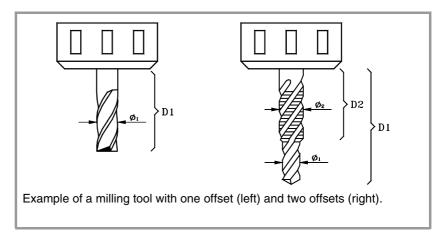
### **Tool offsets**

**Tool name** 

**Tool family** 

Number of tool offsets. Each tool offset has different geometry and monitory data associated with it.

A tool can have up to 8 offsets. When a tool has several offsets, their numbers must be correlative (non-skipping).



### **Tool status**

It indicates the tool status. The possible status are:

Status	Meaning
Available	The tool is available.
Worn out	The "real life" is greater than the "nominal life".
Rejected	The tool has been rejected by the PLC.

When using tool life monitoring, the "worn-out" and "rejected" indicators are also set by the CNC when any of the previous cases occur.



**CNC 8070** 

### **Tool geometry**

This area shows the data about the tool type and its dimensions. The geometry data depends on the type of tool. The table only shows the data that makes sense for the selected tool.

While defining the data, it shows various information graphics depending on the data being defined. On the other hand, the bottom of the screen shows the description of the data currently selected.

The data related to the geometry may be accessed with the following hotkeys:

Hotkey	Access
L	Length, length wear and edge length.
R	Radius, radius wear, nose (tip) radius and nose (tip) radius wear.
Α	Penetration angle
0	Offsets on each axis.

### Offset selection

The geometry data is associated with the tool offset. If the tool has been defined with several offsets, it shows the number of the offset whose data is displayed, and it also allows selecting the previous or next offset. To change offsets, place the focus on the buttons and press [SPACE].



Offset number and selection of the previous or next offset. In this case, it will display the data of the second tool offset.

### Operation type and tool

Regardless of the software installed, it is possible to define both milling and lathe tools. The tool is defined depending on the operation it can carry out. If it is not the right tool for any of the proposed operations, it must be assigned the operation "Others".

Once the operation has been selected, the screen will show the available tools. It will show a help graphic with the selected tool type.

The data to define the geometry of the tool will be relevant to the selected tool. It will only show the data that makes sense for the selected tool.

Operation	Tool type			
Milling (A) Flat endmill. (B) Toric endmill. (C) Ball endmill. (D) Disk endmill.	(A)	(B)	(C)	(D)
Reaming (A) Reamer.	(A)			
Grooving/cutoff (A) Square.	(A)			
Drilling (A) Drill bit.	(A)			
Boring (A) Quill.	(A)			
Turning (A) Diamond. (B) Square. (C) Round.	(A)	(B)	(C)	

**Fool table** 



**CNC 8070** 



**CNC 8070** 

(REF: 0809)

Operation	Tool type
Surface milling (A) Surface milling endmill.	(A)
Tapping (A) Cutter. (B) Tap.	(A) (B)
Measuring probe	
Others	To define the tools that do are not suitable for the proposed operations.

### Axis orientation in turning.

It is defined with an icon that is only displayed when defining a turning (lathe) tool.

The orientation of the axes is determined by the type of lathe (horizontal or vertical), the position of the turret and the spindle position (on the right or on the left).

### Location code or tool calibration point.

It is defined with an icon that is only displayed when defining a turning (lathe) tool.

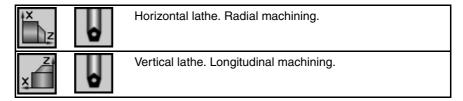
The location code indicates which is the calibrated tool tip and, therefore, the point controlled by the CNC to apply radius compensation. The location code depends on the orientation of the machine axes.

### Tool-holder orientation.

It is defined by an icon that is displayed in the following cases.

- · On a lathe model, for all the tools.
- On a mill model, only for turning (lathe) tools.

The orientation of the tool holder indicates whether it is a tool for horizontal or radial machining. For turning tools, the meaning of this icon depends on the orientation of the tool axes.



### Spindle turning direction.

Use the following icons to define the spindle turning direction.

Icon	Spindle turning direction.
	Undefined spindle turning direction.
	Clockwise spindle turning direction.
	Counterclockwise spindle turning direction.

The new tools do not have a pre-determined spindle turning direction; during execution, the spindle turns in the programmed direction (M03/04).

When assigning a turning direction to a tool 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.

### "L" - Tool length

This data is only shown on tools that are not for turning. The dimensions of the turning tools are defined with the offsets.

### "R" - Tool radius

This data is only shown on tools that are not for turning. The dimensions of the turning tools are defined with the offsets.



- L Tool length.
  - Tool radius.
- Lc Cutting length.

### "LW" "RW" - Tool length and radius wear

R

Tool radius wear and length wear offset. The CNC adds the wear value to the nominal length and radius to calculate the real tool length (L+LW) and real tool radius (R+RW).

Initial length	Wear	Final length
50	0.2	50.2
50	-0.2	49.8
-50	0.2	-49.8
-50	-0.2	-50.2

In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute. In either case, deleting the wear value or setting it to 0 implies resetting the amount of wear to 0. See "Select the type of wear values to enter, incremental or absolute." on page 207.

Using incremental wear, the value entered by the user will be added (or subtracted if it is negative) to the absolute value of the wear. After pressing [ENTER] to accept the new value, the wear field will show the resulting absolute value.

Initial wear	Incremental wear	Total wear
1	0.2	1.2
1	-0.2	0.8
-1	0.2	-0.8
-1	-0.2	-1.2

### "A" - Cutter angle

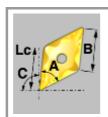
This data is only shown for turning tools.

### "C" - Cutting angle

This data is only shown for turning tools.

### "B" - Cutter width

This data is only shown for turning tools.



- A Cutter angle.
- C Cutting angle.
- B Cutter width.
- Lc Cutting length.

**12.** 

VE TABLE
Tool table



**CNC 8070** 

### "Rp" - Tool tip radius

Tool tip radius.

### "RpW" - Tool tip radius wear

Tool tip radius wear. The CNC adds the wear value to the nominal tool tip radius to calculate the actual (real) tool tip radius (Rp+RpW).

In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute. In either case, deleting the wear value or setting it to 0 implies resetting the amount of wear to 0. See "Select the type of wear values to enter, incremental or absolute." on page 207.

Using incremental wear, the value entered by the user will be added (or subtracted if it is negative) to the absolute value of the wear. After pressing [ENTER] to accept the new value, the wear field will show the resulting absolute value.

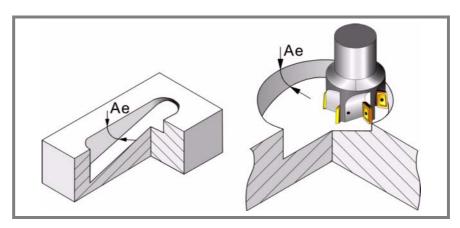
### "Lc" - Cutting length

Length of the cutting edge.

### "Ae" - Entry angle

This data is only shown on tools that are not for turning.

Penetration angle for pocket milling.



### Tool (length) offsets and offset wear.

The offsets are used to define the tool dimensions in each axis. The dimensions of the turning tools are defined using these offsets; either these offsets or tool length and radius may be used for the dimensions of the rest of the tools.

On tools that are not just for turning, e.g. endmills and drill bits, the offsets may also be used to define the tool position when using a tool holder or an intermediate tool. In this case, the tool dimensions are defined with the radius and the length.

The sign criterion for the offsets and their wear is established by machine parameter TOOLOFSG.

TOOLOFSG	Meaning.
Negative.    X(+)   OffX   OffX   OffZ(-)	Tool calibration returns a negative offset. The offset wear must be entered with a positive value.
Positive.    X(+)   OffIX     OffIZ (+)	Tool calibration returns a positive offset. The offset wear must be entered with a negative value.





**CNC 8070** 

In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute. In either case, deleting the wear value or setting it to 0 implies resetting the amount of wear to 0. See "Select the type of wear values to enter, incremental or absolute." on page 207.

Using incremental wear, the value entered by the user will be added (or subtracted if it is negative) to the absolute value of the wear. After pressing [ENTER] to accept the new value, the wear field will show the resulting absolute value.

Initial wear	Incremental wear	Total wear
1	0.2	1.2
1	-0.2	0.8
-1	0.2	-0.8
-1	-0.2	-1.2

### Select the type of wear values to enter, incremental or absolute.

The tool offset wear (for length, radius, tip radius and offsets) may be entered with either absolute or incremental values. The value shown by their respective fields in the tool table are always absolute.



After pressing this icon, the CNC shows the dialog box for configuring the tool table. Here, the user can define whether the wear values being entered are incremental (box checked) or absolute (empty unchecked). By default, the wear value being entered will be absolute.

When selecting an incremental wear, it is possible to define the maximum increment possible; by default 0.5 mm (0.019685 inch). This data limits the wear increment to be edited, not the maximum absolute wear possible. If no value is defined, empty data, there will be no limit for the wear increment to be entered.

The option selected in the tool table will also be applied in the tool calibration mode.

### Tool life monitoring

This group shows the data related to tool life monitoring. If a tool has been defined with several offsets, it is possible to manage the status of each one.

When managing the live of two or more offsets of the same tool, the list of the magazine positions (magazine table) only shows whether the tool has been rejected or not.

### **Type**

It is used to activate and select the type of tool life monitoring (in time or number of operations).

### Nominal life

Machining time (in minutes) or number of operations that the tool may carry out.

### Real (actual) life

Machining time or number of operation the tool has carried out. The CNC updates this value when the tool is being used.

### Tool magazine

This information cannot be modified if the tool is in the spindle, on the changer arm or in the magazine.

**12.** 

NE TABLE
Tool table

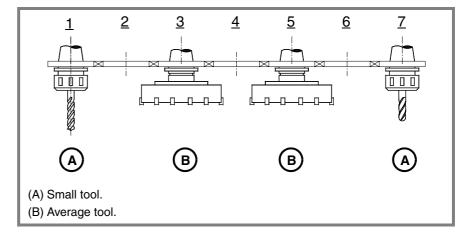


**CNC 8070** 

### Size

Tool size. The size determines the number of positions (pockets) the tool occupies in the magazine.

Size	Magazine positions it occupies.
Small	It only occupies its position. It takes no extra position.
Medium	It takes half of an additional position to the right and to the left.
Large	It takes a full additional position to the right and to the left.
Custom	The user defines the number of additional positions the tool occupies to its right and to its left.



### Positions to the right / Positions to the left

Space reserved in the magazine for the tool to the right and to the left of its position.

This data can be defined when the tool size is "custom".

### Special

The tool always occupies the same position in the magazine.

### Custom

Data defined by the manufacturer.

### Data 1 / Data 2

These data show, in numerical format, the information selected by the manufacturer.

### Data 3 / Data 4

These data show, in binary format, the information selected by the manufacturer.

### Comment

Comment associated with the tool.



**CNC 8070** 

### 12

## FOOL AND MAGAZINE TABLE Operations with the tool table

### 12.3 Operations with the tool table

### 12.3.1 Editing the tool table

### Initialize table

The table can only be initialized when the CNC is "READY". The table is initialized from the vertical softkey menu.



When resetting the table, all the tools are deleted from the list, including those in the spindle and on the tool changer arm. It also initializes the active-tools table and the magazine tables because the available tools have been erased.

### Adding/removing a tool to/from the list

A tool may be added or deleted only when it is selected on the tool list. Both operations are carried out from the vertical softkey menu.



### Add a tool to the list.

It adds a new tool to the list. The tool is added to the list at the first free position.



### Remove a tool from the list.

It removes a tool from the list. A tool cannot be removed if it is in the magazine, in the spindle or on the tool changer arm.

### **Data editing**

Proceed as follows to fill out the tool table data:

- 1. Select the tool to be set from the list and press [ENTER] to access its data.
- 2. Configure the table to show only the data that may be defined hiding the rest.
- Define the tool data. Every time a new value is defined, press [ENTER] to validate it.





When a tool has several offsets, all the data of an offset may be copied to another offset. This operation is carried out from the softkey menu.

These softkeys may be used to copy on to the clipboard the data of the offset being displayed and then paste it to another offset.

4. Make a safety backup copy of the tool data out to a file.

Keys for too	Keys for tool table editing		
	To move the cursor through the table data.		
ENTER -	To validate the new value.		
SPACE	To activate/deactivate the selection boxes and open/close drop lists.		
	To move the cursor whitin a data.		
	To select the line before or after the comment.		
End Home	To move the cursor to the beginning or end of the comment line.		



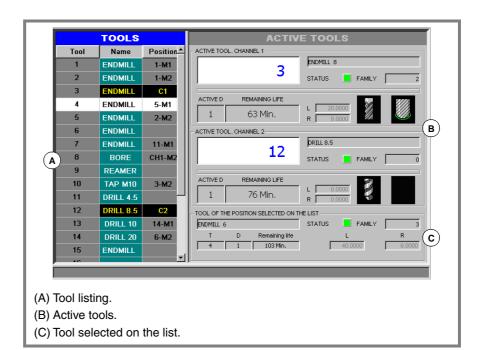
**CNC 8070** 

This table shows the list of available tools and which one is active in each channel.

The table is divided in two panels. To switch panels, press the panel-change key.

12.

TOOL AND MAGAZINE TABLE
Active-tools table



The left panel shows the list of the available tools and the right panel shows the data of the active tool in each channel.

### **Tool listing**

For each tool, it indicates the position and the magazine where it is located, whether it is a ground tool or the tool is active in a channel. The CNC updates data of the list every time a tool change is carried out.

It is the same list that appears in the tool table. See "12.2.2 The tool list" on page 200.

### **Active tools**

It shows the data of the active tool in each channel and also the data of the tool selected on the list. It is also possible to change the tool of the spindle. See *"12.4.2 Changing the tool of the spindle"* on page 213.

The tool data cannot be edited on this screen. The data shown here is defined in the tool table.



**CNC 8070** 

### 12.4.1 Description of the icons of the vertical softkey menu

The options of the vertical softkey menu depend on which table is active.



### Display units (millimeters/inches)

It is used to change the units for data display. The change of units is only valid for display and data entry. For programming, the CNC assumes the units set by the active function G70 or G71.

The icon highlights the units currently selected (millimeters or inches).



### Initializing the table

It initializes the tool table. The CNC will request confirmation of the command.

Initializing the tables eliminates all the tools from the list. It also initializes the active-tools table and the magazine tables because the available tools have been erased.



### Save table

It saves the table data in a file. See "12.1.3 Save and load the tables" on page 194.



### Recall table

It recalls the table data previously saved in a file. See "12.1.3 Save and load the tables" on page 194.

Bear in mind that loading the tool table initializes the magazine tables and the active-tools table.



### **Print table**

This softkey may be used to print the table in the pre-determined printer or save it as a file (prn format) at the CNC. See "12.1.4 Printing the tables" on page 196.



**CNC 8070** 

**FAGOR** 

It is possible to change the tool of the spindles from the active tools panel. The tool to be placed must be defined in the tool table.

To change the active tool, follow these steps.

- 1. Use the cursor to select the active tool to be modified and enter the number of the new tool.
- 2. Press the [CYCLE START] key to load the tool automatically or the [ENTER] key to update the positions list after a manual tool change.
- **3.** The tool list will show that the tool is in the spindle.



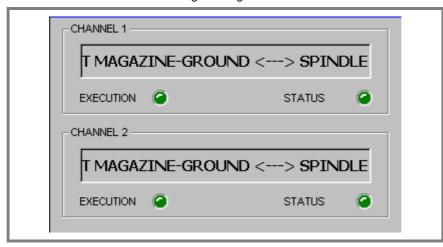
TOOL AND MAGAZINE TABLE
Active-tools table



**CNC 8070** 

### Table for the status of the tool change process 12.5

This screen monitors the tool changes being executed in each channel.



It indicates the following for each tool change process:

• The type of change selected and the magazine involved.

The type of change refers to the type of operation being carried out or to be carried out when executing function M06. The possible types of change are load a tool into the spindle or load and unload a ground tool from the magazine through the spindle.

The "execution " LED informs that the tool change process is running and that the magazine is carrying out an operation.

• The "status" LED informs on whether the magazine is in an error state or not. To eliminate the error condition, press the reset softkey in the magazine table.

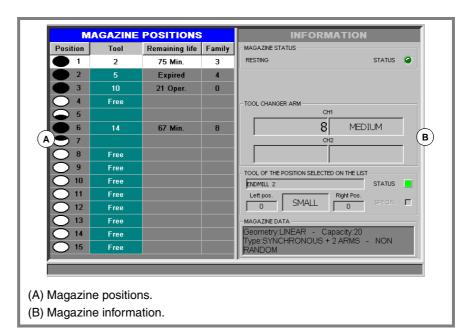
**TOOL AND MAGAZINE TABLE** Table for the status of the tool change process



**CNC 8070** 

Up to four different magazines may be configured. Each magazine has a table showing the tool distribution in the magazine and which table is in the spindle and on each holder of the changer arm (if any).

The CNC updates the table data every time a tool is changed.



The table is divided in two panels.

The left panel shows the list of magazine positions. For each position, it indicates whether it is free, disabled or has a tool. For each tool, it shows the remaining life time (when using tool life monitoring) and the family it belongs to.

The right panel is informative only; it cannot be selected. This panel shows information about the magazine status, about the tools in the tool changer arm and about the tool selected on the list. Optionally, it may show information on the type and configuration of the magazine.



TOOL AND MAGAZINE TABLE
Magazine table



**CNC 8070** 

## TOOL AND MAGAZINE TABLE

### 12.6.1 Description of the icons of the vertical softkey menu

The options of the vertical menu of softkeys depend on which table is active.



The vertical softkey menu shows all the options for the selected table. The group of options shown on this menu may be expanded with the softkey associated with the "+" icon that offers a different set of options.

The options shown for the tool table are.



### Load / unload a tool to / from the tool changer arm

(This icon is only available in magazines with tool changer arm).

It may be used to load and download a tool into the changer arm. See "12.7.2 Load / unload a tool to / from the tool changer arm" on page 223.



### Find text

To search for a text on a tool list. See "12.1.2 Search for a text in the tables" on page 193.



### Loading a tool into the magazine

The tool may be loaded into the magazine in two ways.

- Automatic loading (pressing [START]). A tool previously defined in the table is loaded into the magazine. It is loaded from ground through the spindle.
- Manual loading (pressing [ENTER]). It assumes that the tool defined in the table
  has already been manually loaded into the magazine. It updates the magazine
  list

See "12.7.1 Loading / unloading tools to / from the magazine" on page 221.



### Unloading the tool from the magazine

The tool may be unloaded into the magazine in two ways.

- Automatic unload (pressing [START]). A tool defined in the table is unloaded from the magazine. It is unloaded to ground through the spindle.
- Manual loading (pressing [ENTER]). It assumes that the tool defined in the table
  has already been manually loaded from the magazine. It updates the magazine
  liet

See "12.7.1 Loading / unloading tools to / from the magazine" on page 221.



### Initializing the table

Initialize (reset) the magazine table. The CNC will request confirmation of the command.

Initializing the tables eliminates all the tools from the list. It also initializes the active-tools table because the tool arrangement in the magazine may have changed.



### Save table

It saves the table data in a file. See "12.1.3 Save and load the tables" on page 194.



### Recall table

It recalls the table data previously saved in a file. See "12.1.3 Save and load the tables" on page 194.

Bear in mind that loading the magazine tool initializes the active-tools table.



FAGOR

**CNC 8070** 



### Print table

This softkey may be used to print the table in the pre-determined printer or save it as a file (prn format) at the CNC. See "12.1.4 Printing the tables" on page 196.



### Enabling and disabling a magazine position

It enables or disables the magazine position selected with the cursor.



### Resetting the magazine

It eliminates the error status of the tool manager. This icon is only available when an error occurs at the tool manager.



### **Magazine information**

It shows or hides the magazine data.



### Initialize the magazine data

Initialize (reset) the magazine table. The CNC will request confirmation of the command.

It initializes all the magazine positions assigning T1 to position 1, T2 to position 2 and so on. The tools must exist and must not be in another magazine.

It also initializes the list of tools and the active-tools table because the tool arrangement in the magazine may have changed.



**CNC 8070** 

The list of magazine positions appears on the left panel of the tool magazine table. For each position, it indicates whether it is free, disabled or has a tool. For each tool, it shows the remaining life time (when using tool life monitoring) and the family it belongs to.

### Magazine position

The status of a position is indicated with a symbol next to the position number.

- The position is free (white filled circle).
- The position is semi-free (black-and-white filled circle).
- The position is occupied (black filled circle).
- The position is disabled (red filled circle).

### **Tool number**

Number of the tool occupying the magazine position.

### Remaining life

If the tool life monitoring is active, shows the remaining life, either in machining time or in the number of operations to be carried out or the tool status (rejected or worn out).

### **Tool family**

Family the tool belongs to, defined by the user in the tool table.

12.

TOOL AND MAGAZINE TABLE

Magazine table



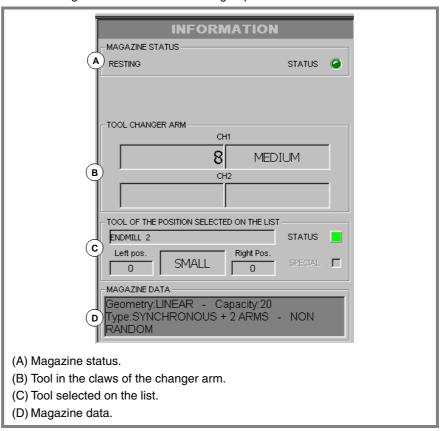
**CNC 8070** 

10

TOOL AND MAGAZINE TABLE

Magazine table

The right panel shows different information about the status of the tool magazine and the tool changer arm. This information is grouped as follows:



### Magazine status

This area shows the operation being carried out in the magazine.

In standby: The magazine is in standby.

Loading: A tool is being loaded into the magazine.

Unloading: A tool is being unloaded from the magazine.

**0+0** 

The "Status" led informs the user whether the magazine is in an error state or not. To eliminate the error condition, press the reset softkey in the magazine table.

### Change status

If the magazine is involved in a tool change at a given time, it shows the information about the status of that change; the operation is carried out when executing an M06, manager status (in execution or at rest) and the status of the change process (whether it is in error or not).

### Tools in the changer arm

This area shows which are the tools that occupy each claw (holder) of the changer. It shows the number and size of each tool.



**CNC 8070** 

### **Tool information**

It shows the data related to the magazine of the tool selected from the list. The "status" led informs about the tool status.

Green The tool is available.

Yellow The tool life is about to expire.

Red The tool is expired (worn out) or has been rejected by the PLC.

### Magazine data

1

Description of the type of magazine.

To show and hide this information, press the information softkey.

**12.** 

TOOL AND MAGAZINE TABLE

Magazine table



**CNC 8070** 

### 12.7 Operations with the magazine table

### 12.7.1 Loading / unloading tools to / from the magazine

The tools may be loaded and unloaded to/from the magazine manually or automatically. When done manually, the positions list must be updated.

### Manual load/unload

The operator places the tools directly in the magazine without using the CNC. Then, the positions list must be updated.

### Automatic load/unload

The operator places the tools in the spindle and the CNC places the tool in the magazine. The positions list is updated automatically.

### Initialize table

The table can only be initialized when the CNC is "READY". The table is initialized from the vertical softkey menu. The CNC offers two ways to initialize the table.



Initializing the table eliminates all the data about the position of the tools in the magazine. It also initializes the active-tools table because the tool arrangement in the magazine has been changed.



When initializing the table, It initializes the magazine positions assigning T1 to position 1, T2 to position 2 and so on. The tools must exist and must not be in another magazine.

It also initializes the active-tools table because the tool arrangement in the magazine may have changed.

### Loading a tool



The tool is loaded into the magazine using the vertical softkey menu. Only the tools defined in the tool table may be loaded into the magazine and they must be defined as ground tools. In other words, they must not be in any position of the magazine, of the spindle or in the claws of the tool changer.



Even if the magazines are configured for not admitting ground tools while machining, they may be loaded into the magazine through this maneuver.

Depending on tool size, when it is loaded into the magazine, it may affect several positions.

### Magazine management using the "Load tool" softkey.

This type of management is valid for loading tools both automatically and manually. After pressing the softkey, the user is asked what kind of loading to do.

- 1. Press the softkey associated with tool loading.
- **2.** Enter the tool number and the magazine position to insert it. By default, it offers the position selected on the list with the cursor.
- **3.** Press the [CYCLE START] key to load the tool automatically by means of the spindle or the [ENTER] key to update the positions list after loading manually.



**CNC 8070** 

### Magazine management using the positions list.

This type of management is only valid for loading tools manually. It lets update the list of positions after having placed the tools directly in the magazine, without using the CNC.

- 1. Select a magazine position from the list and enter the number of the tool it occupies it.
- 2. Press [ENTER] to update the positions list.

### Unloading a tool



The tool is unloaded into the magazine using the vertical softkey menu. After unloading a tool from the magazine, it becomes a ground tool.

### Magazine management using the "Unload tool" softkey.

This type of management is valid for unloading tools both automatically and manually. After pressing the softkey, the user is asked what kind of loading to do.

- 1. Press the softkey associated with tool unloading.
- 2. Enter the number of the tool to be unloaded.
- 3. Press the [CYCLE START] key to unload the tool automatically to the spindle or the [ENTER] key to update the positions list after unloading manually.

### Magazine management using the positions list.

This type of management is only valid for unloading tools manually. It lets update the list of positions after having removed the tools directly from the magazine, without using the CNC.

- Select a magazine position from the list and delete the number of the tool it occupies it.
- 2. Press [ENTER] to update the positions list.

table

FAGOR

**CNC 8070** 

### 12.7.2 Load / unload a tool to / from the tool changer arm



A tool is loaded/unloaded to/from the claws (holders) of the changer arm using the vertical softkey menu. To insert a tool in the holders of the changer arm (when available), it must be placed in the magazine. Ground tools cannot be placed in the claws (holders) of the tool changer arm.

Nor can the tools that are in the holders of the tool changer arm be inserted in the spindle.

To load or unload one of these tools:

- **1.** Press the softkey associated with the operations with the changer arm.
- 2. Define the tool number in the positions of the tool changer arm. When using a two-holder changer arm, there may not be tools in the spindle and in the second holder of the changer arm at the same time.
- 3. Press [ENTER] to update the table.



TOOL AND MAGAZINE TABLE
Operations with the magazine table



**CNC 8070** 

12.

TOOL AND MAGAZINE TABLE
Operations with the magazine table



**CNC 8070** 

TOOL AND MAGAZINE TABLE
Operations with the magazine table



**CNC 8070** 

### **UTILITIES MODE**

### **Utilities mode selection**



The utilities mode may be accessed from any other operating mode, even while a program is in execution, by pressing the relevant key.

### Operations in this work mode

This work mode shows the files that may be accessed from the CNC and may be used for the following operations:

- See the files stored at the CNC, in a floppy disk or at another CNC (or PC) connected via Ethernet.
- · Create new folders to save files.
- Select a file group and carry out operations such as copy, rename or delete files.
- · Change file attributes.
- Do a file search based on a text already defined in them.
- Set a password to restrict the access to the customizing tool FGUIM, to machine parameters and to the PLC.
- Make or restore a backup copy of the CNC data.



**CNC 8070** 

### Appearance of the utilities mode 13.1

When this operating mode is active, it will be indicated at the top right side of the general status bar. This mode only has one screen.

### Standard configuration

The standard configuration of this work mode, described in this manual corresponds to the one supplied by Fagor. In some cases, the manual shows a sample screen, instead of that of the CNC in order to make it easier to understand.

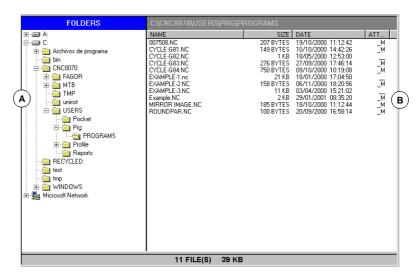


Appearance of the utilities mode



**CNC 8070** 

The utilities mode screen shows the following information:



A. Folder tree.

It may show the folders of the CNC, of a floppy disk or another CNC (or PC) connected via Ethernet.

B. List of files saved in the selected folder.

To switch windows, press the relevant key.

### Softkey menu

When selecting the utilities mode, the horizontal softkey menu will show the options related to this work mode.

13.

UTILITIES MODE
Appearance of the utilities mode



**CNC 8070** 

### 13.1.2 Window description

### Folder tree

This window shows the folders that may be accessed from the CNC, as well their structure. Besides the CNC folders, it may show those of a floppy disk or another CNC (or PC) connected via Ethernet.

The folders may contain files and other folders.

Icon	Description
	It contains no folders.
+	It contains no folders and they are not shown here.
⊟…	It contains folders and they are shown here.
	The folder has been selected.

To select a folder from the list:

- Move the cursor with the relevant keys.
- Using the alphanumeric keyboard, pressing a key will select the first element starting with that letter. Pressing it again will select the second one and so on.

Keys to select a folder		
<b>+</b>	To switch windows (folders or files).	
	To close and open a folder submenu.	
	To move the cursor thru the elements of the window.	
	To move the cursor page by page.	
End Home	To move the cursor to the beginning or end of the list.	

When selecting a folder, the CNC shows a list of the files it contains (See the "13.2.2 Options" section of this chapter). Likewise, the bottom of the window will show the number of files contained in the folder and the total size (bytes) they amount to.

### List of files

This window shows the files stored in the selected folder. Any of them may be deleted, renamed, its protections changed and copied within the same device (CNC, PC, floppy disk) or between two different devices.

This window offers the following information:

- The file name.
- The file size, in bytes.
- The time and date when the file was edited last.
- . The attributes of the file.

### Attributes. Modifiable and hidden files

Only the letters of the attributes currently selected will be shown. Those not selected will appear as " $\_$ " .

- H The file is hidden and it will not appear when selecting a program for editing or execution.
  - Remember that a hidden file can still be deleted if its name is known. To prevent that (if so wished) its "modifiable" attribute should be removed.
- M The file may be modified; in other words, edited, copied, deleted, etc.

13.

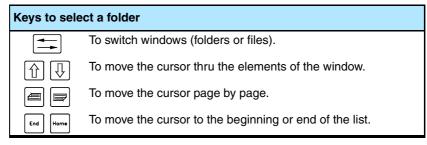
UTILITIES MODE Appearance of the utilities mode



**CNC 8070** 

To copy, rename, delete, etc. the files must be previously selected on the list. A file may be selected from the list:

- Move the cursor with the relevant keys.
- Using the alphanumeric keyboard, pressing a key will select the first element starting with that letter. Pressing it again will select the second one and so on.
- Using the "file search" option of the softkey menu permits looking for all the files that contain the indicated text.



To select a group of files, keep the [SHIFT] key pressed while the moving the cursor. To add or remove a file from the selection, keep the [CTRL] key pressed and place the cursor on the file and press [SPACE] key.

13.

UTILITIES MODE
Appearance of the utilities mode



**CNC 8070** 

### 13.1.3 Vertical softkey menu (icons)



The icon menu always shows all the icons related to the Utilities mode. The group of icons of this menu may be expanded with the softkey associated with the "+" icon that offers a new set of icons.



**UTILITIES MODE** 

Appearance of the utilities mode

### X

### Cut

Copies the selected files on the clipboard. After pasting the content of the clipboard, the files are deleted from the folder.

(This icon is only available when there is a file selected).



### Copy

Copies the selected files on the clipboard.

(This icon is only available when there is a file selected).



### **Paste**

Pastes the files from the clipboard into the selected folder. If the files were placed using the "Cut" option, they will be removed from their original location.

The contents of the clipboard are not eliminated after "pasting". Therefore, this pasting operation may be repeated as often as you wish.



### Rename

It is used to change the name of the folder or file currently selected.

- When renaming a folder, if there is one already with the new name, the proposed name change will be ignored and the folder will keep its original name.
- If when renaming a file, there is a previous file with the same name in the folder, the new file replaces the previous one.
- The file selected in automatic cannot be renamed.



### Modifiable file

It is used to change the "modifiable" attribute of the selected files. This attribute allows protecting the files so they cannot be modified in the editing mode.

The attributes column shows the letter -M- indicating that the program may be modified.

When a program is NOT modifiable, its contents may be viewed; but cannot be modified.



### Hidden file

It is used to changed the "hidden" attribute of the selected files. This attribute allows protecting the files so they are not displayed when selecting a program to be edited or executed.

The attributes column shows the letter -H- indicating that the program will be hidden (not visible).

However, a hidden program may be deleted if its number is known; therefore, it is recommended to remove the modifiable attribute (M) in order to avoid deleting it.



### Remove file

It is used for deleting the selected folder or files. To delete the files, the CNC will show a dialog box requesting confirmation of the command whereas the empty folders will be deleted directly without requesting confirmation.

The folders can only be deleted if they are empty.

The file selected in automatic cannot be deleted.



**CNC 8070** 

### 13.2 Utilities (Softkeys)

### 13.2.1 Sorted by

It is used to order the list of files.



When selecting this option, the softkey menu shows the different ordering options. To return to the previous menu, press the key for the previous menu.

The file list may be ordered alphabetically, by size, by data or by type. When pressing the same ordering criteria twice, the CNC toggles from ascending to descending and vice versa.

The softkey menu shows the ordering criteria currently selected (the softkey will appear pressed).

### **13.2.2 Options**

It is used to personalize how the program listing will be displayed on the screen.



When selecting this option, the softkey menu shows the following personalizing options. To return to the previous menu, press the key for the previous menu.

### **Update**

It updates the list of files showing the files of the folder currently selected. Only when the "auto-update" option is not active.

### **Auto-update**

When this option is selected (the softkey will appear pressed), every time a folder is selected, the CNC will automatically update the list of files.

### Column adjust

When this option is selected (the softkey will appear pressed), the columns of the file lists will adjust to the text they contain so as to show the text that may be truncated because it is too long.

### Reset all

It closes the folder tree (layout) and it only shows the devices (floppy disk, Ethernet network, etc.) connected to the CNC.

### Show hidden files

When this option is selected (the softkey will appear pressed), the file list shows all the files of the selected folder even those having the "hidden" attributes. Otherwise, these files will not be shown.

13.

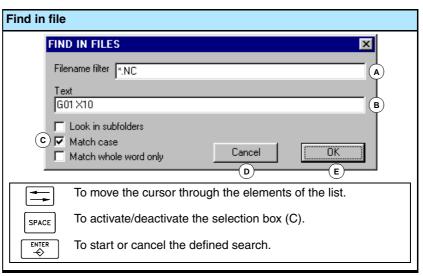
UTILITIES MODE
Utilities (Softkeys)



**CNC 8070** 

### 13.2.3 Search in files

It is used to search files. When selecting this option, the CNC shows a dialog box where the following data may be defined:



- **A.** Description of the files to be searched. The "\*" and "?" wild characters may be used on the search, meaning:
  - \* Any character string.
  - ? Any character.

In such a way that:

- \* Looks for all the files.
- \*.NC Looks for all the files having the extension "NC".
- exa\*.\* Looks for all the files beginning with "exa".
- **B.** Text included within the files.
- C. Defines the search criteria.
  - Look in the sub-folders.
  - Ignore uppercase and lowercase.
  - · Consider the text to find as a whole word.
- **D.** Start the defined search.
- E. Cancel the defined search.

After defining the search options, place the cursor on one of the lower buttons to accept (D) or cancel (E) the defined search and press [ENTER].

The file window will show the list of the programs found.

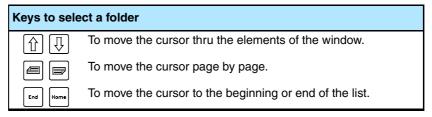


**CNC 8070** 

### 13.2.4 Select all

It selects all the files of the list. The selection will be canceled by moving the cursor.

To add or remove a file from the selection, keep the [CTRL] key pressed and place the cursor on the file and press [SPACE] key.

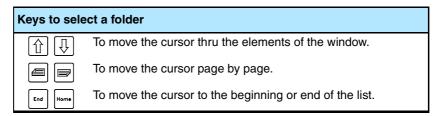


Having selected a group of files, they may be deleted, copied, cut or their attributes may be changed by pressing the relevant icon.

### 13.2.5 Invert selection

It inverts the file selection made selecting the files that appeared unselected and vice versa.

To add or remove a file from the selection, keep the [CTRL] key pressed and place the cursor on the file and press [SPACE] key.

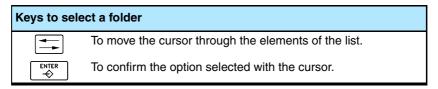


Having selected a group of files, they may be deleted, copied, cut or their attributes may be changed by pressing the relevant icon.

### 13.2.6 New folder

It is used to create a new folder to store files.

When selecting this option, the CNC shows a dialog box requesting the file name. Write the new name and select one of the buttons to accept it or ignore it.



If when defining the folder name, there is already a folder with the same name, it will NOT create the new folder.

13.

UTILITIES MODE
Utilities (Softkeys)



**CNC 8070** 

### 13.2.7 Protection passwords

The passwords may be used to define each of the codes that the user will have to enter in order to access certain CNC functions. If entered correctly, it stores it and it does not request it again unless the CNC is turned off. If the password is wrong, the requested action cannot be carried out and it requests it again every time.

In a CNC with a write-protected disk, when the CNC is powered up in setup mode, it does not request the protection passwords. When the CNC is powered up in user mode, it requests the protection passwords.

Press the "Passwords" softkey to access the password setting screen. The following passwords may be defined.

- · General password.
- PLC.
- · Machine parameters.
- · Customizing.
- Machine parameters for kinetics.

### General password

It is requested when trying to access this password setting screen.

### **PLC**

It is requested when trying to carry out the following actions at the PLC:

- Editing the PLC program. When entering the wrong password, the PLC program opens as read-only.
- · Adding a file to the project.
- · Deleting a file.
- Renaming a file.
- Editing PLC messages. When entering the wrong password, the PLC messages may be neither displayed nor edited.
- · Generate PLC.
- When accessing to the "Commands" service options.
- In monitoring, when modifying the status of a resource.

### **Machine parameters**

It is requested when trying to carry out the following actions in the machine parameter table:

- · Modifying the value of a parameter.
- Initialize a table.
- · Loading a table.
- When starting the CNC application and the unit is powered up in setup mode.

### Customizing

It is requested when entering the FGUIM application. When entering the wrong password, it will not be possible to access the application.

### Machine parameters for kinetics

It is requested when trying to carry out the following actions in the machine parameter tables for the kinematics: If not set, it will apply the password used in the rest of the machine parameters.

- · Modifying the value of a parameter.
- Initialize a table.
- · Loading a table.





**CNC 8070** 

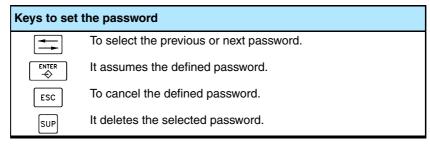
### Administrator mode

It is requested to start the unit up in administrator mode. The access to the administrator mode is enabled with the validation code. If you don't have this software option, you will not be able to access the administrator mode.

### How to set the protection passwords

Press the "Passwords" softkey to access the password setting screen. From this screen, it is possible to define, modify or delete the passwords. If this screen is protected, pressing the softkey will request the general password.

Each password may be up to 10 ASCII characters long. It is case sensitive.



The "Delete all" softkey deletes all the passwords defined.

13.

UTILITIES MODE
Utilities (Softkeys)



**CNC 8070** 

Utilities (Softkeys)

### 13.2.8 Data safety backup. Backup - Restore

This option may be used to make a backup copy of the data of the MTB folder in order to be able to restore this configuration later on if necessary. The backup is saved in a single file.

Both the folder being backed up and the file name it is saved in may be configured. When not indicated otherwise, it makes a backup of the "MTB" folder and it saves it in the *mtb.r* file in unit –A:–.

### Configuring backup options

The backup options may be defined using the "Backup options" softkey. As backup options, it is possible to define both the folder being backed up and the name of the file containing the data. By default, it makes a backup of the "MTB" folder and it saves it in the *mtb.r* file in unit –A:—.

The CNC saves the defined options in a file called *backup.ini*. If this file does not exist when making or restoring a backup, the default values will be used.

### Making a data backup

The backup is made using the "automatic backup" softkey. Pressing this softkey makes a safety copy of the data of the indicated folder. The data is saved in the selected file.

When not indicated otherwise or if the *backup.ini* file does not exist, it will make a backup of the "MTB" folder and will save it in the *mtb.r* file in unit –A:–

### Restoring data from a backup file

Backup data is restored using the "Restore backup" softkey. Pressing this key copes the data from the indicated file to its original folder; i. E. To the folder that was backed up. If this folder does not exist, it creates it and goes on with the operation.

If the files being restored already exist, they will be overwritten; i.e. the ones in the folder will be replaced by the ones in the backup.

The name of the file from which the backup is restored is defined in the *backup.ini* file. If this file does not exist, it proceeds as follows:

- **1.** It looks for the *mtb.r* file in unit –A:– and it restores it.
- 2. If this file does not exist, it restores the first backup file (extension -r-) of unit -A:-
- 3. If there is no backup file, it shows the relevant error message.



**CNC 8070** 

### **PLC**

In this operating mode, the PLC may be accessed to check its operation or the status of the different PLC variables. It is also possible to edit and analyze the PLC program as well as the message and error files of the PLC.

At a CNC with a write-protected (read-only) disk, working in user mode, the PLC program is write-protected and any changes made to it are temporary; i.e. it will disappear the next time the unit is turned on. To make the changes permanent, unprotect the PLC program by starting the unit up in setup mode and validating the changes made.

### **MDI** mode selection



The PLC mode may be accessed either using the user key customized for it or from the tasks window. The task window may be accessed with key combination [CTRL]+[A].

### What can you do in this work mode?

In short, the PLC mode offers the following operations:

- Edit, modify or view the contents of the PLC project.
- Debug the PLC program by executing the different parts of the program separately (first cycle, main module, periodic module).
- Analyze the behavior of the logic signals of the PLC program depending on a time base and triggering conditions set by the user.
- · Monitor the status of PLC resources and variables.
- See all the resources used in the PLC project and in which program of the project they are being used.
- Run statistics on execution times of the PLC project and of the modules that form it.
- Edit, modify or view the texts associated with the messages and errors of the PLC.



**CNC 8070** 

### 14.1 Appearance of the PLC mode

When this operating mode is active, it will be indicated at the top right side of the general status bar.

This operating mode may be displayed with several screens. Each of them corresponds to a PLC service. The exchange between the different services is carried out with the list of services.



The different services overlap each other. If several services active, it is possible to switch from one to another using the relevant key, if it has been set accordingly. The services are switched in a rotary manner, in such a way that when pressing on the last active service, it shows the first one.

### Standard configuration

The standard configuration of this work mode, described in this manual corresponds to the one supplied by Fagor. In some cases, the manual shows a sample screen, instead of that of the CNC in order to make it easier to understand.



- PLC

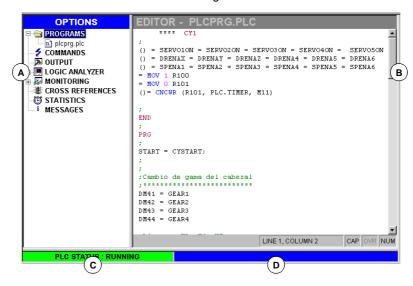
Appearance of the PLC mode



**CNC 8070** 

### 14.1.1 Screen description

The PLC mode screen shows the following information.



A. Service window.

It shows a list of the services available in the PLC environment.

B. Data window.

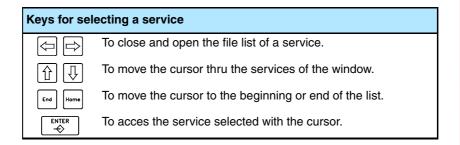


It shows the screen associated with the selected service, if any. The screens of the different services overlap each other. If several services active, it is possible to switch from one to another using the change key, if it has been set accordingly. To close a screen, press [ESC].

- C. It shows the PLC status, running or stopped.
- D. CNC messages.



To switch windows, press the relevant key.



### Softkey menu

When selecting the PLC mode, the horizontal softkey menu will show the options associated with the PLC service currently selected. The set of options offered by this menu may be changed using the "+" softkey which offers a new set of options.

14.

PLC
Appearance of the PLC mode



**CNC 8070** 

The icons are activated with their associated softkey (F8 to F12).

### List of icons.



Appearance of the PLC mode



### Show/hide the service window

It toggles between the shared usage of the area of the PLC environment (that displays both the service window and the data window) and the full work screen (hiding the service window so the data window is expanded covering the whole area of the PLC environment).



### Find text

It is used for searching a text in all the files of the project. The result will be shown at the "Outputs" service.



### Go to file

(It only appears when working with the "Outputs" or "Cross references" service).

If the cursor is positioned on the result of a text search, a compiling error or a PLC resource, after selecting this option, it opens the corresponding file and the cursor goes to the line it refers to.



### Mnemonic language or contact language.

(It only appears in the "Programs" service when monitoring a program).

It toggles the display between the mnemonic language and the contact language (ladder diagram).

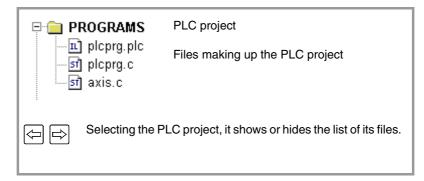


**CNC 8070** 

### 14.2 "Programs" service

This service is used for managing the PLC project and its files. The PLC project may consist of several files edited in the following languages:

- Mnemonics or equations language (files with "plc" extension).
- Contac language (ladder diagram) (files with "ld" extension).
- C language (files with "c" extension).



### **PLC** project

The PLC project is a set of files that, once compiled, generate the PLC program.

When selecting the PLC project, the softkey menu will show the options to manage that project. Among these options, it is possible to add files to the PLC project and compile it.

### Files of the PLC project

As mentioned earlier, the PLC project may consist of one or several files edited in the following languages:

- · Mnemonics or equations language.
- · Contact language (ladder diagram).
- · C language.

The PLC project may consist of several files in the same language. Two files having the same name and different extension cannot belong to the same project.

When selecting a file, the softkey menu shows the options to manage that file. Among these options, it is possible to edit and monitor the file. These options will show the editing and monitoring screens.

14.

PLC

"Programs" servic



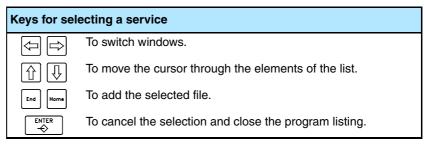
**CNC 8070** 

### 14.2.1 PLC project (softkeys)

These options are used to create a PLC project and compile it to create the executable PLC program.

### PLC project "Add file"

It is used to add a file to the PLC project. This file may be a new one or an existing one. When selecting this option, the CNC shows a list of the available files.



To add a file to the PLC project.

- 1. Select the file from the list or write its name in the bottom window.
- 2. Press [ENTER] to accept the selection and add the file or [ESC] to cancel the selection and close the file listing.

When accepting the selection, the selected file will appear on the list that make up the PLC project.

### PLC project "Generate PLC"

It compiles and loads the PLC program based on the files that make up the PLC project. If an error occurs during compilation, the program will not be generated and the CNC will display a list of the detected errors.

The proper compilation of a file in mnemonic language or contact (ladder) language generates the equivalent files in both languages. That is why two files having the same name and different extension cannot belong to the same project because, when compiled, they are the same file.

The following expressions in mnemonic language cannot be translated directly into contact (ladder) language and the CNC will translate them as follows.

- Expressions with the XOR operator.
- Expressions where the NOT operator affects several contacts.

Original expression.	Transformed expression.
a XOR b	(a OR b) AND (NOT a OR NOT b)
NOT (a AND b)	(NOT a OR NOT b)
NOT (a OR b)	(NOT a AND NOT b)



**CNC 8070** 

### 14.2.2 Files of the PLC project PLC (Softkey)

When selecting a program of the PLC project at the service window, the softkey menu will offers the following options.

### Files of the PLC project "Edit"

This softkey shows the editing window. See "14.3 Program editing" on page 244.

### Files of the PLC project "Monitoring"

This softkey shows the monitoring window. See "14.6 Program monitoring" on page 257.

### Files of the PLC project "Eliminate"

This softkey deletes the selected file of the PLC project (the file will still be available in the CNC's hard disk).

### Files of the PLC project "Rename"

This softkey may be used to change the name of the selected file.

### Files of the PLC project "Copy"

This softkey may be used to make a copy of the selected file. When selecting this option, the CNC shows a list with the programs stored at the CNC. To make a copy of the file:

- 1. Select the destination folder for the copy.
- 2. Define the file name at the bottom window. To replace an existing file, select it from the list.
- **3.** Press [ENTER] to copy the file or [ESC] to cancel the selection and close the file listing.



### Files of the PLC project "Move up"

When the PLC project contains several files, this softkey may be used to move the selected file up.



### Files of the PLC project "Move down"

When the PLC project contains several files, this softkey may be used to move the selected file down.

14.

PLC

"Programs" servic



**CNC 8070** 

### 14.3 Program editing

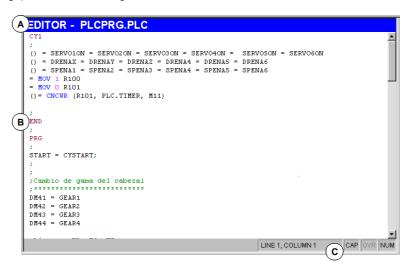
To edit a program, select it from the "Programs" service list and select the "Edit" option of the softkey menu. The PLC will show the right editor for the language of the selected program; a text editor if the program is in mnemonic language or C language or a contact (ladder) editor if the program is edited in contact (ladder) language.

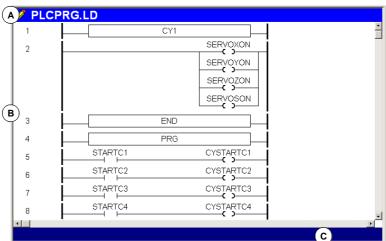
- Mnemonics or equations language (files with "plc" extension).
- Contac language (ladder diagram) (files with "ld" extension).
- C language (files with "c" extension).

To close the edit window, press [ESC].

### **Editor description**

The text editor window (top image) and that of the contact (ladder) editor (bottom image) show the following information.





### A. Title bar.

Name of the program selected for editing. An "\*" next to the program name means that the changes made to the program have not been saved (the program must be saved so they do not get lost).

### B. Edit area.

Line number and area for editing the program. A program edited in contact (ladder) language will contain numbered blocks representing the various elements.



PLC rogram editing



**CNC 8070** 

### C. Status bar.

In a program edited in C language or mnemonic language, the bar shows information about the cursor position and the status of the editor options.

CAP Capital letters. When active, the text is always written in capital letters.

OVR Overwrite text. It toggles between overwriting and inserting text.

When active, it overwrites the existing text.

NUM Numeric keypad active.

In a program edited in contact (ladder) language, the bar shows the comments of a contact and other messages.

### Program editor, Unicode format.

The editor of the CNC admits Unicode characters. When saving a program, the editor respects its original format, ANSI or Unicode, except when editing a Unicode character, in which case the CNC will always save the program in Unicode format.



Programs in Unicode format are not compatible with versions older than V4.0. A program in Unicode format cannot be edited or executed in a version older than V4.0.

A third-party text editor (like Windows notepad) may be used to convert Unicode format programs into ANSI format; but special characters that have no ANSI equivalent will be lost in the process.

### Functionalities of the editor.

### Zoom at the editor.

The editor has the following hotkeys to increase or decrease the size of the editor font. If the CNC has a mouse with a wheel, the [CTRL] key combined with this wheel can also be used to increase and decrease the size of the text font.

[CTRL]+[+] Zoom in. [CTRL]+[-] Zoom out.

### Multi-line blocks.

POCK.) L ₽ .C.M=0 ₽ The editor adjusts the long blocks to the size of the window dividing the block into several lines. On the right side of each cut line, the editor shows a symbol to indicate that the block continues in the next line.

### Softkey menu

When accessing the PLC program editor, the horizontal softkey menu will show all the options associated with editing a file depending on the editing language.

14.

PLC rogram editing



**CNC 8070** 

### 14.4 Editing in C language or mnemonic language (softkeys).

The following softkeys may be useful when editing.

Hotkey.	Function.
[CTRL]+[C]	Copy the selected text.
[CTRL]+[X]	Cut the selected text.
[CTRL]+[V]	Paste the selected text.
[CTRL]+[Z]	Undo the last change.
[CTRL]+[Y]	Redo the selected text.
[CTRL]+[G]	Save the program/Recover the original program.
[CTRL]+[+]	Zoom in.
[CTRL]+[-]	Zoom out.
[CTRL]+[HOME]	Move the cursor to the beginning of the program.
[CTRL]+[END]	Move the program to the end of the program.

### 14.4.1 Analyze

It analyzes the program searching for errors. The errors found will be displayed in the "Outputs" service window.



To close this window and return to the editor, press [ESC]. To return to the editor without closing the window, press the screen switching key.

### 14.4.2 File

It is used to restore, save, import or print a file.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

### File "Restore original"

This option will only be available when the "Save always" option is active (the section on "14.4.6 Customizing" in this chapter).

It is used to recover the original file without the changes made since the last time it was opened. When selecting this option, the CNC requests confirmation of the command.

### File "Save"

This option will only be available when the "Save always" option is not active (the section on "14.4.6 Customizing" in this chapter).

It is used for saving the file that is being edited.

### File "Save as"

It is used to save the file, that is being edited, with a different name. After saving the file, one keeps editing the new file.

### File "Include program"

It is used to import the contents of a file saved at the CNC into the program that is being edited.

### File "Print"

This softkey may be used to print the program in the pre-determined printer.



**CNC 8070** 

# Editing in C language or mnemonic language (softkeys).

### 14.4.3 Undo

This softkey may be used to "undo" the last modifications made. The modifications are undone one by one starting from the most recent one. The CNC offers the following keyboard shortcuts to undo and redo the operations.

[CTRL]+[Z] Undo the last change.

[CTRL]+[Y] Redo the selected text.

### 14.4.4 Operations with blocks

It is used to copy, cut and paste the information of a block or set of blocks and export this information as an independent program.

This option is only available when there is a text selected in the program or on the clipboard. To select a text in the program, keep the [SHIFT] key pressed while moving the cursor.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

### Operations with blocks "Copy"

It copies the selected text onto the clipboard.

### Operations with blocks "Cut"

Copies the selected text onto the clipboard and deletes it from the program.

### Operations with blocks "Paste"

Pastes the contents of the clipboard into the program.

### Operations with blocks "Copy to program"

Saves the selected texts as an independent program.

When selecting this option, the CNC shows a list of the available programs. To save the text as a program:

- 1. Select the destination folder.
- 2. Define the file name at the bottom window. To replace an existing file, select it from the list.
- **3.** Press [ENTER] to save the program or [ESC] to cancel the selection and close the program listing.

14.

PLC

FAGOR

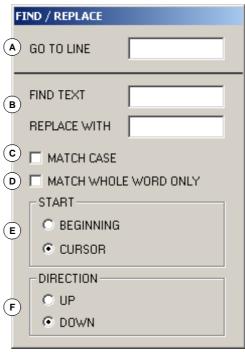
**CNC 8070** 

### 14.4.5 Find/Replace

It is used for searching for a line or a text in a program. It can also replace a text with another text.

When selecting this option, the CNC shows a dialog box requesting the line number or the text to look for. When defining a text search, certain options may also be defining that allow:

- A. Go to a line of the program.
- **B.** Replacing the text being searched with another in the program.
- C. Ignore uppercase and lowercase.
- **D.** Consider the text to find as a whole word.
- **E.** Select whether the search starts at the beginning of the program or at the cursor position.
- F. Direction of the search.



After defining the search options, press [ENTER] to start the search or [ESC] to cancel it. The text found in the program will be highlighted and the softkey menu will show the following options:

- "Replace" option, to replace the highlighted text.
- "Replace all" option, to replace the text throughout the whole program.
- "Find next" option, to skip this text and keep on searching.
- "Find previous" option, to look for the text without replacing it.

To end the search, press [ESC].



PLC

Editing in C language or mnemonic language (softkeys)



**CNC 8070** 

### 14.4.6 Customizing

It is used for personalizing the appearance and properties of the editing window.

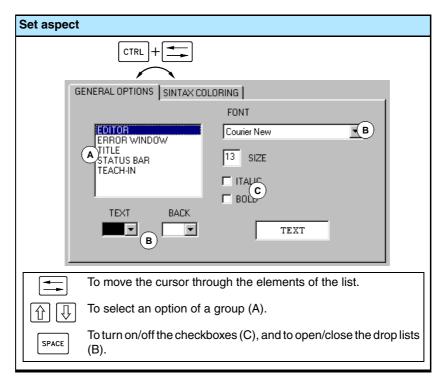


When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

### Customizing "Appearance" (looks)

It is used for personalizing the look of the different elements of the editing window and the colors of the elements of the program. In large programs (more than 200 kB), the editor cancels the syntax coloring.

When selecting this option, the CNC shows the dialog box with two panels. The first one has the elements of the editing window that may be customized and the second one those of the program. The panel is selected by the key combination [CTRL]+[TAB].



After defining the new look, use the cursor to select one of the buttons here below to accept or ignore the changes made and press [ENTER]. The dialog box may also be closed directly without accepting the changes by pressing [ESC].

### Screen editor "Save always"

This softkey is used to activate the automatic program saving feature. The softkey appears pressed when this option is enabled.

- When this option is active, the CNC will automatically save the program every time the cursor changes blocks. In large programs (more than 200 kB), the CNC saves the program when the user has not modified the program for about 5 seconds.
- If this option is not active, it is up to the user to save the program using the softkey menu.

14.

PLC

FAGOR

CNC 8070

### 14.5 Editing in contact (ladder) language (softkeys).

The following softkeys may be useful when editing.

Hotkey.	Function.
[+][-]	Window zoom.
[CTRL][+] [CTRL][-]	Cell zoom.
[ENTER]	It edits the element on which the cursor is located. If the cursor is on the left column, it will show the dialog for editing the block comment.
[♠][♠][♠][♣]	It moves the cursor in all 4 directions.
[SHIFT][♠] [SHIFT][♠] [SHIFT][♠] [SHIFT][♣]	It changes the selection.

### 14.5.1 Analyze

It analyzes the program searching for errors. The errors found will be displayed in the "Outputs" service window.



To close this window and return to the editor, press [ESC]. To return to the editor without closing the window, press the screen switching key.

### 14.5.2 File

It is used to restore, import or export a file.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

### File "Save"

This softkey saves the file that is being edited.

### File "Save as"

This softkey saves the file, that is being edited, with a different name. After saving the file, one keeps editing the new file.

### Softkey "Import file"

This softkey imports the translation of a program edited in mnemonic language into the program that is now being edited. This softkey is useful for converting a mnemonic-language file into a contact (ladder)-language file without having to compile the PLC project.

The following expressions in mnemonic language cannot be translated directly into contact (ladder) language and the CNC will translate them as follows.

- Expressions with the XOR operator.
- Expressions where the NOT operator affects several contacts.

Original expression.	Transformed expression.
a XOR b	(a OR b) AND (NOT a OR NOT b)
NOT (a AND b)	(NOT a OR NOT b)
NOT (a OR b)	(NOT a AND NOT b)

### File "Export file"

This softkey exports to a file the translation into mnemonic language of the program that is now being edited.



**CNC 8070** 

# 14.5.3 **Editing**

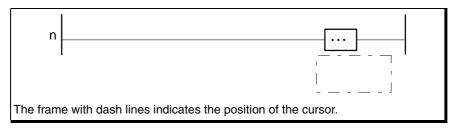
It may be used to edit the selected program and copy, cut and paste a block or group of blocks.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

#### Edit "New block".

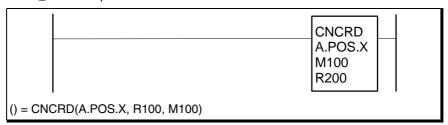
This softkey adds a new block with an empty contact. This contact will be the action for a consultation and "n" the number identifying the block. The new block is inserted above the block where the cursor is located.



Positioning the cursor on the action\_contact and pressing [ENTER] displays a dialog box for associating an instruction and a comment with it. The "Expected" field offers a context help with valid instructions.



The instruction must be written in mnemonic language. Placing the cursor on action\_contact and using the "Parallel" softkey, it is possible to add other action\_contact in parallel.



To define a directing instruction, activate the "Directing" field in the dialog box and then write an instruction. Directing instructions can only be programmed in a block with their action\_contact empty and without consultation\_contact.



#### Edit "Cut"

It cuts the selected contact or block.

## Edit "Copy"

It copies the selected contacts or blocks.

#### Edit "Paste"

It pastes the contacts or blocks that have been previously cut or copied. It pastes them in parallel with the selection.





**CNC 8070** 

# Edit "Delete"

It deletes the selected contacts or blocks.

#### Edit "Undo"

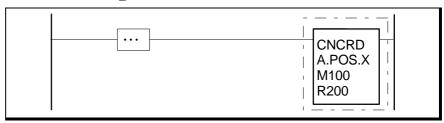
It may be used to "undo" the modifications made. The modifications are undone one by one starting from the most recent one.

#### Edit "Redo'

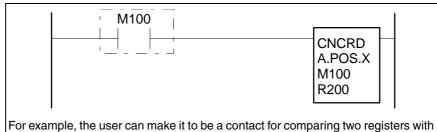
It is active after using "Undo" and restores one by one the previous status of the program, before "Undo" was used.

#### Edit "Left series"

It adds a consultation\_contact to the left of the contact selected with the cursor.



Positioning the cursor on the consultation\_contact and pressing [ENTER] displays a dialog box for associating an instruction and a comment with it. The "Expected" field offers a context help with valid instructions.

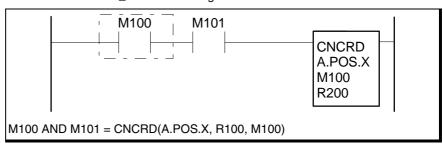


the instruction "CPS R1 EQ R2" or a simple contact of the mark "M100"

In mnemonics, it is the same as M100 = CNCRD(A.POS.X, R100, M100)

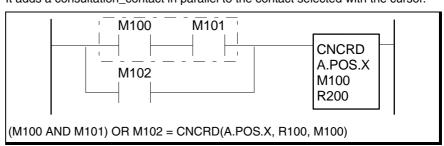
#### Edit "Right series"

It adds a consultation\_contact to the right of the contact selected with the cursor.



#### Edit "Parallel"

It adds a consultation\_contact in parallel to the contact selected with the cursor.

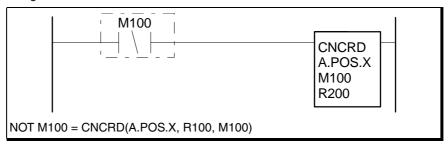




**CNC 8070** 

# Edit "Negate"

It negates the consultation\_contact selected with the cursor.



**PLC** Editing in contact (ladder) language (softkeys).



**CNC 8070** 

It may be used to enlarge or reduce the size of the contacts and of the text.

It increases the size of the font as well as the height and width of the contacts.

View "Zoom -"

It reduces the size of the font as well as the height and width of the contacts.

View "Cell +"

It increases the width of the contacts.

View "Cell -"

It reduces the width of the contacts.

# 14.5.5 Marks

It may be used to add or remove follow-up marks in the program.

Ð

When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

#### Marks "Set / remove"

It places or removes a mark on the block where the cursor is located.

Marks "Next mark"

Places the cursor on the next marked block.

Marks "Previous mark"

Places the cursor on the previous marked block.

Marks "Remove all"

Removes all the marks.



**CNC 8070** 

(REF: 0809)

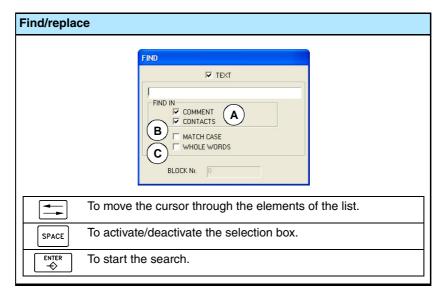
**PLC** Editing in contact (ladder) language (softkeys).

It is used to find a text or a block by its number.

#### Find "Find"

When selecting this option, the CNC shows a dialog box requesting the block number or the text to look for. When defining a text search, certain options may also be defining that allow:

- A. Select the types of fields in which to search.
- **B.** Ignore uppercase and lowercase.
- C. Consider the text to find as a whole word.



After defining the search options, press [ENTER] to do the search or [ESC] to cancel it. Pressing [ENTER] positions the cursor in the first field that matches the search parameters.

#### Find "Find next"

It positions the cursor in the next field that matches the search parameters.

#### Find "Find previous"

It positions the cursor in the previous field that matches the search parameters.

14.

PLC

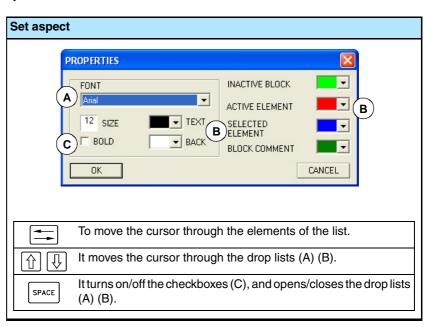
Editing in contact (ladder) language (softkeys).



**CNC 8070** 

# 14.5.7 Customizing

It is used for personalizing the appearance and properties of the editing window. When selecting this option, the CNC displays a dialog box showing the elements that may be set.



After defining the new look, use the cursor to select one of the buttons here below to accept or ignore the changes made and press [ENTER]. The dialog box may also be closed directly without accepting the changes by pressing [ESC].



PLC Editing in contact (ladder) language (softkeys).



**CNC 8070** 

# 14.6 Program monitoring

To monitor a program, select it from the list of the service window and select the "Monitoring" option of the softkey menu. The CNC will display the monitoring window and it will show the instructions of the selected program.

To close the monitoring window, press [ESC].

# Description of the monitoring window

The editing window shows the following information:



#### A. Title bar.

It shows the name of the program that is being monitored.

B. Monitoring area.

It shows the status of the instructions being executed. It is a real-time monitoring. Only the instructions that are being executed are monitored.

The PLC will show, in a different color, the instructions that are not being executed such as the first cycle or those subroutines whose call is not active. The default colors are:

Red Active variables.

Green Instruction that is not being executed.

Black Inactive variable or comment.

**C.** In a program edited in C language or in mnemonic language, it is the data entry zone. This zone is used to modify the values of the PLC resources.

In a program edited in contact (ladder) language, it is the zone that shows the comments of a contact and other messages. In the "Monitoring" service, it is not possible to change the names or comments associated with the contacts.

14.

PLC Program monitoring



**CNC 8070** 

# 14.6.1 Monitoring in C language or mnemonic language (softkeys).



This softkey toggles the display between the mnemonic language and the contact language (ladder diagram). The softkeys of the horizontal menu change according to the selected language.

In a program edited in C language or in mnemonic language, the softkey menu shows the following options.

#### Monitoring "Size +"

Every time this softkey is pressed, the size of the text will increase.

#### Monitoring "Size -"

Every time this softkey is pressed, the size of the text will decrease.

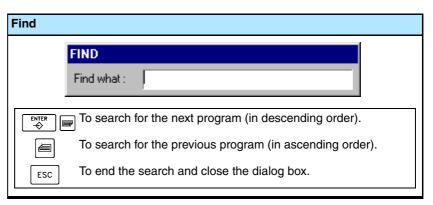
#### Monitoring "Bold"

When selecting this option, the text of the program appears in bold.

#### Monitoring "Find"

To find text in a program.

When selecting this option, the CNC shows a dialog box requesting the text to be found. Key in the text and press [ENTER] to begin the search.



As the text is being searched, the cursor will position on the match found. To end the search, press [ESC].

#### Monitoring "Customize"

It is used to personalize certain functions of the monitoring window. When selecting this option, the CNC shows a dialog box with the options available for customizing.



**CNC 8070** 

# 14.6.2 Monitoring in contact (ladder) language (softkeys).



This softkey toggles the display between the mnemonic language and the contact language (ladder diagram). The softkeys of the horizontal menu change according to the selected language.

In a program edited in contact (ladder) language, the softkey menu shows the following options.

## View

It may be used to enlarge or reduce the size of the contacts and of the text.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

#### View "Zoom +"

It increases the size of the font as well as the height and width of the contacts.

#### View "Zoom -"

It reduces the size of the font as well as the height and width of the contacts.

#### View "Cell +"

It increases the width of the contacts.

#### View "Cell -"

It reduces the width of the contacts.

#### Marks

It may be used to add or remove follow-up marks in the program.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

#### Marks "Set / remove"

It places or removes a mark on the block where the cursor is located.

#### Marks "Next mark"

Places the cursor on the next marked block.

#### Marks "Previous mark"

Places the cursor on the previous marked block.

#### Marks "Remove all"

Removes all the marks.

# Find

It is used to find a text or a block by its number.

#### Find "Find"

When selecting this option, the CNC shows a dialog box requesting the block number or the text to look for.

#### Find "Find next"

It positions the cursor in the next field that matches the search parameters.

14.

PLC



CNC 8070

## Find "Find previous"

It positions the cursor in the previous field that matches the search parameters.

# Customizing

It is used for personalizing the appearance and properties of the editing window. When selecting this option, the CNC displays a dialog box showing the elements that may be set.

## **Activate**

It is used to turn monitoring on or off. The program is not monitored until the "Activate" option is pressed.



PLC ogram monitoring



**CNC 8070** 

# 14.7 "Commands" service

The "Commands" service is used to debug the PLC program execution by taking advantage of the possibility to execute the various parts of the program separately (first cycle, main module and periodic module). This service also offers the possibility of starting up and interrupting the execution of the PLC program.

When selecting this service, the horizontal softkey menu offers all the options associated with this service.

# 14.7.1 Options of the "Commands" service (softkeys)

#### Commands "Run"

It starts the execution of the PLC program.

- 1. The first cycle (CY1) is executed once.
- 2. Cyclic execution of the main program (PRG) and the periodic module (PE).

The main program is executed according to the frequency defined by machine parameter.

The periodic module is executed according to the frequency defined in the program.

#### Commands "Stop"

It interrupts the execution of the PLC program.

#### Commands "CY1"

It executes the part of the program corresponding to the first cycle (CY1). The CNC does not execute this option when the PLC program is running.

#### Commands "Cycle"

It executes the main program (PRG) once. The CNC does not execute this option when the PLC program is running.

## Commands "Resume"

It resumes the cyclic execution of the PLC program. The CNC does not execute this option when the PLC program is running.

#### Commands "PE module"

It executes the periodic module (PE) once. The CNC does not execute this option when the PLC program is running.

14.

PL( mands" servic

FAGOR

**CNC 8070** 

# 14.8 "Outputs" service

The "Outputs" service shows the following information:

- The result of generating (compiling) the PLC program.
- The result of analyzing a file of mnemonics.
- The result of text search in the files making up the PLC project.

Pressing the "Show" softkey accesses the screen of this service that shows this information. To close the window, press [ESC].

## Description of the outputs service

The screen of this service looks like this:

#### A. Title bar.

It shows the type of information displayed in the window (information about the compilation, file analysis or a search).

B. Requested information.

It shows the requested information.



**CNC 8070** 

# 14.8.1 Options of the "Outputs" service

When accessing the "Outputs" service, the softkeys will show the following option.

#### Outputs "Compiling info"

It shows the result of compiling (generating) the PLC program. The screen shows a list of warning or errors detected when compiling the PLC program.



After selecting a warning or error from the list, it is possible to go to the erroneous line by pressing the relevant icon.

# Keys to move the cursor To select the previous or next message or error. To select the previous or next information page. To select the first or last message or error from the list.

# Outputs "Checking info"

It shows the result of analyzing the file of mnemonics. The screen shows a list of the error detected when analyzing the program.

After selecting a warning or error from the list, it is possible to go to the erroneous line by pressing the relevant icon.

#### Outputs "Search info"

It shows the result of searching text in the programs making up the PLC project.

After selecting an element from the list, it is possible to go to its line by pressing the relevant icon.

14.

PLC



**CNC 8070** 

# 14.9 "Logic analyzer" service

The "Logic analyzer" service is used to analyze the behavior of the logic PLC signals according to a time base and certain trigger conditions set by the user.

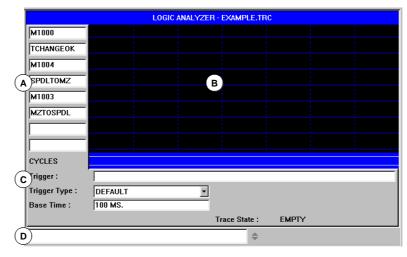
Up to 8 PLC variables or expressions may be analyzed at the same time and the result of the analysis (the traces) will appear on a graphic interface that facilitates the interpretation of the data. Once the done capturing data, the user may:

- Modify the time base to display different zooms of the traces.
- Move along the traces to display points, times, time differences, etc.

To close the window, press [ESC].

# Description of the logic analizer

The logic analyzer looks like this:



- **A.** The data area is used to define the variables or expressions to be analyzed. Up to 8 PLC variables or expressions may be defined.
- **B.** The graphic area shows the traces for the defined PLC variables or expressions and a trace of the PLC cycles with the indicated conditions.

Also, a vertical red line is displayed to indicate the trigger point (if any) and a green line to indicate the cursor position.

- **C.** The information area is used to define the data that conditions the display of the traces (trigger, type of trigger, time base and trace status).
- **D.** The data entry area is used to modify the PLC resources.

## Softkey menus

When selecting the logic analyzer, the horizontal softkey menu offers all the options associated with this service. The set of options offered by this menu may be changed using the "+" softkey which offers a new set of options.



PLC

"Logic analyzer" servic

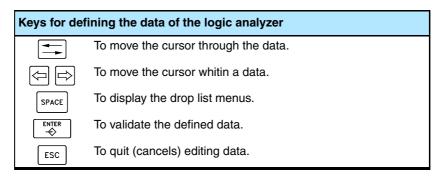


**CNC 8070** 

# 14.9.1 Editing logic analyzer data

In order to capture trace data, the user must enter the variables or expressions to be analyzed, the trigger type and conditions as well as the time based to be used to display the captured values.

To edit this data, select the graphic area of the logic analyzer and press the "Edit view" softkey to place the cursor on the data editing area.



#### **Definition of variables**

Up to 8 PLC variables or expressions may be defined to obtain their trace. The definition of the expression cannot exceed 80 characters.

When modifying a variable, if there was already a trace of that expression, the trace will be deleted when validating the new edition.

#### **Trigger condition**

A trigger condition is the one used to capture data and may be defined with a PLC variable or expression. The definition of the expression cannot exceed 80 characters.

If the trigger condition is modified after a data capture, all the traces will be deleted when validating the new condition.

#### **Trigger type**

It sets whether the data is to be captured "before", "after" or "before and after" the selected trigger condition is met.

option to execute and stop the trace.

Before The data capture starts when the trigger condition is met and ends

when the operator selects the option to stop the trace.

Once the trace has been executed, the trigger signal will be shown

at the beginning of the trace.

After The data capture starts when the operator selects the option to

execute the trace and ends when the trigger condition is met.

Once the trace has been executed, the trigger signal will appear

at the end of the trace.

In the middle 
The data capture starts and ends when the operator selects the

option to execute and stop the trace.

Once the trace has been executed, the trigger signal will be shown

at the center of the trace.

#### Time base

The operator may use this parameter to assign the timeframe to each vertical division. The size of these divisions, therefore the resolution of these signals will be determined by this time base. Consequently, the smaller the time base, the greater the signal resolution.

14.

PLC

"Logic analyzer" service



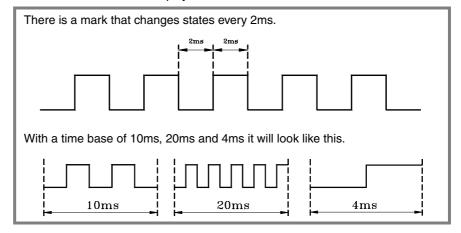
**CNC 8070** 

14.

PLC

"Logic analyzer" service

The value is given in milliseconds or microseconds according to the active units and the selected value will be displayed in the information area.



While the graphic window is selected, use the "+" and "-" keys to divide or multiply this time base by two.

#### Start distance / Trigger distance

This data is only shown when there is a trace.

Start distance It represents the time difference between the indicator

cursor and the base point of the trace (beginning of the trace if there is no trigger point or it has not been

reached).

Trigger distance It represents the time difference between the indicator

cursor and the trigger point (if it has taken place).

The value is updated when moving the indicator cursor of the graphic area. Its value may also be edited, thus updating the cursor position in the graphic area.

#### Reference distance

This data is only shown when there is a trace already and the operator has put a reference signal. It represents the time difference between the indicator cursor and the reference signal.

The value is updated when moving the indicator cursor of the graphic area. Its value may also be edited, thus updating the cursor position in the graphic area.

#### **Trace status**

This element cannot be edited, it automatically reflects the status of the trace. The possible messages are:

- Empty.
- Beginning.
- Executing.
- Stopping.
- Full.



**CNC 8070** 

# 14.9.2 Save, load and reset the analyzer configuration

#### Save the configuration

It is used to save the current configuration of the logic analyzer (PLC variables and expressions, trigger conditions, graphic traces) into a file.

When selecting this option, the CNC shows a list of trace files (TRC) already saved. To save the current configuration:

- 1. Define the file name at the bottom window. To replace an existing file, select it from the list.
- 2. Press [ENTER] to save the configuration or [ESC] to return to the logic analyzer without saving the configuration.

#### Load the configuration

It is used to recover a logic analyzer configuration that has been stored earlier.

When selecting this option, the CNC shows a list of trace files (TRC) already saved. To load one of these files:

- 1. Define the file name at the bottom window or select it from the list.
- 2. Press [ENTER] to save the configuration or [ESC] to return to the logic analyzer without loading the configuration.

#### Initialize the configuration

It is used to reset all the analyzer data eliminating the defined variables or expressions as well as the trigger condition and trigger type. The traces are also deleted because there are no variables to be analyzed.

14.

"Logic analyzer" service



**CNC 8070** 

# 14.9.3 Execute and analyze trace

#### **Execute trace / Stop trace**

It used to start or stop the data capture to display the traces.

#### Analyze trace

It is used to analyze the different graphic aspects of the trace.



When selecting this option, the softkey menu shows the options available for analyzing the trace. To return to the main menu, press the key for the previous menu.

· Go to the beginning

It moves the indicator cursor and the current graphic view to the beginning of the trace.

· Go to the end

It moves the indicator cursor and the current graphic view to the end of the trace.

· Go to the time

It moves the indicator cursor and the current graphic view to the time value set by the operator with respect to the base point of the trace (starting point of the trace if there is no trigger or to the trigger point if there is one).

· Go to trigger

It moves the indicator cursor and the current graphic view to the trigger point.

· Go to reference

It moves the indicator cursor and the current graphic view to the reference point defined earlier.

Set reference

It sets a reference point at the current cursor position for time difference calculation.

· Remove reference

Removes the reference point defined earlier.

#### Trace view / Edit view

It toggles between the graphic window and the data and display conditions area.

#### MS (milliseconds) / US (microseconds)

It toggles between milliseconds (0.001<sup>-3</sup> seconds) and microseconds (10<sup>-6</sup> seconds).

#### **Print**

It is used to print the graph on a printer connected to the CNC or as a BMP file in the CNC. When printing to a file, it is saved in the folder:

"C:\Cnc8070\Mtb\Plc\Watch\\*.bmp"



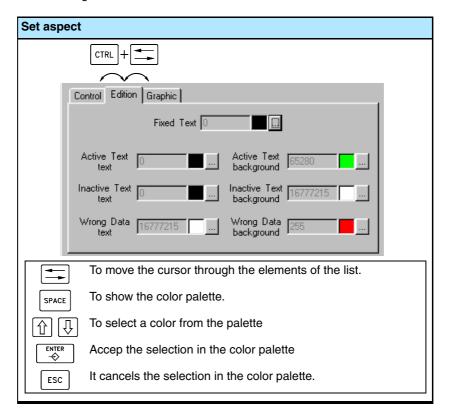


**CNC 8070** 

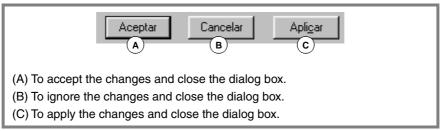
# 14.9.4 Customize the appearance of the logic analyzer

It is used to customize the looks of the different elements of the logic analyzer.

When selecting this option, the CNC shows a dialog box with the options available for customizing.



After defining the new look, use the cursor to select one of the buttons here below to accept or ignore the changes made and press [ENTER].



The dialog box may also be closed directly without accepting the changes by pressing [ESC].



analyzer" servic



**CNC 8070** 

# 14.10 "Monitoring" service

This service is used to analyze the status of the various PLC variables and resources.

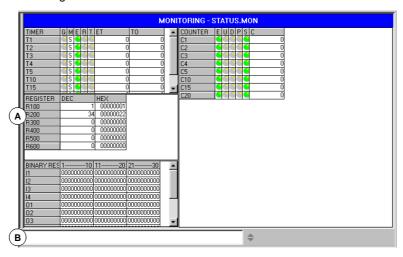
When selecting the "Monitoring" option at the service window, it will show a list with the three resource files used last. This will let you recover one of them without having to define it again.

Pressing the "Show" softkey access the service screen. If a file was selected from the list, it will show the resources defined in it. To close the monitoring window, press [ESC].

## Description of the monitoring service

This service screen is divided into several windows that may be accessed with the window switching key.

The monitoring screen looks like this:



#### A. Resources selected for monitoring.

It shows the user resources and symbols being analyzed. They are shown grouped in the following tables:

Timers Counters

Registers Binary resources

Each of them shows information on the status of those resources.

## B. Data entry area.

It is used to define the user resources and symbols to be analyzed and modify their values by directly assigning a value to them.

# Softkey menus

When selecting this service, the horizontal softkey menu will show all the options associated with resource monitoring. The set of options offered by this menu may be changed using the "+" softkey which offers a new set of options.



Monitorina" servic



**CNC 8070** 

# 14.10.1 Description of resource tables

As mentioned earlier, the user resources and symbols defined are displayed in different tables. Each table shows the following information.

#### **Timers**

This table shows the timers and user defined symbols for the timers and their status.

This table has the following fields:

- G It indicates whether any of the trigger inputs of the timer is active (TG1, TG2, TG3 or TG4).
- M Timer status:
  - S Stopped.
  - T Timing.
  - D Disabled.
- E Status of the Enable input (TEN).
- R Status of the Reset input (TRS)
- T Status output.
- ET Elapsed time.
- TO Remaining time.

The inputs and outputs that are high will be indicated with a green symbol.

#### Registers

This table shows the registers and user symbols defined for the registers as well as their values.

The values may be shown in decimal, hexadecimal or binary format.

#### **Binary resources**

This table shows the binary resources (inputs, outputs, messages, errors) and the user defined symbols for the binary resources as well as information on the their status.

#### Counters

This table shows the counters and user symbols defined for the counters as well as their status.

This table has the following fields:

- E Status of the enable input (CEN).
- U Status of the Count-up input (CUP).
- D Status of the Countdown input (CDW).
- P Status of the preset input (CPR).
- S Status output.
- C Value of the internal count of the counter.

The inputs and outputs that are high will be indicated with a green symbol.

14.

Monitoring" service



**CNC 8070** 

# 14.10.2 Definition of the table resources

Each table shows the following resources and user symbols:

Timer table

- Timers (T1...T256).
- · User symbols defined for the timers.

#### Counter table

- Counters (C1...C256).
- User symbols defined for the counters.

#### Register table

- Registers (R1...R1024).
- User symbols defined for the registers.
- PLC-CNC register variables.

#### Binary resource table

- Inputs (I1...I1024) and outputs (O1...O1024).
- Marks (M1...M8192).
- Messages (MSG1...MSG1024) and errors (ERR1...ERR1024).
- User symbols defined for the binary resources.
- · CNC-PLC boolean variables.

When defining a resource or user symbol, it will be added to the corresponding table.

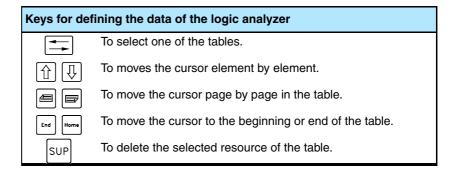
# **Data entry**

The resources and user symbols of the tables are defined at the data entry area. Their values may be changed by directly assigning a value to them (M110=1, R300=34). If it is a hexadecimal value, it must be preceded by the "\$" sign (M10=\$1, R200=\$20).

The window saves the last "N" assignments so they can be recalled later on. In order to make selecting easier, it is possible to display a window with the list of the assignments already made.

# 

To remove a resource or user symbol from a table, select that element and press [SUP].



14.

PL( Monitorina" servic



**CNC 8070** 

# 14.10.3 Options of the "Monitoring" service (softkeys).

When accessing the "Monitoring" option of the softkey menu, the following options will be available.

#### Save set

It is used to save the set of defined resources into a file.

When selecting this option, the CNC displays a window with the list of the files (.MON) currently saved. To save a set of defined resources:

- Define the file name at the bottom window. To replace an existing one, select it from the list.
- **2.** Press [ENTER] to save the program or [ESC] to cancel the selection and close the program listing.

#### Load set

It is used to recover the set of resources previously saved into a file.

When selecting this option, the CNC shows a window with the list of available files. To load one of these files:

- 1. Define the file name at the bottom window or select it from the list.
- 2. Press [ENTER] to load the selected file or [ESC] to cancel the operation and close the program listing.

#### Binary

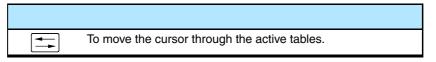
It toggles between displaying the register values in decimal and hexadecimal format or only in binary.

#### Visibility

It shows or hides certain resource tables.

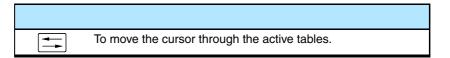
#### Add row

It increases the size of the active window (where the cursor is) by adding a file to it.



#### Remove row

It decreases the size of the active window (where the cursor is) by removing a row from it.



#### **Data input**

It selects the data entry area.

#### Clear all

It deletes all the resources from the tables.

14.

onitoring" service



**CNC 8070** 

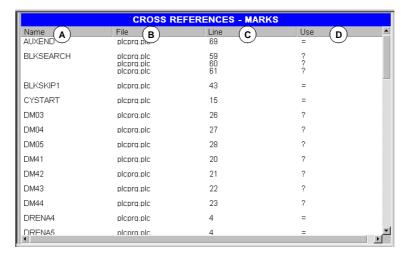
# 14.11 "Cross references" service

The "Cross references" service may be used to obtain information about the PLC resources being used in the PLC project.

Pressing the "Show" softkey accesses the window of this service. To close the cross-reference window, press [ESC].

# **Description**

The screen of this service looks like this:



- A. Name of the resources being used by the PLC project.
- **B.** File of the PLC project where the resource is being used.
- C. Line, within the file, containing the resource.
- D. How the resource is being used.

For the inputs, outputs, marks and registers:

- ? The resource is used as a consultation.
- = The resource is used as an action.

For the timers.

TG Timer trigger.

T Status consultation.

TEN Enable input.

TRS Reset input.

For the counters.

C Status consultation.

CPR Counter preset.

CEN Enable input.

CUP Count-up input.

CDW Countdown input.

14.

PLC 'Cross references" service



**CNC 8070** 

# 14.11.1 Options of the "Cross reference" service (softkeys)

When accessing the "Cross reference" service, the softkey menu will offer the following options:

#### Inputs

It shows information about the inputs.

#### **Outputs**

It shows information about the outputs.

#### Marks

It shows information about the marks.

#### Registers

It shows information about the registers.

#### **Timers**

It shows information about the timers.

#### Counters

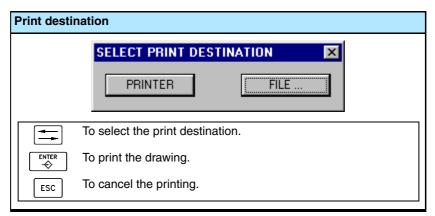
It shows information about the counters.

#### **Print**

It is used to print the cross reference tables out to a printer connected to the CNC or as a "\*.PRN" file at the CNC. When printing to a file, it is saved in the folder:

"C:\Cnc8070\Users\Reports\\*.prn"

When selecting this option, the CNC shows a dialog box requesting the destination for the file (printer or file). When printing to a file, it is possible to select the name and location of the file.



After selecting the destination, press [ENTER] to print the table or [ESC] to cancel it.



PLC Cross references" service



**CNC 8070** 

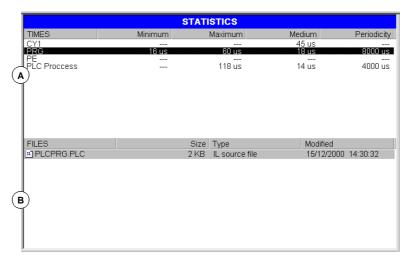
# 14.12 "Statistics" service

The "Statistics" service may be used to obtain information about the execution times of the PLC and the files making up the PLC project.

Pressing the "Show" softkey access the service screen. To close the statistics screen, press [ESC].

# **Description**

The screen of this service looks like this:



#### A. Execution time-table.

This table shows the following data (from left to right):

- Modules making up the PLC program.
- Minimum module execution time.
- Maximum module execution time.
- Average module execution time.
- Module execution frequency.

#### B. PLC file table.

This table shows the following data (from left to right):

- Files making up the PLC project.
- · Size of each file.
- File type.
- Date last modified.



**CNC 8070** 

# 14.12.1 Options of the "Statistics" service (softkeys)

#### Refresh

It refreshes the table information.

#### Print

This softkey may be used to print the table in the pre-determined printer or save it as a file (prn format) at the CNC. When selecting the "File" option, it will be saved in the folder "C:\Cnc8070\Users\Reports\".

When selecting this option, the CNC will show a dialog box requesting the print destination (printer or file). After selecting the destination, press [ENTER] to print it or [ESC] to cancel it.

14.

'Statistics" service



**CNC 8070** 

# 14.13 "Messages" service

The "Messages" service may be used to edit the texts associated with PLC messages and errors. These files may also be displayed and edited (and thus translated into other languages) using any text editor.

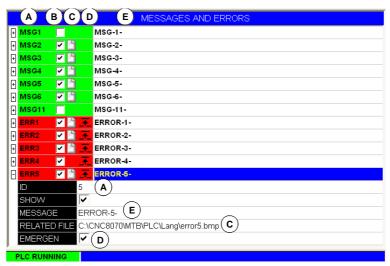
The message and error files are stored in the folder: "C:\CNC8070\MTB\PLC\LANG\<language>" corresponding to the language active at the CNC. To use messages and errors in other languages, copy the files to the folder of the desired language.

On CNC power-up, the messages and errors are loaded from the folder corresponding to the language active at the CNC.

Pressing the "Show" softkey access the service screen. To close the message window, press [ESC].

# **Description**

This screen shows the messages (MSG) and errors (ERR) defined at the PLC. The screen shows the following table.



A. Message or error identifier.

Number, between 1 and 1024, that identifies the message (MSG) or the error (ERR). The numbering needs not be sequential; i.e. number skipping is allowed.

- **B.** Display of the additional information files. This field indicates whether to show or not the file for additional information when displaying the message or the error.
- **C.** Address of the additional data file which may be a text file (\*.txt), image file (\*.bmp, \*.ipg), a combination of both (\*.htm, \*.html) or a video (\*.avi).
- D. Activate the emergency signal (\_EMERGEN) or not when activating the error. This field appears selected by default when editing a new error. When editing messages, this field does not appear because PLC messages do not activate the emergency signal.
- E. Text of the message or error displayed for the user.



PLo Messages" servic



**CNC 8070** 

# 14.13.1 Options of the "Messages" service (softkeys)

#### Save

It saves the message and error table into a file in ASCII format (\*.MEF).

The contents of the table may be saved at the CNC, in a floppy disk or in another CNC (or PC) via Ethernet.

#### Load

It recovers the table values saved earlier in the hard disk of the CNC.

#### **Print**

It prints the message and error table out to a printer connected to the CNC or as a file (\*.PRN format) at the CNC.

#### New message

It is used to define a new message in the table.

#### **New error**

It is used to define a new error in the table.

#### Select information

It selects the file for additional information.

#### Delete

It deletes the selected message or error from the table.

14.

PLC

"Messages" service



**CNC 8070** 

# 14.13.2 Editing the message and error table

To add a new message or error to the table, press the "New message" or "New error" softkey. The CNC adds a new row to the table and it identifies it with the label -MSG-or -ERR-, indicating that it corresponds to a message or to an error. To delete a message or an error from the table, select it with the cursor and press the "Delete" softkey.

To edit or modify the table data, use the cursor to select the field whose value is to be changed and define the data as follows:

- The number and text of the message or error is entered directly from the keyboard.
- The "show" option is turned on and off with the [SPACE] key.
- The "EMERGEN" option is turned on and off with the [SPACE] key. This option only appears when editing errors.
- The file for additional information may be entered directly or may be selected with the "Associate file" softkey.

Press [ENTER] every time a data is edited to accept the new value or [ESC] to cancel it



PL(



**CNC 8070** 

# 14.13.3 Displaying PLC messages

When activating a message (marks "MSG1" through "MSG1024"), the PLC message window (top right) shows the message number and its associated text. If the message has been defined so it shows a file with additional information, it will be displayed at full screen (if the file does not exist, a blue screen will be displayed).

If there are several active messages, it will display the one with highest priority (the one with lowest number). It will also show the "+" sign next to the PLC message window to indicate that there are more messages activated by the PLC.

# Message window

In order to expand the PLC message window and display the active message list, press the key combination [CTRL]+[M]. The list will show a symbol next to each message to indicate that it has a file with additional information associated with it.



It does not have a file with additional information.



It has a file with additional information.

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. To close the additional information window, press [ESC].

#### File with additional information

The files with additional information, defined by the machine manufacturer, may be displayed automatically when activating the message (if so defined in the table for messages and errors) or when requested by the user (at the message window).

The following keys are available to move around in the additional information window depending on the type of file displayed (TXT, BMP, JPG, HTM, HTML or AVI):

• TXT, BMP, JPG, HTM, HTML file.

· AVI file.

Home

Home

To stop and moves the video forward or backward 1 frame, 5 frames or the whole video.

SPACE

To stop the video.

ENTER 🔷

To play the video.

14.

PLC

"Messages" service



**CNC 8070** 

# 14.13.4 Displaying PLC errors

When activating an error (marks "ERR1" through "ERR1024"), part-program execution is interrupted and the center of the screen displays a window with the error number and its associated text. If the error has been defined so it shows a file with additional information, it will be displayed at full screen. If the error has the "Emergen" field selected, the error will open the emergency relay of the CNC.

#### Error window.



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].

When there is an active error, no other action but eliminating the error state is allowed. Although the window displaying the errors may be closed by pressing [ESC], it does not mean that the error status has been taken care of. To do that, press [RESET]. Pressing the [RESET] key assumes the initial conditions.

#### File with additional information

The files with additional information, defined by the machine manufacturer, may be displayed automatically when activating the error (if so defined in the table for messages and errors) or when requested by the user (at the error window).

The following keys are available to move around in the additional information window depending on the type of file displayed (TXT, BMP, JPG, HTM, HTML or AVI):

• TXT, BMP, JPG, HTM, HTML file.



To move the window contents up or down one line, one page or everything.

AVI file.



To stop and moves the video forward or backward 1 frame, 5 frames or the whole video.



To stop the video.



To play the video.



**CNC 8070** 

# 14.13.5 Grouping the additional information text files in a single file.

PLC messages and errors can show an additional information file in text format. The PLC allows grouping several or all these files into a single file as follows.

#### Defining the additional information file.

It must be a text file (extension txt) and may have any name. The information of each message and error must be structured in the following format:

[<id>] <text>

The <id> field, keeping the brackets, will be the code that identifies the help text inside the file, which needs not be the same as the number of the error or message it will be associated with. The <text> field will be the information text that may have up to 500 characters including line feeds.

For example, the OEM.txt file will have the following structure.

```
[10]
Help text.
[27]
Help text.
[33]
Help text.
```

#### Calling the texts from the PLC message or from the PLC error.

To associate the help message with a PLC message or PLC error, the "associated file" field must be defined like <file>#<id>. The <file> field will be the path and the name of the file. The <id> field will be the code that identifies the help text inside the file.

For example, the "associated file" will be defined as follows.

C:\CNC8070\MTB\PLC\LANG\OEM.txt#27

14.

PLC



**CNC 8070** 

# 14.13.6 Save, load and print a message and error table

#### Save table

To save the table data, press the "Save" softkey and the CNC will show a list with the files saved at the CNC.

To save the table data:

- 1. Select the destination folder.
- 2. Define the file name at the bottom window. To replace an existing file, select it from the list.
- 3. Press [ENTER] to save the file or [ESC] to cancel the operation.

The file will be saved with the extension: \*.MEF.

#### Load table

To recover the table data, press the "Load" softkey, the CNC will show a list with the files available at the CNC.

To load the table data:

- 1. Select the folder where the file is.
- 2. Select the file or write its name at the bottom window.
- **3.** Press [ENTER] to accept the selection or [ESC] to cancel it and close the file listing.

#### Print table

This softkey may be used to print the table of messages and errors in the predetermined printer or save it as a file (prn format) at the CNC. When selecting the "File" option, it will be saved in the folder "C:\Cnc8070\Users\Reports\".

When selecting this option, the CNC will show a dialog box requesting the print destination (printer or file). After selecting the destination, press [ENTER] to print it or [ESC] to cancel it.



**CNC 8070** 

# **MACHINE PARAMETERS**

In order for the machine tool to be able to properly execute the programmed instructions, the CNC must know the specific data of the machine such as feedrates, accelerations, feedback, automatic tool change, etc. This data is set by the machine manufacturer and must be defined in the machine parameter table.

At a CNC with a write-protected (read-only) disk, working in user mode, the machine parameters are write-protected and any changes made to it are temporary; i.e. it will disappear the next time the unit is turned on. To make the changes permanent, unprotect the machine parameters by starting the unit up in setup mode and validating the changes made.

# Selecting the machine parameter tables



The machine parameter tables may be accessed either using the user key if it has been set accordingly or from the task window. The task window may be accessed with key combination [CTRL]+[A].

# What can you do in this work mode?

The machine parameter operating mode offers the following operations:

- Edit and modify the machine parameter tables.
- Validate the machine parameter tables.
- · Saving and restoring the table contents.
- · Printing the table contents.
- · Initializing the tables.

The machine parameter tables may be edited in this mode or be copied from a floppy disk or other peripheral connected to the CNC via Ethernet.

#### **Parameter setting**

To set the machine parameter tables, proceed as follows:

1. Edit the values of the tables.

First, set the general machine parameters because they define the axes of the machine, thus the axis machine parameters.

The general machine parameters can also be used to define the data for the cross compensation.

Use the axis machine parameters to determine whether leadscrew error compensation will be applied or not and define the data of the corresponding table.

2. Validate the table data.

This operation is a must after defining the parameters of the tables and every time any of the data is modified.

3. Save the data of the table.

It is recommended to save the data of the machine parameter table so as to have a backup copy of them. The data may be stored at the CNC, in a floppy disk or at another CNC (or PC) connected through Ethernet.



**CNC 8070** 

# 15.1 Appearance of the machine parameter tables

When this mode is active, it will be indicated at the top right side of the general status bar.



This operating mode consists of several tables. The various tables may be selected using the horizontal softkeys. If while one of the tables is selected, the previous menu key is pressed, that table will be deselected.

The available machine parameter tables are:

- · General parameters.
- Axis and spindle parameters.
- · Jog parameters.
- Parameters related to the "M" functions.
- · Parameters for kinetics.
- · Parameters for the magazines.
- · HMI parameters.
- · OEM parameters.

# Standard configuration

The standard configuration of this work mode, described in this manual corresponds to the one supplied by Fagor. In some cases, the manual shows a sample screen, instead of that of the CNC in order to make it easier to understand.



**CNC 8070** 

MACHINE PARAMETERS

#### 15.1.1 Icon description (vertical softkeys)

The icons are activated with their associated softkey (F8 to F12).



The icon menu offers all the options for the selected table. The group of icons of this menu may be expanded with the softkey associated with the "+" icon that offers a new set of icons.

#### List of icons.



#### Display units (millimeters/inches)

To toggle between the two types of units used to represent the parameters that depend on those units (coordinates, feedrates, etc.).

The icon highlights the units currently selected (millimeters or inches).



#### Initialize table

It resets all the parameters to their default values.

The CNC will request confirmation of the command.



#### Find text

It is used to find a text or a value in the table.

When selecting this option, the CNC shows a dialog box requesting the text to be found.



#### Validate table

For validating the table data. If there is no table selected, all the data of all the tables will be validated.

This table is necessary after changing a data. In certain cases, the CNC will have to turned off and back on in order for the data to be validated.



#### Save tool

It saves the values of the table into a file. If there is no table selected, all the data of all the tables will be saved.

This file may be saved at the CNC, in a floppy disk or at another CNC (or PC) connected through Ethernet.



#### Recall table

It restores the values of the table previously saved into a file. If there is no table selected, all the data of all the tables will be recalled.



#### Print table

This softkey may be used to print the table in the pre-determined printer or save it as a file (prn format) at the CNC.



#### **Activate TEACH-IN**

It is used for activating the TEACH-IN work mode in order to set the leadscrew and cross compensation tables.



**CNC 8070** 



**CNC 8070** 

(REF: 0809)



## Exporting the table for cross compensation and leadscrew error compensation.

This icon may be used to export the cross compensation and leadscrew error compensation tables out to an ASCII file. The files will have the extension mp.

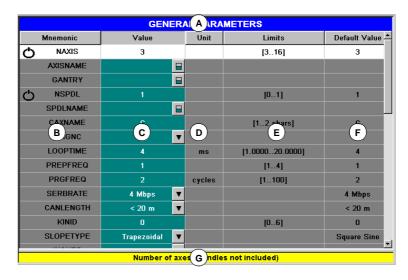


## Importing the table for cross compensation and leadscrew error compensation.

This icon may be used to import the cross compensation and leadscrew error compensation tables that were previously saved in a file.

#### 15.2 Parameter table description

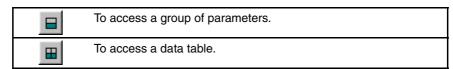
The machine parameter tables have the following structure:



- A. Name of the selected machine parameter table.
- B. Parameter list.
- C. Parameter value, set by the user.
- **D.** Units used to define the parameter.
- E. Limit values for the parameter.
- F. Default value of the parameter.
- G. Parameter description.

The section on "15.3.1 Data editing and validation" in this chapter shows how to edit the parameters of the tables.

Certain parameters cannot be edited; they only access a set of parameters or a data table. These parameters will be identified as follows.



To access the data associated with the parameter, select the parameter with the cursor and press [ENTER]. To return to the previous parameter table, press [ESC].



To obtain further information about the data appearing in the table, refer to the chapter on machine parameters of the Installation and Start-up Manual.

**15.** 

MACHINE PARAMETERS
Parameter table description

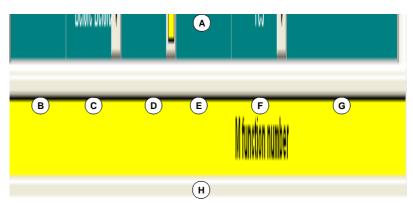


**CNC 8070** 

MACHINE PARAMETERS
Parameter table description

#### 15.2.1 "M" function setting table

This table is accessed using the M-function machine parameter "DATA" and it has the following structure:



- A. Name of the selected machine parameter table.
- B. List of user defined "M" functions.
- C. Type of synchronism.
- **D.** Associated subroutine.
- E. Time for execution time estimate.
- **F.** Send the M function to the PLC during block search.
- G. Function describing comment.
- H. Parameter description.

The section on "15.3.1 Data editing and validation" in this chapter shows how to edit the parameters of the tables.



To obtain further information about the data appearing in the table, refer to the chapter on machine parameters of the installation manual.



**CNC 8070** 

There are two types of compensation table:

- Leadscrew error compensation, that is accessed with the axis machine parameter "LSCRWDATA" when the compensation is active.
- Cross compensation, which is accessed with the general machine parameter "CROSSCOMP".

These compensation tables have the following structure:

GENERAL PARAMETERS : CASS COMPENSATION 1 VALUES				
Mnemonic	POSITION	POSERROR	NEGERROR	
CROSS COMP 0	12.2409	0.5849	0.3541	
CROSS COMP 1	15.7370	0.4511	0.5240	
CROSS COMP 2	21.2319	0.3257	0.8409	
CROSS COMP 3	28.2250	0.5417	0.9840	
CROS OMP 4	<b>C</b> 890	<b>D</b> 59	0. E	
CROS COMP 5	35.9840	155.0000	0.5-10/	
CROSS COMP 6	38.9810	0.6840	0.8970	
CROSS COMP 7	44.7079	0.6780	0.1409	
CROSS COMP 8	48.2049	0.4683	0.9799	
CROSS COMP 9	50.4690	0.9869	0.9847	
	Valor F orrección			

- A. Name of the selected machine parameter table.
- B. Compensation point.
- C. User defined axis position.
- **D.** Amount of error to compensate in the positive direction.
- **E.** Amount of error to compensate in the negative direction.
- F. Parameter description.

Sections "15.3.1 Data editing and validation" and "15.3.4 Importing and exporting compensation tables." of this chapter show how to edit the parameters of the tables.



To obtain further information about the data appearing in the table, refer to the chapter on machine parameters of the installation manual.

15.

MACHINE PARAMETERS
Parameter table description



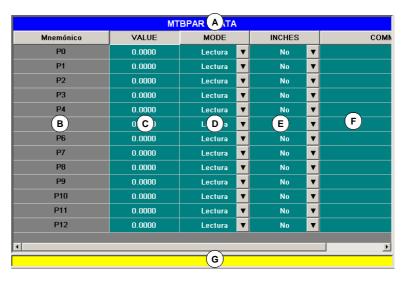
**CNC 8070** 

MACHINE PARAMETERS
Parameter table description

#### 15.2.3 OEM parameters

The OEM parameters may be read and written directly in the table or through their associated variables.

The "SIZE" parameter indicates the number of parameters in the table. The table, which is accessed with the general machine parameter "DATA", has the following structure.



- A. Name of the selected machine parameter table.
- B. Name of the parameter.
- C. Value of the parameter.
- **D.** Mode for accessing the parameter from its variable.
- E. The parameter is affected by the change of units.
- F. Function describing comment.

The values of the table may be modified at any time. The new values are assumed immediately without having to validate the parameters. This means that the values and permissions of the table may be modified while executing a program.

The simulation environment has a copy of this table. On CNC power-up, the values of the parameters of the real table are copied into the simulation table and from there on, they become different in the writing of the variables of both tables.

In the simulation table, only the parameter values may be modified, not the rest of the permissions. The values of the simulation table can only be read or modified through their variable.



To obtain further information about the data appearing in the table, refer to the chapter on machine parameters of the installation manual.



**CNC 8070** 

#### 15.3 Operations with tables

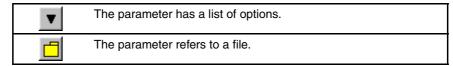
#### 15.3.1 Data editing and validation

Use the softkey menu to select the desired machine parameter table. To edit the table data, proceed as follows:

- 1. Use the cursor to select the parameter whose value is to be set.
- 2. Key in the desired value. If [SUP] is pressed, the parameter will assume the default value.
- 3. Press [ENTER] to accept the new value or [ESC] to ignore the new value and recover the previous one.

#### Parameter types

Certain parameters can only assume specific values. These parameters will be identified as follows.



Depending on the type of values they can assume, they will be defined as follows:

 If the parameter may assume a value between specific limits, enter the desired value.

If the value exceeds those limits, the parameter will assume its default value.

- If the parameter contains a list of options, press [ENTER] to display the drop list and select one of those options.
- If the parameter refers to a file, press [ENTER] and the CNC will show the list of available files.

#### Data validation.



After editing or modifying the table data, the new values must be validated so they are assumed by the CNC. To validate the data, press the VALIDATE icon.

To validate certain parameters, restart the CNC. These parameters will be identified with the following symbol next to their name:



Restarting the CNC required.

**15.** 

MACHINE PARAMETERS

Operations with tables



CNC 8070

### CNC 8070

**FAGOR** 

(REF: 0809)

#### Saving a table

It may be used to save the table data, in ASCII format, in a file.

Use the softkey menu to select the table whose data is to be saved. If no table is selected, all the data of all the tables will be saved.



After selecting the table whose data is to be saved, press the SAVE icon and the CNC shows a list with the tables that are already saved.

# Keys for selecting a data file To switch windows. To close and open a folder submenu. To move the cursor thru the elements of the window. ENTER To save the new program. ESC To cancel the selection and close the programs listing.

To save the table data, proceed as follows:

- 1. Select the destination folder.
- 2. Define the file name at the bottom window. To replace an existing file, select it from the list.
- 3. Press [ENTER] to save the file or [ESC] to return to the tables without saving the file

The files are saved with the extension \*.MP.

#### Recall a table

It may be used to restore the table data, in ASCII format, from a file.

Use the softkey menu to select the table whose data is to be recalled (restored). If no table is selected, all the data of all the tables will be recalled.



After selecting the table whose data is to be restored, press the LOAD icon and the CNC shows a list with the tables that are already saved.

To recall the table data, proceed as follows:

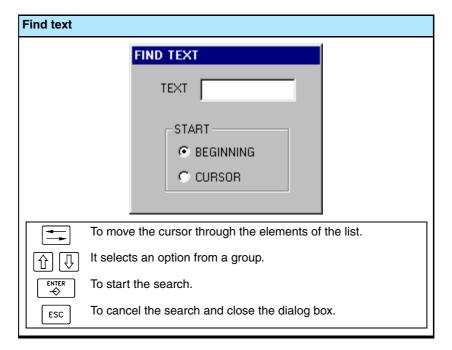
- 1. Select the folder containing the file.
- 2. Select the file from the list or write its name in the bottom window.
- 3. Press [ENTER] to recall the data or [ESC] to return to the tables without recalling the data.

#### 15.3.3 Find text

It is used to find text or a value in the table.



To perform the search, press the FIND icon and the CNC will show a dialog box requesting the text or value to be searched. It is also possible to select whether the search must start at the beginning of the table or at the current cursor position.



To start the search, press [ENTER] and the cursor will position on the text found. Pressing this FIND icon again will allow repeating the search or defining a new one.

**15.** 

MACHINE PARAMETERS

Operations with tables



**CNC 8070** 

#### 15.3.4 Importing and exporting compensation tables.

The cross compensation and leadscrew error compensation tables are saved and restored with the rest of general parameters and axis parameters using the softkeys "Save table" "Restore table". These tables may be exported and imported individually using the softkeys "Export table" and "Import table".



This softkey may be used to save the machine parameter table in an ASCII file. This softkey saves all the parameters of the table.



This softkey may be used to restore the machine parameter table from an ASCII file. This softkey restores all the parameters of the table.

When loading general parameters or axis parameters, neither the cross compensation tables nor the leadscrew error compensation tables are used.



This softkey may be used to export the cross compensation or leadscrew error compensation table being displayed out to an ASCII file.



This softkey may be used to import the cross compensation and leadscrew error compensation table from an ASCII file.

When saving the cross compensation and leadscrew error compensation tables, we recommend to use the axis name as the file name in order to identify it more easily; for example, LSCRWX.mp for leadscrew error compensation tables and CROSS\_XY.mp for cross compensation tables.

## Generating the ASCII file for cross compensation and leadscrew error compensation manually.

In order to save time and eliminate transcription errors, instead of entering the data manually, it is possible to adapt the format of a text file that contains the result of the measurement and import it later on. The file of points must have the extension mp and meet the following requirements.

- The points of the table must be ordered by their position on the axis and the table
  must begin by the most negative point (or least positive) to be compensated. For
  axis positioning outside this area, the CNC will apply the compensation that was
  defined for the nearest end.
- The machine reference point, whose position is indicated in parameter REFVALUE, must have  $\cdot 0 \cdot$  error.
- For the leadscrew error compensation table, the maximum slope allowed is ·1·; i.e. the compensation steps must not exceed the position step between two consecutive points.

The format of the file must be like in the following example. The first column, as well as the titles of the columns, are optional and do not appear in the mp file when exporting the tables. The data of the tables may be separated by a blank space or by a tabulator.

POINT	POSITION	POSERROR
0	-10,0000	-0,0023
1	0,0000	-0,0007
2	10,0000	0,0043
3	20,0000	0,0036

When using bi-directional compensation, a new column must be inserted in the table with the amount of error to be compensated when moving in the negative direction.



**CNC 8070** 

## SETUP ASSISTANCE

The set of utilities of the setup assistance is meant to speed up and simplify the machine setup procedure. The setup assistance offers the following utilities.

- · The oscilloscope.
- The circularity (roundness) test.
- The bode diagram.

It must be borne in mind that these utilities are assistance tools that only show the system's response to the various adjustments; it is always up to the technician to choose the best adjustment.

#### The oscilloscope

The oscilloscope is an assistance tool for adjusting the CNC and the drives. This utility can graphically show 4 previously selected variables, modify their values and check the different responses of the system.

Using the oscilloscope, it is possible to edit the CNC's machine parameters that affect the adjustment; it is also possible to modify at the Sercos<sup>®</sup> drives the variables that have writing permission. In any case, a protection password is required for modifying the CNC's machine parameters or the drive's variables.

#### The Bode diagram

The Bode diagram is an assistance tool for adjusting the CNC and the drives. This utility may be used to obtain the Bode diagram of the response amplitude depending on the frequency of the input signal.

With this tool, it is possible to insert a sinusoidal signal into an axis, so that changing the frequency and analyzing the response provides the behavior of the axes at different frequencies and will give the user an idea of how it behaves with any signal.

This diagram may be used to check the system's gain, the bandwidth and the mechanical resonance. All this helps achieve the proper adjustment of the loops, analyze mechanical problems and check the final features.

#### The circularity (roundness) test

This tool helps improve and adjust the axis reversal peak, an effect that appears when the axes reverse their moving direction. With this utility, a circle may be executed with the axes and graphically show the result. This graphic shows the difference between the actual (real) coordinate obtained from the feedback device and the theoretical coordinate calculated at each point.

The same screen showing the graphics also shows the machine parameters that affect the movement reversal. These parameters may be modified while running the test, thus being possible to evaluate the response of the system to these changes and optimize the adjustment on the go.



**CNC 8070** 

#### Selection of setup assistance



Setup assistance may be accessed either using the user key if it has been set accordingly or from the task window (by selecting the *Tuning* option). The task window may be accessed with key combination [CTRL]+[A].

The different tools are accessed from the horizontal softkey menu.



The oscilloscope.



The Bode diagram.



The circularity (roundness) test.



**CNC 8070** 

The oscilloscope is an assistance tool for adjusting the CNC and the drives. This utility can graphically show 4 previously selected variables, modify their values and check the different responses of the system.

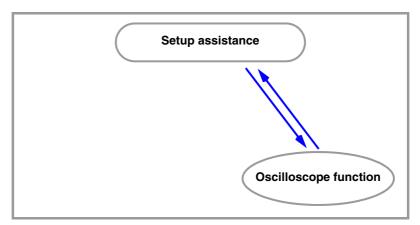
Using the oscilloscope, it is possible to edit the CNC's machine parameters that affect the adjustment; it is also possible to modify at the Sercos<sup>®</sup> drives the variables that have writing permission. In any case, a protection password is required for modifying the CNC's machine parameters or the drive's variables.

The oscilloscope is an assistance tool that only shows the response of the system to the various settings (adjustments); it is up to the technician to choose the best adjustment.

#### How to access the oscilloscope



The oscilloscope may be accessed from the softkey menu for setup assistance.



This utility is one of the features of setup assistance. Together with the circularity test and the bode diagram, it helps speed up and simplify the machine setup process.



The oscilloscope function requires Sercos communication and a drive version V06.01 or later. Otherwise, the CNC will show the error "Access denied".

16.

SETUP ASSISTANCE
Oscilloscope

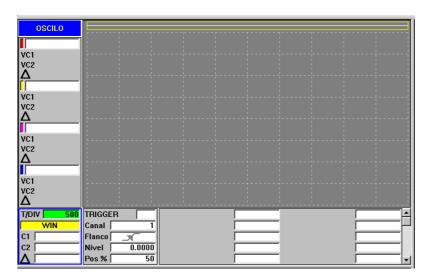


(REF: 0809)

**CNC 8070** 

#### 16.1.1 Interface description

The screen of the oscilloscope function looks like this, with two clearly different parts. The left side and the lower side show the data of the variables to be shown, the trigger data, the time base and the parameters that may be acted upon. The top right side shows the variables graphically.



#### The graphics window

It is the area showing a graphic representation of the selected variables. For each variable, it shows the scale used to draw it.

#### The data area

It is the area where the user interacts with the system. It allows defining the graphic environment, the variables to be display, the trigger configuration, the time base configuration and the values of the parameters involved.

Some of this data may also be defined at the configuration screen. See "Configuration screen" on page 305.

#### Variables to be displayed

It is the area where the user selects the variables to be shown in the graphics window. Up to 4 variables may be displayed at the same time.

This window does not allow selecting the color of each variable being displayed. To change the color of any variable, it must be done at the configuration screen.

#### Trigger data

In this area, the user may select the trigger conditions. The options offered by this trigger data area are:

#### Trigger

It indicates when the data capture begins.

If not selected, the data capture begins as soon as the user gives the command to begin. The "Flank", "Level" and "Position" data are ignored.

If selected, the trigger condition must be specified with the "Flank", "Level" and "Position" data.



SETUP ASSISTANCE
Oscilloscope



**CNC 8070** 

It indicates which variable or channel (CH1, CH2, CH3, CH4) is to be used as reference or trigger condition.

#### Flank

It may be up-flank (leading edge) or down-flank (trailing edge).

It is taken into account when trigger has been selected.

#### Trigger level

It sets the value to be assumed by the variable to start capturing data.

It is taken into account when trigger has been selected.

#### Position (%)

It is taken into account when Trigger has been selected.

It is defined as a percentage, between 0% and 100%. It indicates the number of samples to be taken before the Trigger. For example, a 10% position indicates that 10% of the total number of programmed samples will be taken before the Trigger and the remaining 90% after that instant.

The Trigger condition is evaluated after gathering the indicated % of samples. If the position is defined as 50% and the Trigger condition takes place after taking 10% of the samples, it is ignored until capturing 50% of the samples.

#### Machine parameters to be set

In this area of the data window, it is possible to directly change the values of up to 20 machine parameters to be adjusted.

#### Horizontal softkey menu

The horizontal softkey menu offers the following options:

· Softkey "Config".

It access the configuration screen. It may be used to define which variables will be displayed, their scale, colors, time base, the Trigger, the trigger condition, the parameters to act upon, etc.

· Softkey "Data".

It accesses the data processing submenu. This submenu serves to save and load the captured trace as well as the configuration defined for it.

It may also be used to send data of the captured trace out to printer or to a file.



· Enlarge screen

With this option, it is possible to enlarge the graphics window to the left, using the area that shows the variables to be displayed. Pressing the same softkey returns the screen to the previous size.

• Softkey "Overlap channels".

Wit this option, several channels may be overlapped.

When this option is selected, the softkey color changes to red.

· Softkey "Autoscale".

When autoscaling a channel, the system determines the proper vertical scale and offset so the signal appears as enlarged as possible inside its graphic area.



· Softkey "Freeze screen".

The display is "frozen" without interrupting the data capture. When this option is selected, the softkey shows "Resume".

16.

SETUP ASSISTANCE
Oscilloscope



**CNC 8070** 

This softkey only appears on the horizontal softkey menu if the capture is in continuous mode. To set the capture in continuous mode, change the "mode" option of the configuration screen.



· Softkey "Begin capture".

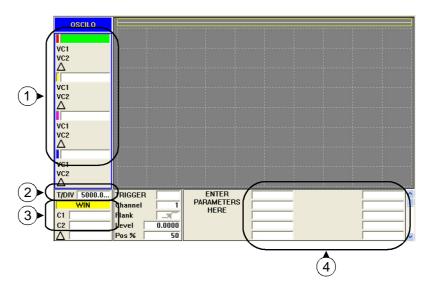
Start taking data to show it graphically.

When this option is selected, the softkey shows "Stop".

#### Vertical softkey menu

Depending on which data has the focus, the vertical softkeys show different options.

- Zone 1: Fields of variables.
- Zone 2: T"T/DIV" field.
- Zone 3: "WIN", "C1" and "C2" fields.
- Zone 4: Fields of machine parameters.



#### Zone 1. Focus on a variable of a channel.

When the focus is on any of the variables, the vertical softkeys will be:



The scale of the graphic may be changed with the "scale" softkeys. The new scale value will appear on the graphic next to the corresponding variable.





The graphics of the desired channel may be moved up and down using the offset softkeys.



This softkey may also be used to center the graphics.



When the focus is on any of the variables, a new variable name may be either keyed in or selected without having to go to "CONFIG".

16.

SETUP ASSISTANCE
Oscilloscope

FAGOR

**CNC 8070** 

#### Zone 2. Focus on the "T/Div" field.

When the focus is on the "T/Div" field, the following softkeys will appear.



Using the autoscale softkey, the oscilloscope selects the right time scale so the signal appears as large as possible inside its corresponding graphic area.





It increases or decreases the time scale.

#### Zone 3. Focus on the "WIN", "C1" and "C2" field.

With "WIN", it shows the oscilloscope window where it is. Cursors "C1" and "C2" may be used to analyze each signal of the last data capture. These two cursors may be used to obtain the position in milliseconds of each signal and the time difference between them " $\Delta$ ".

When the focus is on one of these fields, the following vertical softkeys will be shown.



It takes the window to the beginning.



It takes the window to the end.



It takes the window to the trigger. If there is no trigger, it will go to the center of the window.



Moves the window to the left.



Moves the window to the right.

#### Zone 4. Focus on the "parameter name" field.

As with variables, by selecting the various parameters, it is possible to change their names and values.

When pressing [ENTER] after changing the name of a parameter, use the [♠][♠][♠][♣] keys to move the focus to the next field to change the value of that parameter. After changing the parameter value and pressing [ENTER], the focus does not go on to the next field. Use the arrow keys to go on to the next parameter. When the focus is on this field, the 4 vertical softkeys will be:



It increases the value of the parameter in the amount indicated on the configuration screen.



It decreases the value of the parameter in the amount indicated on the configuration screen.



It resets the parameter to its default value.



It validates the value given to the parameter.

SETUP ASSISTANCE Oscilloscope



**CNC 8070** 

Using the "Config" softkey, a screen may be accessed where it is possible to define the list of parameters to be displayed.

SETUP ASSISTANCE
Oscilloscope

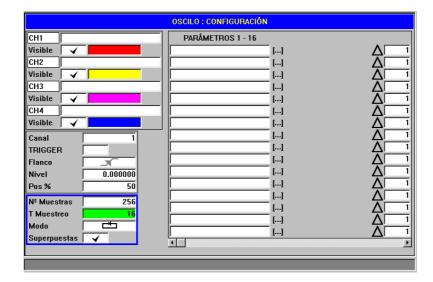


**CNC 8070** 

#### 16.1.2 Configuration screen

The configuration screen may be accessed with the horizontal softkey "CONFIG".

This screen looks like this with two clearly different areas. The left side shows the data of the variables to be displayed, the trigger data and the time base. The right side shows the parameters to be acted upon and the amount to increase/decrease the values of those parameters.



## Selection of the variables to be displayed, trigger and time base

It is the area where the user selects the variables to be displayed at the graphics window, the trigger options and the time base. Up to 4 variables may be displayed at the same time.

When typing the name of any variable incorrectly, the CNC will show the error "Access denied".

Besides the trigger data and the base time, this window may be used to select the color used to display the variable and whether it will be visible or not.

#### The sampling period.

The sampling period must be multiple of the machine parameter LOOPTIME; if a wrong value is entered, the CNC will set it to the right one.

When capturing only two variables of the same drive, it is possible to set a sampling time shorter than LOOPTIME. In this case, the sampling period must be multiple of 0.0625 ms which is the minimum allowed. Not all the variables of the drive accept this sampling time. The CNC will issue an error message when trying to represent a wrong variable.

#### Parameter selection and increase/decrease

It is the area where the user selects the parameters upon which to act. To the right of each parameter, there is a square used to increase/decrease of its value.

This way, when the focus is on the value of a variable on the main screen, pressing the vertical softkey "V+" will increase the parameter value in the amount set on the configuration screen.

By default, the increase/decrease value of a parameter is initialized to 1.

**16.** 

SETUP ASSISTANCE
Oscilloscope



**CNC 8070** 

#### 16.1.3 Configure and execute the oscilloscope function

In the oscilloscope mode, from a single screen, it is possible to select the variables to be analyzed, the trigger conditions and the machine parameters of the CNC or the drive that may be modified.

#### Operating procedure

Execute a part-program to move the axis or axes to be adjusted. Once the program is running, go into the adjustment mode and access the oscilloscope function.

Define the variables to be analyzed, the trigger conditions and the machine parameters of the CNC or the drive that may be modified.



When capturing 1 or 2 variables of the same drive, the capture will be done by the drive itself and the CNC will be in charge of drawing the results.

When capturing more than 2 variables or 2 variables from different drives, in order to synchronize data, the capture will be done by the CNC and the sampling period will be the CNC's cycle time.

Data capture and later analysis of the data.

Once the data capture has been interrupted or completed, it is possible to analyze the signals and modify the parameters that have been previously selected in order to improve the machining conditions.

Repeat the capture, the analysis and the modification of parameters until obtaining the best machining conditions.

#### Recommendations

Execute endless repetitive movements. After adjusting the axes individually, readjust the interpolating axes together.

The user must determine which is the best adjustment, the oscilloscope function is just an assistance tool.

#### Operation

To enter or modify a data on the screens, it must be selected; in other words, it must have the editing focus on it.

Use the  $[\begin{cases} \begin{cases} \begi$ 

Not all the fields are editable, only the ones that may be selected can be edited, those having the focus. There are two types of editable fields:

· Editable values:

They may be assigned a value, either a two-digit number or alphanumeric (numbers and letters). Before validating the data, it is checked out and if it is not correct, it will not be accepted and a warning message will be issued.

• Values that may be selected:

The possible data are fixed one of them must be selected. Use the  $[\clubsuit]$  [�] keys to see the possible values. For this type of values that are icons, the [SPACE] key has the same effect as the  $[\clubsuit]$  key.

16.

SETUP ASSISTANCE
Oscilloscope



**CNC 8070** 

Here is a list of machine parameters and variables of the drive that may be modified from the oscilloscope.

The new values are assumed by the CNC after pressing [ENTER] and they are maintained until the unit is turned off. Once the setup is completed, the changes must be saved so they can be effective next time the CNC is turned on.



Use the "Validate" softkey to save the data. The CNC saves the new values in the CNC tables and/or in the drive tables accordingly.

#### Axis or spindle machine parameters that may be edited

The table indicates the parameters that will be assumed at the beginning of the next block. For this to be so, the instruction "#SYNC POS" must be programmed after each motion block.

CNC	Drive	Associated variable	Update
PRELOAD		MPG.PRELOAD[tandem]	Immediate
PRELFITI		MPG.PRELFITI[tandem]	Immediate
TPROGAIN		MPG.TPROGAIN[tandem]	Immediate
TINTIME		MPG.TINTIME[tandem]	Immediate
TCOMPLIM		MPG.TCOMPLIM[tandem]	Immediate
INPOSW		MPA.INPOSW.[set].axis	Immediate
BACKLASH	PP58	MPA.BACKLASH.[set].axis	Immediate
G00FEED		MPA.G00FEED.[set].axis	Beginning of next block
MAXVOLT		MPA.MAXVOLT.[set].axis	Immediate
PROGAIN	PP104	MPA.PROGAIN.[set].axis	Immediate
FFGAIN	PP216	MPA.FFGAIN.[set].axis	Immediate
MANFFGAIN	PP216	MPA.MANFFGAIN.[set].axis	Immediate
ACFWFACTOR		MPA.ACFWFACTOR.[set].axis	Immediate
ACFGAIN	PP217	MPA.ACFGAIN.[set].axis	Immediate
MANACFGAIN	PP217	MPA.MANACFGAIN.[set].axis	Immediate
LACC1		MPA.LACC1.[set].axis	Beginning of next block
LACC2		MPA.LACC2.[set].axis	Beginning of next block
LFEED		MPA.LFEED.[set].axis	Beginning of next block
ACCEL	PP42	MPA.ACCEL.[set].axis	Beginning of next block
DECEL	PP42	MPA.DECEL.[set].axis	Beginning of next block
ACCJERK		MPA.ACCJERK.[set].axis	Beginning of next block
DECJERK		MPA.DECJERK.[set].axis	Beginning of next block
ESTDELAY		MPA.ESTDELAY.[set].axis	Immediate
SERVOOFF		MPA.SERVOOFF.[set].axis	Immediate
MINANOUT		MPA.MINAOUT[.set].axis	Immediate
BAKANOUT	PP2	MPA.BAKANOUT.[set].axis	Immediate
BAKTIME	PP3	MPA.BAKTIME.[set].axis	Immediate

If the machine parameters are protected, when accessing any of them, the CNC will display a window requesting a password.

16.

SETUP ASSISTANCE
Oscilloscope



**CNC 8070** 

#### Variables that may be modified at the Sercos drive

Drive	Name	Update
CP1	CP1.axis	Immediate
CP2	CP2.axis	Immediate
CP3	CP3.axis	Immediate
CP30	CP30[set].axis	Immediate
CP31	CP31[set].axis	Immediate
CP32	CP32[set].axis	Immediate
NP1	NP1.axis	Immediate
RP1	RP1.axis	Immediate
RP2	RP2.axis	Immediate
RP3	RP3.axis	Immediate
RP4	RP4.axis	Immediate
RP51	RP51.axis	Immediate
RP52	RP52.axis	Immediate
RP53	RP53.axis	Immediate
RP54	RP54.axis	Immediate
SP1	SP1[set].axis	Immediate
SP2	SP2[set].axis	Immediate
SP50	SP50.axis	Immediate
SP51	SP51.axis	Immediate
TP10	TP10.axis	Immediate
TP11	TP11.axis	Immediate
TP12	TP12.axis	Immediate
TP13	TP13.axis	Immediate
TP14	TP14.axis	Immediate

Besides the parameters shown in the previous table, the user may access any variable having a writing permission if he knows its name. To do this, use the following nomenclature:

Indicate the axis, the name of the parameter and the set separated by a period.

When modifying any parameter of the previous table, the change is effective immediately without having to reset or restart the unit even when there is a program in execution.

The drive parameters modified with this method are functional immediately, but these values will be lost when the drive is turned off. Use the "Save to flash" option to maintain the modified values of those machine parameters.



SETUP ASSISTANCE
Oscilloscope



**CNC 8070** 

#### Saving, loading and resetting the condigurations

When defining a new configuration of variables to be displayed and parameters modified at the oscilloscope, the system allows saving that configuration into a file.

Likewise, the system allows loading a configuration that was saved previously or resetting the current configuration.

#### Save

To save the current configuration, go to the configuration screen and press the "Save" softkey.

After this, a new screen appears showing the available configurations with the focus on the last one.

Also, it is possible to rename the new configuration whose extension must be "osc".

#### Load

To load a previously saved configuration, go to the configuration screen and press the "Load" softkey.

Then, a new screen appears showing a list of previously saved configurations, being possible to select any of them.

#### Reset

Pressing the "Reset" softkey of the configuration screen deletes or initializes the current configuration. There are neither variables nor parameters selected and the rest of conditions (colors, trigger, etc.) assume the values assigned by default.

16.

SETUP ASSISTANCE
Oscilloscope



**CNC 8070** 

SETUP ASSISTANCE
The Bode diagram

### 16.2 The Bode diagram

The Bode diagram is an assistance tool for adjusting the CNC and the drives. This utility may be used to obtain the Bode diagram of the response amplitude depending on the frequency of the input signal.

With this tool, it is possible to insert a PRBS signal into an axis, so that changing the frequency and analyzing the response provides the behavior of the axes at different frequencies and will give the user an idea of how it behaves with any signal.

This diagram may be used to check the system's gain, the bandwidth and the mechanical resonance. All this helps achieve the proper adjustment of the loops, analyze mechanical problems and check the final features.

The Bode diagram is an assistance tool that only shows the response of the system to the various settings (adjustments); it is up to the technician to choose the best adjustment.

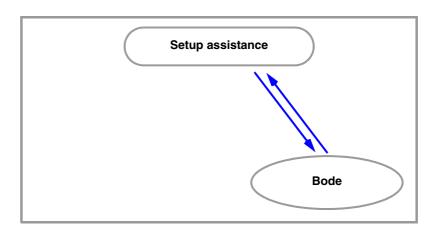


The Bode diagram requires a drive version V6.11 or newer.

#### How to access the Bode diagram



The Bode diagram may be accessed from the softkey menu for setup assistance.

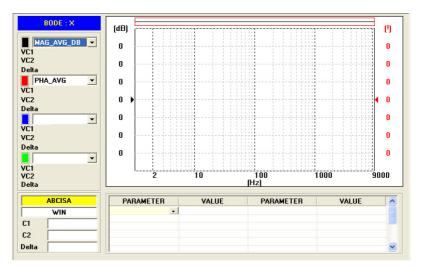




**CNC 8070** 

#### 16.2.1 Interface description

The screen of the Bode diagram is similar to that of the oscilloscope and looks like this, with two clearly different parts. The left side and the lower side show the selected axis, the various ways to display the captured data, the time base or frequency base and the parameters that may be acted upon. The top right side shows the data graphically.



The graphics window

It is the area showing a graph of the selected data. For each graph, it shows the scale used to draw it.

#### The data area

It is the area where the user interacts with the system. It allows defining the graphic environment, the way the graphics will be displayed, the configuration for the time base or the frequency base and the values of the parameters involved.

#### Ways to display

It is the area where the user selects the how to display the captured data in the graphics window. The user can change the type of graph without having to do a new capture. Up to 4 graphs may be displayed at the same time.

This window does not allow selecting the color of each graph being displayed. To change the color of any graph, it must be done at the configuration screen.

#### Machine parameters to be set

In this area of the data window, it is possible to directly change the values of up to 22 machine parameters to be adjusted.

**16.** 

SETUP ASSISTANCE
The Bode diagram



**CNC 8070** 

# 16.

SETUP ASSISTANCE
The Bode diagram

10.

Horizontal softkey menu

The horizontal softkey menu offers the following options;

· Softkey "Config".

This softkey accesses the configuration screen. It may be used to define the axis to obtain its Bode, the two variables (IN/OUT) that will be displayed, the types of graph to show the variables, its colors, the capture configuration and the movement configuration.

The configuration screen offers the options to save, load or reset the current configuration and also load the reference Bode configuration.

· Softkey "Data".

It accesses the data processing submenu. This submenu serves to save and load the captured trace as well as the configuration defined for it.

The following softkeys are also displayed.



#### Print.

It may be used to send data of the captured trace out to printer or to a file



#### Convert into reference capture.

It may be used to convert the data of the capture channels into a reference Bode with which to compare the capture just done or another one that is located in a file.

· Softkey "Views".

It accesses the submenu to display the graph. This submenu shows has the following softkeys.

#### Overlap channels.

Wit this option, several channels may be overlapped. When this option is selected, the softkey color changes to red.



#### Enlarge screen

With this option, it is possible to enlarge the graphics window to the left, using the area that shows the data of the graphs to be displayed. Pressing the same softkey returns the screen to the previous size.



#### Activate/Cancel reference capture.

It may be used to activate or cancel the reference Bode with which to compare the capture just done or another one that is located in a file.

· Softkey "Scales".

It accesses the submenu to configure the abscissa and ordinate axes. This submenu may be used to activate the logarithmic scale on the abscissa axis.



• Softkey "Begin capture".

Start taking data to show it graphically. When this option is selected, the softkey shows "Stop".

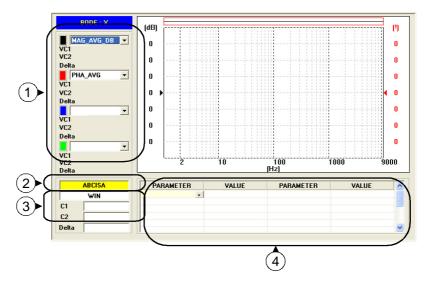
#### Vertical softkey menu

Depending on which data has the focus, the vertical softkeys show different options.

- Zone 1: Fields of variables.
- Zone 2: T"T/DIV" field.
- Zone 3: "WIN", "C1" and "C2" fields.
- Zone 4: Fields of machine parameters.



**CNC 8070** 



#### Zone 1. Focus on a type of graph.

When the focus is on any type graph, the following vertical softkeys will be shown.



The scale of the graphic may be changed with the "scale" softkeys. The new scale value will appear on the graphic, in the same color as the type of graph.



The desired graph may be moved up and down using the offset softkeys.



This softkey may also be used to center the graphics.



When the focus is on any of the variables, any type of graph may be either selected without having to go to the configuration screen or having to do another capture.

#### Zone 2. Focus on the "ABSCISSA" field

When the focus is on the "ABSCISSA" field, the following softkeys will appear.



Using the autoscale softkey, the Bode selects the right time scale frequency scale so the signal appears as large as possible inside its corresponding graphic area.





It increases or decreases the time scale or frequency scale.

#### Zone 3. Focus on the "WIN", "C1" and "C2" field.

Cursors "C1" and "C2" may be used to analyze each signal of the last data capture. These two cursors may be used to obtain the position in milliseconds or Hertz of each signal and the time difference between them "" $\Delta$ ".

When the focus is on one of these fields, the following vertical softkeys will be shown.



It takes the window to the beginning.



It takes the window to the end.

16.

SETUP ASSISTANCE
The Bode diagram



**CNC 8070** 

SETUP ASSISTANCE The Bode diagram It takes the window to the center.



Moves the window to the left.



Moves the window to the right.

#### Zone 4. Focus on the "parameter name" field.

Its value may be modified by selecting the various parameters. When pressing [ENTER] after selecting a parameter, use the [♠][♠][♠][♠] keys to move the focus to the next field to change the value of that parameter. After changing the parameter value and pressing [ENTER], the focus does not go on to the next field. Use the arrow keys to go on to the next parameter. When the focus is on this field, the 4 vertical softkeys will be:



It increases the value of the parameter in the amount indicated on the configuration screen.



It decreases the value of the parameter in the amount indicated on the configuration screen.



It resets the parameter to its default value.



It validates the value given to the parameter.



**CNC 8070** 

Here is a list of machine parameters and variables of the drive that may be modified from the Bode.

The new values are assumed by the CNC after pressing [ENTER] and they are maintained until the unit is turned off. Once the setup is completed, the changes must be saved so they can be effective next time the CNC is turned on.



Use the "Validate" softkey to save the data. The CNC saves the new values in the CNC tables and/or in the drive tables accordingly.

#### Axis or spindle machine parameters that may be edited

The table indicates the parameters that will be assumed at the beginning of the next block. For this to be so, the instruction "#SYNC POS" must be programmed after each motion block.

CNC	Drive	Associated variable	Update
PRELOAD		MPG.PRELOAD[tandem]	Immediate
PRELFITI		MPG.PRELFITI[tandem]	Immediate
TPROGAIN		MPG.TPROGAIN[tandem]	Immediate
TINTIME		MPG.TINTIME[tandem]	Immediate
TCOMPLIM		MPG.TCOMPLIM[tandem]	Immediate
INPOSW		MPA.INPOSW.[set].axis	Immediate
BACKLASH	PP58	MPA.BACKLASH.[set].axis	Immediate
G00FEED		MPA.G00FEED.[set].axis	Beginning of next block
PROGAIN	PP104	MPA.PROGAIN.[set].axis	Immediate
FFGAIN	PP216	MPA.FFGAIN.[set].axis	Immediate
MANFFGAIN	PP216	MPA.MANFFGAIN.[set].axis	Immediate
ACFWFACTOR		MPA.ACFWFACTOR.[set].axis	Immediate
MANACFGAIN	PP217	MPA.MANACFGAIN.[set].axis	Immediate
LACC1		MPA.LACC1.[set].axis	Beginning of next block
LACC2		MPA.LACC2.[set].axis	Beginning of next block
LFEED		MPA.LFEED.[set].axis	Beginning of next block
ACCEL	PP42	MPA.ACCEL.[set].axis	Beginning of next block
DECEL	PP42	MPA.DECEL.[set].axis	Beginning of next block
ACCJERK		MPA.ACCJERK.[set].axis	Beginning of next block
DECJERK		MPA.DECJERK.[set].axis	Beginning of next block
SERVOOFF		MPA.SERVOOFF.[set].axis	Immediate
BAKTIME	PP3	MPA.BAKTIME.[set].axis	Immediate

If the machine parameters are protected, when accessing any of them, the CNC will display a window requesting a password.

16.

SETUP ASSISTANCE
The Bode diagram



**CNC 8070** 

#### Variables that may be modified at the Sercos drive

Drive	Name	Update
CP1	CP1.axis	Immediate
CP2	CP2.axis	Immediate
CP3	CP3.axis	Immediate
CP30	CP30[set].axis	Immediate
CP31	CP31[set].axis	Immediate
CP32	CP32[set].axis	Immediate
NP1	NP1.axis	Immediate
RP1	RP1.axis	Immediate
RP2	RP2.axis	Immediate
RP3	RP3.axis	Immediate
RP4	RP4.axis	Immediate
RP51	RP51.axis	Immediate
RP52	RP52.axis	Immediate
RP53	RP53.axis	Immediate
RP54	RP54.axis	Immediate
SP1	SP1[set].axis	Immediate
SP2	SP2[set].axis	Immediate
SP50	SP50.axis	Immediate
SP51	SP51.axis	Immediate
TP10	TP10.axis	Immediate
TP11	TP11.axis	Immediate
TP12	TP12.axis	Immediate
TP13	TP13.axis	Immediate
TP14	TP14.axis	Immediate

Besides the parameters shown in the previous table, the user may access any variable having a writing permission if he knows its name. To do this, use the following nomenclature:

Indicate the axis, the name of the parameter and the set separated by a period.

When modifying any parameter of the previous table, the change is effective immediately without having to reset or restart the unit even when there is a program in execution.

The drive parameters modified with this method are functional immediately, but these values will be lost when the drive is turned off. Use the "Save to flash" option to maintain the modified values of those machine parameters.



SETUP ASSISTANCE
The Bode diagram

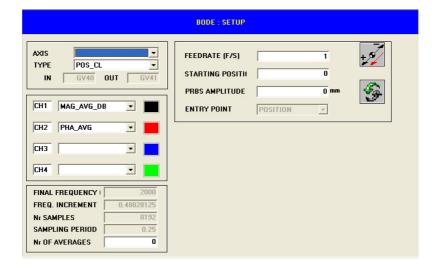


**CNC 8070** 

#### 16.2.3 Configuration screen

The configuration screen may be accessed with the "Config" softkey. This screen shows four different areas.

- The selected axis and the two variables to be displayed.
- The selected types of graph.
- The capture configuration data.
- The data of the movement and of the excitation signal.



#### **Axis**

Axis whose Bode is to be obtained.

#### Type of capture

The variables that are captured (In/Out) are preset in all types of capture except in the "ADVANCED" type where they are set by the user.

Variable.	Meaning.
ADVANCED	"In/Out" variables that may be set by the user.
CUR_CL	Closed current loop. Current command / current feedback.
VEL_CL	Closed velocity loop. Velocity command / velocity feedback.
POS_CL	Closed position loop. Position command / position feedback.
CUR_OL	Open current loop.
VEL_OL	Open velocity loop.
POS_OL	Open position loop.
TF_CURCOM_MOTSPE	Transfer function. Current command / motor speed.
TF_MOTSPE_LOASPE	Transfer function.  Motor speed / load speed.

16.

SETUP ASSISTANCE
The Bode diagram



**CNC 8070** 



**CNC 8070** 

(REF: 0809)

#### Channels

Different ways to display the graph of the obtained data. The selected channels must have the same units in the abscissa axis, "time" graphs and "frequency" graphs cannot be displayed at the same time.

• "IN, OUT"

The captured signals themselves. They are "time" graphs; in other words, the scale of the abscissa axis is given in time units.

• "MAG, PHASE"

Gain and phase after transferring the "In/Out" variables. They are "frequency" graphs; in other words, the scale of the abscissa axis is given in frequency units (Hz).

Channel.	Meaning.
MAG_AVG PHASE_AVG	Average of the captures done so far.
MAG_DB PHASE_DB	Data in decibels.
MAG_OL PHASE_OL	Data in closed loop to which a mathematical transformation is applied to represent the open loop system.

• "COHERENCE"

Coherence after transferring the "In/Out" variables. It is a "frequency" graph; in other words, the scale of the abscissa axis is given in frequency units (Hz).

#### Capture data.

Except the "Number of averages" field that can always be edited, the rest of the fields can only be edited by the user when the type of capture is "ADVANCED".

"FINAL FREQUENCY"

Maximum frequency displayed. It is directly related to the sampling period (1 / 2\*SamplingPeriod)).

• "FREQUENCY INCREMENT"

Step between frequencies. It is directly related to the final frequency and the number of samples (FinalFrequency / NumberOfSamples).

- "SAMPLING PERIOD"
- "NUMBER OF SAMPLES"
- ""NUMBER OF AVERAGES"

Number of consecutive captures performed automatically. Each capture involves the movement of the axis, the application of the PRBS and the capture itself.

#### Movement

"FEEDRATE (F/S)"

Moving speed.



It changes the axis moving direction, positive or negative.

• "INITIAL POSITION"

Position where each capture begins.

• "PRBS AMPLITUDE"

Amplitude of the excitation signal.



It changes the units of the amplitude of the excitation signal.

#### ENTRY POINT

It may be used to select the point where to apply the excitation signal, only when the type of capture is "ADVANCED".

#### Horizontal softkey menu

When defining a new graphic configuration and modified parameters, the system permits saving that configuration in a file. Likewise, the system allows loading a configuration that was saved previously or resetting the current configuration.

Save the current configuration.

After pressing the "Save" softkey, the CNC shows the list of available configurations and lets you save the current configuration. The file extension must be bod.

• It loads a previously saved configuration.

After pressing the "Load" softkey, the CNC shows the list of previously saved configurations and lets you select any of the them.

• Reset or initialize the current configuration.

Pressing the "Reset" softkey of the configuration screen deletes or initializes the current configuration. There are neither variables nor parameters selected and the rest of conditions assume the values assigned by default.

• Activate/Cancel reference capture.

It may be used to activate or cancel the configuration of the reference Bode.

16.

SETUP ASSISTANCE
The Bode diagram



**CNC 8070** 

16.

SETUP ASSISTANCE
The circularity (roundness) test

This tool helps improve and adjust the axis reversal peak, an effect that appears when the axes reverse their moving direction. With this utility, a circle may be executed with the axes and graphically show the result. This graphic shows the difference between the actual (real) coordinate obtained from the feedback device and the theoretical coordinate calculated at each point. This difference is shown projected radially.

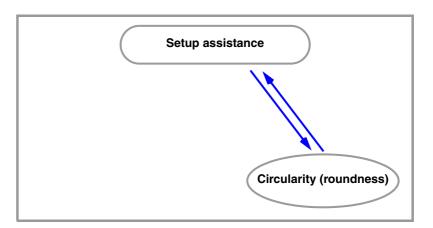
The same screen showing the graphics also shows the machine parameters that affect the movement reversal. These parameters may be modified while running the test, thus being possible to evaluate the response of the system to these changes and optimize the adjustment on the go.

The circularity (roundness) test diagram is an assistance tool that only shows the response of the system to the various settings (adjustments); it is up to the technician to choose the best adjustment.

#### How to access the circularity (roundness) test



The circularity (roundness) test may be accessed from the softkey menu for setup assistance.

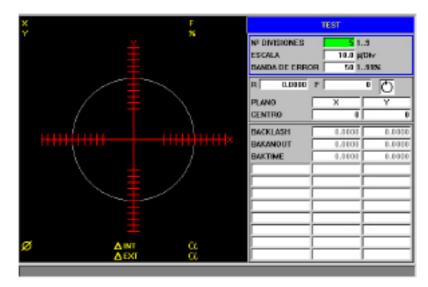




**CNC 8070** 

#### 16.3.1 Interface description

The screen of the circularity test looks like this, with two clearly different parts. A graphics window that shows the result of the test and a data area for interacting with the system.



The graphics window

It is the area that shows, graphically, the result of the test.

The graphic shows the two moving axes and the theoretical circle of the interpolation that will be carried out. As the test is being run, the positioning error at each point is drawn on the circle. This error is shown projected radially.

Superimposed on the graphics, it shows the following additional data that is updated by the CNC.

- The real coordinates of the axes.
- Programmed feedrate and % applied.
- · Diameter of the displayed circle.
- Maximum and minimum error over the theoretical radius and angular position where it has been detected.

#### The data area

It is the area where the user interacts with the system. It offers a set of data for defining the graphic environment, the subroutine that will be used to generate the machine movement and the machine parameters involved in the adjustment. Selecting one of the data frames the group of parameters it belongs to.

#### Graphic environment data to be configured

- Number of divisions on both sides of the theoretical circle.
- · Scale or value in microns of each division.
- Error margin. Percentage of the area that is occupied by the error margin (divisions area).

See "Machine parameter editing" on page 307.

16.

SETUP ASSISTANCE
The circularity (roundness) test



**CNC 8070** 



**CNC 8070** 

(REF: 0809)

#### **Movement subroutine**

The circular interpolation is carried out using a subroutine. See "Define and execute the movement subroutine" on page 326.

The following data is taken into consideration when executing the subroutine.

- · Plane where the circle is executed.
- · Circle center coordinates.
- · Circle radius.
- · Programmed feedrate.
- Turning direction.

#### Parameters to be set

To make the adjustment, it is possible to change the CNC machine parameters or the drive machine parameters involved. See "Adjustment of the machine parameters involved" on page 328.

It is possible to interact with up to 11 different machine parameters. Some of these parameters are always visible, but some may be defined at will. For each parameter, it shows its value on each axis of the work plane.

To see the whole list of parameters that may be defined, see section "16.3.8 Machine parameters that may be modified".

#### Horizontal softkey menu

The horizontal softkey menu offers the following options.

- · Softkey "Simple".
  - It captures the data of a full circle.
- Softkey "Continuous".
  - It captures data continuously.
- · Softkey "Stop".

It stops the data capture. This softkey is shown when data capture is running.



This softkey only stops the data capture. It does not stop the movement of the axes. To stop the movement of the axes, press the [STOP] key of the operator panel.

- · Softkey "Clear".
  - It deletes the graphic representation. It may be done while the capture is running; in that case, it goes on with the graphic representation.
- · Softkey "Load".
  - It loads a previously saved configuration.
- · Softkey "Save".

It saves the configuration defined.

### Vertical softkey menu

The vertical softkey menu offers the following options.



· Softkey "Validate".

It saves the values of the machine parameters in the tables of the CNC and/or the drive accordingly.

When changing the values of the machine parameters, the changes are effective immediately. However, these changes are maintained only during the current session of the CNC; i.e. until the unit is turned off. On power-up, it assumes the values stored in the tables.



· Softkey "Initialize".

It resets the window data to its default values.

16.

SETUP ASSISTANCE
The circularity (roundness) test



**CNC 8070** 

### 16.3.2 Configuring and executing the circularity (roundness) test

The process consists in executing a circle with the axes of the machine and verifying it on its graphic representation. The graphic shows the difference between the actual (real) coordinate obtained from the feedback device and the theoretical coordinate calculated in that point. This difference is shown projected radially.

The interpolation of the machine axes and the point capture for the graphics are initiated separately. The data capture is handled from the softkey menu whereas the movement of the axes is controlled from the operator panel.

The adjustment process is repeated until obtaining the best adjustment of the axes. The circularity (roundness) test diagram is an assistance tool that only shows the response of the system to the various settings (adjustments); it is up to the technician to choose the best adjustment.

### How to execute the circularity (roundness) test

Basically it is done as follows. Each of these points is described in more detail in the indicated chapter.

The configuration used may be saved in a file for later.

- Before initiating the process, certain aspects must be configured as well as the graphic environment, the circular interpolation, the values of the parameters involved, etc. See "16.1.4 Machine parameter editing" on page 307.
  - A previously saved configuration may be used.
- Execute a circular interpolation of the machine axes. The axes must be moving before the data capture starts. See "16.3.4 Define and execute the movement subroutine" on page 326.
- 3. Initiate the data capture and the graphic simulation. Once the axes are moving, the data capture for the graphics may begin. While capturing data, the position error at each point is drawn on the graphics. See "16.3.5 Data capture for the graphic" on page 327.
- 4. Optimize the adjustment by changing some machine parameters of the CNC or of the drive. This makes it possible to evaluate the system response to these changes and optimize the adjustment. See "16.3.6 Adjustment of the machine parameters involved" on page 328.
- **5.** Save the modified values and the configuration used. See "16.3.7 Validate the changes and save the configuration used" on page 330.

### Move through the data

To enter or modify a data, it must be selected; i.e. it must have the editing focus on it. To select a data, use the  $[\clubsuit][\clubsuit][\clubsuit][\clubsuit]$  keys.

The data are accepted with the [ENTER] key. Before accepting a data, it is checked out and if the data is not correct, it is not accepted.



**CNC 8070** 

### 16.3.3 Configure the graphic environment

This operation may be carried out before or during the test. If they are modified during the test, the screen is cleared and the graphic representation goes on.

The following graphic characteristics may be defined from the data window.

- Number of divisions on both sides of the theoretical circle.
- · Scale or value in microns of each division.
- Error margin. Percentage of the area that is occupied by the error margin (divisions area).

### Initialize the data

When accessing the circularity test, it assumes the values used last. Pressing the "Initialize" softkey restores the default values.

Number of divisions 5

Scale: 10 microns/division

Error margin: 50 %



Bear in mind that the softkey <u>INITIALIZE</u> initializes all the data of the window, including the values of the machine parameters.

**16.** 

SETUP ASSISTANCE
The circularity (roundness) test



**CNC 8070** 

### 16.3.4 Define and execute the movement subroutine

In order to run the test, the axes of the machine must be executing a circular interpolation. The axes must be moving before initiating the point capture and they must keep moving during the whole process. To achieve this, a repetitive movement must be executed.



The movement of the axes is managed from the operator panel. The softkey menu only controls the data capture for the graphics.

### Subroutine associated with the circularity test

The circular interpolation is executed with the subroutine associated with the circularity test. This subroutine is located in the folder "C:\CNC8070 \MTB \SUB \testcirc\_vx.nc", where *vx* indicates the subroutine version and neither its name nor its location must be changed. This subroutine may be modified by the OEM to adapt it to his needs.

Some data of the subroutine are defined in this window.

- Plane where the circle is executed.
- · Circle center coordinates.
- · Circle radius.
- · Programmed feedrate.
- Turning direction of the axes. The turning direction is given by an icon. To change the turning direction, place the focus on the icon and press [SPACE].

### **Execute the associated subroutine**

Press the [START] key to execute the subroutine. The subroutine data cannot be modified while it is being executed. The movement must be stopped before editing any of the data.

When trying to execute the subroutine, if it is not in its pre-established location, the CNC interprets that it does not exist and it creates a predefined one.

### Example of a subroutine

The following example executes a repetitive circular movement in the XY plane. In this case, the repetitive movement is carried out in an endless loop using the "GOTO" command. The movement is stopped with the [STOP] key of the operator panel.

```
%testcirc_V1
G1 G5 X0 Y0 F1000
N10:
    G2 X0 Y0 I10 J0
$GOTO N10
M29
```



**CNC 8070** 

Once the graphic representation has been defined and the machine axes are being interpolated, the point capture for the graphics may begin.

The point capture is initiated from the softkey menu. The point capture may be either simple, where the graph is made once (a single whole circle), or continuous where the graph is redrawn for every interpolation of the machine (after every whole circle).



When capturing points in the circularity test, they are not drawn on the graphics of the CNC.

The machine parameters may be modified while capturing points. See "Adjustment of the machine parameters involved" on page 328.

### Simple data capture

Data capture is initiated with the "Simple" softkey. It deletes the graph and starts drawing the circularity error enlarged to the defined scale.

The capture ends after a whole circle or when the "Stop" softkey is pressed.

### Continuous point capture

Data capture is initiated with the "Continuous" softkey. It deletes the graph and starts drawing the circularity error enlarged to the defined scale.

The capture ends when the "Stop" softkey is pressed.

When modifying the machine parameters, it is possible to see the new graph over the previous one or press the "Clear" key to only see the new one.

### Stop the point capture

Press the "Stop" softkey to stop the data capture. It stops the simple or continuous data capture at any time.



This softkey only stops the data capture. It does not stop the movement of the axes. To stop the movement of the axes, press the [STOP] key of the operator panel.

Once the data capture is completed, two lines are drawn on the graph indicating the angular position of the maximum and minimum error.

### Delete the graphic representation

Use the "Clear" softkey to delete the graph. It deletes the graph and the data shown over it.

It may be pressed at any time, even while it is being drawn.

16.

SETUP ASSISTANCE
The circularity (roundness) test



**CNC 8070** 

The best adjustment of the backlash peak may require modifying the value of certain machine parameters. This may be done directly in this window, thus without having to go to the machine parameter table.

When turning the CNC on, it assumes the new values in the CNC tables and/or in the drive tables accordingly. When changing these values, the changes are effective immediately and are assumed by the CNC until the unit is turned off. Once the setup is completed, the changes must be validated so they can be effective next time the CNC is turned on.

The parameters may be modified before or during the test.

### Selection of the parameters to be displayed

It is possible to interact with up to 11 different machine parameters. Some of these parameters are always visible, but some may be defined at will. However, the CNC will only admit the valid parameters for this type of adjustment.

**BACKLASH** 

BAKANOUT Additional velocity command pulse

BAKTIME Duration of the additional velocity command pulse

To see the whole list of parameters that may be defined, see section "16.3.8 Machine parameters that may be modified". When using Fagor Sercos drives, the drive parameters may also be adjusted.

### How to select the parameters

The parameter setting area shows three data columns. The parameters are defined in the first column. The rest of the fields show the parameter value for each axis.

The list of valid parameters is shown when editing a parameter or pressing [SPACE]. Use the [♠][♣] keys to move through the list and [ENTER] to select one of them. Once selected, the rest of the fields show the parameter value for each axis.

To select the parameters of the drive, simply write their names.

### Save the value of the parameters

The new data is assumed by the CNC until the unit is turned off. Once the setup is completed, the changes must be saved so they can be effective next time the CNC is turned on.

Use the "Validate" softkey to save the data. The CNC saves the new values in the CNC tables and/or in the drive tables accordingly.

When quitting the circularity test without having saved the parameters, the CNC will show a message warning about it and will give a chance to save them.

### Password-protected parameters

If the machine parameters are protected, a password will be requested when trying to modify them. If entered correctly, it stores it and it does not request it again unless the CNC is turned off. If the password is wrong, the values cannot be modified and it requests it again every time.

The test may be executed even when not knowing the access password, but the machine parameters cannot be changed.

The access passwords are determined from the utilities mode.

16.

SETUP ASSISTANCE
The circularity (roundness) test



**CNC 8070** 

### Initialize the value of the parameters

When turning the CNC on, it assumes the new values set by the manufacturer in the CNC tables and/or in the drive tables accordingly. The values used the last time will be kept for when accessing the test again later.

Pressing the "Initialize" softkey sets the values assigned to the parameters as "default values" in the tables, not those set by the manufacturer.



Bear in mind that the **INITIALIZE** softkey initializes all the data of the window.

**16.** 

SETUP ASSISTANCE
The circularity (roundness) test



**CNC 8070** 

### Validate the parameters

The new data is assumed by the CNC until the unit is turned off. Once the setup is completed, the changes must be saved so they can be effective next time the CNC is turned on.

Use the "Validate" softkey to save the data. The CNC saves the new values in the CNC tables and/or in the drive tables accordingly.

When quitting the circularity test without having saved the parameters, the CNC will show a message warning about it and will give a chance to save them.

### Save and load the configuration

The system permits saving the current configuration into a file in ASCII format (extension "TST"). This file only contains the configuration. It contains neither the graphics nor the values of the machine parameters. When loading a configuration, the parameters assume the value they have at the time.



The configuration saving option does not update the machine parameter table. To do that, use the VALIDATE softkey.

### Save the configuration

Press the "Save" softkey to save the current configuration. Select the folder and the file name and press [ENTER]. If there is a configuration already saved with the same name, it will ask whether it is to be replaced or not.

By default, the configuration is saved in the folder "C:\CNC8070 \MTB \DATA" or in the last folder selected by the user.

### Load the configuration

Press the "Load" softkey to load a previously saved configuration. Select the folder and the file name and press [ENTER].



**CNC 8070** 

### 16.3.8 Machine parameters that may be modified

### General machine parameters

Parameter	Units	Update
PRELFITI	ms	Immediate
PRELOAD	%	Immediate
TCOMPLIM	%	Immediate
TINTIME	ms	Immediate
TPROGAIN	%	Immediate

### Axis machine parameter

Parameter	Units	Update		
BACKLASH	mm(inch) or degrees	Immediate		
BAKANOUT	rpm	Immediate		
BAKTIME	ms	Immediate		
ACCEL	mm(inch)/s <sup>2</sup> or degrees/s <sup>2</sup>	Beginning of the next block		
ACCJERK	mm(inch)/s <sup>3</sup> or degrees/s <sup>3</sup>	Beginning of the next block		
ACFGAIN	%	Immediate		
ACFWFACTOR	ms	Immediate		
DECEL	mm(inch)/s <sup>2</sup> or degrees/s <sup>2</sup>	Beginning of the next block		
DECJERK	mm(inch)/s <sup>3</sup> or degrees/s <sup>3</sup>	Beginning of the next block		
FFGAIN	%	Immediate		
G00FEED	mm(inch) or degrees	Beginning of the next block		
INPOSW	mm(inch) or degrees	Immediate		
LACC1	mm(inch)/s <sup>2</sup> or degrees/s <sup>2</sup>	Beginning of the next block		
LACC2	mm(inch)/s <sup>2</sup> or degrees/s <sup>2</sup>	Beginning of the next block		
LFEED	mm(inch) or degrees	Beginning of the next block		
MANACFGAIN	%	Immediate		
MANFFGAIN	%	Immediate		
MAXVOLT	mV	Immediate		
PROGAIN	1000/min	Immediate		
SERVOOFF		Immediate		

16.

SETUP ASSISTANCE
The circularity (roundness) test



**CNC 8070** 

SETUP ASSISTANCE
The circularity (roundness) test



**CNC 8070** 

### **DDSSETUP**

17

This operating mode may be used to set up Sercos drives by editing their parameters and variables.

### **DDSSetup mode selection**



The DDSSetup mode may be accessed either using the user key customized for it or from the tasks window. The task window may be accessed with key combination [CTRL]+[A].

### What can you do in this work mode?

In short, this operating mode offers the following operations:

- See the list of Sercos drives connected to the CNC.
- Change the access level for drive parameters.
- View and change the parameters and variables of the drives.
- Display the list of errors being generated by the drive.
- Monitor in real time the value of the drive variables.
- View the status of the operation being carried out at the drive and the status of the digital inputs and outputs.
- Generate velocity commands.



**CNC 8070** 

### 17.1 Appearance of the DDSSetup mode

When this operating mode is active, it will be indicated at the top right side of the general status bar.



This operating mode consists of several services. As these services are being activated, they overlap each other. If several services active, it is possible to switch from one to another using the relevant key, if it has been set accordingly.

The active services are switched in a rotary manner, in such a way that when pressing on the last active service, it shows the first one.



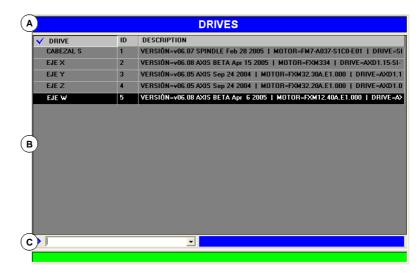
Appearance of the DDSSetup mode



**CNC 8070** 

### 17.1.1 Screen description

The DDSSetup screen shows the following information.



- A. Title bar that shows the name of the active service.
- B. Main window.
- C. Command bar.

**17.** 

**DDSSETUP**Appearance of the DDSSetup mode



**CNC 8070** 



### List of drives.

Shows the list of Sercos drives connected to the CNC. It shows a short description of each drive.



### Drive accessing level change.

Change the access level for drive parameters and variables. This softkey shows a window that requests the password to quit the basic level and access a new level.



### "Parameters and variables" service.

Shows the list of parameters and variables of the drive. You may choose whether to show the list of parameters, the list of variables or both.



### "Error management" service.

It shows the list of errors detected by the drive. The errors are shown in the order they come up.

It is also possible to disable some errors.



### "Monitoring" service.

It shows a list of variables that are selected by the user.



### "Information" service.

This service monitors the status of the operation being carried out at the drive and the status of the digital inputs and outputs of the drive.



### Command simulator.

This option may be used to configure and start up the internal command generator of the drive.



### Select the command bar.

It directly selects the command bar of the DDSSetup. From the command bar, it is possible to execute different commands of the drive, check variables and modify their value is they may be written.



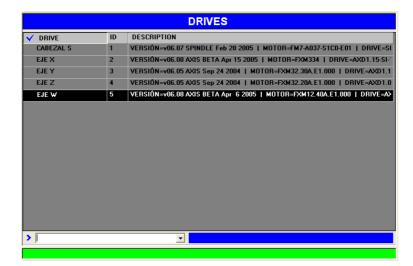
**CNC 8070** 

### 17.2 List of drives

This service shows a list of Sercos drives connected to the CNC. For each drive, it shows the name of the axis associated with it, its identifier and a short description for it, such as software version, motor ID and the ID of the drive itself.



This screen is the one shown by default when accessing the DDSSetup. It can also be accessed from any other service using the relevant softkey.



The command bar is located at the bottom of the window. See "17.4 Command bar" on page 339.

**17.** 

**DDSSETUP** List of drives



**CNC 8070** 

Saving and loading the parameters of all the drives.

From the DDSSetup mode it is possible to save and load the configuration of all the drives using a single softkey.



### Saving the data of all the drives.

This softkey saves the parameters and variables of all the drives into a file.



### Restoring the data of all the drives.

This softkey loads the parameters and variables of all the drives from a file. This process also sends the command "save the parameters into flash".



### Update the descriptor of the parameters and variables.

It updates the file that describes the parameters and variables of the drive.

If when restoring the data, a writing error occurs at a drive, the CNC will warn about it and it will give the option to either resume or abort the process.

If when restoring the data, the number of files does not match the number of Sercos axes available, the CNC will warn about it and it will give the option to either resume or abort the process.



**CNC 8070** 

### 17.4 Command bar

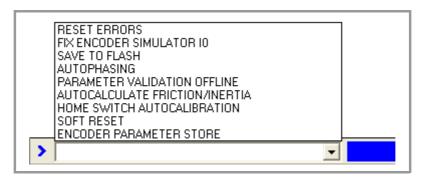
The command bar is available in various services of the DDSSetup mode. From the command bar, it is possible to execute different commands of the drive, check variables and modify their value is they may be written.

**③** 

The command bar may be selected with the window changing key, same as the rest of the screen elements or using its associated softkey.

### **Executing commands of the drive**

The drop list offers the following commands. To execute one of them, select it from the list and press [ENTER].



### Consulting and modifying drive variables

Read-only variables may be consulted (those with the "R" indicator on the attributes column). Read-write variables may be consulted and/or edited (those with the "RW" indicator on the attributes column). See "17.6 –Parameters and variables—service." on page 341.

If a variable has been written, pressing [ENTER] reads its value. If a value is assigned to a variable, when pressing [ENTER], it assumes the new value.

Variables may be identified using their mnemonic or their Sercos identifier. Refer to the drive's manual to know the list of available variables.

PV210 Consulting the PV210 variable.

S36 Consulting the variable SV1 (Sercos ID = 36).

RV9 = 23 It assigns the value of 23 to the variable RV9.

S36 = 100 It assigns the value of 100 to variable S36 (Sercos ID = 36).

If not indicated otherwise, it accesses the variables of the drive selected in the service. To access those of a particular drive, select it in the service or add a suffix with the axis name to the mnemonic (or Sercos ID).

PV210.Y It consults the variable PV210 of the Y axis drive.

S36.S Consulting variable (Sercos ID = 16) of the spindle drive.

**17.** 

**DDSSETUP**Command bar



**CNC 8070** 

Changing the drives accessing level

### 17.5 Changing the drives accessing level

The parameters, variables and commands of the drive are organized by access levels. To access each parameter, the drive must be configured at the access level required by that parameter. A password is needed to access each level.

8

This softkey is used to change the access level for drive parameters. Pressing this softkey shows a window that requests the password to quit the basic level and access a new level.

The available access levels are the following:

• Basic level (USER)

It is the default level, it is accessed when entering the DDSSetup mode. It requires no password.

The user level gives access to a group of parameters that slightly changes the operation of the drive.

• Intermediate level (OEM)

This level gives access to a large group of parameters that depend on the motor connected, they set how the electronics of the drive is adapted to that motor.

Maximum level (FAGOR)

This level gives full access to all the variables, parameters and commands of the system.



The Fagor level gives access to a group of parameters depending on the drive's electronics and they factory set.

Only technicians from Fagor Automation are granted a FAGOR access level.



**CNC 8070** 

### 17.6 -Parameters and variables - service.



This service shows the list of parameters and/or variables of the drive. It is selected with the "View" softkey of the horizontal menu.

PARÁMETERS						
2) DATOS INTERNOS SET						
VALUE	UNIT	RANGE	SID	NAME		
0		[01]	F-0-52	%Р1		
18.77		[0.00327.67]	S-0-119	%P111		
1.775	mS	[0.00032.767]	S-0-120	%P112		
50.0	%	[10.01000.0]	F-0-28	%P114		
100.0	%	[10.01000.0]	F-0-29	%P115		
	VALUE 0 18.77 1.775 50.0	VALUE UNIT  0  18.77  1.775 mS  50.0 %	VALUE         UNIT         RANGE           0         [01]           18.77         [0.00,.327.67]           1.775         mS         [0.00,.32.767]           50.0         ≈         [10.0,.1000.0]	VALUE         UNIT         RANGE         SID           0         [01]         F-0-52           18.77         [0.00.327.67]         \$-0-119           1.775         mS         [0.000.32.767]         \$-0-120           50.0         \$\mathref{z}\$         [10.0.1000.0]         F-0-28	VALUE         UNIT         RANGE         SID         NAME           0         [01]         F-0-52         2P1           18.77         [0.00.327.67]         S-0-119         2P111           1.775         mS         [0.000.32.767]         S-0-120         2P112           50.0         %         [10.0.1000.0]         F-0-28         2P114	

The command bar is located at the bottom of the window. See "17.4 Command bar" on page 339.

### Selection of the elements to be displayed

The "View" softkey of the horizontal menu may be used to choose displaying either the parameters and variables, only the parameters or only the variables.

From the top of the window, it is possible to select:

- The drive whose parameters and/or variables are to be consulted.
- The group of parameters and/or variables to work with. Once selected, the list will show the elements of the selected group.
- The set of parameters to be consulted. Those parameters and/or variables that have sets will display the values of that set. Also, its identifier will show the selected set (e.g. for set 4 of the variable SP2, it will show SP2.4)

Parameters with a set will not be affected by the change.

### List of variables and parameters

The list of variables and parameters offers the following information.

- Mnemonic of the variable or of the parameter.
- · Current value at the drive.
- · Units being used.
- · Valid range of values. If it is a string of characters, it will show the maximum length permitted.
- · Sercos identifier.
- · Name of the variable or of the parameter.

### **Editing the values**

The values, of parameters or variables with write permission are edited in the Value field. If the new value is accepted, the list shows the change.

The changes are saved in the drive's RAM memory. To save the change permanently, use the "Data" + "Flash" option of the horizontal menu.

### How to make a backup copy of the data

The "Data" softkey of the horizontal menu offers a series of options to make a backup copy of the values.

- The "Save" softkey saves the values of the parameters and variables into a file.
- The "Load" softkey restores the values of the parameters and variables from a previously saved file.

-Parameters and variables- service.

**DDSSETUP** 



**CNC 8070** 

### 17.7 -Error management- service.



This service shows the list of errors detected by the drive. The errors are shown in the order they come up.

Once the cause of the error has been corrected, the errors may be reset with the "Error reset" softkey of the horizontal menu.



The command bar is located at the bottom of the window. See "17.4 Command bar" on page 339.

### Disabling the errors

To disable an error of the drive, use the "Error listing" softkey of the horizontal menu. This softkey shows the whole list of possible errors. To disable the desired ones, checkmark the box on the left.

Then, press the "Send" softkey to send the list to the drive. To make the error disabling effective, execute the command DC1 (reset errors) from the command bar.

Disabling the errors requires an OEM or FAGOR access level. See "17.5 Changing the drives accessing level" on page 340. Even then, only resettable errors may be disabled, even if the list also shows the non-resettable errors.



DDSSETUP Error management- service.



**CNC 8070** 

### 17.8 -Monitoring- service



This service monitors the value of the parameters and variables of the drive. The user selects the variables and parameters to be monitored.

MONITORING				
VARIABLE	VALUE			
† GV25.5 (GV25)				
GV26.5 (GV26)				
, ,				

The command bar is located at the bottom of the window. See "17.4 Command bar" on page 339.

### List of variables or parameters

The "Add" softkey shows a window where one can select the variables or parameters to be monitored and they will be shown on the screen. Variables and parameters of any drive may be selected.

Use the "Initialize" softkey to clear the whole list. To delete a variable or parameter from the list, select it and press the [DEL] key.

The "Save" softkey saves the list into a file and the "Load" softkey restores the list from a file.

### Monitoring the status of variables or parameters

Press the "Start" softkey of the horizontal menu to start monitoring. Press the "Stop" softkey to stop monitoring.

**17.** 

**DDSSETUP**-Monitoring- service



**CNC 8070** 

### 17.9 -Information- service

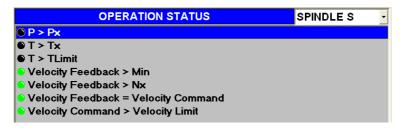


This service monitors two types of information. It shows the status of the operation being carried out at the drive, on the other hand it shows the status of the digital inputs and outputs of the drive.

The command bar is located at the bottom of the window. See "17.4 Command bar" on page 339.

### Monitoring the status of the operations of the drive

The "Operation status" softkey of the horizontal menu gives access to this option.



This screen monitors the status of the operation being carried out at the drive. The drive whose operations are to be monitored is selected at the top of the window.

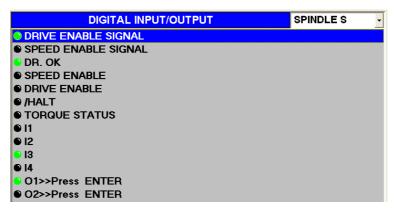
There is a led for each operation and it shows the status of the operation at the drive.

Green led Operation active.

Black led Operation inactive.

### Monitoring the status of the operations of the drive

The "Digital I/O" softkey of the horizontal menu gives access to this option.



This screen monitors the status of the digital inputs and outputs of the drive. The drive whose digital I/O's are to be monitored is selected at the top of the window.

There is a led for each digital input and output of the drive and it shows its status.

Green led Input or output active.

Black led Input or output inactive.



**DDSSETUP**—Information—service



**CNC 8070** 

### Type of inputs and outputs

The inputs and outputs may be grouped as follows.

• Inputs and outputs that cannot be assigned.

These inputs and outputs indicate the status of the drive's control signals.

Speed enable Drive enable Error Reset
LSC Status DR. OK PROG. OK
Master Speed Enable Master Drive Enable Master Halt

**Torque Status** 

• Inputs and outputs that may be assigned.

Variables and/or parameters of the drive may be associated with inputs and outputs (analog and digital).

Those digital outputs that have not been associated with any parameter or variable may be forced to become active or inactive using the button on the left of their led.

**17.** 

**DDSSETUP**-Information- service



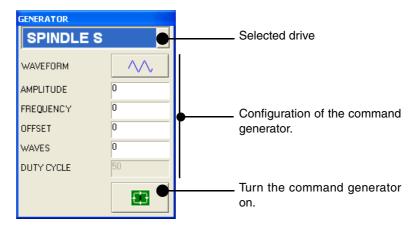
**CNC 8070** 

### 17.10 Command generator



This option may be used to configure and start up the internal command generator of the drive. In the configuration, it is possible to establish the shape of the signal, its amplitude, period, etc.





The fields to be configured at the generator are:

Shape To select the type of signal to be generated with the drive

(sinusoidal or square).

Amplitude Signal amplitude.

Frequency Signal frequency.

Offset Signal Offset.

Waves Number of waves to generate. When set to 0 (zero), the

waves are generated continuously, without interruption.

Duty Cycle Percentage of the wave at 1/0 (only on squarewaves).

The internal command generator is turned on and off using the lower buttons.



It turns the command generator on.



It turns the command generator off.



**CNC 8070** 

### **DIAGNOSIS**

This operating mode may be used to test the hardware and software of the PC that the CNC is based upon.

- System information.
- Information about the modules that make up the CNC software.
- Information about the elements connected to the CNC through the CAN bus and SERCOS.

The diagnosis is a testing and displaying tool; it cannot be used to modify the displayed values.

### Diagnosis mode selection



The diagnosis mode may be accessed either using the user key customized for it or from the tasks window. The task window may be accessed with key combination [CTRL]+[A].



**CNC 8070** 

### Appearance of the diagnosis mode 18.1

When this operating mode is active, it will be indicated at the top right side of the general status bar.

This operating mode may be displayed with several screens. Each one corresponds to an element of the CNC configuration.

### Standard configuration

The standard configuration of this work mode, described in this manual corresponds to the one supplied by Fagor. In some cases, the manual shows a sample screen, instead of that of the CNC in order to make it easier to understand.

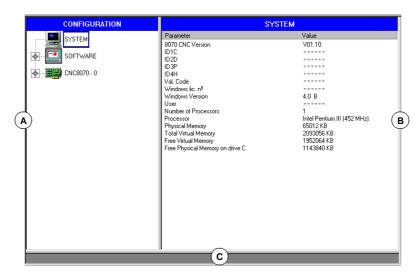


Appearance of the diagnosis mode



**CNC 8070** 

The diagnosis screen shows the following information.



A. It shows the hardware and software configuration of the CNC.

System diagnosis: Information on the system elements; CNC version, user name, microprocessor(s) used, status of the various system memories, etc.

Software diagnosis: Information about the modules that make up the CNC software and the software options installed.

CNC8070 diagnosis: Information about the elements connected to the CNC through the Can bus and Sercos (drives, input/output cards, etc.).

- B. Diagnosis of the selected element.
- C. CNC messages.

To switch windows, press the relevant key.

### After selecting an element of the configuration Hide or display the list of resources of an element. To move the cursor through the elements of the list. ESC To move the cursor to the beginning or end of the list.

18.

DIAGNOSIS
Appearance of the diagnosis mode



**CNC 8070** 

**DIAGNOSIS**Appearance of the diagnosis mode

### 18.1.2 Description of the softkey menus.

The softkey menus of the the diagnosis mode offer the following options.

### Horizontal softkey menu.



Generate the reportfagor.zip file with all the relevant information for proper error diagnosis and, if necessary, the end user can send it to Fagor Automation.



View the error and warning history issued by the CNC.

### Vertical softkey menu.



### Update data

This softkey is used to update (refresh) the data of the diagnosis window.



### Print table

This softkey may be used to print the configuration in the pre-determined printer or save it as a file (prn format) at the CNC.

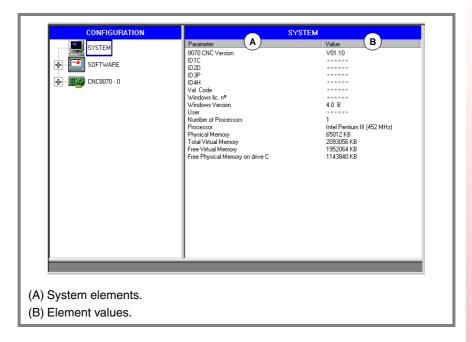


**CNC 8070** 

### 18.2 Configuration diagnosis

### 18.2.1 System diagnosis

When selecting this configuration element, the diagnosis window shows the list of the system elements and their values.



18.

**DIAGNOSIS**Configuration diagnosis



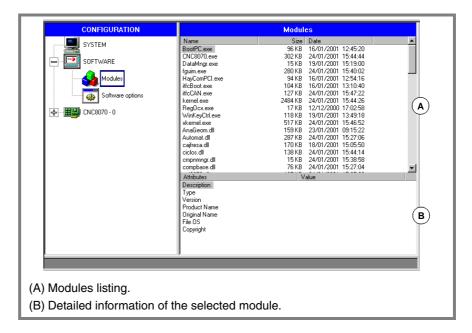
**CNC 8070** 

### 18.2.2 Software diagnosis

This option may be used to analyze the modules that make up the CNC software and the software options installed.

### Module information

When selecting this configuration element, the diagnosis window shows the list of the modules that make up the CNC software.



It shows the size of each module (in bytes) and the date it was created. When selecting a module from the list, the bottom of the screen shows more detailed information.

### Software options

When selecting this element of the configuration, the diagnosis window shows the software options currently installed.

It also shows the horizontal softkey "Validation code" for entering the resulting validation code according to the hardware ID of the unit and the desired options.



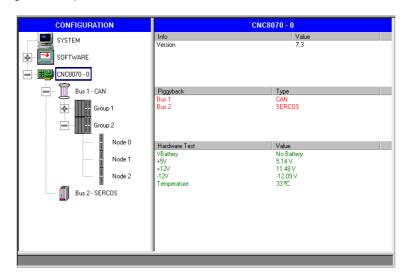
**DIAGNOSIS**Configuration diagnosis



**CNC 8070** 

### 18.2.3 Hardware diagnosis. CAN bus and SERCOS

When selecting this element of the configuration, the diagnosis window shows the software identification of the CNC's communications board (version, boot, checksum) and the type of buses connected to it. Likewise, it monitors the different voltage and temperature alarms.



Sercos bus configuration

• Listing of nodes and drive associated with each one of them.

For each drive, it indicates its software version, type of drive and the motor connected to it.

### Can bus configuration

- · Resources connected and their status.
- Groups connected and number of nodes that make up each group (detected and active).

### Detailed information of the selected group.

- Group resources and their status.
- Information about the nodes that make up the group.

### Detailed information of the selected node.

- Software version, boot and checksum of the code.
- · Node status.
- · Resources of each module.

The symbol next to the node indicates its status and its possible association.



Active.



Active, but in error.



Inactive.

18.

**DIAGNOSIS**Configuration diagnosis



**CNC 8070** 

### 18.3 Sercos diagnosis

### General information about the bus

General purpose, it shows the following information about the Sercos bus.

Version SERCON chip version.

Loop time Time between two synchronization messages (MST). It is the same as general parameter

LOOPTIME.

T2 Time from the MST until the CNC starts sending the telegram with the position commands

(MDT).

T3 Time from the MST until the drives have the position commands.

T4 Time from the MST when the drives read the feedback values.

TL Time from the MST until the RT IT takes place at the CNC.

Distortion errors Distortion error counter at the CNC.

Ring errors Sercos ring error counter.

Phase Current Sercos phase.

Phase status Phase status (0 = OK).

Logic ID and name Name and driveID (number of the rotary switch) of the Sercos axes.

### General information about the bus

It shows the following information for each drive.

### **General information**

T1 Time from the MST until the drive starts sending its telegram (AT).

Control word Contents of the drive's control word.

Status word Contents of the drive's status word.

Name of the Sercos axis.

### **Error information**

"Reset Hard" counter Number of times that the drive has been reset.

Distortion errors Counter of distortion errors detected at the drive.

AT errors Counters of AT reception errors of each drive.

### Information on the last operation carried out by the service channel

ID Processed Sercos ID.

Element Processed data (Name, Units, min/max Value, Status, Value, Attributes).

Process Operation carried out (Read, Write).

Command status In processes with commands, it indicates its status.

Error code Operation error code (0 = OK).



**CNC 8070** 

## Save the CAN configuration for the start-up test.

### 18.4 Save the CAN configuration for the start-up test.

Saving a CAN configuration to a file serves to check the CAN configuration is correct when starting the system up. The CAN configuration must only be saved to a file after having verified that it is correct.

### Save the CAN configuration.

When the CAN bus level is selected, it shows the "Save" softkey that may be used to save the current CAN configuration. The configuration is saved in the file C:\Cnc8070 \mtb \data \canData.tab.

This file contains the number of groups present in the bus, the number of modules, the number of inputs and outputs, etc.

### Verify the CAN configuration during power-up.

To verify the configuration, the information must be previously saved to the file *canData.tab*.

During the power-up process, the data saved in the *canData.tab* file is compared with the data read during power-up. If when comparing the data, they are not the same, the corresponding error message is issued.

"The CAN configuration is not the same as the one saved in the file"

Once the error is detected, it will keep coming up even after a reset. There two ways to correct the error:

- Power up the CNC again with the original CAN configuration; the one saved in the canData.tab file
- Save the new CAN configuration. The CAN configuration must only be saved to a file after having verified that it is correct.

At a CNC with a write-protected (read-only) disk, when it is started up in user mode, the MTB folder is protected and the changes made to it are temporary until the unit is turned off. In order for the CAN configuration file not to be eliminated when turning the unit off, it must be started up in setup mode and then save the CAN configuration.

placenosis start-up test.

FAGOR

**CNC 8070** 

### 18.5 Report generation

### Automatic reports generated when accessing the diagnosis mode

Every time when accessing the diagnosis mode, the CNC generates the following reports. These reports are saved in the folder C:\CNC8070 \Diagnosis\Blackbox.

SystemInfo.txt Validation code.

SercosInfo.txt Version of drives, motors and feedback.

### Generate a report with the system configuration

A report with all the information offered by the diagnosis mode may be obtained from any section of the diagnosis mode. This produces a document with all the information about the FAGOR system installed on a machine such as the software version, status of the remote modules, installed drives and motors, etc.



To print the information, use the "Print" softkey. Pressing this softkey, the CNC will request the data it must include on the report. The CNC can create a report with the data shown on the screen or with the whole configuration of the system.

This information may be printed or saved to a file. When printing to a file, it is possible to select the name and location of the file.

### Generate the warranty registration report.

The CNC can generate a template so the end user can send to Fagor the documentation with the serial numbers of all the elements in order to obtain the corresponding warranty. This template, one printed out, contains the fields that the user must fill out and then send it to Fagor Automation.



To print the warranty registration form, use the "Print" softkey. After pressing this softkey, select "warranty registration report".

The report may be printed or saved to a file. When printing to a file, it is possible to select the name and location of the file.



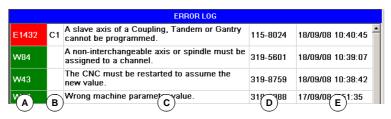
**CNC 8070** 

# View the history of errors and warnings issued by the CNC.

### 18.6 View the history of errors and warnings issued by the CNC.



The softkey menu shows a window with the history of errors and warnings issued by the CNC. Pressing this softkey displays the following window.



- A. Error or warning number.
- **B.** Number of the channel where the error or warning occurred; this cell does not show any value when it is a system error or system warning.
- C. Text of the error or warning.
- D. Internal code of the error or warning.
- E. Date when the error or warning occurred.

18.

DIAGNOSIS and warnings issued by the CNC.

FAGOR

**CNC 8070** 

### Generating the Fagor file for error diagnosis. 18.7



The softkey menu offers the possibility to generate the reportfagor.zip file with all the relevant information for proper error diagnosis and, if necessary, the end user can send it to Fagor Automation.

After pressing this softkey, the CNC creates the reportfagor.zip file and saves it in the "C:\Cnc8070\Diagnosis" folder. When pressing this softkey, the CNC also generates the following files in the "C:\Cnc8070\Users\Reports" folder and includes them in the reportfagor.zip file.

Diagcnc.txt CNC diagnosis report.

Hardware.txt Hardware configuration report.

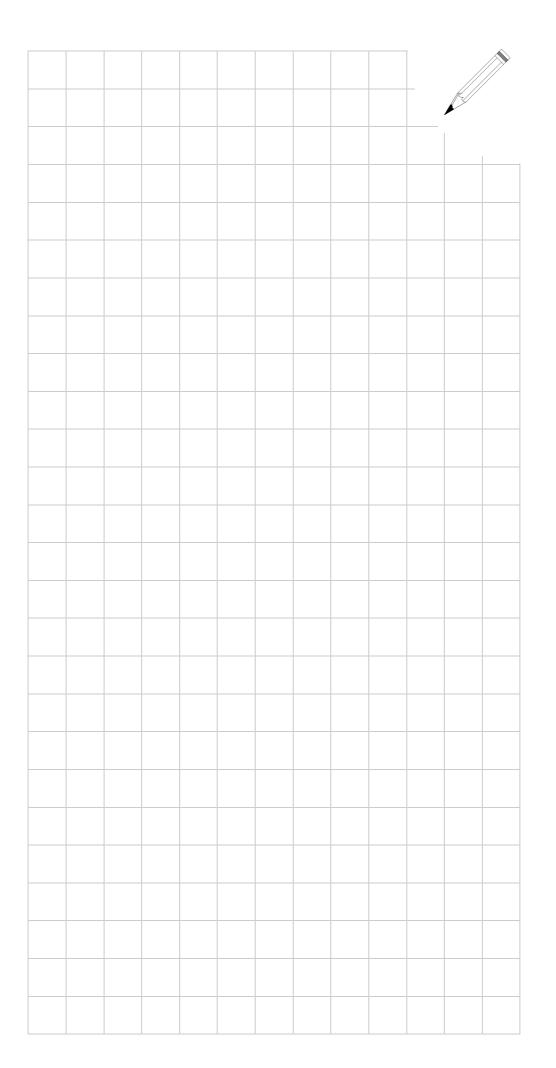
Times.txt Time statistics report.



DIAGNOSIS
Generating the Fagor file for error diagnosis.

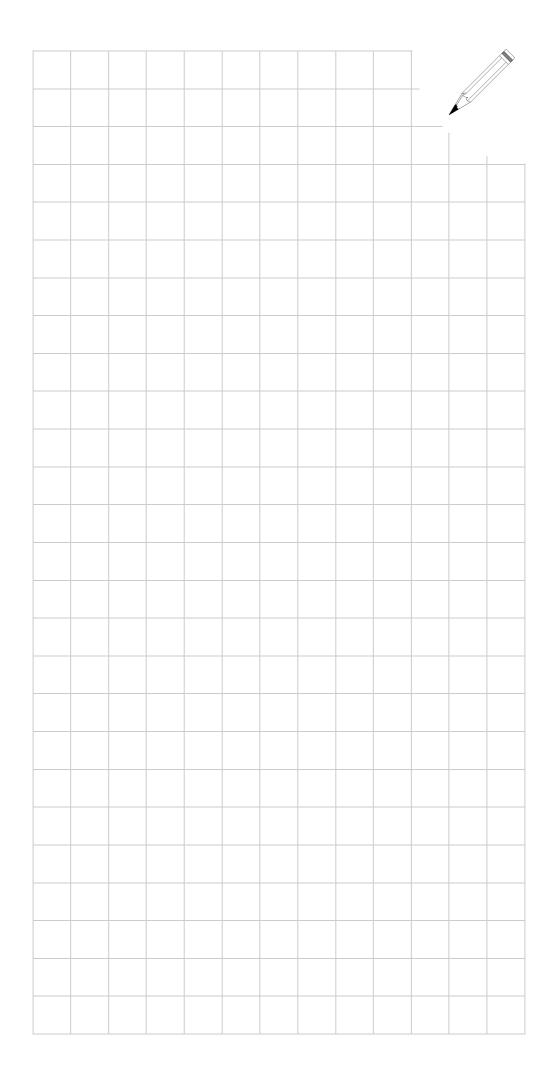


**CNC 8070** 





**CNC 8070** 





**CNC 8070**