CNC 8060 L 8070 L

Operating and programming manual.

(Ref: 1707)

FAGOR AUTOMATION

MACHINE SAFETY

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

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

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

HARDWARE EXPANSIONS

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

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

COMPUTER VIRUSES

FAGOR AUTOMATION guarantees that the software installed contains no computer viruses. It is up to the user to keep the unit virus free in order to guarantee its proper operation. Computer viruses at the CNC may cause it to malfunction.

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.

DUAL-USE PRODUCTS

Products manufactured by FAGOR AUTOMATION since April 1st 2014 will include "-MDU" in their identification if they are included on the list of dual-use products according to regulation UE 428/2009 and require an export license depending on destination.

FAGOR AUTOMATION

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

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

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

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

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

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

INDEX

About the product 7
EC Declaration of conformity and Warranty conditions 9
Version history 11
Safety conditions 13
Material returning terms 17
CNC maintenance 19

CHAPTER 1 DESCRIPTION OF THE KEYS

1.1 Monitor LCD-15. (8070). 23
1.2 Monitor LCD-10K. (8060/8070). 24
1.3 HORIZONTAL KEYB / HORIZONTAL KEYB + MOUSE. (8070). 26
1.4 VERTICAL KEYB / VERTICAL KEYB + MOUSE. (8070). 27
1.5 OP PANEL / OP PANEL + SPDL RATE. (8070). 28
1.6 OP PANEL. (8060). 29
1.7 Keyboard shortcuts. 30

CHAPTER 2 INTRODUCTION. THE CNC.

2.1 Connectivity. 31
2.1.1 Mouse (only 8070). 31
2.1.2 USB. 31
2.1.3 Ethernet. 31
2.1.4 Compact flash (only 8060). 32
2.2 Turning the CNC on and off. 33
2.2.1 Turning the CNC on. 33
2.2.2 Turning the CNC off. 34
2.2.3 Emergency shutdown with battery (8070 only). 34
2.2.4 Create a backup copy of the CNC data. 35
2.3 Access modes and software protection on the CNC. 36
2.3.1 Administrator mode. 37
2.3.2 Setup mode. 37
2.3.3 User mode. 38
2.3.4 Restrictions for working in setup mode and switching to user mode. 38
2.4 The interface. 39
2.4.1 The CNC status bar. 40
2.4.2 Task window. 41
2.4.3 Channel synchronization window. 42
2.4.4 PLC message window. 43
2.4.5 Horizontal softkey menu. 44
2.4.6 CNC warnings and errors window. 45
2.4.7 PLC error window. 46
2.4.8 Dialog boxes. 47
2.5 Operating modes. 48
2.6 File selection window. 49
2.7 Calculator. 52
2.7.1 Defining expressions. 54

CHAPTER 3 AUTOMATIC MODE: CUT_VIEW.

3.1 Select the piercing points. 60
3.2 Program execution. 61
3.2.1 Program selection. 61
3.2.2 Execute a program. 62
3.3 Tool inspection. 63
3.4 Change the machining conditions (MHFS mode). 65
3.5 Display/hide the dynamic override. 65

CHAPTER 4 AUTOMATIC MODE: PROGRAM.

4.1 Program execution. 69
4.1.1 Program selection. 69
4.1.2 Execute a program. 70
4.2 Cancel the execution and resume from another block while keeping the history. 71
4.3 Tool inspection. 72

FAGOR

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

4.4 Change the machining conditions (MHFS mode). 74
4.5 Display/hide the dynamic override. 74

CHAPTER 5 MANUAL MODE.

5.1 Operations with the axes. 77
5.1.1 Machine reference zero (home) search. 77
5.1.2 Jog 78
5.1.3 Jogging the axes with handwheels 80
5.1.4 Moving an axis to a particular position (coordinate) 82
5.1.5 Coordinate preset 82
5.2 Setting the feedrate. 83
5.3 Setting and activating the zero offsets and the fixture offsets. 83

CHAPTER 6 EDISIMU MODE.

6.1 Functionalities of the editor. 87
6.1.1 Contextual programming assistance. 88
6.1.2 Help for programming subroutines. 89
6.2 Select a program. 90
6.3 Operations with blocks. 90
6.3.1 Copy, cut and paste. 90
6.3.2 Find a line or a text in the program. 91
6.4 Undo and redo. 91
6.5 Operations with files. 92
6.5.1 Import DXF files 93
6.6 Syntax errors when editing 95

CHAPTER 7 MDI/MDA MODE.

7.1 Edit and execute individual blocks. 99
7.2 Block history. 100

CHAPTER 8 TECHNOLOGY TABLES.

8.1 Material files. 102
8.1.1 Where the CNC saves the material files (material_file.xml). 103
8.1.2 Selecting a material file. 103
8.1.3 Creating, renaming and duplicating a material file. 104
8.1.4 Deleting a material file. 104
8.1.5 Importing and exporting material files. 104
8.1.6 Protecting technology tables with a password. 105
8.2 Cutting and piercing parameters. 107
8.2.1 Selecting and activating a material file. 109
8.2.2 Adding or deleting parameters (only OEM). 109
8.2.3 Modifying the properties of the parameters. 109
8.2.4 Setting the properties of the material file parameters. 110
8.2.5 Enter password. 111
8.2.6 Operations with material files. 112

CHAPTER 9 MACHINE STATUS.

9.1 Monitoring of digital and analog inputs. 113
9.2 Monitoring of digital and analog outputs. 114
9.3 Status of the axes. 114
9.4 Laser status. 115

CHAPTER 10 USER TABLES.

10.1 Zero offset tables 118
10.2 Fixture table 120
10.3 Arithmetic parameter tables 121
10.4 Operations with tables 122
10.4.1 Data editing 122
10.4.2 Save and recall tables 122
10.4.3 Find text. 123

CHAPTER 11 UTILITIES MODE.

11.1 Set how to display the list of programs. 128
11.2 Select files and create folders. 129
11.3 Find in files 130
11.4 Protection passwords 131

FAGOR

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

11.5 Data safety backup. Backup - Restore 132
11.6 Encrypting files..... 134

CHAPTER 12 PROGRAM THE POWER AND TURN THE LASER ON AND OFF.

12.1 Program the power. 137
12.2 G192. Limiting power. 138
12.3 Turning the laser on and off. 139
12.4 Variables related to power. 140

CHAPTER 13 RADIUS COMPENSATION.

13.1 Defining the laser radius. 142
13.2 G40/G41/G42. Activating/canceling the laser radius compensation. 143
13.3 G136/G137. Selecting the type of transition between blocks. 144
13.4 G137/G138. Strategy for starting and ending radius compensation. 145
13.5 Starting, ending and joining paths with radius compensation. 147
13.5.1 Starting radius compensation..... 147
13.5.2 Joining two paths with radius compensation. 150
13.5.3 Changing the type of radius compensation (G41/G42) when selected. 154
13.5.4 Canceling radius compensation. 156
13.6 Variables. 159

CHAPTER 14 INITIAL AND FINAL SUBROUTINES.

14.1 Executing the initial subroutine 162
14.2 Executing the final subroutine. 162

CHAPTER 15 CHANGING THE STATUS OF THE PLC MARKS.

15.1 Changing the status of the PLC marks. 163

CHAPTER 16 SELECT A MATERIAL FROM THE TECHNOLOGICAL TABLE.

16.1 Select a material. 165

CHAPTER 17 PIERCING (INITIAL PIERCING).

17.1 Activating the piercing operation. 168
17.2 Subroutine associated with the piercing. 170
17.3 Variables associated with the piercing operation. 171

CHAPTER 18 CUTTING (SHEET CUTTING FOLLOWING A PROFILE).

18.1 Activating the cutting operation. 174
18.2 Canceling the cutting operation. 176
18.3 Subroutines associated with cutting. 176
18.4 Variables associated with the cutting operation. 177

CHAPTER 19 SYNCHRONIZED SWITCHING (GRID PATTERN CUTTING)

19.1 Activate synchronized switching. 180
19.2 Cancel synchronized switching. 182
19.3 Variables related to synchronized switching. 183

CHAPTER 20 PWM (PULSE-WIDTH MODULATION).

20.1 Activate the PWM. 186
20.2 Cancel the PWM. 188
20.3 Variables related to the PWM. 188

CHAPTER 21 POWER CONTROL BASED ON FEED.

21.1 Activating power control. 190
21.2 Canceling the power control. 193
21.3 Variables associated with the power control. 194

CHAPTER 22 GAP CONTROL.

22.1 Activating gap control. 198
22.2 Canceling gap control. 200
22.3 Variables associated with the gap control. 200

FAGOR

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

CHAPTER 23 LEAPFROG.

23.1 Activating the leapfrog. 202
23.2 Variables associated with the leapfrog. 207

CHAPTER 24 CO2 LASER PATH DISPERSION COMPENSATION.

24.1 Activating dispersion compensation. 210
24.2 Canceling dispersion compensation. 212
24.3 Associated variables. 212

CHAPTER 25 PROGRAMMING EXAMPLE.

CHAPTER 26 CNC VARIABLES.

26.1 Understanding how variables work. 215
26.1.1 Accessing numeric variables from the PLC. 217
26.2 Variables in a single-channel system. 218
26.3 Variables in a multi-channel system. 221
26.4 Variables associated with machine parameters. 224
26.4.1 General machine parameters. 224
26.4.2 General machine parameters. Execution channels. 232
26.5 Variables associated with the logic signals of the PLC. 234
26.5.1 PLC consulting logic signals; laser. 234
26.5.2 PLC modifiable logic signals; laser. 237
26.6 Variables associated with the active material. 240
26.7 Variables associated with the active cutting. 240
26.8 Variables associated with the active piercing. 240
26.9 Variables associated with the technological tables. 241
26.9.1 Common parameters. 241
26.9.2 Piercing technological table. 242
26.9.3 Cutting technological table. 245
26.10 Variables related to power. 248
26.11 Variables related to the PWM. 253
26.12 Variables associated with the power control. 255
26.13 Variables associated with the leapfrog. 259
26.14 Variables related to synchronized switching. 261
26.15 Variables associated with the compensation of the CO2 laser dispersion. 263
26.16 Variables associated with the gap control. 264
26.17 Variables associated with the subroutine call parameters. 266
26.18 Variables defined by the user. 267

FAGOR

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

ABOUT THE PRODUCT

BASIC CHARACTERISTICS.

Basic characteristics.	8060 L	8070 L
PC-based system.	Closed system. Without access to the administrator mode.	Open system. Access to the administrator mode.
Number of axes.	3 to 6	3 to 31
Number of spindles.	1	1 to 6
Number of tool magazines.	1	1 to 4
Number of execution channels.	1	1 to 4
Number of handwheels.	1 to 3	1 to 12
Type of servo system.	Analog / Digital Sercos	
Communications.	RS485 / RS422 / RS232 Ethernet	
Integrated PLC. PLC execution time. Digital inputs / Digital outputs. Marks / Registers. Timers / Counters. Symbols.	 < 1ms/K 1024 / 1024 8192 / 1024 512 / 256 Unlimited	
Block processing time.	< 1 ms	< 1 ms

Remote modules.	RIOW	RIO5	RIOR	RCS-S
Valid for CNC.	8070 8060	8070 8060	8070 8060	8070 8060
Communication with the remote modules.	CANopen	CANopen	CANopen	Sercos
Digital inputs per module.	8	24 / 48	48	- - -
Digital outputs per module.	8	16 / 32	32	- - -
Analog inputs per module.	4	4	- - -	- - -
Analog outputs per module.	4	4	- - -	4
Inputs for PT100 temperature sensors.	2	2	- - -	- - -
Feedback inputs.	- - -	- - -	- - -	4 (*)

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

Customizing (only 8070).
PC-based open system, fully customizable. INI configuration files. FGUIM visual configuration tool. Visual Basic®, Visual C++®, etc. Internal databases in Microsoft® Access. OPC compatible interface

SOFTWARE OPTIONS.

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

	8060 L	8070 L
Open system. Access to the administrator mode.	- - -	Option
Editing and simulation environment.	Standard	Standard
Number of execution channels.	1	1 to 4
Number of axes.	3 to 6	3 to 31
Number of spindles.	1	1 to 6
Number of tool magazines.	1	1 to 4
Number of interpolated axes (maximum).	4	3 to 31
Maximun axes and spindles.	7	32
Limited to 4 interpolated axes.	- - -	Option
Tool radius compensation.	Option	Option
HSSA-I machining system.	Option	- - -
HSSA-II machining system.	- - -	Option
Probing canned cycles. (In laser model, oonly G100; whitout cycles).	Option	Standard
Profile editor.	Option	Option
"C" axis.	Option	Option
Tandem axes.	Option	Option
Axis and spindle synchronization.	Option	Option
Tangential control.	- - -	Option
Dynamic RTCP.	Option	Option
Third-party CANopen.	Option	Option
Volumetric compensation (up to 10 m^3).	- - -	Option
Volumetric compensation (more than 10 m^3).	- - -	Option
PWM (Pulse-Width Modulation).	Standard	
Gap control.	Standard	Standard

EC DECLARATION OF CONFORMITY AND WARRANTY CONDITIONS

DECLARATION OF CONFORMITY

The declaration of conformity for the CNC is available in the downloads section of FAGOR'S corporate website at http://www.fagorautomation.com. (Type of file: Declaration of conformity).

WARRANTY TERMS

The warranty conditions for the CNC are available in the downloads section of FAGOR's corporate website at http://www.fagorautomation.com. (Type of file: General sales-warranty conditions.

VERSION HISTORY

Here is a list of the features added to each manual reference. Each manual reference is valid for the indicated software version and newer versions.

Ref. 1604

Software 8060 V01.50.51 8070 V05.50.51
First version. Laser model.

Ref. 1703

Software 8060 V01.50.51 8070 V05.50.51
Improvements in the gap control. • The machine parameter GAPSENSORDELAY is no longer functional.
Improvements in the leapfrog.

Ref. 1707

Software 8060 V01.50.53 8070 V05.50.53	
Improved leapfrog. • New TIME command in the instruction #LEAP, to determine the dwell before the leap.	• Commands: TIME • Variables: (V.)G.LEAPTIME
Technological tables (common parameters). • Sheet metal thickness.	• Variables: (V.)TT.THICKNESS

FAGOR

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

BLANK PAGE

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 2006/42/EC Directive.

PRECAUTIONS BEFORE CLEANING THE UNIT

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

PRECAUTIONS DURING REPAIR

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

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

PRECAUTIONS AGAINST PERSONAL DAMAGE

Interconnection of modules.	Use the connection cables provided with the unit.
Use proper cables.	To prevent risks, only use cables and Sercos fiber recommended for this unit. To prevent a risk of electrical shock at the central unit, use the proper connector (supplied by Fagor); use a three-prong power cable (one of them being ground).
Avoid electrical overloads.	To prevent electrical shock and fire risk, do not apply electrical voltage out of the indicated range.
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 with a relative humidity (non-condensing).
Do not work in explosive environments.	In order to avoid risks or damages, do no work in explosive environments.

PRECAUTIONS AGAINST PRODUCT DAMAGE

Working environment.
This unit is ready to be used in industrial environments complying with the directives and regulations effective in the European Community.
Fagor Automation shall not be held responsible for any damage suffered or caused by the CNC when installed in other environments (residential, homes, etc.).

Install the unit in the right place.
It is recommended, whenever possible, to install the CNC away from coolants, chemical product, blows, etc. that could damage it.
This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as:

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

Enclosures.
The manufacturer is responsible of assuring that the enclosure involving the equipment meets all the currently effective directives of the European Community.

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

Use the proper power supply.
Use an external regulated 24 Vdc power supply for the keyboard, operator panel and the remote modules.

Grounding of the power supply.
The zero volt point of the external power supply must be connected to the main ground point of the machine.

Analog inputs and outputs connection.
Use shielded cables connecting all their meshes to the corresponding pin.

Ambient conditions.
Maintain the CNC within the recommended temperature range, both when running and not running. See the corresponding chapter in the hardware manual.

Central unit enclosure.
To maintain the right ambient conditions in the enclosure of the central unit, it must meet the requirements indicated by Fagor. See the corresponding chapter in the hardware manual.

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.

SAFETY SYMBOLS

Symbols that may appear on the manual.

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

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

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

Information symbol.
This symbol indicates notes, warnings and advises.

Symbol for additional documentation.
This symbol indicates that there is another document with more detailed and specific information.

Symbols that the product may carry.

Ground symbol.
This symbol indicates that that point must be under voltage.

ESD components.
This symbol identifies the cards as ESD components (sensitive to electrostatic discharges).

MATERIAL RETURNING TERMS

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

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

CNC MAINTENANCE

CLEANING

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

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

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

PRECAUTIONS BEFORE CLEANING THE UNIT

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

- Do not handle the connectors with the unit supplied with power. Before handling these connectors (I/O, feedback, etc.), make sure that the unit is not powered.
- Do not get into the inside of the unit. Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

OPERATING.

I

FAGOR

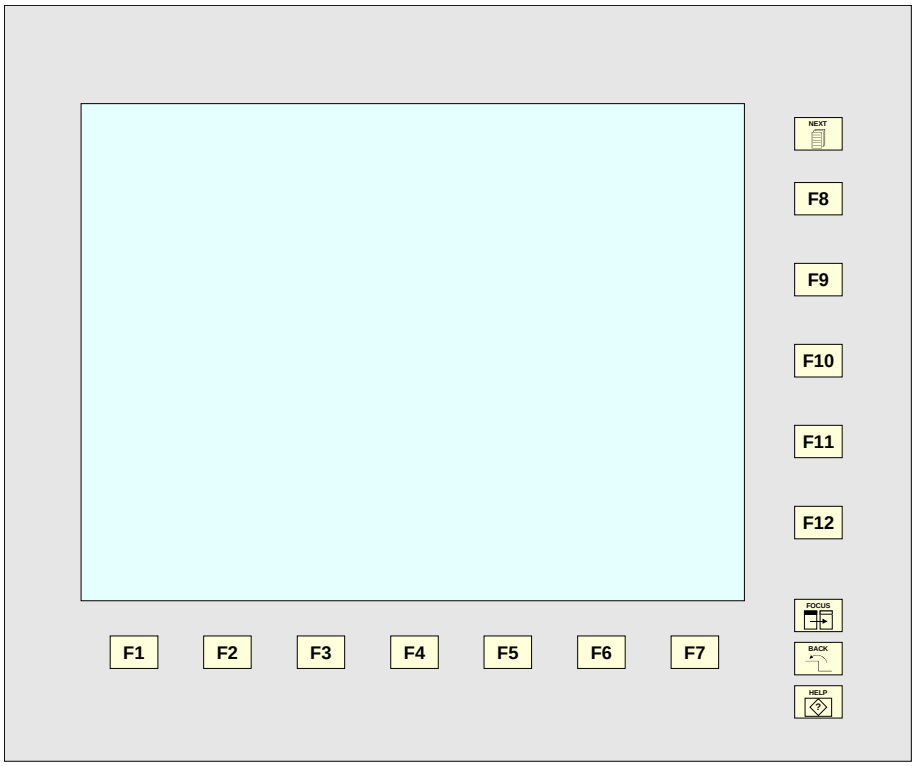
FAGOR AUTOMATION

BLANK PAGE

DESCRIPTION OF THE KEYS

1

1.1 Monitor LCD-15. (8070).



Function keys.

F1 **Softkeys.**
Keys F1 through F12 select the options of the softkey menus.

Browsing keys.

NEXT **NEXT key.**
Configurable key that can carry out one of the following actions.

- 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.
- Sequentially access the different channels.
- The horizontal softkey menu shows the available screens and the vertical menu shows the available channels.

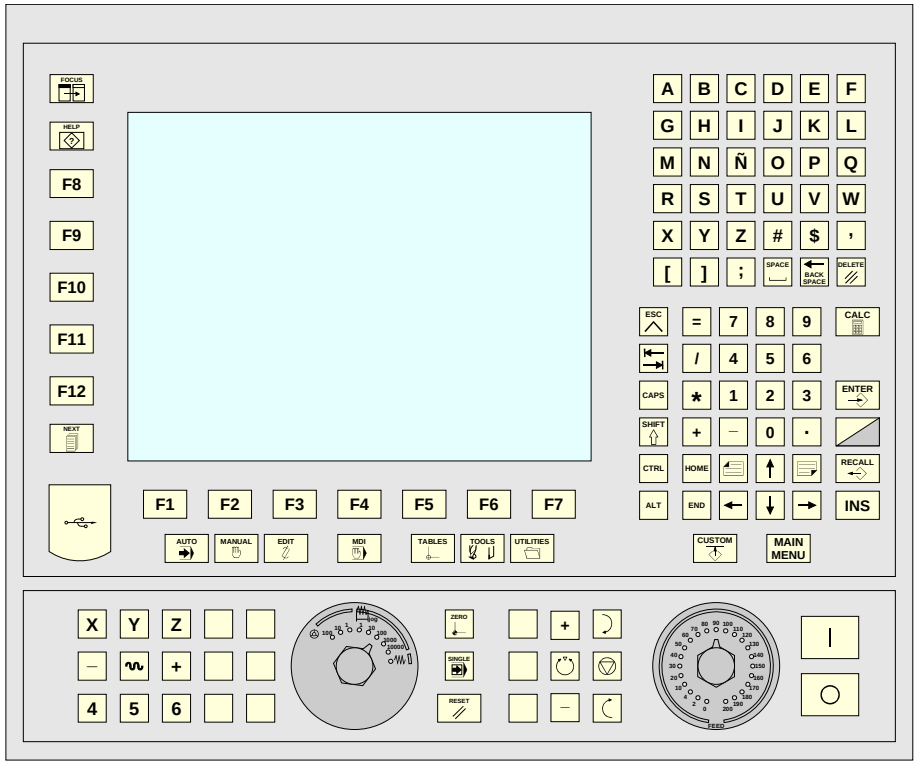
FOCUS **FOCUS key.**
It is used to switch between the different windows of the screen.

BACK **BACK key.**
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.

HELP **HELP key.**
It accesses the CNC's system help.

1.2 Monitor LCD-10K. (8060/8070).



Function keys.

Softkeys.
Keys F1 through F12 select the options of the softkey menus.

Browsing keys.

NEXT key.
Configurable key that can carry out one of the following actions.

- 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.
- Sequentially access the different channels.
- The horizontal softkey menu shows the available screens and the vertical menu shows the available channels.

FOCUS key.
It is used to switch between the different windows of the screen.

Help key.

HELP key.
It accesses the CNC's system help.

Browsing keys.

MAIN MENU — Main menu.

Work modes.

AUTO — Automatic mode.

MANUAL — Manual mode.

EDIT — EDISIMU mode.

MDI — MDI/MDA mode.

TABLES — User tables (zero offsets, fixtures and arithmetic parameters).

TOOLS — Technology tables.

UTILITIES — Utilities mode.

Change the status of an icon.

CUSTOM — Configurable mode.
OEM-configurable key that can carry out one of the following actions.

- Access a CNC work mode.
- Execute an application.
- Access the operating system.
- Carry out no function at all.

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.

Keys to move the cursor.

The arrow keys move the cursor one position to the left, right, up or down.

The previous-page or next-page keys show the previous or next page at the part-program or PLC program editor.

HOME END The home and end keys move the cursor the beginning or end of the line.

The tab key moves the cursor to the next field of the active menu.

Editing keys.

SHIFT Shift.
Hold this key down to write upper case letters. When combining this key with a cursor moving key, it selects the text the cursor slides on.

CAPS Shift lock.
It toggles between uppercase and lowercase letters.

ALT ASCII characters.
Hold this key down and key in the corresponding ASCII code.

BACK SPACE Delete.

DELETE Delete.

INS Insert or overwrite.

ESC Escape key, to cancel the current action without assuming the changes.

ENTER Key to validate commands, data and program blocks of the editor.

RECALL Recover data.
While the Teach-in mode is active, this key enters the axes and their current position into the block.
When selecting a profile or conversational canned cycle in the part-program, the key accesses either the profile editor or canned cycle editor accordingly.

CALC Calculator.

Jog keyboard for jogging the axes.

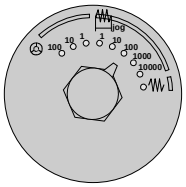
X+ 7+ Keys to select axes and jog them in the positive direction.

X- 7- Keys to select axes and jog them in the negative direction.

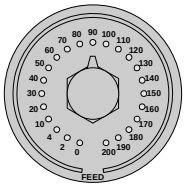
X 7 + – Keys to select the axes and keys to select the jogging direction. Both keys (axis and direction) must be pressed to jog the axis.

Rapid key. When pressing this key while moving an axis, the CNC applies the rapid feedrate.

Feed selectors.



Sector for the type of jog; continuous / incremental jog or handwheels.
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.



Selector of percentage of feedrate override, between 0% and 200%, for jog and automatic movements.

Execution keys.

Cycle start key (START).
Execute the selected program in automatic mode, a block in MDI/MDA mode, etc.

Cycle stop key (STOP).
Interrupt the execution of the CNC.

RESET Reset key.
It initializes the system setting the initial conditions as defined by machine parameters.

SINGLE Single block execution mode.
When selecting the "single block" execution mode, the program simulation will be interrupted at the end of each block.

ZERO Machine reference zero (home) search.

Spindle control.

Turn on the laser.

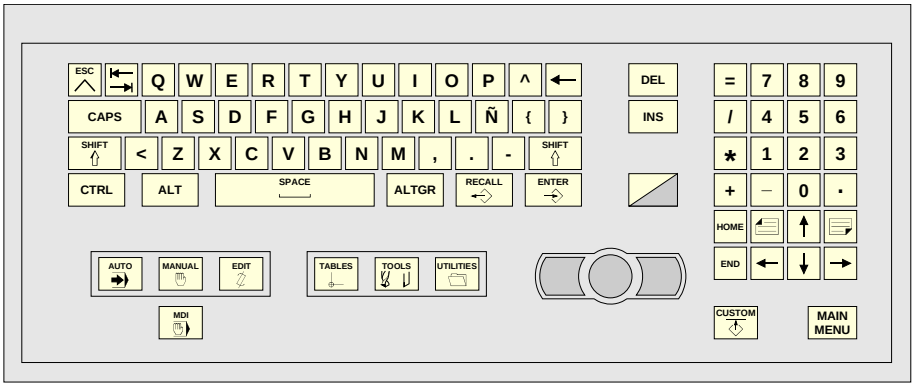
Turn off the laser.

Turn on the laser.

\+ – Percentage variation of the laser power.

Not used.

1.3 HORIZONTAL KEYB / HORIZONTAL KEYB + MOUSE. (8070).



Browsing keys.

MAIN MENU — Main menu.

Work modes.

AUTO — Automatic mode.

MANUAL — Manual mode.

EDIT — EDISIMU mode.

MDI — MDI/MDA mode.

TABLES — User tables (zero offsets, fixtures and arithmetic parameters).

TOOLS — Technology tables.

UTILITIES — Utilities mode.

Change the status of an icon.

CUSTOM — Configurable mode.
OEM-configurable key that can carry out one of the following actions.

- Access a CNC work mode.
- Execute an application.
- Access the operating system.
- Carry out no function at all.

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.

Keys to move the cursor.

The arrow keys move the cursor one position to the left, right, up or down.

The previous-page or next-page keys show the previous or next page at the part-program or PLC program editor.

HOME END — The home and end keys move the cursor the beginning or end of the line.

The tab key moves the cursor to the next field of the active menu.

Editing keys.

SHIFT — Shift.
Hold this key down to write upper case letters. When combining this key with a cursor moving key, it selects the text the cursor slides on.

CAPS — Shift lock.
It toggles between uppercase and lowercase letters.

ALTGR — Alternate characters.
Hold this key down to access the alternate character set of the keyboard.

ALT — ASCII characters.
Hold this key down and key in the corresponding ASCII code.

Delete.

DEL — Delete.

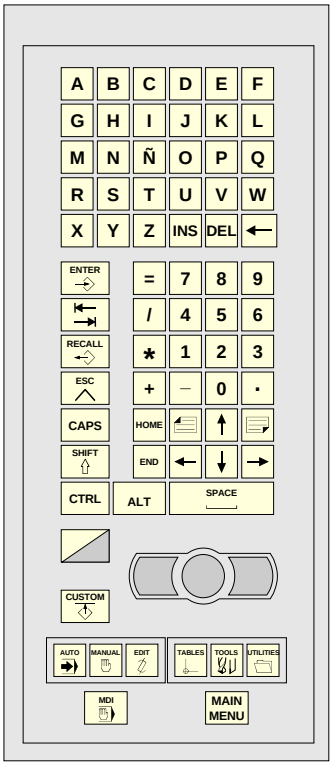
INS — Insert or overwrite.

ESC — Escape key, to cancel the current action without assuming the changes.

ENTER — Key to validate commands, data and program blocks of the editor.

RECALL — Recover data.
While the Teach-in mode is active, this key enters the axes and their current position into the block.
When selecting a profile or conversational canned cycle in the part-program, the key accesses either the profile editor or canned cycle editor accordingly.

1.4 VERTICAL KEYB / VERTICAL KEYB + MOUSE. (8070).



Browsing keys.

MAIN MENU — Main menu.

Work modes.

AUTO — Automatic mode.

MANUAL — Manual mode.

EDIT — EDISIMU mode.

MDI — MDI/MDA mode.

TABLES — User tables (zero offsets, fixtures and arithmetic parameters).

TOOLS — Technology tables.

UTILITIES — Utilities mode.

Change the status of an icon.

CUSTOM — Configurable mode.

OEM-configurable key that can carry out one of the following actions.

- Access a CNC work mode.
- Execute an application.
- Access the operating system.
- Carry out no function at all.

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.

Keys to move the cursor.

The arrow keys move the cursor one position to the left, right, up or down.

The previous-page or next-page keys show the previous or next page at the part-program or PLC program editor.

HOME END — The home and end keys move the cursor the beginning or end of the line.

The tab key moves the cursor to the next field of the active menu.

Editing keys.

SHIFT — Shift.

Hold this key down to write upper case letters. When combining this key with a cursor moving key, it selects the text the cursor slides on.

CAPS — Shift lock.

It toggles between uppercase and lowercase letters.

ALT — ASCII characters.

Hold this key down and key in the corresponding ASCII code.

Delete.

DEL — Delete.

INS — Insert or overwrite.

ESC — Escape key, to cancel the current action without assuming the changes.

ENTER — Key to validate commands, data and program blocks of the editor.

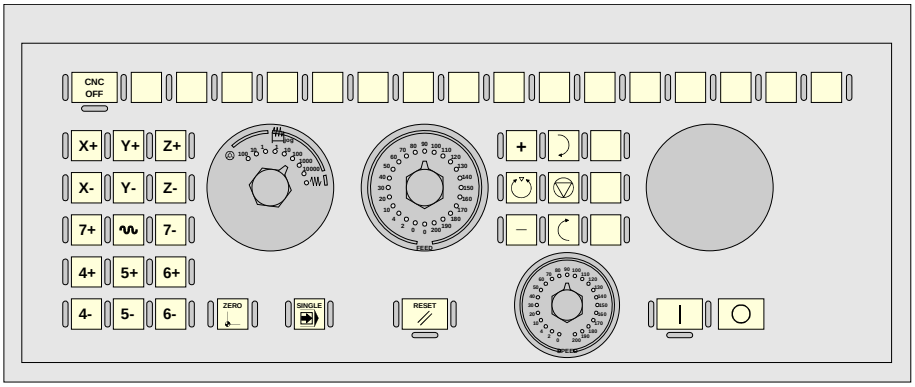
RECALL — Recover data.

While the Teach-in mode is active, this key enters the axes and their current position into the block.

When selecting a profile or conversational canned cycle in the part-program, the key accesses either the profile editor or canned cycle editor accordingly.

FAGOR AUTOMATION
CNC 8060 L
CNC 8070 L
(REF: 1707)

1.5 OP PANEL / OP PANEL + SPDL RATE. (8070).



Jog keyboard for jogging the axes.

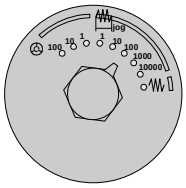
X+ 7+ Keys to select axes and jog them in the positive direction.

X- 7- Keys to select axes and jog them in the negative direction.

X 7 + − Keys to select the axes and keys to select the jogging direction. Both keys (axis and direction) must be pressed to jog the axis.

Rapid key. When pressing this key while moving an axis, the CNC applies the rapid feedrate.

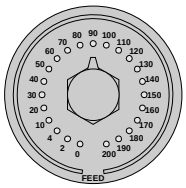
Feed selectors.



Sector for the type of jog; continuous / incremental jog or handwheels.

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.



Selector of percentage of feedrate override, between 0% and 200%, for jog and automatic movements.

Execution keys.

Cycle start key (START).
Execute the selected program in automatic mode, a block in MDI/MDA mode, etc.

Cycle stop key (STOP).
Interrupt the execution of the CNC.

RESET Reset key.
It initializes the system setting the initial conditions as defined by machine parameters.

SINGLE Single block execution mode.
When selecting the "single block" execution mode, the program simulation will be interrupted at the end of each block.

ZERO Machine reference zero (home) search.

Spindle control.

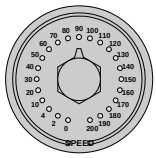
Turn on the laser.

Turn off the laser.

Turn on the laser.

\+ − Percentage variation of the laser power.

Not used.

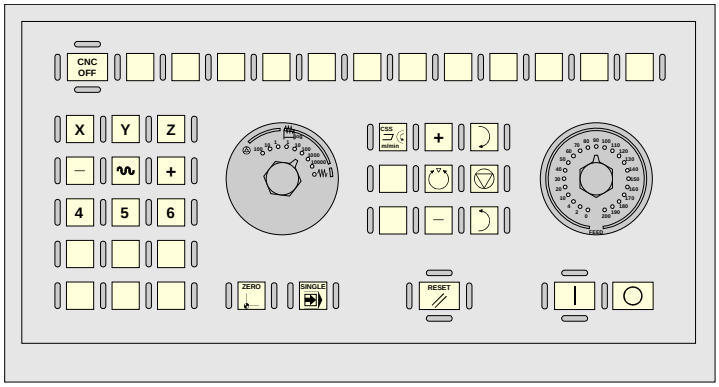


Laser power selector, between 0% and 200%.

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.).

1.6 OP PANEL. (8060).



Jog keyboard for jogging the axes.

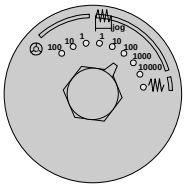
X+ 7+ Keys to select axes and jog them in the positive direction.

X- 7- Keys to select axes and jog them in the negative direction.

X 7 + – Keys to select the axes and keys to select the jogging direction. Both keys (axis and direction) must be pressed to jog the axis.

Rapid key. When pressing this key while moving an axis, the CNC applies the rapid feedrate.

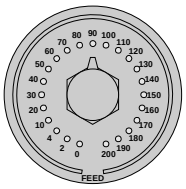
Feed selectors.



Sector for the type of jog; continuous / incremental jog or handwheels.

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.



Selector of percentage of feedrate override, between 0% and 200%, for jog and automatic movements.

Execution keys.

Cycle start key (START).
Execute the selected program in automatic mode, a block in MDI/MDA mode, etc.

Cycle stop key (STOP).
Interrupt the execution of the CNC.

RESET Reset key.
It initializes the system setting the initial conditions as defined by machine parameters.

SINGLE Single block execution mode.
When selecting the "single block" execution mode, the program simulation will be interrupted at the end of each block.

ZERO Machine reference zero (home) search.

Spindle control.

Turn on the laser.

Turn off the laser.

Turn on the laser.

+ – Percentage variation of the laser power.

Not used.

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.).

1.7 Keyboard shortcuts.

Operations at the interface.
[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.
[ALT] + [W] Show / hide the window for errors and warnings.
[ALT] + [F4] Turn the CNC off.

Work modes.
[CTRL] + [A] To show the task window.
[CTRL] + [SHIFT] + [F1] Main menu.
[CTRL] + [F6] Automatic mode.
[CTRL] + [F7] Manual mode.
[CTRL] + [F9] EDISIMU mode.
[CTRL] + [F8] MDI mode.
[CTRL] + [F10] User tables.
[CTRL] + [F11] Technology tables.
[CTRL] + [F12] Utilities mode.
[ALT]+[B] Change the status of an icon.
[CTRL] + [K] Calculator.

Browsing keys.
[CTRL]+[F1] Previous menu.
[CTRL]+[F2] Switch window.
[CTRL]+[F3] Switch screens.

Execution keys.
[CTRL]+[S] Cycle start key (START).
[CTRL]+[P] Cycle stop key (STOP).
[CTRL]+[R] Reset key.
[CTRL]+[B] Single block execution mode.

The shortcuts for the [START] [STOP] and [RESET] keys are only available when the CNC is installed as simulator on a PC.

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.

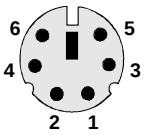
[CTRL]+[F5] [RECALL] key.

INTRODUCTION. THE CNC.

2

2.1 Connectivity.

2.1.1 Mouse (only 8070).



Mouse connection with a PS-2 connector.

2.1.2 USB.

USB support for the following devices. The rest of the devices are not available.

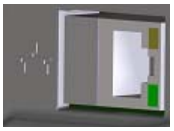
- Pendrive.
- Keyboard and mouse.

No USB devices must be used nor connected while executing a part program. Using USB devices may slow down the execution or cause some "RT-IT overflow" type error.

DO NOT connect any kind of hub to the USB port.

USB devices must comply with the generic regulation IEC 61000-6-2:2005 on immunity for industrial use. We recommend to also comply with the European Directive 2004/108/EC on EMC for industrial use.

2.1.3 Ethernet.



Ethernet support for the following features, through the connector of the central unit.

- Point-to-point connection.
- Microsoft company networks.
- Internet for telediagnosis.
- TeamViewer server.

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.

2.1.4 Compact flash (only 8060).

Additional storage space for user programs.

Fagor Automation offers several flash compact in its catalog to increase storage space; if you are going to use other compact flashes, always use industrial grade compact flash SLC; they support temperatures between -40ºC and +80ºC (-40 ºF and 176 ºF) and last 27 years with 1000 writes per day.

Fagor Automation shall not be held responsible for any problems caused by using other lower-quality compact flash.

2.2 Turning the CNC on and off.

2.2.1 Turning the CNC on.

After powering up the unit, the operating system will start up first. Then, and depending on how the manufacturer has set it, the CNC 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 selected by the machine manufacturer.

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.

Restore a back-up copy.

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.

Further information: See *"2.2.4 Create a backup copy of the CNC data."* on page 35.

Peculiarities of start-up in setup or user modes.

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.

2.2.2 Turning the CNC off.

Use one of the following methods to turn off the CNC. The CNC must not be turned off if there is any program in execution.

- Press the [ALT]+[F4] key combination.
- Use the [CNC OFF] button available on some control panels.
- Use the "Turn off CNC" softkey available in manual mode.

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 not be turned off with the on/off switch without having closed the application using one of the aforementioned methods. Turning the unit off incorrectly may cause the loss of information/the following:

- *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.

2.2.3 Emergency shutdown with battery (8070 only).

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 CNC application, then the whole system and finally turns the battery off.

2.2.4 Create a backup copy of the CNC data.

- 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.

2.3 Access modes and software protection on 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 access mode with an icon on the task bar of the operating system, next to the clock. When the CNC is turned on, the status bar shows the active access mode with icons.

Icon.	Access mode.
	Administrator mode.
	Setup mode.
	User mode.

Access mode change (DiskMonitor window).

- With the CNC application turned off, the access mode is changed using the icon indicating the active access mode, on the task bar of the operating system (next to the clock). When pressing this icon, the unit shows the DiskMonitor window to switch from one access 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.
- With the CNC started up in any access mode, the [ALT][D] hotkey shows the DiskMonitor window to change from one access 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.
- The desired access mode may be selected from the utilities mode, from the softkey menu. 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.

In the previous window, 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.

2.3.1 Administrator mode.

This mode must only be used to install non-Fagor software, to install the CNC (also possible from the setup mode), to update the operative system or change the system configuration. The CNC application does not start up in this mode.

The unit shows a message on the desktop, indicating the active access mode and warning that it is not a protected mode.

The access to the administrator mode is enabled with the validation code ("Open system" software option). If you don't have this software option, (i.e. you have a "closed system") you will not be able to access the administrator mode and, therefore, you will not be able to install third-party software.

Protection level.

There is not protection level in administrator mode, the whole disk is unprotected.

Protection password.

The access to this mode is protected with the password "administrator mode", defined in the utilities mode. When starting the unit up in this access mode, it will request the access password.

2.3.2 Setup mode.

The setup mode must only be used to update the CNC software and to set up the machine; it does not allow installing third-party software. This mode may be used to access the operative system.

The unit shows a message on the desktop, indicating the active access mode and warning that it is not a protected mode.

Protection level.

The setup mode has an intermediate protection level where everything that may be changed while setting the machine up is unprotected; folders ..\MTB, ..\USERS, ..\DIAGNOSIS and the Windows register.

Protection password.

The access to this mode is protected with the password "machine parameters", defined in the utilities mode. When starting the unit up in this access mode, it will request the access password.

2.3.3 User mode.

It is the usual work mode for the user, once the setup is completed. This mode does not allow updating the CNC or accessing the operative system. Some utilities of the operative system (task manager, clock) will be available from the diagnosis mode.

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. Files from flash, pendrives, ethernet, etc. can only be managed from the explorer of the utilities mode.

Protection level.

The user mode has the maximum protection level where only the folders and files that may be changed during the normal operation of the machine are unprotected.

Protection password.

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

2.3.4 Restrictions for working in setup mode and switching to user mode.

The purpose of the following restrictions is for the OEM to deliver its machines with the CNC in access protected mode (User mode).

Setup mode.

On power-up, the CNC will show a message indicating that it is in an unprotected mode and that the setup has not been completed yet. In this situation, the CNC is no longer under Fagor warranty. After a certain period of time, with the next reset, the CNC shows the message again.

When closing the application, the CNC asks if the setup is completed..

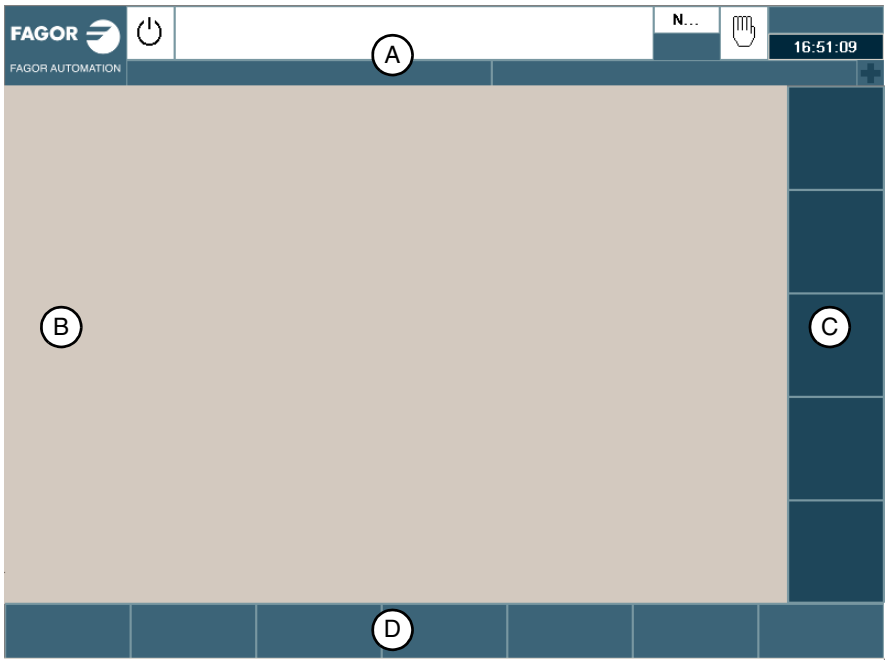
- If YES is selected and there are passwords, the CNC makes a backup copy and switches over to user mode (process OK).
- If YES is selected and there are no passwords, the CNC issues a warning message and does not close the application.

User mode.

On power-up, the CNC checks for passwords and a backup copy. If any of these two is missing, the CNC will show a message indicating that setup has not been completed yet. In this situation, the CNC is no longer under Fagor warranty. The CNC checks this at every reset.

This situation can be reached when accessing the user mode from setup mode through "DiskMonitor".

2.4 The interface.



A General CNC-status bar.

This area shows the information on the program being executed, the active operating mode, the PLC messages, etc. See *"2.4.1 The CNC status bar."* on page 40.

B Pages or windows of the active operating mode.

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

C Vertical softkey menu.

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

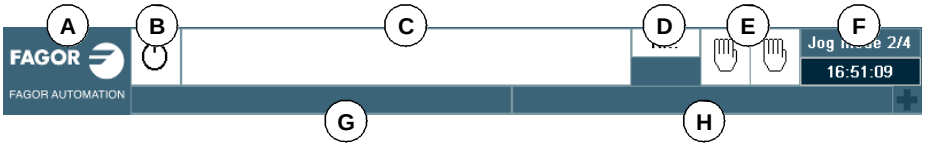
This menu may appear to the left or right of the interface depending on how the OEM has configured the machine.

D Horizontal softkey menu.

The menu options change, depending on the active operating mode. The different options are selected using softkeys F1 through F7. See *"2.4.5 Horizontal softkey menu."* on page 44.

2.4.1 The CNC status bar.

The status bar of the CNC (top of the screen) shows the following information.



A Task window. Clicking with the mouse or pressing on a touch-screen, the CNC shows the task window (same as pressing the keystroke sequence [CTRL]+[A]) that shows the list of the work modes and hotkeys of the CNC. See *"2.4.2 Task window."* on page 41.

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

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

C Program selected in the active channel for execution.

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

E Information about the channels. Number of channels available and active channel (indicated in blue). Icons show which operating mode each channel is in. The channel synchronization window may be expanded using the [ALT]+[S] keys. See *"2.4.3 Channel synchronization window."* on page 42.

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

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

F Active operating mode (automatic, manual, etc.) selected page number and total number of pages available. System clock.

Click with the mouse or press on a touchscreen to display the list of pages available and also to configure which are hidden and which are visible.

G Active CNC message. For each channel, this area 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.

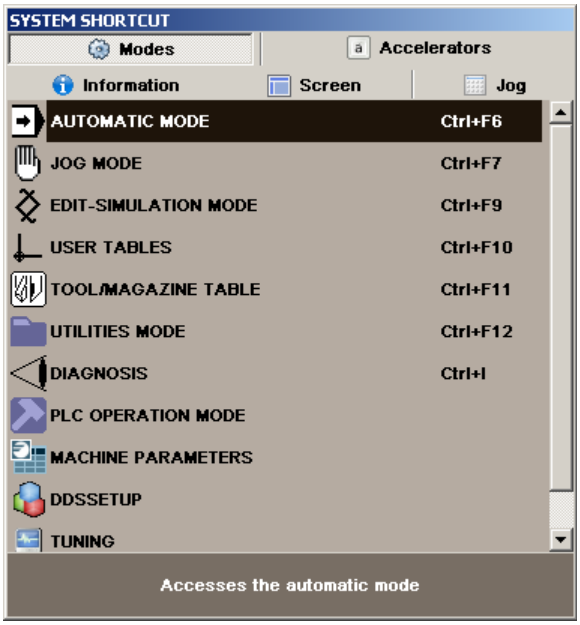
The list of messages shows, next to each message, the channel where it is active.

H PLC messages.

Further information: See *"2.4.4 PLC message window."* on page 43.

2.4.2 Task window.

The task window shows the list of all the operating modes of the CNC and the list of available hotkeys. To open the task window, use the key combination [CTRL]+[A] or click with the mouse (or press on a touchscreen) on the OEM icon on the status bar (top left of the screen). 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.

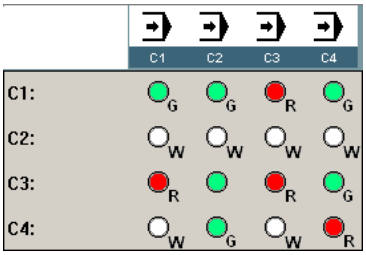
[⬆] [⬇]	Moves the cursor through the window.
[CTRL]+[TAB]	Changes the panel.
[ENTER]	It accesses the selected mode.

2.4.3 Channel synchronization window.

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

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

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



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

Channel 2 is not expecting any synchronism mark.

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

2.4.4 PLC message window.

When the PLC activates a message, the CNC shows on the general status bar (lower right) 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. To display the list of active messages, press the key combination [CTRL]+[M] or click on the PLC message line.

List of active PLC messages.

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. To close the additional data window, press [ESC].

Icon.	Meaning.
	The message does not have a file with additional information.
	The message has a file with additional information.

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.

Key.	Meaning.
	Scroll the window line by line.
	Scroll the window page by page.
	Take the window to the beginning or to the end.

AVI file.

Key.	Meaning.
	Stop the video and move to next frame or previous frame.
	Stop the video and move to the next 5 frames or to the previous 5 frames.
	Stop the video and move to the end or to the beginning.
	Stop the video.
	Play the video.

2.4.5 Horizontal softkey menu.

The machine manufacturer may have configured how to use the horizontal softkey menu (parameter SFTYPE) in one of the following ways.

Standard softkey menu.

Softkey tree based on menus and submenus; i.e., there are different softkey levels within the same operating mode. To return to the main menu, press the [BACK] key (previous menu).

Key.	Meaning.
	Previous menu.

Pop-up softkey menu.

Softkey tree based on popup menus so there is only one softkey level. There are no softkey submenu, which simplifies the operation.

The softkey menu expands and shrinks with keys [F1] through [F7]. The softkey of the pop-up menu are selected with the [ENTER] key or from the numeric keypad using the number that comes with each softkey.

2.4.6 CNC warnings and errors window

When an error occurs, the CNC displays a window with the error category, its number and its associated text. There are three error categories, each one of them shows a window in a different color.

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 user can toggle the different active errors or messages using the [⬆][⬇] keys.

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

Included in this category are program syntax errors, errors generated by the PLC, etc. These errors stop the execution of the program and are errors that must be corrected.

While the error window is active, no other action will be possible but removing it (it is not possible to change operating modes in the channel). Some errors may be eliminated by pressing the [ESC] key, whereas for others, the [ESC] key only closes the window that shows it and the [RESET] key must be pressed to eliminate the error state. After pressing [RESET] the CNC assumes the initial conditions set by the machine manufacturer with the machine parameters.

The [RESET] key is needed to eliminate the errors that open the emergency relay, errors occurred in execution, loop errors, bus errors, PLC errors, hardware errors, etc.

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.

2.4.7 PLC error window.

When the PLC activates an error, the CNC interrupts the execution of the part-program 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 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.

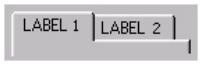
Key.	Meaning.
↑ ↓	Scroll the window line by line.
PgUp PgDn	Scroll the window page by page.
HOME END	Take the window to the beginning or to the end.

AVI file.

Key.	Meaning.
← →	Stop the video and move to next frame or previous frame.
PgUp PgDn	Stop the video and move to the next 5 frames or to the previous 5 frames.
HOME END	Stop the video and move to the end or to the beginning.
SPACE	Stop the video.
ENTER	Play the video.

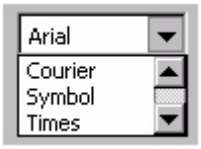
2.4.8 Dialog boxes.

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.

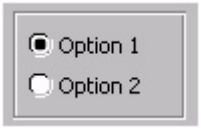


- Drop list.

  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 palette.

  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.

2.5 Operating modes.

Access to the operating modes.

The CNC operating modes may be accessed from the keyboard or from the task window. To open the task window, use the key combination [CTRL]+[A] or click with the mouse (or press on a touchscreen) on the OEM icon on the status bar (left part).

From the task window, it is possible to access all the operating modes of the CNC, whereas from the keyboard it is only possible to access the specific operating modes of the OEM (PLC, machine parameters, etc.). When this operating mode is active, it will be indicated at the top right side of the general status bar.

Operating modes accessible from the keyboard.

Key.	Operation mode.
AUTO	Automatic mode; PROGRAM or CUT_VIEW.
MANUAL	Manual mode.
EDIT	EDISIMU mode.
MDI	MDI/MDA mode.
TABLES	User tables (zero offsets, fixtures and arithmetic parameters).
TOOLS	Technology tables.
UTILITIES	Utilities mode.
CUSTOM	Configurable mode. OEM-configurable key that can can access to one of the following operating modes. • Machine parameters. • PLC. • Diagnosis mode. • Setup assistance. • Machine status.

The [CUSTOM] key, depending on how the OEM has configured it, can also execute an application (FGUIM), access the operating system or not carry any action out.

Screens or pages the make up the work modes.

Each operating mode may consist of several screens or pages. The current screen or page and the total number of available screens are shown at the upper right hand side of the general status bar.

NEXT

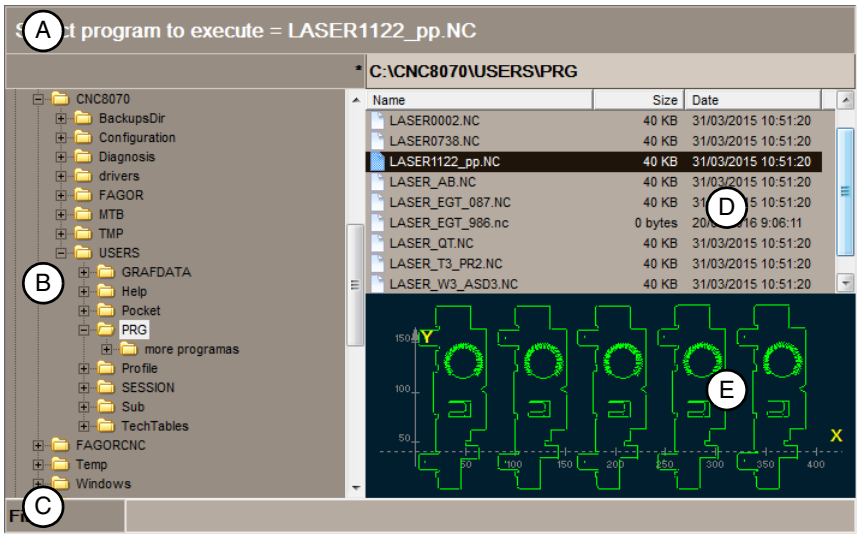
The exchange between the different screens of an operating mode is done using the key to access that operating mode or with the [NEXT] key (if the OEM has configured it that way). When pressing one of these keys, the CNC will show the next screen and when pressing them together with the [SHIFT] key, 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.

By clicking on the active operating mode, the CNC shows the list of available pages and allows you to configure which ones are hidden and which ones are visible.

2.6 File selection window.

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 and operation to carry out (open, save, etc.).

B Folder tree.

C Area to define the name of the new files. A file can also be selected when knowing its name.

D Files contained in the selected folder.

E Preview of the selected file.

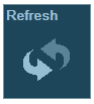
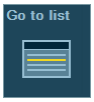
Horizontal softkey menu.

The horizontal softkey menu shows the following options.

Softkey.	Description.
Sort by name.	Every time the same softkey is selected, the CNC toggles the sorting criteria from ascending to descending and vice versa.
Sort by size.	
Sort by date.	
Sort by type.	
Search for file.	Search for a file by name.
	Show or hide the preview window.

Vertical softkey menu.

The vertical softkey menu shows the following options.

Softkey.	Description.
	Refresh the file window.
	Block/unblock the selected folder. This softkey hides the folder tree and expands the file and preview windows.
	Select the folder tree.
	Select the files window.
	Select the editing window.

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.

To select a file from the list.

A file may be selected 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. To make searching easier, the file list may be sorted according to different criteria.

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.

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.

Find file.

The option "file search" of the softkey menu may be used to look, in the selected folder, for all the files whose name contain the indicated text. When selecting this option, the CNC shows a dialog box requesting the text to be found. The programs are searched one by one.

When defining a search, the softkey menu shows the "Next" (up) and "Previous" (down) options. To end the search, press [ESC].

2.7 Calculator.

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 accessed from an editable element, the result shown by the calculated may be inserted into it. 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. The editing window 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 [⬆][⬇]. After selecting an expression from the window, press the [ENTER] key to recover it. The [ESC] key closes the history window.

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.

2.7.1 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.

The calculator offers up to 26 variables (from "A" to "Z", except "Ñ",) 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.

Arithmetic operations.

x + *y*	Add	
x - *y*	Subtract	
x * *y*	Multiply	
x / *y*	Division	
x% *y*	Percentage	10%50 = 5
x^ *y*	Power	2^3 = 2 * 2 * 2 = 8
x!	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.

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

Extended functions.

SQRT x	Square root	SQRT 16
LN x	Neperian log	LN 100
LOG x	Decimal log.	LOG 100
e ^ x	"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.

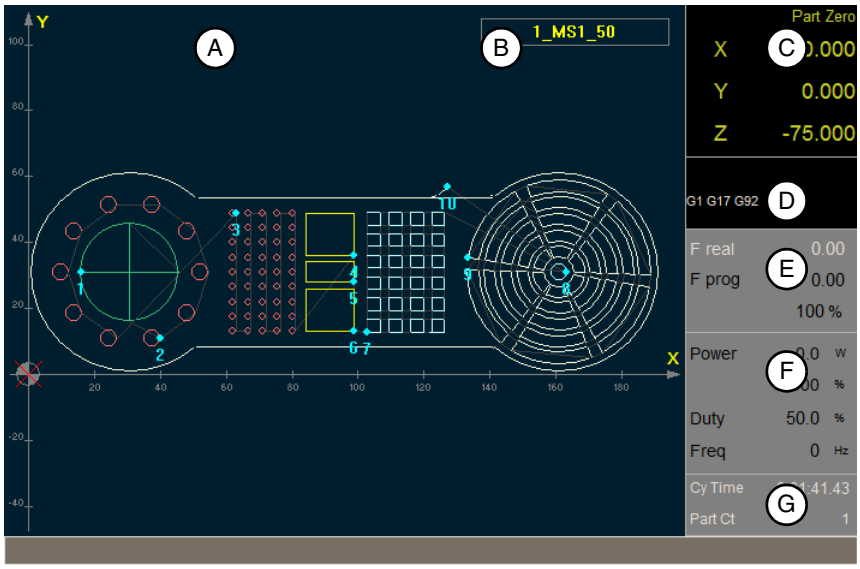
The "E" variable defines the error allowed so the result of the function with the calculated value is smaller than "E". This variable is useful when not knowing what value to assign to "N". In that case, it is recommended to assign to "N" a very large value and to "E" the error allowed.

AUTOMATIC MODE: CUT_VIEW.

3

The [AUTO] button is used to access the pages CUT_VIEW and STANDARD in automatic mode and allows you to toggle between them.

The page CUT_VIEW provides two functions: before the cutting operation (before pressing the [START] button), it displays a preview of the part to be cut, as well as the piercing points. During the cut (after pressing the [START] button), the CNC displays the laser path in red on top of the original preview.



A Graphic window.

B Active material.

C The current position of the axes as compared to part zero. If the name of the axis appears in red, it means that the PLC is inhibiting the movement of the axes (INHIBIT mark active).

D Active "M" and "G" functions and high-level commands.

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

"F real" Actual feedrate of the axes.

"F prog" Programmed feedrate and feedrate percentage applied as compared to the programmed. When setting a new feedrate in the MDI/MDA mode, it will become the new feedrate for the automatic mode.

If the text "F real" appears in red, it means that the PLC is inhibiting the movement of the axes (FEEDHOL mark active).

F Information on the laser generator power.

"Power" Laser generator power and percentage of power applied.

"Duty" Percentage of work cycle (duty) of the PWM.

"Freq" PWM frequency.

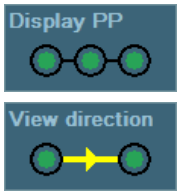
G Execution (cycle) time "Cy Time" of the program and number of parts made "Part Ct".

The graphics window.

Information from the graphic window before the cutting operation.

Before the cutting operation (before pressing the [START] button), the window displays a graphic representation of the part to be cut. The horizontal softkey menu allows the piercing points to be displayed/hidden and to select the piercing to be used in the cut. The paths are represented with the following colors.

- Non-cutting paths (G00) are shown in grey.
- Cutting paths (G01, G02, G03, etc.) are shown in the color defined in the technology table.

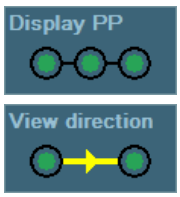


- The piercing points are displayed in light blue and are numbered. These points are hidden by default; use the "Display PP" softkey in the horizontal menu to display them.
- Display or hide the cutting direction in each block.

Information from the graphic window during the cutting operation.

During the cutting operation (after pressing the [START] button), this window displays the laser path. The paths are represented with the following colors.

- Non-cutting paths (G00) are shown in grey.
- During the cutting, the cursor advances according to the real position of the cut. The CNC shows the path of the laser in red (machined blocks). The rest of the blocks are shown in their original color, as defined in the technology table.



- The piercing points are displayed in light blue and are numbered. These points are hidden by default; use the "Display PP" softkey in the horizontal menu to display them.
- Display or hide the cutting direction in each block.

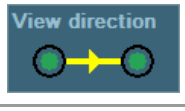
Zoom in the graphic window.

The zoom option is available using the mouse or the keyboard.

Keys.	Mouse.	Action.
[+][–]	Wheel.	Zoom in or out.
[⬅][➡][⬆][⬇]	Right button,	Move the graphic.
	Left button.	Zoom in on a certain area. Press the left button and drag the mouse to create a window with the area to expand. Release the button to zoom in.
[HOME]	Left button. (Double click)	Best zoom (initial zoom).

Horizontal softkey menu.

The horizontal softkey menu shows the following options.

Softkey.	Description.
	Show or hide the piercing points.
	Select the piercing points to use.
	Display or hide the cutting direction in each block.

The status of the first two softkeys depends on the presence of the instructions #PIERCING and #CUTTING in the program.

- Program without #PIERCING; both softkeys are disabled.
- Program with #PIERCING, but without #CUTTING; piercing points can be shown. The softkeys for selecting the piercing points are disabled.
- Program with #PIERCING and #CUTTING; both softkeys are disabled.

Vertical softkey menu.

The vertical softkey menu shows the following options.

Softkey.	Description.
	Select a program for execution.
	Begin tool inspection. Tool inspection is only available when program execution is interrupted.
	Select the program that is being edited.

3.1 Select the piercing points.

The horizontal softkey menu allows the selection of the piercing points to be used. The selection is only active for a single execution; after finishing the execution or a reset, it is canceled. After pressing the "Select PP" softkey, the menu displays the following options.

- Initial piercing point.

  Select a piercing point to start the execution. The CNC uses a dotted line to show the paths that are not machined and a solid line to show those that are.

  The CNC executes all the blocks from the start of the program to the block that calls up the initial subroutine (#INITIALSUB). After executing the subroutine, the execution continues at the selected piercing point. The movement at this point is carried out in G00.

If there is no block to call up the initial subroutine in the program (#CALL INITIALSUB), the CNC does not allow an initial piercing point to be selected.

- Final piercing point.

  Select a piercing point to end the execution. The CNC uses a dotted line to show the paths that are not machined and a solid line to show those that are.

  After executing the command #CUTTING OFF for the selected piercing point, the execution continues from the block that calls up the final subroutine (#FINALSUB) until block M30. If there is no block that calls up the final subroutine, the execution ends at block M30.

- Sole piercing point.

  Select a piercing point and machine only that part. This option is the equivalent of selecting the same piercing point as both the initial and the final point.

- Select all.

  Select all the piercing points; i.e., execute the entire program.

3.2 Program execution.

3.2.1 Program selection.

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

Softkey.	Meaning.
	This softkey shows the list of available programs.
	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.

3.2.2 Execute a program.

The name of the program selected in the channel for execution appears on the general status bar. 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.

Start executing

To start the execution of the program, press [START] on the operator panel.

Every time [START] is pressed, the CNC checks that the room temperature does not exceed 65 ºC (149 ºF) and, if it does, the CNC does not let run the program and issues the corresponding error message.

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.

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.

Activate rapid traverse while executing a program.

If the OEM has set it this way (parameter RAPIDEN), while executing the program it will be possible to apply a fast feedrate to programmed movements (except to G00 movements). Depending on how this option has been configured, rapid traverse may be applied from the PLC (for example using user keys) or with the "rapid" key of the jog panel or, on the other hand, it may be necessary to use both.

When only one of the two options is necessary to activate rapid traverse and the CNC has several channels, activating rapid traverse from the PLC only affects the corresponding channel. The rapid key, on the other hand, affects simultaneously all the channels that may be affected at the time. If the active channel is in jog mode and another channel is executing a program, when pressing the rapid key in the active channel (jog mode), the rapid traverse will also be applied in the channel that is executing the program.

The feedrate for these movements is set by the FRAPIDEN parameters of the axes and of the channel. If these parameters are set to zero, the CNC does not limit the feedrate and assumes the one set for G00. The feedrate cannot exceed the maximum feedrate set by PLC (variable (V.)PLC.G00FEED), but it can exceed the maximum machining feedrate (parameter MAXFEED of the axes and of the channel) and the active feedrate set by PLC (variable (V.)PLC.F).

3.3 Tool inspection.

When the execution is interrupted, tool inspection makes it possible to jog the axes, execute blocks from MDI/MDA mode, etc. At the conclusion of the tool inspection, this function enables the axes to be repositioned at the point of interruption or at the initial point of the interrupted block and to continue with the program execution.

If an error occurs in tool inspection mode and it may be eliminated by pressing [ESC], it will not affect the inspection process. If eliminating the error requires a reset, the CNC will request confirmation because a reset cancels the inspection.

The way to operate in tool inspection may be summarized in the following steps.

1. Interrupt the execution of the program and start tool inspection.
2. Run the operations of tool inspection, such as jogging the axes, executing blocks in MDI/MDA mode, etc.
3. Repositioning the axes.
4. Resume the execution of the program.

Begin tool inspection.

Tool inspection may be accessed from the vertical softkey menu only when the execution of the program has been interrupted ([STOP] key). Activating tool inspection makes the following operations possible:

- Jog the axes with the JOG keys.
- Execute blocks in MDI/MDA mode.

Once tool inspection has been completed and before resuming the execution of the program, the axes must be repositioned.

Execute blocks in MDI/MDA mode.

Any program block may be executed in MDI/MDA mode. The conditions when entering the MDI/MDA mode will be those of the interruption point; i.e., the CNC maintains the history of active G and M functions, feedrate, tool and other commands that were programmed. However, the CNC treats the following functions and commands in a special way.

- The CNC assumes function G1 when resuming execution after tool inspection, restores the function that was active at the interruption point.
- The CNC activates the suspended angular transformation (#ANGAX); it restores them when resuming execution after tool inspection.
- The CNC cancels tangential control (#TANGCTRL); it restores it when resuming execution after tool inspection.
- The CNC assumes function G40 when resuming execution after tool inspection, restores the function that was active at the interruption point. If either G40, G41 or G42 is programmed in MDI, the CNC stores it without making it effective; it applies it when resuming execution after tool inspection.
- If at the time of the interruption, there is a #TOOL ORI movement in execution or pending, it is not made effective in blocks programmed in MDI; the movement is executed when resuming execution after tool inspection.
- The CNC does not execute modal subroutines that are active in the interrupted program, nor calls to other subroutines.
- If the active mirror functions are changed in MDI mode, the changes will affect the following blocks edited in MDI; but not the movements in jog, or repositioning or the end point of the interrupted block. The interrupted program assumes the changes from the next block on.
- The CNC cancels functions G200 and G201 (manual intervention); it restores them when resuming execution after tool inspection.

In general, all the changes made in MDI/MDA mode are kept active when resuming the program after tool inspection except the following functions that are restored at the time of interruption.

- The CNC restores the type of interpolation G00, G01, G02, G03, G33 or G63 that was active at the time of the interruption.
- The CNC restores function G90 or G91 that was active at the time of the interruption.
- The CNC restores the status of function #MCS that was active at the time of the interruption.

Repositioning the axes.

To complete the tool inspection and before resuming the execution of the program, the axes must be repositioned. The vertical softkey menu offers two options.

Softkey.	Reposition point.
	Repositioning the axes at the interruption point.
	Repositioning the axes at the starting point of the interrupted block.

After selecting one of these two options, the vertical softkey menu shows a list of axes that are out of position.

If the execution is interrupted during a polynomial interpolation (#POLY), the axes must be repositioned to the beginning of the interrupted block in order to be able to redo the same path.

Repositioning the axes.

The CNC allows repositioning the axes either one by one or in groups. Use the vertical softkeys to select the axes to be repositioned and press [START]. The CNC will reposition the axes at the selected point (according to the softkey selected earlier) at the feedrate set by the machine manufacturer. Once one axis has reached its position, it will no longer be available.

The movement of the axes may be interrupted with the [STOP] key, after which it is possible to jog the axes using the jog keyboard. After interrupting a movement, it is necessary to select again the axes to be repositioned.

Canceling repositioning.

The CNC admits the possibility to end tool inspection before it is completed; i.e. without having repositioned all the axes. The vertical softkey menu, next to the list of axes, shows the following softkey to cancel repositioning.

Softkey.	Meaning.
	Canceling repositioning.

Resuming the execution of the program.

Once all the axes are repositioned or after canceling repositioning, press [START] to resume program execution.

- If tool inspection has ended by repositioning all the axes, when pressing [START], the CNC completes the interrupted path and goes on with the rest of the program.
- If tool inspection has ended after canceling the repositioning of the axes, when pressing [START], the axes move from their current position to the end point of the interrupted path and then the CNC goes on with the rest of the program.

3.4 Change the machining conditions (MHFS mode).

MHFS

When the execution is interrupted, the "MHSF" softkey allows the machine conditions to be changed by executing any M, H, F or S function. The MHFS mode is only available when the independent interpolator is involved in the movement of one of the axes (#MOVE, #FOLLOW, #CAM, etc).

After this softkey is pressed, the CNC allows the feedrate value and the M and H functions to be edited. To assign the new values, press [START]. 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.

3.5 Display/hide the dynamic override.

From this page during the machining, the user can change the dynamics set by program.

The [CTRL][H] hotkey shows/hides the dynamic override of the HSC at the bottom of the screen. On this bar, a cursor may be used to change the percentage of the dynamics set by program. With the bar, it also shows the upper and lower limits (range) between which the value may be varied. Over the cursor, it shows the percentage being applied. The cursor may be moved with the mouse or with the moving arrows of the keyboard. The [ESC] key also hides the bar.

3.

AUTOMATIC MODE: CUT_VIEW.
Display/hide the dynamic override.

FAGOR AUTOMATION

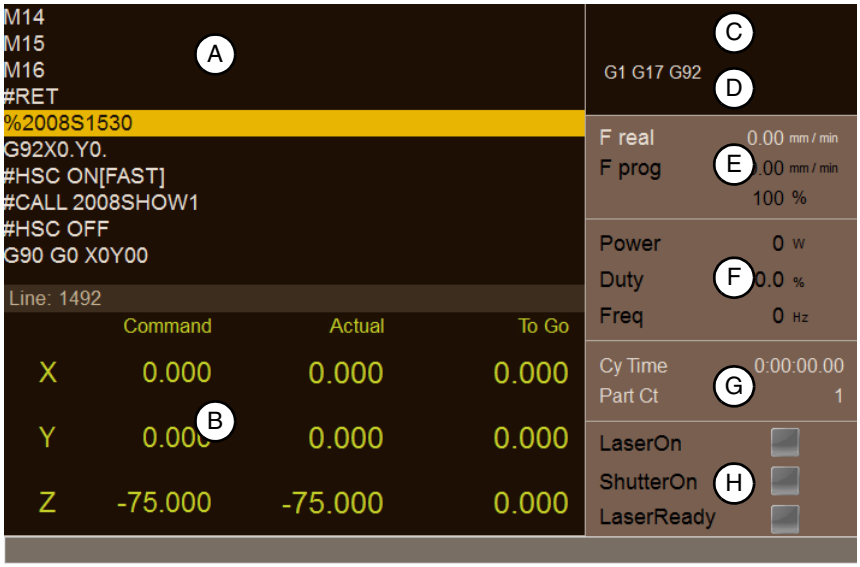
CNC 8060 L
CNC 8070 L

(REF: 1707)

AUTOMATIC MODE: PROGRAM.

4

The [AUTO] button is used to access the pages CUT_VIEW and STANDARD in automatic mode and allows you to toggle between them.



A Program window. It shows data on the program selected for execution and lets selecting the initial and final execution blocks.

B Information related to the position of the axes.

"Command" The programmed coordinate, in other words, the target position.

"Current" The current position of the axes as compared to part zero.

"To Go" Distance to reach the programmed point.

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

"F real" Actual feedrate of the axes.

"F prog" Programmed feedrate and feedrate percentage applied as compared to the programmed. When setting a new feedrate in the MDI/MDA mode, it will become the new feedrate for the automatic mode.

If the text "F real" appears in red, it means that the PLC is inhibiting the movement of the axes (FEEDHOL mark active).

F Information on the laser generator power.

"Power" Laser generator power.

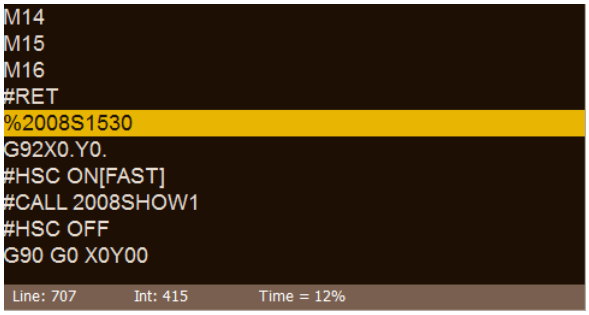
"Duty" Percentage of work cycle (duty) of the PWM.

"Freq" PWM frequency.

G Execution (cycle) time "Cy Time" of the program and number of parts made "Part Ct".

H Laser status. Green LED indicates that the signal status is "Ok" and grey LED indicates otherwise.

Program window.



When accessing the PROGRAM page 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. It shows the following data.

- Blocks of the program or subroutine being executed.
- Line of the program where the cursor is.

During execution, the window shows the following information.

- The cursor shows the block being executed.
- The bottom shows the line number being executed (Line) and, if necessary, the line number where the execution was canceled (INT).
- If the program has been executed before, the bottom shows the percentage of the program that has been executed already (Time). Once a program has been executed, the CNC knows how long it took and in later executions it shows the percentage of the program executed so far. This percentage will be reliable as long as the percentage of feedrate is not changed with respect to when it was executed for the first time.

Vertical softkey menu.

The vertical softkey menu shows the following options.

Softkey.	Description.
	Select a program for execution.
	Begin tool inspection. Tool inspection is only available when program execution is interrupted.
	Select the program that is being edited.

4.1 Program execution.

4.1.1 Program selection.

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

Softkey.	Meaning.
	This softkey shows the list of available programs.
	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.

4.1.2 Execute a program.

The name of the program selected in the channel for execution appears on the general status bar. 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.

Start executing

To start the execution of the program, press [START] on the operator panel.

Every time [START] is pressed, the CNC checks that the room temperature does not exceed 65 ºC (149 ºF) and, if it does, the CNC does not let run the program and issues the corresponding error message.

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.

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.

Activate rapid traverse while executing a program.

If the OEM has set it this way (parameter RAPIDEN), while executing the program it will be possible to apply a fast feedrate to programmed movements (except to G00 movements). Depending on how this option has been configured, rapid traverse may be applied from the PLC (for example using user keys) or with the "rapid" key of the jog panel or, on the other hand, it may be necessary to use both.

When only one of the two options is necessary to activate rapid traverse and the CNC has several channels, activating rapid traverse from the PLC only affects the corresponding channel. The rapid key, on the other hand, affects simultaneously all the channels that may be affected at the time. If the active channel is in jog mode and another channel is executing a program, when pressing the rapid key in the active channel (jog mode), the rapid traverse will also be applied in the channel that is executing the program.

The feedrate for these movements is set by the FRAPIDEN parameters of the axes and of the channel. If these parameters are set to zero, the CNC does not limit the feedrate and assumes the one set for G00. The feedrate cannot exceed the maximum feedrate set by PLC (variable (V.)PLC.G00FEED), but it can exceed the maximum machining feedrate (parameter MAXFEED of the axes and of the channel) and the active feedrate set by PLC (variable (V.)PLC.F).

4.2 Cancel the execution and resume from another block while keeping the history.

The CNC allows canceling the execution of an interrupted program ([STOP] key) keeping the program history (machining conditions) and resume execution at another previous or later block of the same program. Before resuming execution, it is possible to access the MDI/MDA mode and execute blocks. The CNC resumes the program at the selected block with the history active up to the interrupted block plus the changes executed via MDI/MDA.

This option to cancel and resume may be useful when combined with tool inspection, after canceling or finishing the repositioning of the axes. At that moment, it will be possible to select a program block, previous or after the interruption block, and resume program execution from that point on. When linking this option with the end of tool inspection, the repositioning of the axes always corresponds to the starting point of tool inspection and this repositioning is done before selecting the block to resume.

It acts as follows:

1 After interrupting the execution ([STOP] key), the vertical softkey menu of the automatic mode will show the following softkey. Pressing this softkey, the CNC interrupts the execution of the program keeping the history (machining conditions) active at the interruption point. The message bar indicates that a program block may be selected to resume execution.

2 If necessary, access the MDI/MDA mode to execute the necessary blocks to adapt to the new starting point (position the axes, change the machining conditions, etc.).

3 Select the new starting block to resume execution, it may be a block before or after the interruption block. The block is selected by moving the cursor throughout the program or through the search options for text or program line.

4 Once the stating block has been selected, press the [START] key to resume execution at that block. Resuming the program does not initialize or change the program history, nor does it take into consideration the possible changes programmed in the blocks that are not executed between the interruption block and the resuming block of the program. The CNC resumes execution with the history active up to the interrupted block plus the changes executed via MDI/MDA.

The CNC offers no automatic repositioning of axes. For positioning for example, at the starting point of the block where the execution is to be resumed, it must be done via MDI/MDA (if the exact point is known) or select as starting point the motion block right before the desired one.

Considerations before resuming the execution of the program.

It is up to the user to use this feature properly and in the right context, using the MDI/MDA mode to adapt the new starting point to the conditions of the interruption point if necessary. For example:

- When selecting an arc as the starting point, use the MDI/MDA mode to place the axes in their starting point; otherwise the CC will issue programming error messages or it will execute arcs different from the original one. This situation may also be solved by selecting as starting point a previous linear block that positions the axes in the starting point of the right arc.
- If G91 is active in the history of the program, position at the starting point of the original block; otherwise, the resulting path will be different because it will always be incremental with respect to the starting point.
- Inside a loop ($IF, $GOTO, etc), the execution can only be resumed if the resuming block is at the same level as the block where it was canceled.

Modify the interrupted program.

If the interrupted program has a ".mod" extension, it may be modified from the editor while it is interrupted. It is up to the user to check that the changes he makes are coherent.

For the CNC to assume the changes, the modified block must be after the block where it resumes execution.

4.3 Tool inspection.

When the execution is interrupted, tool inspection makes it possible to jog the axes, execute blocks from MDI/MDA mode, etc. At the conclusion of the tool inspection, this function enables the axes to be repositioned at the point of interruption or at the initial point of the interrupted block and to continue with the program execution.

If an error occurs in tool inspection mode and it may be eliminated by pressing [ESC], it will not affect the inspection process. If eliminating the error requires a reset, the CNC will request confirmation because a reset cancels the inspection.

The way to operate in tool inspection may be summarized in the following steps.

1. Interrupt the execution of the program and start tool inspection.
2. Run the operations of tool inspection, such as jogging the axes, executing blocks in MDI/MDA mode, etc.
3. Repositioning the axes.
4. Resume the execution of the program.

Begin tool inspection.

Tool inspection may be accessed from the vertical softkey menu only when the execution of the program has been interrupted ([STOP] key). Activating tool inspection makes the following operations possible:

- Jog the axes with the JOG keys.
- Execute blocks in MDI/MDA mode.

Once tool inspection has been completed and before resuming the execution of the program, the axes must be repositioned.

Execute blocks in MDI/MDA mode.

Any program block may be executed in MDI/MDA mode. The conditions when entering the MDI/MDA mode will be those of the interruption point; i.e., the CNC maintains the history of active G and M functions, feedrate, tool and other commands that were programmed. However, the CNC treats the following functions and commands in a special way.

- The CNC assumes function G1 when resuming execution after tool inspection, restores the function that was active at the interruption point.
- The CNC activates the suspended angular transformation (#ANGAX); it restores them when resuming execution after tool inspection.
- The CNC cancels tangential control (#TANGCTRL); it restores it when resuming execution after tool inspection.
- The CNC assumes function G40 when resuming execution after tool inspection, restores the function that was active at the interruption point. If either G40, G41 or G42 is programmed in MDI, the CNC stores it without making it effective; it applies it when resuming execution after tool inspection.
- If at the time of the interruption, there is a #TOOL ORI movement in execution or pending, it is not made effective in blocks programmed in MDI; the movement is executed when resuming execution after tool inspection.
- The CNC does not execute modal subroutines that are active in the interrupted program, nor calls to other subroutines.
- If the active mirror functions are changed in MDI mode, the changes will affect the following blocks edited in MDI; but not the movements in jog, or repositioning or the end point of the interrupted block. The interrupted program assumes the changes from the next block on.
- The CNC cancels functions G200 and G201 (manual intervention); it restores them when resuming execution after tool inspection.

In general, all the changes made in MDI/MDA mode are kept active when resuming the program after tool inspection except the following functions that are restored at the time of interruption.

- The CNC restores the type of interpolation G00, G01, G02, G03, G33 or G63 that was active at the time of the interruption.
- The CNC restores function G90 or G91 that was active at the time of the interruption.
- The CNC restores the status of function #MCS that was active at the time of the interruption.

Repositioning the axes.

To complete the tool inspection and before resuming the execution of the program, the axes must be repositioned. The vertical softkey menu offers two options.

Softkey.	Reposition point.
	Repositioning the axes at the interruption point.
	Repositioning the axes at the starting point of the interrupted block.

After selecting one of these two options, the vertical softkey menu shows a list of axes that are out of position.

If the execution is interrupted during a polynomial interpolation (#POLY), the axes must be repositioned to the beginning of the interrupted block in order to be able to redo the same path.

Repositioning the axes.

The CNC allows repositioning the axes either one by one or in groups. Use the vertical softkeys to select the axes to be repositioned and press [START]. The CNC will reposition the axes at the selected point (according to the softkey selected earlier) at the feedrate set by the machine manufacturer. Once one axis has reached its position, it will no longer be available.

The movement of the axes may be interrupted with the [STOP] key, after which it is possible to jog the axes using the jog keyboard. After interrupting a movement, it is necessary to select again the axes to be repositioned.

Canceling repositioning.

The CNC admits the possibility to end tool inspection before it is completed; i.e. without having repositioned all the axes. The vertical softkey menu, next to the list of axes, shows the following softkey to cancel repositioning.

Softkey.	Meaning.
	Canceling repositioning.

Resuming the execution of the program.

Once all the axes are repositioned or after canceling repositioning, press [START] to resume program execution.

- If tool inspection has ended by repositioning all the axes, when pressing [START], the CNC completes the interrupted path and goes on with the rest of the program.
- If tool inspection has ended after canceling the repositioning of the axes, when pressing [START], the axes move from their current position to the end point of the interrupted path and then the CNC goes on with the rest of the program.

4.4 Change the machining conditions (MHFS mode).

MHFS

When the execution is interrupted, the "MHSF" softkey allows the machine conditions to be changed by executing any M, H, F or S function. The MHFS mode is only available when the independent interpolator is involved in the movement of one of the axes (#MOVE, #FOLLOW, #CAM, etc).

After this softkey is pressed, the CNC allows the feedrate value and the M and H functions to be edited. To assign the new values, press [START]. 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.

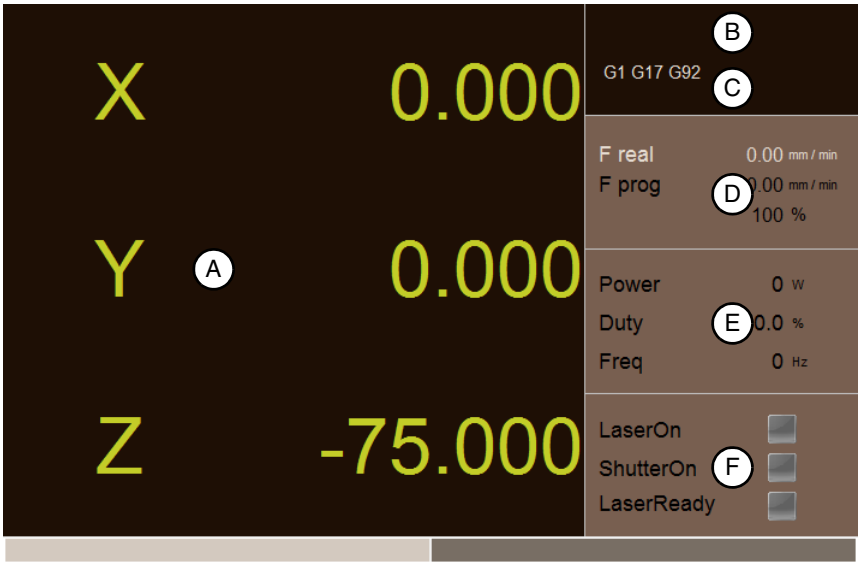
4.5 Display/hide the dynamic override.

From this page during the machining, the user can change the dynamics set by program.

The [CTRL][H] hotkey shows/hides the dynamic override of the HSC at the bottom of the screen. On this bar, a cursor may be used to change the percentage of the dynamics set by program. With the bar, it also shows the upper and lower limits (range) between which the value may be varied. Over the cursor, it shows the percentage being applied. The cursor may be moved with the mouse or with the moving arrows of the keyboard. The [ESC] key also hides the bar.

MANUAL MODE.

The [MANUAL] key accesses MANUAL mode.



A Current position of the axes as compared to part 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 Information related to the feedrate "F" of the axes:

"F real" Actual feedrate of the axes.

"F prog" Programmed feedrate and feedrate percentage applied as compared to the programmed. When setting a new feedrate in the MDI/MDA mode, it will become the new feedrate for the automatic mode.

If the text "F real" appears in red, it means that the PLC is inhibiting the movement of the axes (FEEDHOL mark active).

E Information on the laser generator power.

"Power" Laser generator power.

"Duty" Percentage of work cycle (duty) of the PWM.

"Freq" PWM frequency.

F Laser status. Green LED indicates that the signal status is "Ok" and grey LED indicates otherwise.

Horizontal softkey menu.

The horizontal softkey menu shows the following options.

Softkey.	Description.
Turn the CNC off.	Start the CNC shut-down sequence, it is the same as pressing the key combination [ALT][F4].

Vertical softkey menu.

The vertical softkey menu shows the following options.

Softkey.	Description.
	Change the units for data display. The softkey highlights the units currently selected (millimeters or inches). The selected units are only valid for displaying data. For programming, the CNC assumes the units defined with the active function G70 or G71, or, when not programmed, the units set by the machine manufacturer (INCHES parameter). The CNC will display this softkey or not depending on how machine parameter MMINCHSOFTKEY has been set.
	Setting and activating the zero offsets and the fixture offsets. This softkey shows the zero offsets and the fixture offsets of the system, either to store the active zero offset or to activate a new zero offset.

5.1 Operations with the axes.

5.1.1 Machine reference zero (home) search.

Home search 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 (axis by axis from the operator panel) or automatically (using a subroutine).

Manual home search (one axis at a time).

The axis-by-axis home search cancels the zero offset, the fixture offset and the measuring offset. The CNC assumes the machine reference zero point (home) as the new part zero.

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 can be moved with the [⬆][⬇] 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.

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

Automatic home search (with subroutine).

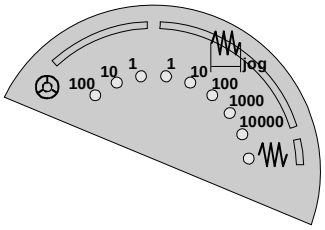
This homing method is only available if the machine manufacturer has previously defined a homing subroutine.

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.

5.1.2 Jog

The axes may be jog using the JOG keyboard on the operator panel. The type of jog is selected with the jog selector switch on the operator panel.



Continuous jog

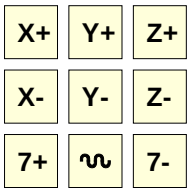
Incremental jog

Handwheels

The jog keyboard and the feedrate selector

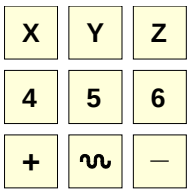
JOG keypad.

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



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.

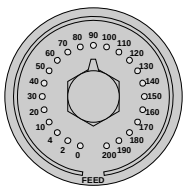
To jog an axis requires activating both the axis key and the moving direction. There are two options, depending on how the jog keyboard has been configured.

- 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.

Feedrate selector.



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.

Movement in 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). If while moving, a second axis is selected and either the '+' key or the '-' key is pressed, the CNC stops the first axis and

de-selects it. The new axis stays selected and moves according to the pressed '+' key or '-' key.

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.

Movement in 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.

1 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 inch.
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.

5.1.3 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 has 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.

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

5.1.4 Moving an axis to a particular position (coordinate)

1 Select the axis to be moved 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 can be moved with the [⬆][⬇] 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. This feedrate may be varied between 0% and 200% using the feedrate override switch on the operator panel. The percentage will be applied on to all the movements carried out in G00 and in G01.

- 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.

5.1.5 Coordinate preset

The coordinates must be preset one axis at a time. The preset may be canceled by homing the axes one by one or by means of function "G53".

1 Use the alphanumeric keyboard to select the axis whose position value (coordinate) is to be preset. 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 can be moved with the [⬆][⬇] keys.

2 Key in the desired preset value.

3 Press [ENTER] to preset the entered value or [ESC] to cancel the operation.

5.2 Setting the feedrate.

Setting a new feedrate in the channel.

The feedrate set in jog mode is only applied in that work mode and for the active channel. When setting a new feedrate in the MDI/MDA mode, it will become the new feedrate for the jog and automatic modes.

Follow these steps to set a new feedrate.

1. Press [F] at the alphanumeric keyboard. The CNC will highlight the relevant data indicating that it is selected.
2. Enter the new feedrate.
3. Press [START] to assume the entered value or [ESC] to cancel the operation.

5.3 Setting and activating the zero offsets and the fixture offsets.

In jog mode, it is possible to save the active offset in the zero offset table or in the fixture offset table (zero offset, coordinate presetting, etc.) and to activate a zero offset already defined in the tables.

This softkey shows the zero offsets and the fixture offsets of the system and their value in each axis of the channel. This list is a brief information of the zero offset tables and fixture offset tables and any change made in jog mode also affects those tables.

Loading a new zero offset or fixture offset into the table.

With an active offset, use the cursor to select an offset from the list and press [ENTER] to save the current offset in that zero offset. The position of all the axes of the channel are updated at the selected zero offset.

Applying a zero offset or fixture offset stored in the table.

Use the cursor to select a zero offset or fixture offset from the list and press the [START] key to activate. The new zero offset is applied to all the axes of the channel.

5.

MANUAL MODE.

Setting and activating the zero offsets and the fixture offsets.

FAGOR

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

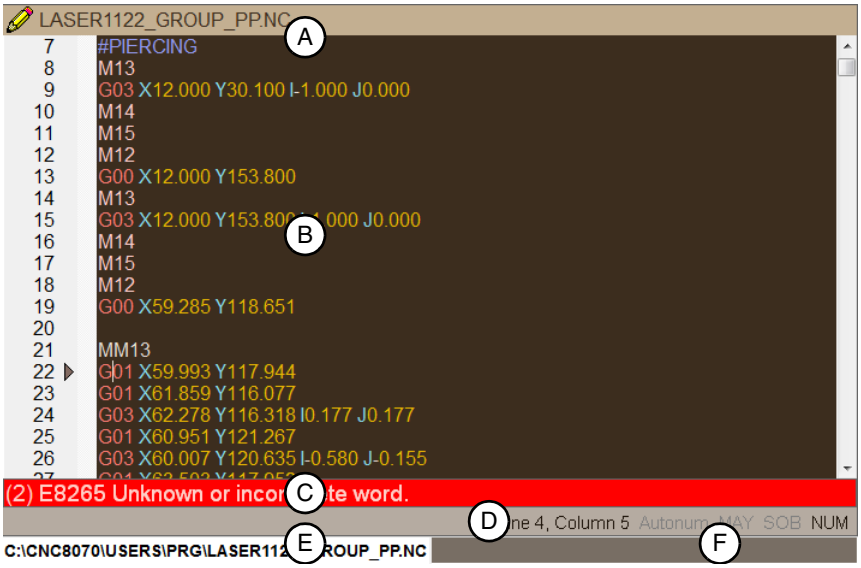
(REF: 1707)

EDISIMU MODE.

6

EDIT

The [EDIT] key accesses the EDITING page.



A Title bar

Name of the program selected for editing. As applicable, it also indicates whether it is a read-only program or not (if read-only 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 displayed in full, place the focus in this area and move the text using the arrow 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.

AUTONUM:	Automatic block numbering. When active, the CNC automatically numbers the new blocks being generated.
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.

E Program name and location.

F CNC messages.

Horizontal softkey menu.

The horizontal softkey menu shows the following options.

Softkey.	Description.
Open program.	Open a program at the editor.
Operations with blocks.	Copy, cut and paste text and blocks as well as copy a block or set of blocks as an independent program. Also find a line or a text in the program and replace a text with another one.
Undo and redo.	Undo the last modifications made.
File.	Recall, 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.

Vertical softkey menu.

The vertical softkey menu shows the following options.

Softkey.	Description.
	Show more options on the softkey menu.
	Analyze 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.

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.

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.

6.1 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.

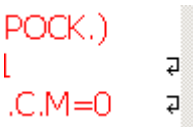
The editor provides the [ALT]+[–] hotkeys to expand and hide grouped blocks. If the CNC has a mouse, click on the symbol located to the right of the cycle, profile or group of blocks to expand them and hide them.

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.

6.1.1 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 [ENTER] key to enter the selected command in 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.

Contextual help is activated from the general customizing options. Contextual assistance is not available when using the 8055 CNC language.

6.1.2 Help for programming subroutines.

Subroutine help files.

Help files may be associated with each OEM subroutine (G180/G189 and G380/G399) 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 G function 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 of the subroutines is only available when the editor uses the CNC language; when the editor is enabled for the 8055 CNC, these helps are not available. The help window of the subroutines is available even when the contextual helps of the editor are disabled.

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.

The help is always active, even when the contextual helps of the editor are disabled.

For further information on how to define the help on subroutines, refer to the programming manual.

6.2 Select a program.

The "Open program" softkey is used to select a program in EDISIMU mode and may be a new program or an existing one. A different program may be edited in each channel. When selecting this option, the CNC shows a list of the available programs. See *"2.6 File selection window."* on page 49.

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 and the CNC will open an empty program.
3. Press [ENTER] to accept the selection and open the program or [ESC] to cancel it and close the program listing.

6.3 Operations with blocks.

6.3.1 Copy, cut and paste.

The "Operations with blocks" softkey 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.

Operations with blocks "Copy"

Copy the selected text onto the clipboard.

Operations with blocks "Cut"

Copy the selected text onto the clipboard and deletes it from the program.

Operations with blocks "Paste"

Paste the contents of the clipboard into the program.

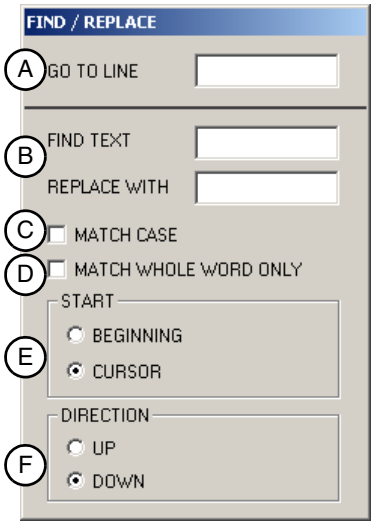
Operations with blocks "Copy to program"

Save 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.

6.3.2 Find a line or a text in the program.

The "Operations with blocks" softkey is used to find a line or a text in the program and replace a text with another one. 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].

6.4 Undo and redo.

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.

6.5 Operations with files.

The "File" softkey is used to recall, save or print the program as well as import dxf, pit or pim files.

File "Restore original"

This softkey 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. In programs larger than 2MB, the editor does not offer the option to recover the original program.

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. Once the program has been saved, the top of the editing window will show the name of 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.
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 return to the editor without saving the program.

File "Print"

This softkey may be used to print the program in the pre-determined printer.

File "Include program"

This softkey 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. The selected program is added to the one being edited after the block indicated by the cursor.

When selecting this option, the CNC shows a list of the programs that may be imported into the one being edited. 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.

File "Import"

This softkey is used to import DXF, PIM and PIT files into the program being edited.

- The DXF format is a 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. See *"6.5.1 Import DXF files"* on page 93.
- PIM and PIT files are part-programs used by the 8055 CNC. When importing this type of file, its programming language is adapted to the one used by the CNC.

When selecting this option, the CNC shows a list of the programs that may be imported into the one being edited. Select the desired file from the list and press [ENTER].

6.5.1 Import DXF files

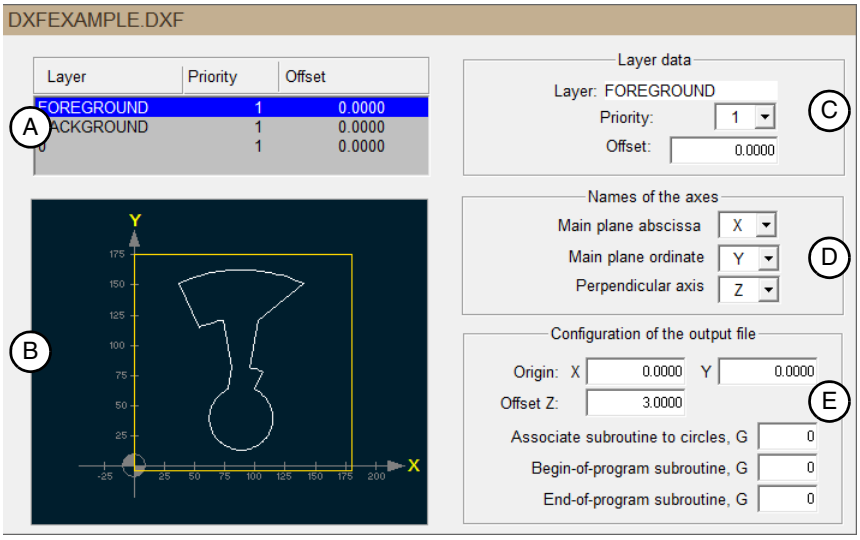
The DXF format is a standard for exchanging graphic files. Importing this type of files makes it possible to generate the part-program directly from a drawing. The DXF file may consist of points, lines and arcs. It can also consist of polylines, but they must be previously uncombined.

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 file 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.

Configuring the import of a DXF file.

When importing a DXF file, it shows the configuration window to define how the different layers of the drawing must be treated.



A List of layers that make up a drawing.

The DXF files may consist of layers and each one has a different part height. All the layers together make up the whole drawing. For each layer, it indicates its priority (order in which they are executed) and offset (height) on the perpendicular axis.

When importing the file, it will be possible to select which layers must be included into the part-program. By default, the CNC includes all the layers; to exclude any of them, select it from the list and press "Disable layer".

B File preview.

The graphic shows all the layers, even if they are disabled and the selected layer is shown in a different color.

C Data of the layer selected by the cursor. For each layer, it is necessary to 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).

D 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.

E Output file configuration.

- Origin XY. Zero offset on the axes of the plane; the indicated offset is added to all the blocks.
- Z offset. Safety distance for approaching to the profile.
- Associate a subroutine to circles. If a valid subroutine has been selected, the dxf converter identifies the circles and converts them into a block with the indicated subroutine number and the position of the center of the circle. If no valid subroutine has been selected, the dxf converter converts the circles to segments with G2/G3. OEM subroutines (G180-189 and G380-G399) and user subroutines are valid .
- Beginning-of-program subroutine.
- End-of-program subroutine.

How to draw and lay the drawing out.

How to draw the contour and other elements (items).

Element.	
Origin (zero point) of the drawing.	The CNC uses the drawing zero point as part zero.
Measuring units.	DXF files do not contain any references to measuring units (mm or inches); therefore, the CNC uses the ones defined in the part-program.
Contours.	The DXF can consist of points, lines, arcs and polylines, but they must be previously decomposed (uncombined). If the file contains polygons, they must also be decomposed; otherwise, they will be ignored.
Holes.	The dxf converter of the CNC lets associate a subroutine with circles. If a valid subroutine has been selected, the dxf converter identifies the circles and converts them into a block with the indicated subroutine number and the position of the center of the circle. If no valid subroutine has been selected, the dxf converter converts the circles to segments with G2/G3.

Laying the drawing out in layers.

A DXF file may be divided into layers offering the designer a way to lay the drawing out. Although each layer can contain any type of information (layers, dimensions, etc.), it must be borne in mind that the CNC uses the layers to define the order of the machining operations and the height where they will be carried out; therefore, the following rules must be followed.

- A layer must not contain profiles located at different heights. When importing the file, the CNC puts all the contents of the layer at the same machining height.
- Profiles placed at the same height may be on different layers.
- The elements that are not part of the machining operation (axes, dimensions, etc.) must be placed on layers that do not contain contours so the CNC can ignore these layers when importing the file.

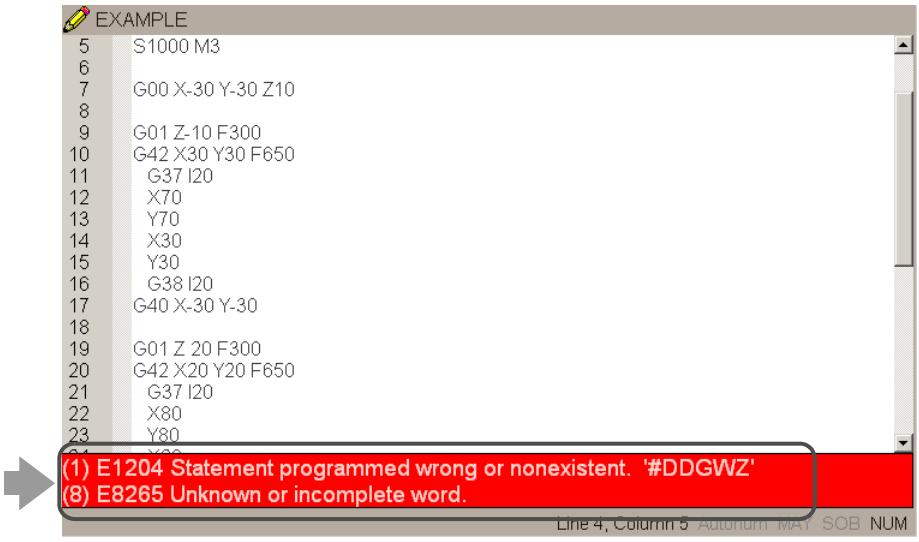
Exporting the file to DXF format.

The DXF file must be in ASCII format, files in Binary format are not permitted. When generating the DXF file from the drawing program, make sure that the file is saved in ASCII format.

When generating the DXF file from the drawing program, select a 4-decimal resolution is the CNC units are millimeters or 5 decimals if the measuring units at the CNC are inches. A greater resolution increases the size of the DXF file unnecessarily because the CNC ignores the excess of resolution.

6.6 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 [⬆][⬇] 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 displayed in full, place the focus in this area and move the text using the arrow keys [⬅][➡][⬆][⬇].

6.

EDISIMU MODE.

Syntax errors when editing

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

MDI/MDA MODE.

7

The [MDI] key accesses the MDI/MDA mode. The MDI mode overlays all other work modes in such a way that when quitting the mode by pressing [ESC], the system returns to the previous work mode.

The channel does not allow accessing its MDI/MDA mode if a program is being executed on the same channel, except when it is interrupted. The blocks executed while a program is interrupted alter the history of the program and these changes are maintained when resuming the execution.

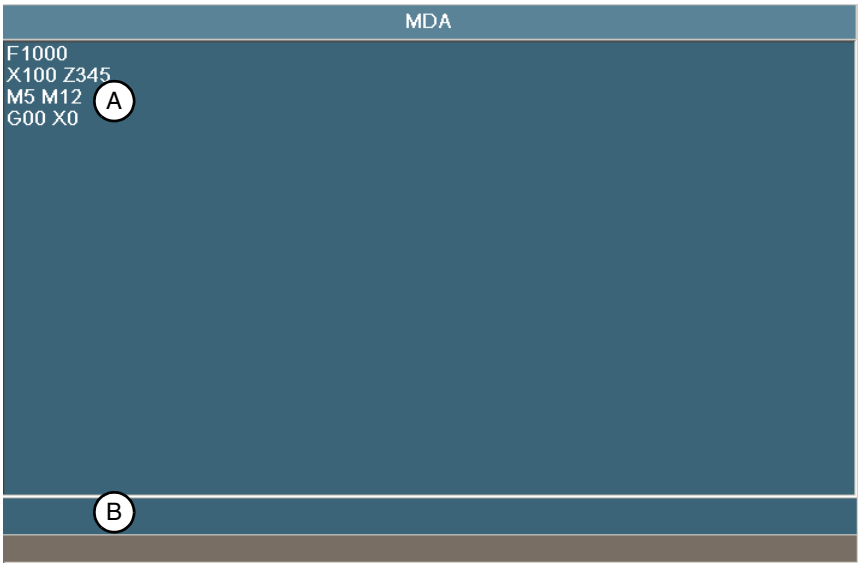
Standard window (MDI).

This window only shows the edit line of the MDI mode. In the MDI window, it is possible to edit and execute new blocks or recover blocks saved in the block history, which may be modified before executing them.

A Window for the MDI mode (Edit line) where the blocks to be executed are edited. Blocks are edited one by one.

Full screen (MDA).

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.



A History of blocks in MDI mode. Every time a new block is edited, it is added to this history.

B Edit line where the blocks to be executed are edited. Blocks are edited one by one.

Horizontal softkey menu (MDA).

The horizontal softkey menu shows the following options.

Softkey.	Description.
New block.	Initiate the editing of a new block.
Modify.	Restore from the history the block selected with the cursor and insert it in the edit line. This option is the same as pressing [ENTER].
Delete.	Delete the block selected with the cursor.
Cancel edit.	Cancel the editing of the block currently being edited. This option is only available when editing a block.
Delete all.	Delete all the blocks from the block history.
Save.	Save the block history as an independent program.
	Move up the block selected with the cursor.
	Move down the block selected with the cursor.

Vertical softkey menu (MDI).

The vertical softkey menu shows the following options.

Softkey.	Description.
	Place the MDI window at the top.
	Place the MDI window at the bottom.
	Shrink the MDI window.
	Expand the MDI window.

7.1 Edit and execute individual blocks.

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.

How to edit and modify blocks.

Both in MDI and in MDA modes, it is possible to edit new blocks or restore blocks that were saved in the history. The blocks restored from the history may be modified as if they were new blocks.

Edit new blocks.

- In MDI mode, the edit line is always visible.
- In MDA mode, one must select the "new block" option from the softkey menu.

Modify a block from the block history.

- In MDI mode, use the [⬆][⬇] keys to display the history and scroll through it. The [ENTER] key restores from the history the block selected with the cursor and insert it in the edit line.
- In MDA mode, use the [⬆][⬇] keys, select a block from the history and use the "modify" option from the softkey menu (or the [ENTER] key) to copy it into the edit line.

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.

Block execution. Changing the feedrate.

When setting a new feedrate in the MDI/MDA mode, it will become the new feedrate for the jog and automatic modes.

7.2 Block history.

Save blocks in block history.

The MDI(MDA mode saves into history all the blocks executed correctly. 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.

Save the blocks as a program.

In MDA mode, to save the block history as an independent program, press the "Save" softkey. The CNC will display a list with the programs stored on it.

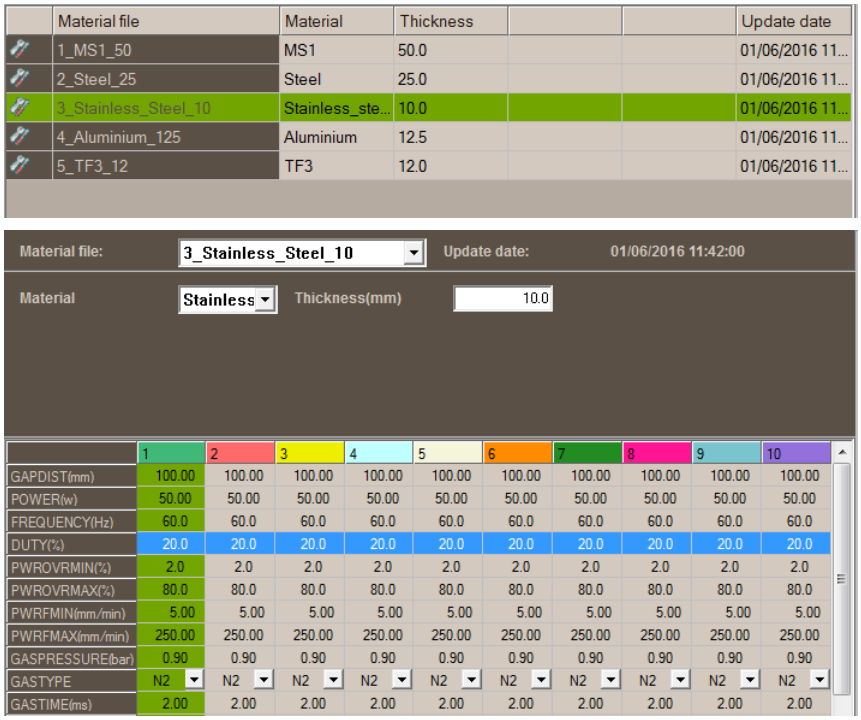
1. Write the program name in the bottom window. To replace an existing program, select it from the list.
2. Press [ENTER] to save the program or [ESC] to return to the MDI mode without saving the program.

TECHNOLOGY TABLES.

8

TOOLS

The [TOOLS] key accesses the technology tables. These tables consist of a series of material files, each of which matches certain material characteristics (type of material, thickness, etc.) with a series of technological parameters for piercing and cutting (power, speed, etc.).



	Material file	Material	Thickness			Update date
	1_MS1_50	MS1	50.0			01/06/2016 11...
	2_Steel_25	Steel	25.0			01/06/2016 11...
	3_Stainless_Steel_10	Stainless_ste...	10.0			01/06/2016 11...
	4_Aluminium_125	Aluminium	12.5			01/06/2016 11...
	5_TF3_12	TF3	12.0			01/06/2016 11...

Material file: 3_Stainless_Steel_10 Update date: 01/06/2016 11:42:00

Material: Stainless Thickness(mm): 10.0

	1	2	3	4	5	6	7	8	9	10
GAPDIST(mm)	100.00	100.00	100.00	100.00	100.00	100.00	100.00	100.00	100.00	100.00
POWER(w)	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00
FREQUENCY(Hz)	60.0	60.0	60.0	60.0	60.0	60.0	60.0	60.0	60.0	60.0
DUTY(%)	20.0	20.0	20.0	20.0	20.0	20.0	20.0	20.0	20.0	20.0
PWROVRMIN(%)	2.0	2.0	2.0	2.0	2.0	2.0	2.0	2.0	2.0	2.0
PWROVRMAX(%)	80.0	80.0	80.0	80.0	80.0	80.0	80.0	80.0	80.0	80.0
PWRFMIN(mm/min)	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00
PWRFMAX(mm/min)	250.00	250.00	250.00	250.00	250.00	250.00	250.00	250.00	250.00	250.00
GASPRESSURE(bar)	0.90	0.90	0.90	0.90	0.90	0.90	0.90	0.90	0.90	0.90
GASTYPE	N2	N2	N2	N2	N2	N2	N2	N2	N2	N2
GASTIME(ms)	2.00	2.00	2.00	2.00	2.00	2.00	2.00	2.00	2.00	2.00

The password and user profiles (normal or high level).

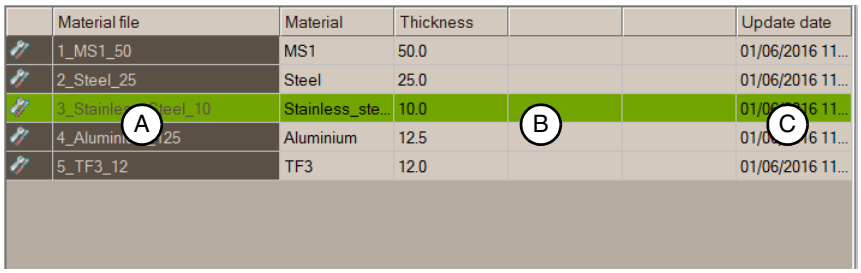
The CNC allows the material files to be password protected. A single password protects all the material files; if the user enters the password, all material files are unlocked. The password does not prevent material files from being displayed or accessed; it merely restricts the operations that users can carry out. See *"8.1.6 Protecting technology tables with a password."* on page 105.

Depending on whether there is a password, the CNC distinguishes between two different user profiles (normal and high level user), in addition to the OEM profile.

Profile.	Characteristics.
OEM.	CNC in setup mode; not password-dependent. The CNC must be started in user mode to have access to both the "Normal user" and "High level user" profiles.
Normal user.	CNC in user mode; password-protected technological tables. The user must enter the password to activate the "High level user" profile.
High-level user.	CNC in user mode; non-password protected technological tables. If the user enters an incorrect password, the CNC activates the "Normal user" password.

8.1 Material files.

The CNC displays the material files defined by the OEM and the user. The active material file appears in green. Use the mouse to double click on a material file or position the cursor over it and click on [ENTER] to access its common piercing and cutting parameters.



A Material files (up to 200). The icon indicates that it is a material file defined by the OEM.

B Material characteristics (type of material, thickness, etc.). The CNC displays the first four common parameters.

C Date updated. The date is updated by the CNC; it is not a material parameter and cannot be modified.

Horizontal softkey menu.

The horizontal softkey menu shows the following options.

Softkey.	Description.
	Defining or changing the password for the material files. The password does not prevent material files from being displayed or accessed; it merely restricts the operations that users can carry out. See *"8.1.6 Protecting technology tables with a password."* on page 105.

Vertical softkey menu.

The vertical softkey menu shows the following options.

Softkey.	Description.
	Adding a new material file. See *"8.1.3 Creating, renaming and duplicating a material file."* on page 104.
	Renaming and duplicating a material file. See *"8.1.3 Creating, renaming and duplicating a material file."* on page 104.
	Importing and exporting material files. See *"8.1.5 Importing and exporting material files."* on page 104.

8.1.1 Where the CNC saves the material files (material_file.xml).

Depending on who creates or modifies the material files, they are saved in different folders. In both cases, the material file is saved in xml format and the name assigned to the material file will be the name with which the data are saved (material_file.xml).

- ..\MTB\Data\TechTables\Material Files.

  Material files created or modified by the OEM (CNC in setup mode). The user cannot modify these files.

- ..\MTB\Data\TechTables\TTDsc.xml.

  Descriptor of the material files created or modified by the OEM.

- ..\USERS\Session\Data\TechTables\Material Files.

  Material files created or modified by the user.

- ..\USERS\Session\Data\TechTables\TTDsc.xml.

  Descriptor of the material files created or modified by the user.

The CNC displays all the material files for the OEM and the user. When the CNC uploads the material files, it looks first in the OEM folder and then in the user folders; if there are duplicate files, the CNC uploads the OEM files.

The descriptor and the material files.

The descriptor (TTDsc.xml) is a file containing the information related to the technological table parameters, such as its name, alias, units, default value, etc. When the OEM or user edits a parameter (adding, modifying or deleting it), they modify this file. The information in this file is common to all the material files.

The material files (material_file.xml) are the different files that appear in the technological tables. These files only contain the parameter values.

8.1.2 Selecting a material file.

Use the mouse to double click on a material file or position the cursor over it and click on [ENTER]; the CNC will display its common piercing and cutting parameters. This action does not activate the material file; to activate a material file, use the appropriate softkey in the piercing or cutting parameters. See *"8.2.1 Selecting and activating a material file."* on page 109.

8.1.3 Creating, renaming and duplicating a material file.

Both the OEM and the high level user can create, rename and duplicate material files. These operations are available in the vertical softkey menu.

The name assigned to a material file will be the name with which the data are saved (material_file.xml). To make it easier to understand, Fagor recommends that the name be a combination of the material, thickness, etc. A material file with an empty name field cannot be created, duplicated or renamed with an already existing name.

Adding a new material file. After the name of the material file has been specified, the CNC adds a new material file to the list and assigns the default values to all parameters.

Renaming a material file. When a material file is renamed, the CNC does not change any of its values.

Duplicating a material file. After defining the new name, the CNC adds a new material file to the list and assigns the same values to all parameters as in the original material file.

After duplicating a material file, the CNC creates the new file in the same folder as the original file. If the file is an OEM file, the CNC creates the new file in the MTB directory. If the file is a user file, the CNC creates the new file in the USER directory.

8.1.4 Deleting a material file.

The OEM can delete any material file. High level users can only delete material files that are not from the OEM. Normal users cannot delete material files. To delete a material file, press the [DEL] key. The CNC will request confirmation.

8.1.5 Importing and exporting material files.

Both the OEM and the high level user can import and export material files. These operations are available in the vertical softkey menu.

The CNC allows material files to be imported from any folder other than the default folders (including from a USB drive). The CNC selects the folder where the files are to be imported according to the active user profile (high level user or OEM). When material files are imported, if a material file with the same name already exists, the CNC will request confirmation from the user.

The CNC allows the material files to be exported to any folder on the CNC. The CNC will export only the material files related to the active user profile (high level user or OEM).

The CNC only imports and exports the values, but not the parameter definitions. When a material file is imported, if the values of any parameter do not match the definitions, the CNC will use the default values for said parameters.

8.1.6 Protecting technology tables with a password.

The CNC allows the material files to be password protected. A single password protects all the material files; if the user enters the password, all material files are unlocked. The password does not prevent material files from being displayed or accessed; it merely restricts the operations that users can carry out.

The password and user profiles (normal or high level).

Depending on whether there is a password, the CNC distinguishes between two different user profiles (normal and high level user), in addition to the OEM profile.

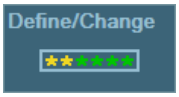
Profile.	Characteristics.
OEM.	• If the CNC powers up in setup mode, the OEM profile is activated. • The password has no effect and it is not possible to access the user profiles. The CNC must be shut down and restarted in user mode to have access to the "Normal user" and "High level user" profiles.
High-level user.	• If the CNC starts up in user mode and no password is set, only the "High level user" profile will exist. The user must set a password to have access to the "Normal user profile". • If the "Normal user" profile is active and the user enters the password, the CNC activates the "High level user" profile.
Normal user.	• If the CNC starts up in user mode and a password has been set, it activates the "Normal user" profile. • If the "High level user" profile is active and the user enters an incorrect or blank password, the CNC activates the "Normal user profile".

Operations permitted, according to user profile.

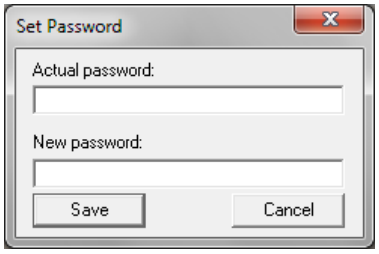
The OEM can modify all the material files (both OEM and user). The user (normal or high level) can never modify the OEM material files.

Operation.	Normal user.	High-level user.	OEM.
Display a new material file.	Yes.	Yes.	Yes.
Accessing common piercing and cutting parameters.	Yes.	Yes.	Yes.
Activating a material file.	Yes.	Yes.	Yes.
Adding material files.	No.	Yes.	Yes.
Adding OEM material files.	No.	No.	Yes.
Renaming a material file.	No.	Yes.	Yes.
Duplicating material files.	No.	Yes.	Yes.
Importing and exporting material files.	No.	Yes.	Yes.
Adding, modifying and deleting parameters.	No.	No.	Yes.
Modifying ENUM parameters.	No.	Yes.	Yes.
Modifying parameter values.	No.	Yes.	Yes.
Validate parameters.	No.	Yes.	Yes.

Defining or changing a password.



The password is defined in the horizontal softkey menu. It is not possible to set the password from the cutting or piercing parameters. After the softkey is pressed, the CNC displays the following dialog box. The password can be 50 characters long and can include capital letters, lowercase letters and numbers (special characters are not allowed).



- Current password. This field is only necessary to delete or change the password. If there is no password, this field will be disabled.
- New password. Set the new password in this field. Leave the field blank to delete the password.

Setting a password for the first time.

Current password.	Since there is no previous password, leave this field blank.
New password.	Setting the password.

Changing the password.

Current password.	Setting the current password.
New password.	Setting the new password.

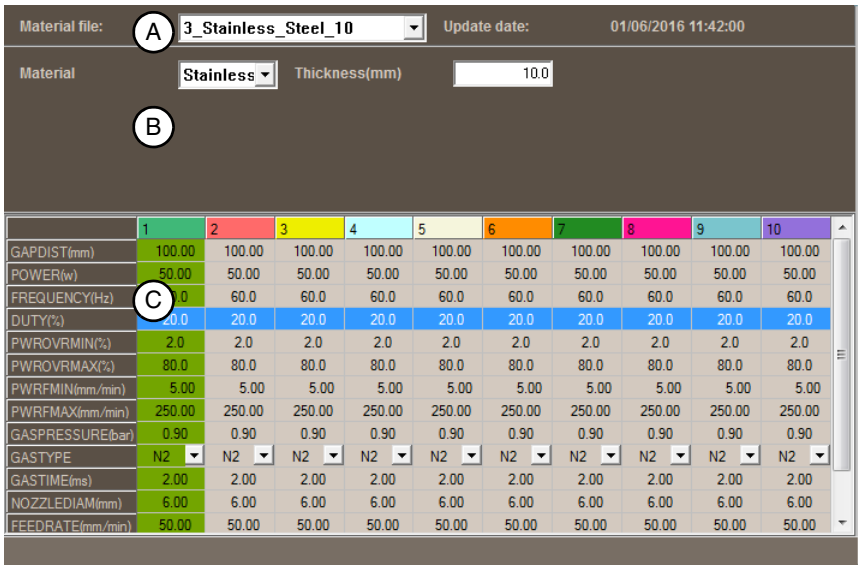
Deleting the password.

Current password.	Setting the current password.
New password.	Leave this field blank.

8.2 Cutting and piercing parameters.

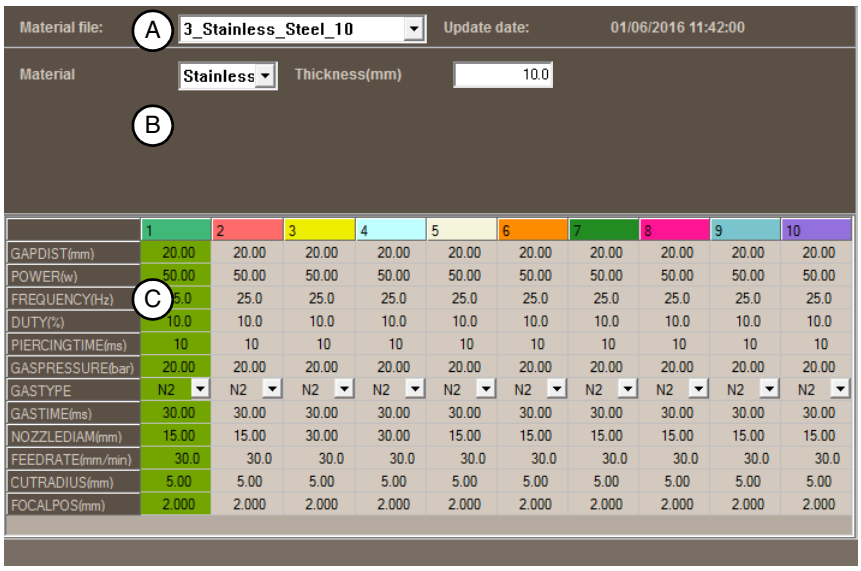
Each material file has certain common parameters (such as the material and thickness) that affect both cutting and piercing operations, and specific parameters associated with these operations. In addition to the material, from the program, a group of cutting and piercing parameters can be activated; it is not possible to select these parameters directly from the technological tables.

Cutting parameters.



	1	2	3	4	5	6	7	8	9	10
GAPDIST(mm)	100.00	100.00	100.00	100.00	100.00	100.00	100.00	100.00	100.00	100.00
POWER(w)	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00
FREQUENCY(Hz)	.0	60.0	60.0	60.0	60.0	60.0	60.0	60.0	60.0	60.0
DUTY(%)	20.0	20.0	20.0	20.0	20.0	20.0	20.0	20.0	20.0	20.0
PWROVRMIN(%)	2.0	2.0	2.0	2.0	2.0	2.0	2.0	2.0	2.0	2.0
PWROVRMAX(%)	80.0	80.0	80.0	80.0	80.0	80.0	80.0	80.0	80.0	80.0
PWRFMIN(mm/min)	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00
PWRFMAX(mm/min)	250.00	250.00	250.00	250.00	250.00	250.00	250.00	250.00	250.00	250.00
GASPRESSURE(bar)	0.90	0.90	0.90	0.90	0.90	0.90	0.90	0.90	0.90	0.90
GASTYPE	N2	N2	N2	N2	N2	N2	N2	N2	N2	N2
GASTIME(ms)	2.00	2.00	2.00	2.00	2.00	2.00	2.00	2.00	2.00	2.00
NOZZLEDIAM(mm)	6.00	6.00	6.00	6.00	6.00	6.00	6.00	6.00	6.00	6.00
FEEDRATE(mm/min)	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00

Piercing parameters.



	1	2	3	4	5	6	7	8	9	10
GAPDIST(mm)	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00
POWER(w)	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00
FREQUENCY(Hz)	.0	25.0	25.0	25.0	25.0	25.0	25.0	25.0	25.0	25.0
DUTY(%)	10.0	10.0	10.0	10.0	10.0	10.0	10.0	10.0	10.0	10.0
PIERCINGTIME(ms)	10	10	10	10	10	10	10	10	10	10
GASPRESSURE(bar)	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00
GASTYPE	N2	N2	N2	N2	N2	N2	N2	N2	N2	N2
GASTIME(ms)	30.00	30.00	30.00	30.00	30.00	30.00	30.00	30.00	30.00	30.00
NOZZLEDIAM(mm)	15.00	15.00	30.00	30.00	15.00	15.00	15.00	15.00	15.00	15.00
FEEDRATE(mm/min)	30.0	30.0	30.0	30.0	30.0	30.0	30.0	30.0	30.0	30.0
CUTRADIUS(mm)	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00
FOCALPOS(mm)	2.000	2.000	2.000	2.000	2.000	2.000	2.000	2.000	2.000	2.000

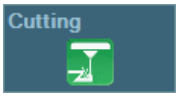
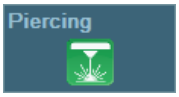
A Material file and date of the last time the table was modified. The date is updated by the CNC. The CNC allows the material file to be changed, even if there is a program running.

B Both the piercing and cutting tables have parameters in common, which represent the characteristics of the material (type of material, thickness, etc.). These parameters can be modified by both the OEM and the high level user.

C Cutting or piercing parameters. These parameters are common to all the materials. Each row represents a cutting parameter, while each column corresponds to a group of cutting conditions. The number of columns cannot be changed and each of them is associated with a color for the graphic representation. The OEM can add, delete and modify the parameters. High level users can modify the parameter values.

Horizontal softkey menu.

The horizontal softkey menu shows the following options.

Softkey.	Description.
	Accesses the list of material files.
	Accesses the common and cutting parameters.
	Accesses the common and piercing parameters.
	Adds a parameter (only OEM).
	Modifies the parameter properties (all for OEM; only ENUM parameters for high level users).
	Deletes a parameter (only OEM).
	Enter password.

Vertical softkey menu.

The vertical softkey menu shows the following options.

Softkey.	Description.
	Sets the selected material file as active material. The CNC does not allow the active material file to be changed if a program is running. The CNC allows a material file to be activated from the interface or the program.
	Saves the material file (OEM or high level user).
	Restores the material file; all parameters (common, piercing and cutting) recover their saved values (OEM or high level user).
	Initializes the material file; sets all parameters (common, piercing and cutting) to their default values (OEM or high level user).
	Validates the material file after adding new parameters (OEM or high level user).

8.2.1 Selecting and activating a material file.

The drop-down menu at the top allows you to select any material file. If the file is not active, the CNC shows the "Activate" softkey to activate it. It is not possible to activate a new file if a program is running.

When an active material file is selected, the parameters show the active values, even if they do not match those saved in the xml file (for example, values validated, but not saved).

The active cutting and piercing parameters column is displayed in green. Only one type of cutting and piercing can be activated from the program; it is not possible to do this from the technological tables.

8.2.2 Adding or deleting parameters (only OEM).

Fagor supplies the material files with certain pre-set parameters. The OEM may add new parameters or delete those already there, including the parameters defined by Fagor. Adding, modifying and deleting parameters affects all the material files; the parameters are unique for all files. The material file can consist of a maximum of 10 common parameters, 40 piercing parameters (Fagor parameters, plus those added by the OEM) and 20 cutting parameters (Fagor parameters plus those added by the OEM).

The Fagor piercing and cutting subroutines are based on the parameters provided by Fagor; if these parameters are modified, the subroutines must also be modified.

Use the horizontal softkey menu to add or delete parameters. After adding, deleting or modifying the parameters, the CNC must be restarted.

Add parameter.

Press the "Add a parameter" softkey and the CNC will display the window to define a parameter. See *"8.2.4 Setting the properties of the material file parameters."* on page 110.

Delete parameter.

Select a parameter and press the "Delete parameter" softkey. The CNC will request confirmation.

8.2.3 Modifying the properties of the parameters.

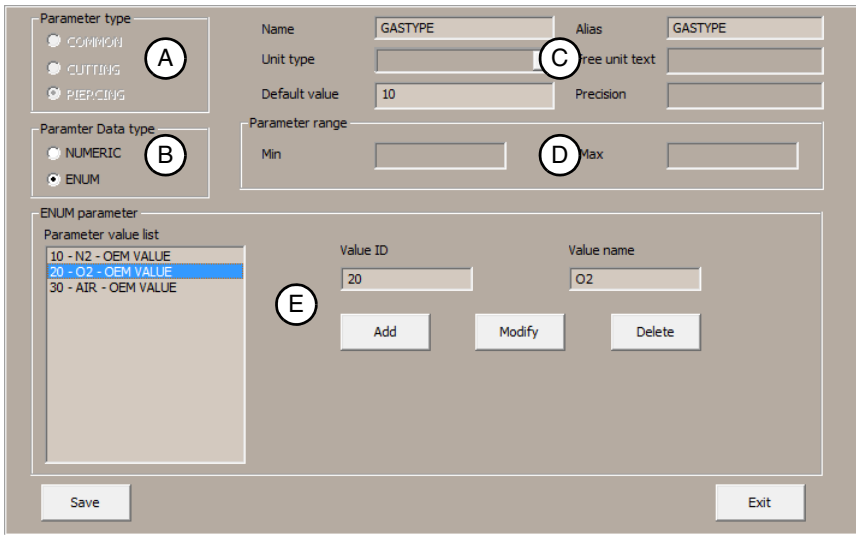
Select a parameter and press the "Modify parameter" softkey; the CNC displays the window to modify the parameter. See *"8.2.4 Setting the properties of the material file parameters."* on page 110.

The OEM can modify the properties of all the parameters. The high level user can only add or delete ENUM parameter options (drop-down list), but cannot ever modify the options set by the OEM. Adding, modifying and deleting parameters affects all the material files; the parameters are unique for all files. After adding, deleting or modifying the parameters, the CNC must be restarted.

8.2.4 Setting the properties of the material file parameters.

The OEM can add, modify or erase all parameters. The high level user can only add or delete ENUM parameter options (drop-down list), but cannot ever modify the options set by the OEM.

For the options add or modify a parameter, the CNC displays the same dialog box. This box will only permit the appropriate options to be modified, depending on the type of parameter selected (numbered or enumerated).



A Parameter type.

Common.	Common parameter, which indicates the characteristics of the material (type of material, thickness, etc).
Cutting.	Cutting parameter.
Piercing.	Piercing parameter.

B Type of parameter data, numbered or enumerated.

C Parameter description.

Name of the parameter.	Name of the variable associated with the parameter and the name used in the xml file. If the parameter alias is not defined, it will also be the name that appears on the material file.
Parameter alias.	Name of the parameter.
Type of units.	Units to display next to the name or alias of the parameter. Only for numerical parameters. See *"Type of units and unit conversion."* on page 111.
Free text for the units.	Parameter units, only when the type of units is "FREE". Only for numerical parameters.
Default value.	Value the parameter assumes when it is created or defined with an incorrect value or without any value.
Accuracy.	Precision of the parameter (number of decimals). Only for numerical parameters. A numeric parameter may have a maximum of 10 whole numbers and 4 decimals.

D Value range. Minimum and maximum values of the parameter. Only for numerical parameters.

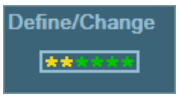
E Enumerated parameter.

List of values.	List of possible parameter values. The ID and name must be defined for each new parameter value.
Value ID.	ID of the new parameter option. ID from 1 to 100 for the values from the OEM and from 101 to 200 for the user values.
Name of the value.	Value of the new parameter option.

Type of units and unit conversion.

Units.	Meaning.
Free.	The OEM defines the units as a text, which will be what appears next to the name or alias of the parameter. This type of variables are not affected by the change of units (millimeters/inches). The PLC reading will display the value multiplied by 10000.
None.	No units. The PLC reading will display the value multiplied by 10000.
Distance.	mm or inches. The CNC will show the correct units according to the general machine parameter INCHES. The PLC reading will display the value multiplied by 10000.
Speed.	In millimeters/minute or inches/minute. The CNC will show the correct units according to the general machine parameter INCHES. The PLC reading will display the value multiplied by 10000.
Acceleration.	Millimeters/minute² or inches/minute². The CNC will show the correct units according to the general machine parameter INCHES. The PLC reading will display the value multiplied by 10000.
Pressure.	BAR o PSI. The CNC will show the correct units according to the general machine parameter PRESSURE. The PLC reading will display the value multiplied by 100.
Percentage.	% The PLC reading will display the value multiplied by 100.
Time.	Seconds. The PLC reading will display the value multiplied by 1000.
Degrees	0 - 360 º. The PLC reading will display the value multiplied by 100.
Frequency.	Herz. The PLC reading will display the value multiplied by 1000.
Voltage.	Volts. The PLC reading will display the value multiplied by 1000.
Power.	Watts. The PLC reading will display the value multiplied by 1000.
Temperature.	º C. The PLC reading will display the value multiplied by 100.

8.2.5 Enter password.



If the technological tables are password protected, this softkey allows it to be entered to activate the "High level user" profile. A single password protects all the material files; if the user enters the password, all material files are unlocked. To reactivate the "Normal user" profile, leave the password blank. See *"8.1.6 Protecting technology tables with a password."* on page 105.

8.2.6 Operations with material files.

Editing the values of the parameters.

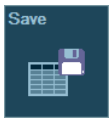
To edit the parameter data, proceed as follows:

1. Use the cursor to select the parameter whose value you wish to set.
2. Editing the new value or selecting it from the list.
3. Press [ENTER] to accept the new value or [ESC] to ignore the new value and recover the previous one.

Restores the default value of a parameter.

Select one or more parameters in a row and press the [DEL] key; the parameters recover their default value.

Saving the material file.



Saves the material file in the xml file.

Restoring the material file.

This softkey restores the original values of the parameters, which are those saved in the xml file.

Resetting the material file.

This softkey resets the parameters to their default values.

Validating the active material file.

This softkey accepts the new parameter values and updates the corresponding variables. This softkey does not save the values in the xml file. This option is only available for the active parameters and during execution.

MACHINE STATUS.

9

This mode displays the status of the I/Os, the axes and the laser. Use the task window ([CTRL]+[A]) to access this mode. Use the horizontal softkey menu to select the different pages in this mode. The [ESC] returns to the last mode used from among CUT-VIEW, PROGRAM and MANUAL.

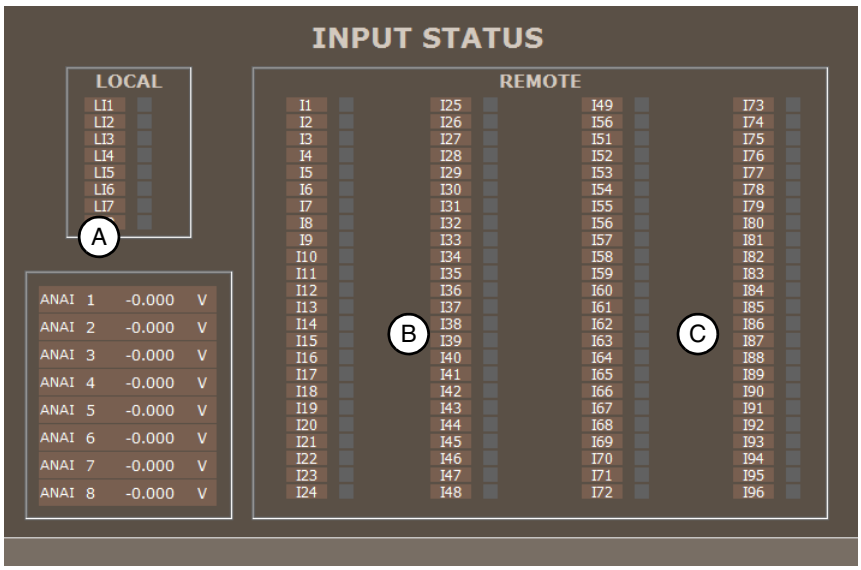
Horizontal softkey menu.

All of the pages in this work mode are available in the horizontal softkey menu.

- Digital and analog inputs.
- Analog and digital outputs.
- Status of the axes.
- Laser status.

9.1 Monitoring of digital and analog inputs.

This page only shows the status of the inputs; it does not allow their value to be modified.



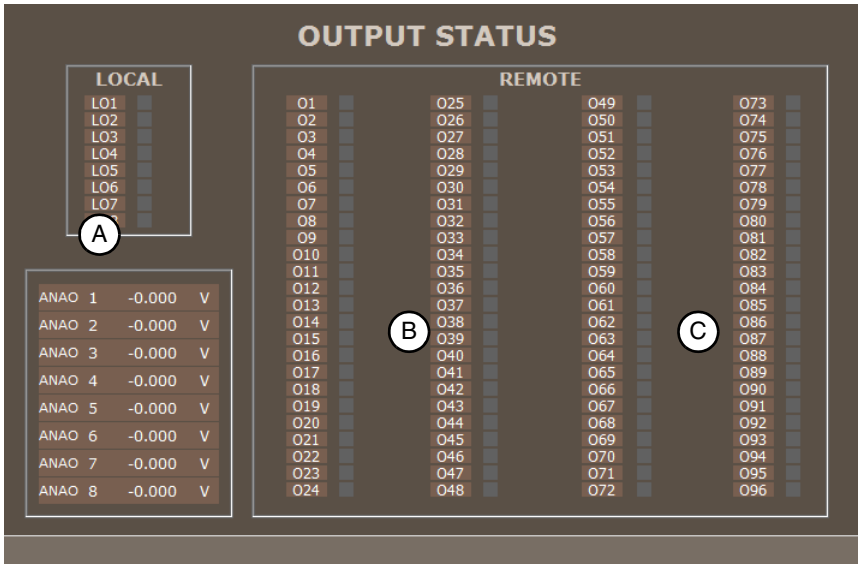
A Available local digital inputs and their status (green LED if active).

B Available remote digital inputs and their status (green LED if active).

C Available remote analog inputs and their value in volts.

9.2 Monitoring of digital and analog outputs.

This page only shows the status of the inputs; it does not allow their value to be modified.

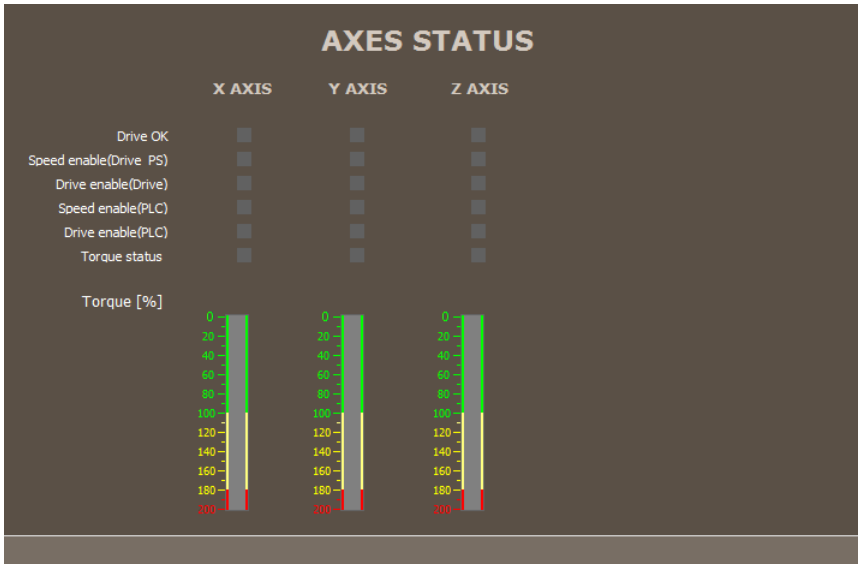


A Available local digital outputs and their status (green LED if active).

B Available remote digital outputs and their status (green LED if active).

C Available remote analog outputs and their value in volts.

9.3 Status of the axes.



By default, the torque bars show the value of the drive BV14 and TV2 variables defined in the OEM machine parameter DRIVEVAR.

DRIVEVAR : DATA

MNEMONIC	AXIS	ID	TYPE	MODE
BV14	*	32972	Asynch	Read
TV2	*	84	Asynch	Read

These variables are only valid for Sercos axes; if any of the axes is analog, its variables must be deleted from this table. The CNC will display error 85 while there are analog axes defined in the OEM machine parameter table.

9.4 Laser status.

After the "laser status" is pressed, the CNC executes the application specified in the Machinestatus.ini file, located in the folder ..\MTB\MMC\Config. In this file, in the [LASERST] section, in the "LaserApp" line, the absolute path of the file to be executed must be specified. If no application is specified, when the softkey is pressed, the CNC will display the error "Laser status application not found".

```
..\MTB\MMC\Config\Machinestatus.ini

[LASERST]
LaserApp = C:\CNC8070\...
```

9.

MACHINE STATUS.
Laser status.

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

USER TABLES.

10

The [TABLES] key accesses the user tables.

The user tables cover the following tables. The various tables may be selected using the horizontal softkeys.

- Zero offset table. There is a table for each channel.
- Clamp tables (fixtures). There is a table for each channel.
- 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 nesting level (7 levels).
- Table of common parameters. The table is common to all the channels.

In order to activate a fixture zero or part zero offset, those values must be previously stored in the relevant CNC table.

Vertical softkey menu.

Softkey.	Description.
	Show more options on the softkey menu.
	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 softkey highlights the units currently selected (millimeters or inches). The selected units are only valid for displaying data. For programming, the CNC assumes the units defined with the active function G70 or G71, or, when not programmed, the units set by the machine manufacturer (INCHES parameter). The CNC will display this softkey or not depending on how machine parameter MMINCHSOFTKEY has been set.
	Initialize the table. Reset all the table data to "0". The CNC will request confirmation of the command.
	Search a text or a value in the table. When selecting this option, the CNC shows a dialog box requesting the text to be found.
	Save the values of the table into a file.
	Restore the values of the table previously saved into a file.
	Print the table in the pre-determined printer or save it as a file (prn format) at the CNC.

10.1 Zero offset tables

This table contains the absolute zero offsets (G54 to G59 and G159), the incremental (G158) and the PLC offset of the axes and spindles that may be activated as the C axis. The table highlights the active offset (both absolute and incremental) in color.

The zero offset may have two different looks, with or without fine setting of absolute zero offset. The type of table depends on the configuration set by the OEM (parameter FINEORG).

The zero offsets associated with the possible C axes are always visible, even when the C axis is not active.

PLC offset (PLC offset).

The PLC offset may not be set directly in the table, its values are set from the PLC or via part-program using variables. This offset is used, for example, to correct deviations originated by machine dilatations. The CNC always adds the PLC offset to the selected zero offset.

Incremental zero offsets.

This zero offset may not be set directly in the table, its values are set via part-program or via variables. The CNC adds the incremental zero offset to the absolute zero offset active at the time.

Absolute zero offsets

Absolute zero offsets besides being set directly in the table may also be set from the PLC or via part-program using 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) or to the machine zero (if the clamp zero is not active). To apply an absolute zero offset, it must be activated via program using the relevant function.

Absolute zero offset table (with fine setting of the zero offset).

Each zero offset has a single value. When activating a zero offset (function G159), the CNC assumes this value as the new zero offset.

Channel 1 : Zero offsets

Origin	X (mm)	A	Z (mm)
PLCOF	00000.0000	00000.0000	00000.0000
G158	00054.5000	00010.0000	00000.0000
G54 (G159=1)	00000.0000	00000.0000	00000.0000
G55 (G159=2)	00000.0000	00000.0000	00000.0000
G56 (G159=3)	00000.0000	00000.0000	00000.0000
G57 (G159=4)	00000.0000	00000.0000	00000.0000
G58 (G159=5)	00000.0000	00000.0000	00000.0000
G59 (G159=6)	00000.0000	00000.0000	00000.0000
G159=7	00000.0000	00000.0000	00000.0000
G159=8	00000.0000	00000.0000	00000.0000
G159=9	00000.0000	00000.0000	00000.0000

B C

A Machine axes and display units (millimeters, inches or degrees).

B Zero offsets.

C Zero offset value in each axes of the channel.

Absolute zero offset table (with fine setting of the zero offset).

Each zero offset has a coarse (or absolute) value and a fine (or incremental) value. Setting the coarse value of an offset deletes its fine value. When activating an offset (function G159), the CNC assumes as new zero offset the sum of both parts.

Channel 1 : Zero offsets

Origin	X (mm)	A	Z (mm)
PLCOF	00000.0000	00000.0000	00000.0000
G158	00054.5000	00010.0000	00000.0000
G54 (G159=1)	00050.0000	00000.0000	00000.0000
△	00003.0000	00000.0000	00000.0000
G55 (G159=2)	00000.0000	00000.0000	00000.0000
△	00000.0000	00000.0000	00000.0000
B (G159=3)	00010.0000	D 0000	00000.0000
C △	00000.0000	00000.0000	00000.0000
G57 (G159=4)	00000.0000	00000.0000	00000.0000
△	00000.0000	00000.0000	00000.0000
G58 (G159=5)	00000.0000	00000.0000	00000.0000

A Machine axes and display units (millimeters, inches or degrees).

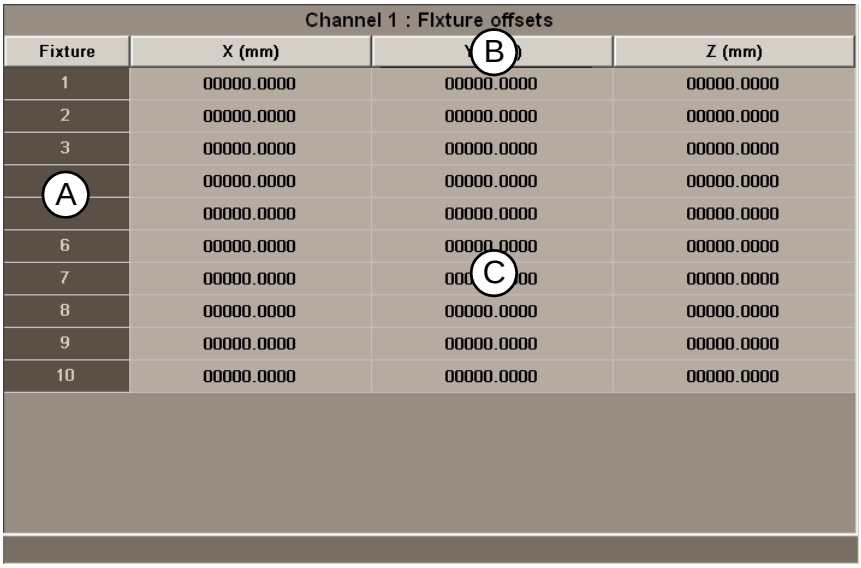
B Zero offsets (coarse part).

C Zero offsets (fine part).

D Zero offset value in each axes of the channel.

10.2 Fixture table

This table stores the clamp offsets for each axis. There are up to 10 clamp offsets. The table highlights in color the active zero offset.



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

The fixture offset besides being set directly in the table may also be set from the PLC or via part-program using 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.

10.3 Arithmetic parameter tables

There are the following arithmetic parameter tables:

- Common parameters. The table is common to all the channels.
- 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).

Local parameters.

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.

Common parameters.

Channel 1 : Common P parameters

Parameter (A)	Value (B)	Comment (C)
P10000	101.025800	Comments
P10001		Comments
P10002	100	Comments
P10003	23	Comments
P10004	1	Comments
P10005	0.168541	Comments
P10006	1.002580	Comments
P10007	-5813	Comments
P10008	-12.025800	Comments
P10009	0	Comments
P10010	0	Comments
P10011	0	Comments
P10012	0	Comments
P10013	0	Comments

A Parameter list.

B Parameter value.

C Parameter describing comment (only in the common-parameters table).

The comment field offers the possibility to associate a short description with the parameter. 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 "Ñ") 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 part-program. 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).

10.4 Operations with tables

10.4.1 Data editing

Select the desired table using the horizontal 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.

How to use the calculator to set the data (table of zero offsets and clamp offsets).

Being the focus on any field of the zero offset table or clamp offset table, press [INS] or [CTRL][K] to access the calculator. The calculator takes the current value of the field and may be used to perform any operation. Press [INS] to load calculated value into the field and close the calculator.

Pressing [ENTER] instead of [INS] calculates the value without inserting it into the field and it allows going on with other operations.

10.4.2 Save and recall tables

Save a table.

This softkey is 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" softkey 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	Arithmetic parameter tables.

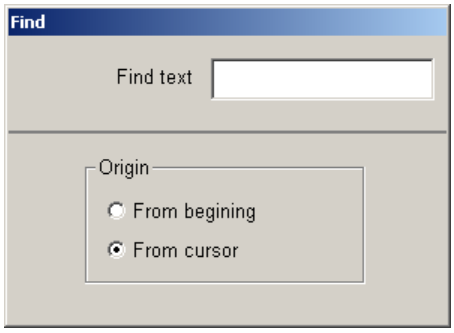
Recall a table.

This softkey is used to restore the table data from an ASCII file. After selecting the table whose data is to be restored, press the "Load" softkey 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.

10.4.3 Find text.

This softkey is used to find text or a value in the table. After pressing this softkey, the CNC shows a dialog box requesting the text or value to be found. It is also possible to select whether the search must start at the beginning of the table or at the current cursor position.



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. Pressing this FIND icon again will allow repeating the search or defining a new one.

10.

USER TABLES.

Operations with tables

FAGOR

FAGOR AUTOMATION

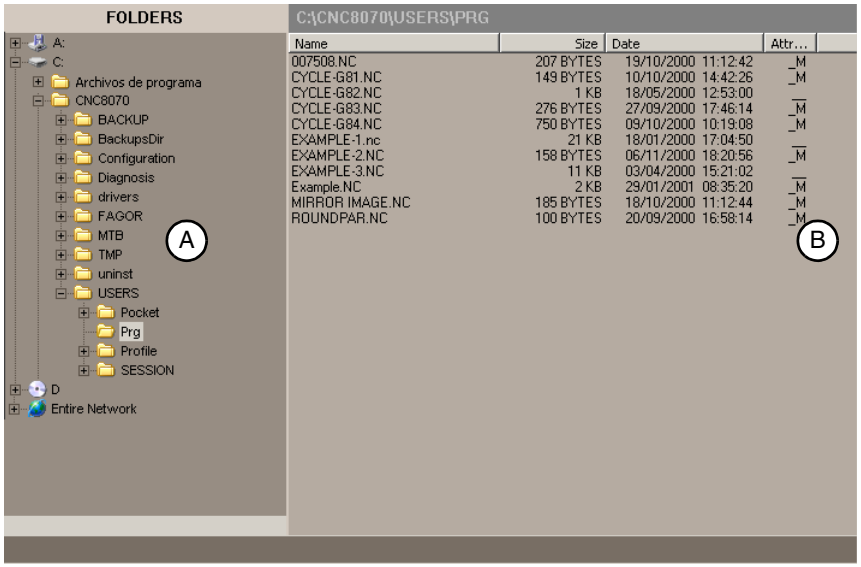
CNC 8060 L
CNC 8070 L

(REF: 1707)

UTILITIES MODE.

11

The [UTILITIES] key accesses UTILITIES mode.



A Folder tree. The tree shows the folders that may be accessed from the CNC, as well their structure.

B List of files saved in the selected folder.

When selecting a folder, 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. Attribute column (modifiable and hidden files).

The attribute column only shows the letters of the attributes currently selected. Those not selected will appear as "_" .

H The file is hidden and it will not appear when selecting a program for editing or execution.

M The file may be modified; in other words, edited, copied, deleted, etc.

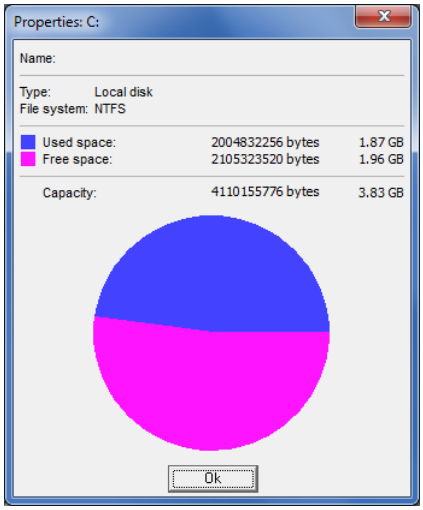
List of files. Open a file.

The [ENTER] key opens the file selected on the list.

- *.exe files: The CNC executes the file.
- Unformatted text files: The CNC opens the file with Notepad.
- Rest of files: The CNC opens the file with its associated program.

Folder tree. Show the properties of an entity or folder.

If a unit or folder has been selected, pressing the "=" will display the properties window. Depending on the size of the folder, the CNC may show a progress bar while it calculates its size.



Horizontal softkey menu.

Softkey.	Description.
Sorted by.	Sorting the list of files.
Options.	Set how the file list will be displayed.
File search.	Search files.
Select all.	Select all the files fom the list.
Invert selection.	Invert file selection.
New folder.	Create a new folder.
Access codes.	Manage CNC passwords.
Automatic backup.	Do a backup of the selected folder.
Restore backup.	Restoring data saved in a backup.
Backup options.	Configuring backup options.

Vertical softkey menu.

The vertical softkey menu always shows all the options associated with this operating mode regardless of the active screen.

Softkey.	Description.
	Show more options on the softkey menu.
	Cut the selected files onto the clipboard. With this option, when pasting the files to their new location, they are erased from the current folder.
	Copy the selected files onto the clipboard.
	Paste 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.

Softkey.	Description.
	Rename the selected folder or file. If there is a folder or file already with the new name, the change will be ignored. The files being used cannot be renamed (for example, the file selected in automatic mode).
	Change the "modifiable" attribute of the selected files. 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.
	Change the "hidden" attribute of the selected files. The attributes column shows the letter -H- indicating that the program will be hidden (not visible). This attribute allows protecting the files so they are not displayed when selecting a program to be edited or executed. However, a hidden program may be deleted if its name is known; therefore, it is recommended to remove the modifiable attribute (M) in order to avoid deleting it.
	Encrypt files. Encrypting may be used to protect any file (part-program, subroutine, etc.) making it illegible so it cannot be used by anyone else.
	Delete 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 files being used cannot be renamed (for example, the file selected in automatic mode).

11.1 Set how to display the list of programs.

Sorting the list of files.

The "Sort by" softkey is used to order the list of files. When selecting this option, the softkey menu shows the different ordering options. To return to the main menu, press the [BACK] key.

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.

Customizing options.

The "Options" softkey 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 main menu, press the [BACK] key.

Update

This option 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, every time a folder is selected, the CNC will automatically update the list of files.

Column adjust

When this option is selected, 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.

Show hidden files

It closes the folder tree (layout) and it only shows the devices accessible from the CNC.

Find in files

When this option is selected, the file list shows all the files of the selected folder even those having the "hidden" attributes. Otherwise, these files will not be shown.

11.2 Select files and create folders.

Having selected a group of files, they may be deleted, copied, cut or their attributes may be changed by pressing the relevant icon.

Select files and folders.

- To select files or folders, besides the mouse, the following keys may be used. To select a group of files from the keyboard, 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.

Key.	Meaning.
	It switches the window focus.
	With the focus on a folder, it closes and opens the submenu of folders.
	To moves the focus element by element.
	To move the focus page by page.
	To move the focus to the beginning or end of the list.

- 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.

Select files from the softkey menu.

Select all.

Select all the files fom the list. The selection will be canceled by moving the cursor.

Having selected a group of files, they may be deleted, copied, cut or their attributes may be changed by pressing the relevant icon.

Invert selection

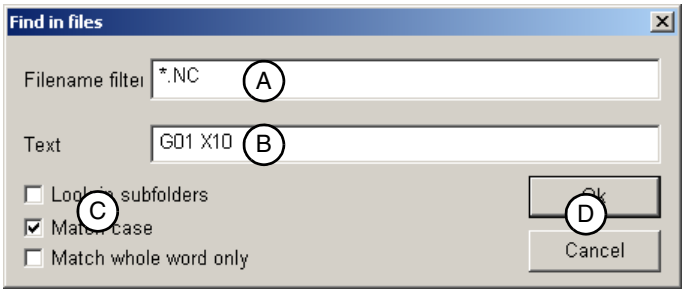
It inverts the file selection made selecting the files that appeared unselected and vice versa.

Create a new folder from the softkey menu.

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.

11.3 Find 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 To start or cancel the defined search.

After defining the search options, place the cursor on one of the lower buttons to accept or cancel the defined search and press [ENTER]. The file window will show the list of the programs found.

11.4 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.

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.

Description of protection passwords.

Password.	Description.
General password.	Password to access this page for setting passwords.
PLC.	Password to carry out the following actions on 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.	Password to carry out the following actions. • 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. • Do a restore of the CNC data.
Customizing.	Password to enter the FGUIM application. When entering the wrong password, it will not be possible to access the application.
Machine parameters for kinetics.	Password to carry out the following actions in the machine kinematics parameter table. 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.
Administrator mode.	Password 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.

11.5 Data safety backup. Backup - Restore

The CNC only allows doing the backup or restore when there is no power (e.g. E-stop button pressed).

This option may be used to make a backup of the CNC configuration (OEM and user data) to be restored later one if necessary.

Softkey.	Meaning.
	Make a backup of CNC data.
	Restore data from the backup .

Make a backup of CNC data.

Data backup may be carried out when the CNC has been started up in SETUP mode or in USER mode.

On Windows explorer, select the folder where the backup will be saved. Press the "BACKUP" softkey and the CNC will display a window with the options to select the data to include in the backup.

Restore data from the backup .

OEM data may only be restored whe the CNC has been started up in SETU mode; if it has been started up in USER mode, only user data may be restored and the rest of the options are disabled. The restore option is protected by the "Machine parameters" password.

On Windows explorer, select the folder where the backup has been saved. Press the "RESTORE" softkey and the CNC will display a window with the options to select the data to include in the backup. If the selected folder does not contain one of the backups, its option will appear disabled. The CNC application must be restarted after restoring a backup.

Select the data to be included in the backup or restore.

The selected options will be kept until they are modified.

OEM data.

- Machine parameters. Backup/restore of CNC machine parameters. The backup saves the data in the BACKUP_OEM_MP.zip file.

  In order for the CNC to properly do the backup and restore of the volumetric compensation tables, it is recommended to save them in the folder ...MTB/SUB.
- Tables. Backup/restore of CNC tables; fixtures, zero offsets and arithmetic parameters. The backup saves the data in the BACKUP_OEM_TABLES.zip file.
- Tools and tool magazines. Backup/restore of tool and tool magazine tables. The backup saves the data in the BACKUP_OEM_MZTOOLS.zip file.
- Servo parameters. Backup/restore of drive machine parameters. The backup saves the data in the BACKUP_OEM_DRIVEMP.zip file.
- Subroutines. Backup/restore of OEM subroutines. The backup saves the data in the BACKUP_OEM_SUB.zip file.
- PLC. Backup/restore of the PLC; PLC programs, messages and errors. The backup saves the data in the BACKUP_OEM_PLC.zip file.
- Configuration and screens. Backup/restore of CNC configuration and screens. The backup saves the data in the BACKUP_OEM_CONFIG.zip file.

User data.

- Programs.

  Backup/restore of user programs, pockets, profiles, etc. The backup saves the data in the BACKUP_USER_PRG.zip file. This backup includes the following folders.

  ..\Users\Prg ..\Users\Pocket ..\Users\Profile ..\Users\CycleData

11.6 Encrypting files

Encrypting may be used to protect any file (part-programs, subroutines, etc.) making it illegible so it cannot be used by anyone else. An encrypted program cannot be edited at the program editor or at the PLC and is not displayed during execution. An encrypted file may be copied, deleted, etc. like any other file.

When pressing the "Encrypt file" softkey, the CNC encrypts the file selected with the cursor. Encrypting maintains the original file and generates a new encrypted file with the same name but with the extension fcr. If the CNC encrypts the file successfully, it asks the user whether or not the original file is to be deleted leaving only the encrypted one. This file cannot be unencrypted at the CNC; therefore, we suggest to save a copy of the original file somewhere safe.

Encrypting files and the manufacturer's (OEM's) password.

The encrypting algorithm takes into account the OEM password (general access code). If this password does not exist when trying to encrypt a file, the CNC will show the relevant warning and will abort the process. If after encrypting a file, the OEM password changes, the CNC will no longer be able to read the file.

When the CNC executes an encrypted program or subroutine, it checks whether the current OEM password matches the one used to encrypt the file. If the two passwords do not match, the CNC will not be able to execute the program or subroutine and will issue the relevant error message. A file encrypted in a CNC may be executed in another CNC if the OEM passwords of the two CNC's are the same.

In the case of a PLC program, there is no need to encrypt the source file because compiling it is enough. The source file may be deleted.

PROGRAMMING.

II

FAGOR

FAGOR AUTOMATION

BLANK PAGE

PROGRAM THE POWER AND TURN THE LASER ON AND OFF.

12

12.1 Program the power.

The power is selected by program using the spindle name followed by the desired power (in watts). The programmed power stays active until another value is programmed. The programmed power can be modified from the control panel (keys or speed override switch), from the PLC or the program.

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

Programming.

Program the spindle name and the power. This function allows programming when the laser is on. In this case, the spindle will assume the new value.

Programming format

The programming format is the following; the arguments appear between curly brackets.

```
S{power}
```

S{power}	Power. Spindle "S". Units: Between 0 and 99999 watts.

```
S1000
```

Turning the laser on and off.

The definition of a power does not imply turning on the laser . The power-up is carried out by functions M03, M04 and M05.

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

The S function is modal. On power-up, after executing an M02 or M30, and after an emergency or reset, the spindle assumes a ·0· speed.

12.2 G192. Limiting power.

The G192 function limits the laser power. If function G192 is not programmed, the power is limited by machine parameter G00FEED of the gear.

Programming.

Program the G192 function and the maximum power in each spindle. This function may be programmed while the laser is on; in this case, the CNC will limit the power to the new programmed value.

Programming format.

```
G192 S..S9={power}
G192 S{power}
```

{power}	Maximum power. Units: Between 0 and 99999 watts.

```
G192 S1000
G192 S1=6000
```

Spindle name.

The spindle name can be any in the range S, S1..S9; for the S spindle, the programming of "=" sign can be omitted.

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

Function G192 is modal. On power-up, the CNC cancels function G192; after executing an M02 or M30, and after an emergency or reset, the CNC maintains function G192.

12.3 Turning the laser on and off.

A power must be set in order to start up a spindle. The definition of a power does not imply turning on the laser . Power-up is carried out through the following auxiliary functions. These functions are modal and incompatible with each other.

M03	Turn on the laser.
M04	Turn on the laser.
M05	Turn off the laser.

Programming. M03/M04. Turn on the laser.

Functions M3 and M4 turn on the laser; both functions are equivalent. These functions may be defined together with the name of a spindle or in a separate block. If the block where they are programmed does not mention any spindle, the CNC will apply them to the master spindle.

Programming format.

The programming format is:

M3
M4
M3.S..S9
M4.S..S9

S1000 M3
("S" spindle; 1000-watt power).

S1=5000 M4
("S1" spindle; 5000-watt power).

M4
(The master spindle turns on the laser).

M3.S S1000
("S" spindle; 1000-watt power).

Programming. M05. Turn off the laser.

Function M05 turns on the laser. This function may be defined together with the spindle name or in a separate block. If the block where they are programmed does not mention any spindle, the CNC will apply it to the master spindle.

Programming format.

The programming format is:

M5
M5.S..S9

M5
(The master spindle turns off the laser).

M5.S2
("S2" spindle; turn off the laser).

12.4 Variables related to power.

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W). The reading of these variables stops the preparation of blocks, except as otherwise indicated.

Variable.	PRG	Meaning.
(V.)[ch].A.SREAL.sn (V.)[ch].SP.SREAL.sn	R	Actual power. Units: Watts.
(V.)[ch].A.SPEED.sn (V.)[ch].SP.SPEED.sn	R	Active power in G97. Units: Watts.
(V.)[ch].PLC.S.sn	R	Active power by PLC for G97. Units: Watts.
(V.)[ch].A.PRGS.sn (V.)[ch].SP.PRGS.sn	R(*)	Active power by program for G97. Units: Watts.
(V.)[ch].A.SLIMIT.sn (V.)[ch].SP.SLIMIT.sn	R	Active power limit. Units: Watts.
(V.)[ch].PLC.SL.sn	R	Active power limit by PLC. Units: Watts.
(V.)[ch].A.PRGSL.sn (V.)[ch].SP.PRGSL.sn	R(*)	Active power limit by program. Units: Watts.
(V.)[ch].A.SSO.sn (V.)[ch].SP.SSO.sn	R	Active power percentage. Units: Percentage.
(V.)[ch].PLC.SSO.sn	R	Power percentage by PLC. Units: Percentage.
(V.)[ch].A.PRGSSO.sn (V.)[ch].SP.PRGSSO.sn	R(*)	Power percentage by program. Units: Percentage.
(V.)[ch].A.CNCSSO.sn (V.)[ch].SP.CNCSSO.sn	R	Power percentage on the control panel. Units: Percentage.

(*) The variable returns the value of block preparation.

Syntax of the variables.

·ch· Channel number.

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

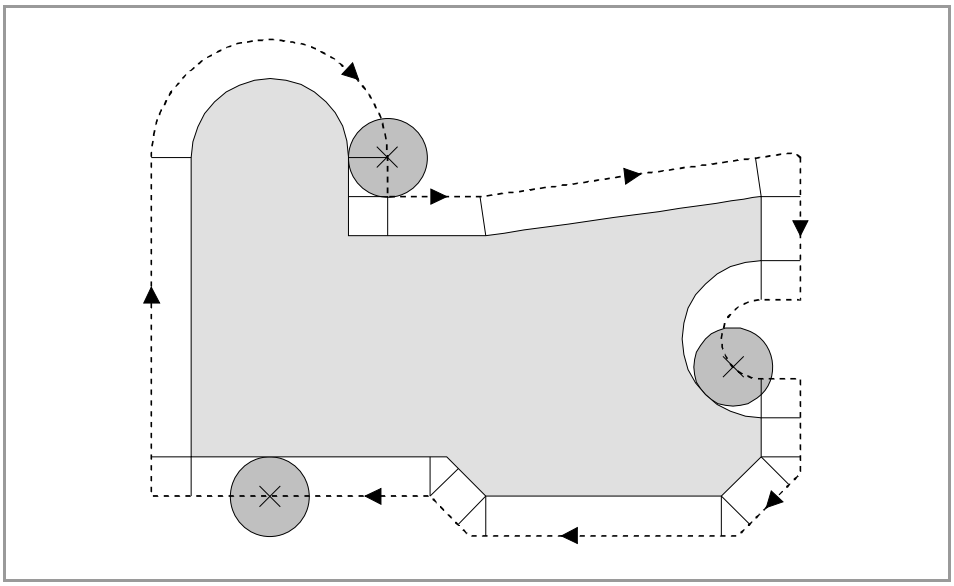
V.A.SREAL.S	Spindle S.
V.SP.SREAL.S	Spindle S.
V.SP.SREAL	Master spindle.
V.A.SREAL.5	Spindle with logic number ·5·.
V.SP.SREAL.2	Spindle with index ·2· in the system.
V.[2].SP.SREAL.1	Spindle with index ·1· in the channel ·2·.
V.PLC.S.S2	Spindle S2.
V.PLC.S.5	Spindle with logic number ·5·.
V.[2].PLC.S.1	Spindle with index ·1· in the channel ·2·.

RADIUS COMPENSATION.

13

Radius compensation allows for programming the contour to be machined based on part dimensions and without taking into account the radius of the laser that will be used later on. When working with radius compensation, the center of the laser follows the programmed path to a distance equal to its radius; this way, the correct dimensions of the programmed part are obtained during the cut.

When the radius compensation is active, the CNC analyzes the blocks to be executed beforehand, in order to detect any compensation errors related to steps, null arcs, etc. If the CNC detects any of these errors, the blocks in which they originate will not be executed and a warning will appear on screen telling the user that the programmed profile has been modified. The CNC will display a warning for every profile correction made.



The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

13.1 Defining the laser radius.

In the radius compensation, the CNC applies as the compensation value the sum of the radius and the radius adjustment. Both numbers are defined according to the following variables. Both variables can be read and written via the program, PLC (asynchronous writing) and interface. The variables return the execution value; reading it interrupts block preparation.

Variables.	Meaning.
(V.)[ch].G.TOR	Laser radio. Units: mm or inches.
(V.)[ch].G.TOI	Radius adjustment: generally, this variable will have a value of 0 and it must only be modified if, for some reason, it is necessary to change the laser radius. Units: mm or inches.

Syntax of the variables.

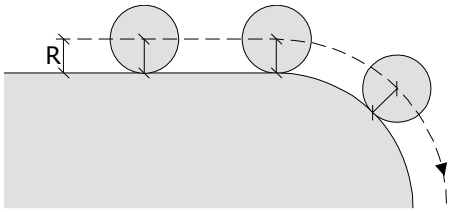
·ch· Channel number.

V.[2].G.TOR	Channel ·2·. Laser radio.
V.[2].G.TOI	Channel ·2·. Radius adjustment.

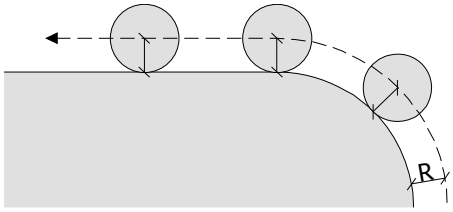
13.2 G40/G41/G42. Activating/canceling the laser radius compensation.

The functions G41 and G42 activate the radius compensation and the function G40 cancels it.

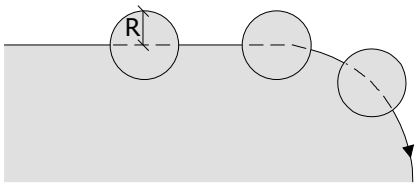
G41 Radius compensation to the left.



G42 Radius compensation to the right.



G40 Canceling radius compensation.



The CNC applies radius compensation on the work plane. Depending on the type of compensation selected (G41/G42), the CNC will place the tool to the left or to the right of the programmed path, in the machining direction, and will apply the compensation value (sum of the laser radius and the radius adjustment). If no tool compensation is selected (G40), the CNC will place the tool center right on the programmed tool path.

Programming.

These functions may be programmed alone in the block or may be added to a movement block.

Programming format.

The programming format is the following; the arguments appear between curly brackets and the optional ones between angle brackets.

```
G40 <X..C{pos}>
G41 <X..C{pos}>
G42 <X..C{pos}>
```

X..C{pos}	Optional. Jogging the axes. Units: ±99999.9999 mm or ±3937.00787 inches.

```
G41 X12345.123 Y12345.1234
G42
G40 12345.1235
```

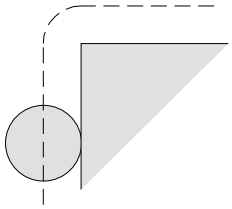
Properties of the functions.

Functions G40,G41 and G42 are modal and incompatible with each other. On power-up, after executing an M02 or M30, and after an emergency or reset, the CNC assumes function G40.

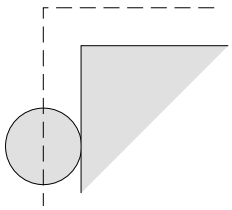
13.3 G136/G137. Selecting the type of transition between blocks.

The functions G136/G137 determine how to join the compensated paths, through circular or linear paths. The functions to select the transition type are as follows:

G136 The CNC joins the compensated paths by means of circular paths.



G137 The CNC joins the compensated paths by means of straight paths.



Programming.

These functions may be programmed alone in the block or may be added to a movement block. These functions may be programmed anywhere in the program, even while the radius compensation is active.

Programming format.

The programming format is the following; the arguments appear between curly brackets and the optional ones between angle brackets.

```
G136 <X..C{pos}>
G137 <X..C{pos}>
```

X..C{pos}	Optional. Jogging the axes. Units: ±99999.9999 mm or ±3937.00787 inches.

```
G136 X12345.123 Y12345.1234
G137
```

Remarks.

Later sections of this chapter offer graphic descriptions of how different paths are joined, depending on the type of transition selected (G136/G137). See *"13.5 Starting, ending and joining paths with radius compensation."* on page 147.

Properties of the functions.

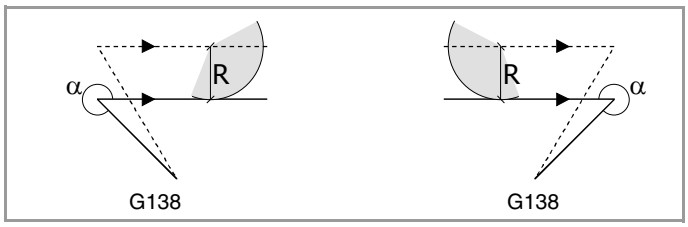
Functions G136 and G137 are modal and incompatible with each other. On power-up, after executing an M02 or M30, and after an emergency or reset, the CNC assumes function G136 or G137, depending on the value of machine parameter IRCOMP.

13.4 G137/G138. Strategy for starting and ending radius compensation.

The functions G137/G138 determine how to start and end the radius compensation. The functions to select the strategy type are as follows:

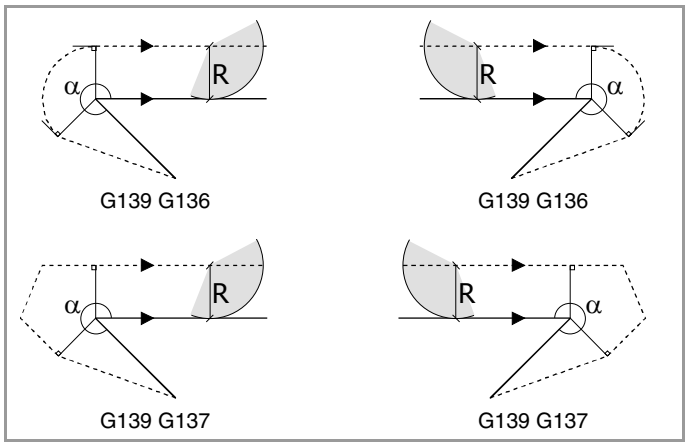
G138 Starting and ending the radius compensation directly.

- When compensation starts, the laser moves directly to the perpendicular of the next path (without contouring the corner).
- When compensation ends, the laser moves directly to the programmed point (without contouring the corner).



G139 Starting and ending the radius compensation compensation indirectly.

- When compensation starts, the laser moves to the perpendicular of the next path, contouring the corner. The way the tool goes around the corner depends on the type of transition selected (G136/G37).
- When compensation ends, the laser moves to the end point, contouring the corner. The way the tool goes around the corner depends on the type of transition selected (G136/G37).



Programming.

These functions may be programmed alone in the block or may be added to a movement block. These functions may be programmed anywhere in the program, even while the radius compensation is active.

Programming format.

The programming format is the following; the arguments appear between curly brackets and the optional ones between angle brackets.

```
G138 <X..C{pos}>
G139 <X..C{pos}>
```

X..C{pos} Optional. Jogging the axes.
Units: ±99999.9999 mm or ±3937.00787 inches.

```
G138 X12345.123 Y12345.1234
G139
```

Remarks.

Later sections of this chapter offer a graphic description of how radius compensation begins and ends, depending on the selected type of strategy (G138/G139). See *"13.5 Starting, ending and joining paths with radius compensation."* on page 147.

Properties of the functions.

Functions G138 and G139 are modal and incompatible with each other. On power-up, after executing an M02 or M30, and after an emergency or reset, the CNC assumes function G139.

13.5 Starting, ending and joining paths with radius compensation.

13.5.1 Starting radius compensation.

After executing function G41 or G42, the CNC activates the radius compensation for the next movement on the work plane, which must be a linear movement. The way radius compensation begins depends on the type of activation strategy (G138/G139) and on the type of transition (G136/G137) selected.

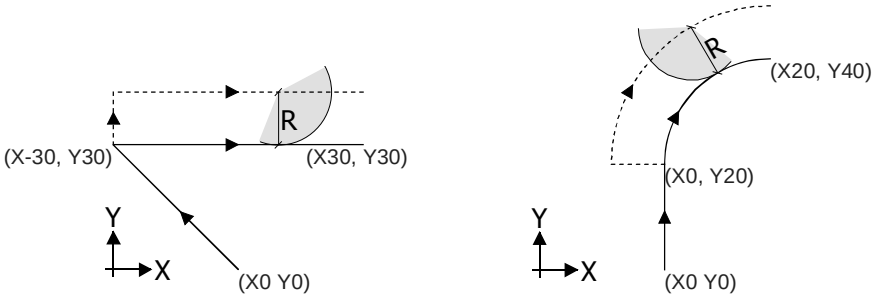
- G139/G136 Laser movement to the perpendicular of the next path, contouring the corner along circular paths.
- G139/G137 Laser movement to the perpendicular of the next path, contouring the corner along linear paths.
- G138 Laser movement directly to the perpendicular of the next path. Regardless of the type of transition (G136/G137) programmed.

The following tables show the different ways tool compensation may begin, depending on the selected functions.

Starting the radius compensation when there is no movement programmed on the plane.

After activating the radius compensation, it may occur that the plane axes are not involved in the first motion block, for example, because they have not been programmed, the same point has been programmed or a null incremental move has been programmed. In this case, the CNC activates the compensation at the laser position as follows.

Depending on the first movement programmed in the plane, the tool moves perpendicular to the path to its starting point. The first movement programmed in the plane may be either linear or circular.



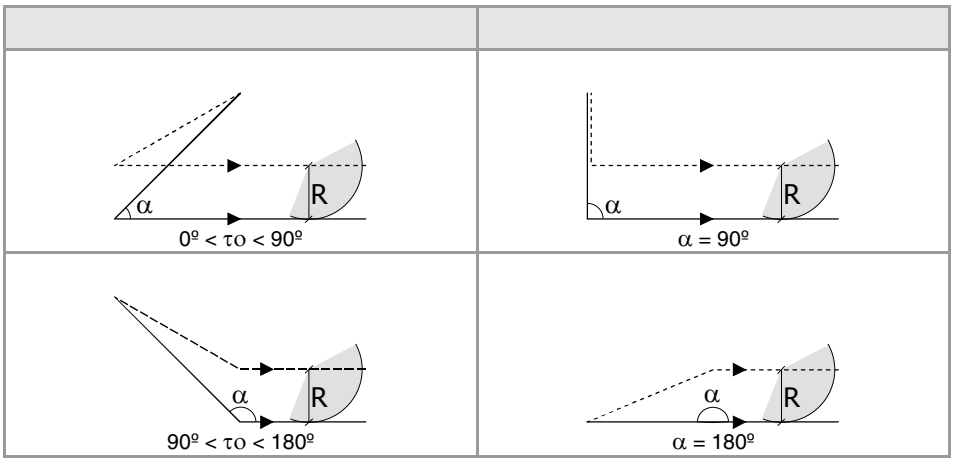

```
...
G90
G01 X-30 Y30
G01 G41 X-30 Y30 Z10
G01 X30
...
```

```
...
G90
G01 Y20
G91 G41 Y0 Z10
G02 X20 Y20 I20 J0
...
```

The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

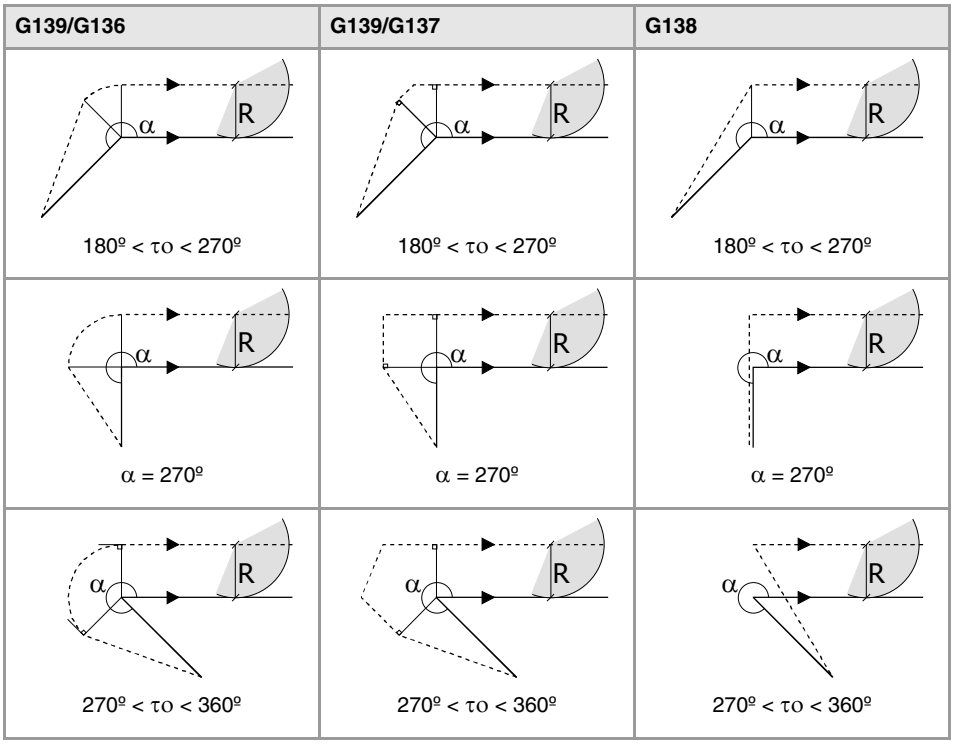
Starting radius compensation; straight path - straight path.

When the angle between paths is less than or equal to 180º, the way in which the compensated paths are joined is independent of the functions G136/G137 and G138/G139 selected.



The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

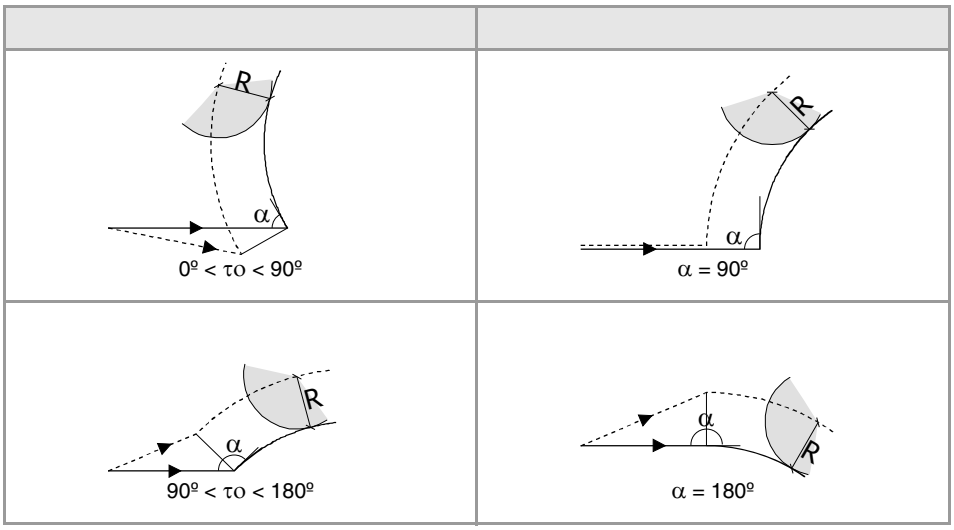
When the angle between the paths is greater than 180º, the way in which the compensated paths are joined depends on the functions G136/G137 and G138/G139 selected.



The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

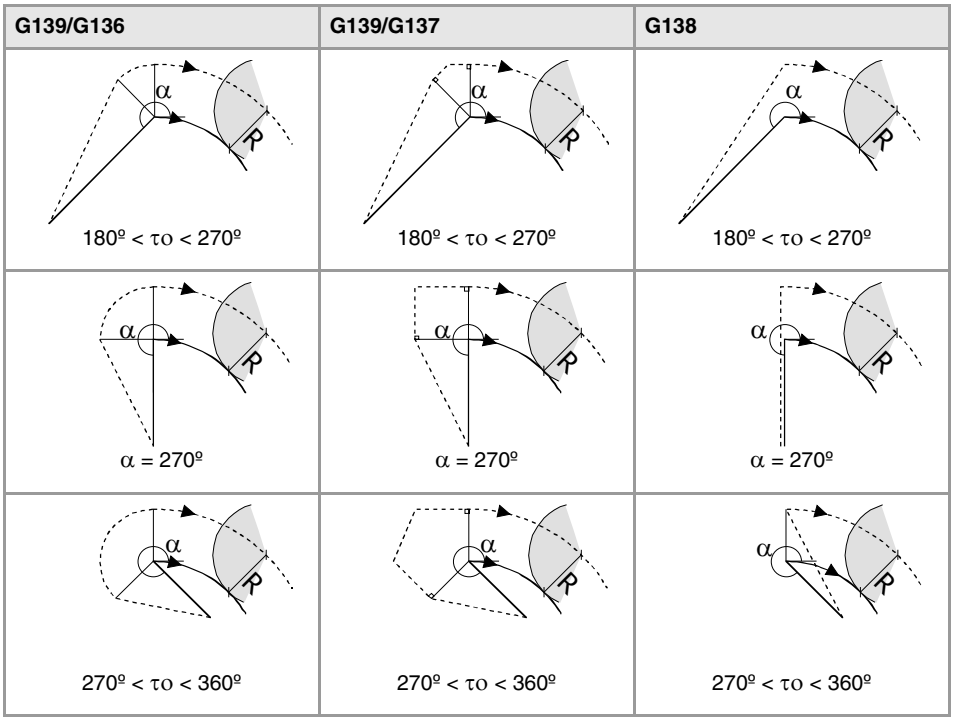
Starting radius compensation; straight path - arc.

When the angle between the straight path and the tangent of the circular path is less than or equal to 180º, the way in which the compensated paths are joined is independent of the functions G136/G137 and G138/G139 selected.



The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

When the angle between the straight path and the tangent of the circular path is greater than 180º, the way in which the compensated paths are joined depends on the G136/G137 and G138/G139 functions selected.



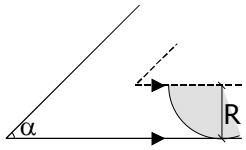
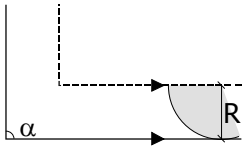
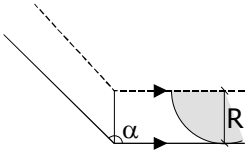
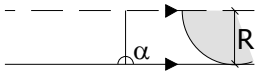
The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

13.5.2 Joining two paths with radius compensation.

The way the compensated paths are joined depends on the type of transition selected (G136/G137). The following tables show the different transition possibilities between various paths depending on the selected function G136 or G137.

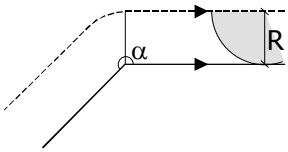
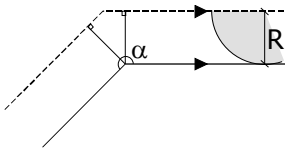
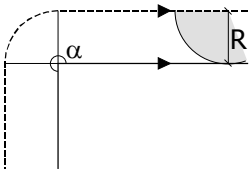
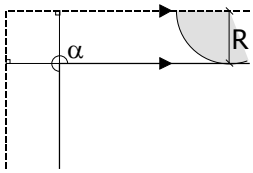
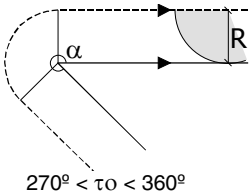
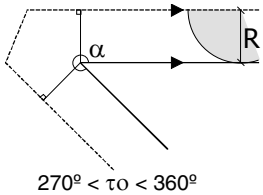
Joining two paths with radius compensation; straight path - straight path.

When the angle between paths is less than or equal to 180º, the way in which the compensated paths are joined is independent of the G136/G137 function selected.

 0º < τo < 90º	 α = 90º
 90º < τo < 180º	 α = 180º

The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

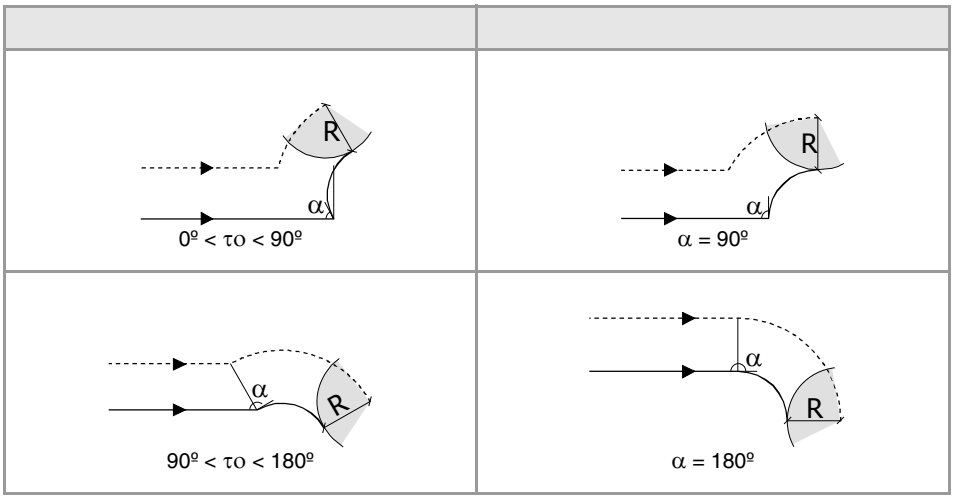
When the angle between the paths is greater than 180º, the way in which the compensated paths are joined depends on the G136/G137 function selected.

G136	G137
 180º < τo < 270º	 180º < τo < 270º
 α = 270º	 α = 270º
 270º < τo < 360º	 270º < τo < 360º

The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

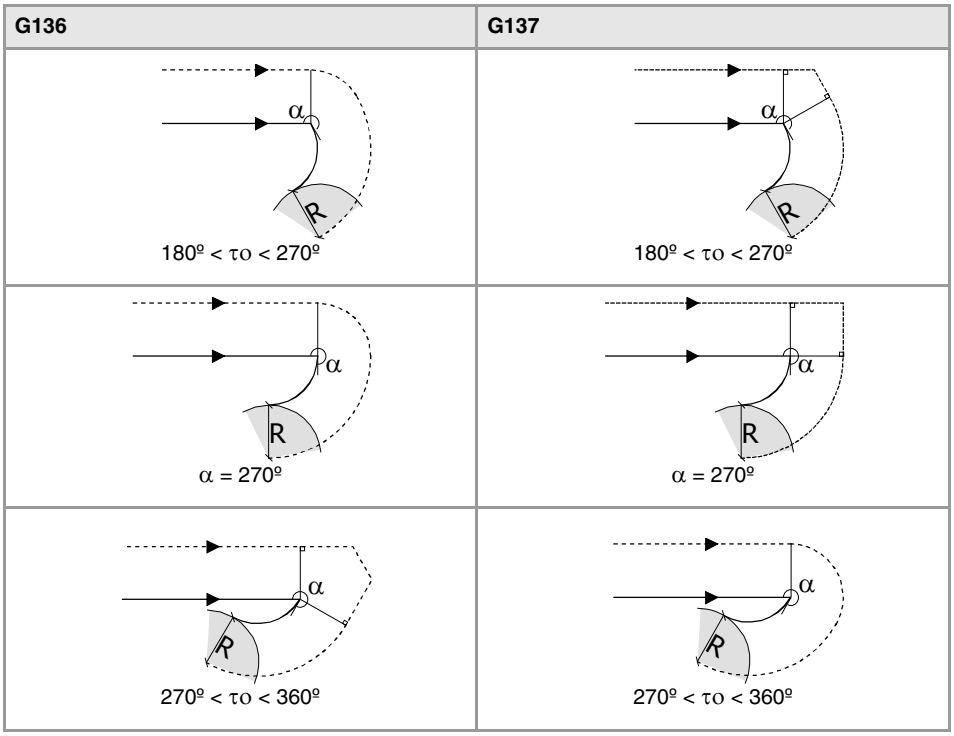
Joining two paths with radius compensation; straight path - arc.

When the angle between the straight path and the tangent of the circular path is less than or equal to 180º, the way in which the compensated paths are joined is independent of the G136/G137 function selected.



The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

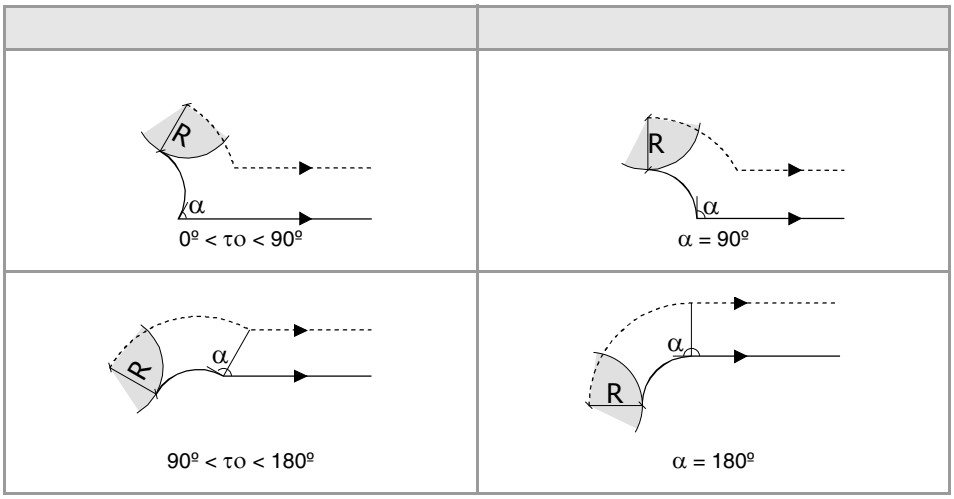
When the angle between the straight path and the tangent of the circular path is greater than 180º, the way in which the compensated paths are joined depends on the G136/G137 function selected.



The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

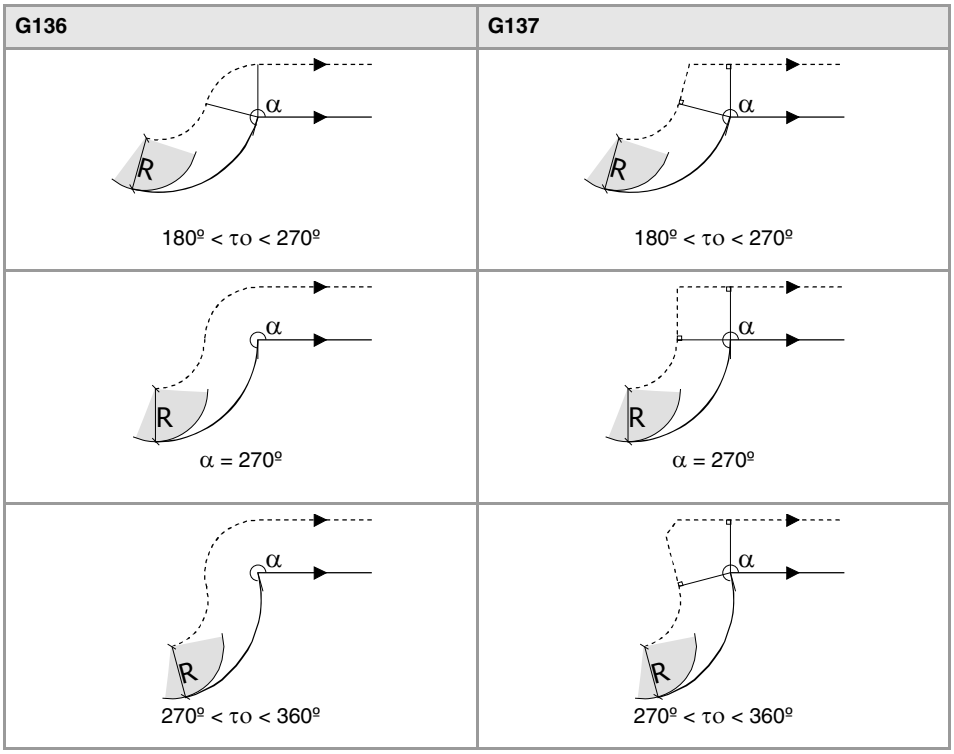
Joining two paths with radius compensation; arc - straight path.

When the angle between the tangent of the circular path and the straight path is less than or equal to 180º, the way in which the compensated paths are joined is independent of the G136/G137 function selected.



The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

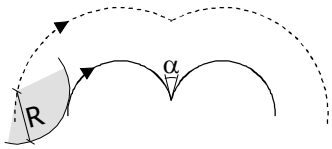
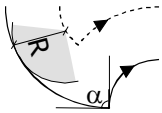
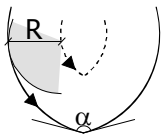
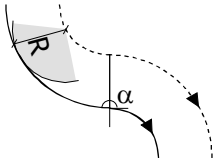
When the angle between the tangent of the circular path and the straight path is greater than 180º, the way in which the compensated paths are joined depends on the G136/G137 function selected.



The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

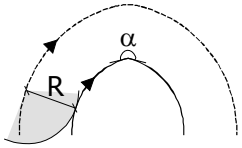
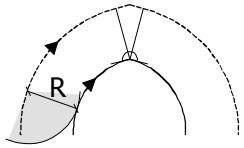
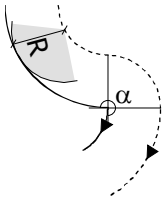
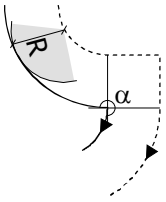
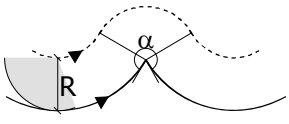
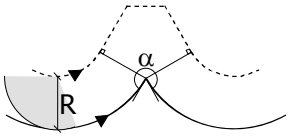
Joining two paths with radius compensation; arc - arc paths.

When the angle between the tangents of the circular paths is less than or equal to 180º, the way in which the compensated paths are joined is independent of the G136/G137 function selected.

 0º < τo < 90º	 α = 90º
 90º < τo < 180º	 α = 180º

The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

When the angle between the tangents of the circular paths is greater than 180º, the way in which the compensated paths are joined depends on the G136/G137 function selected.

G136	G137
 180º < τo < 270º	 180º < τo < 270º
 α = 270º	 α = 270º
 270º < τo < 360º	 270º < τo < 360º

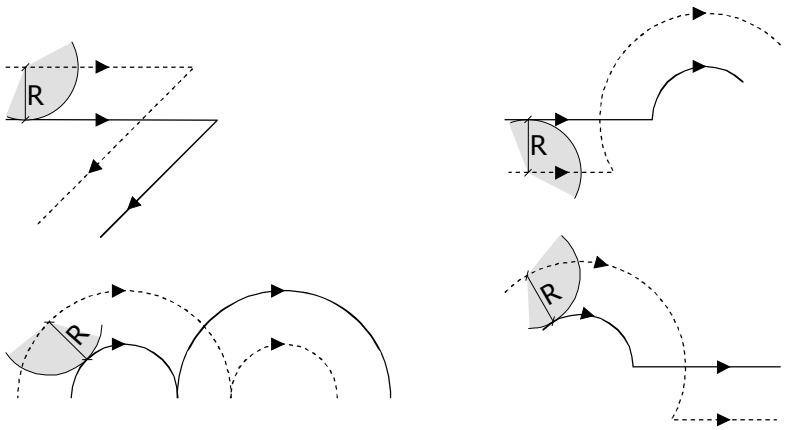
The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

13.5.3 Changing the type of radius compensation (G41/G42) when selected.

The compensation may be changed from G41 to G42 or vice versa without having to cancel it with G40. It may be changed in any motion block or even in a motionless one; i.e. without moving the axis of the plane or by programming the same point twice.

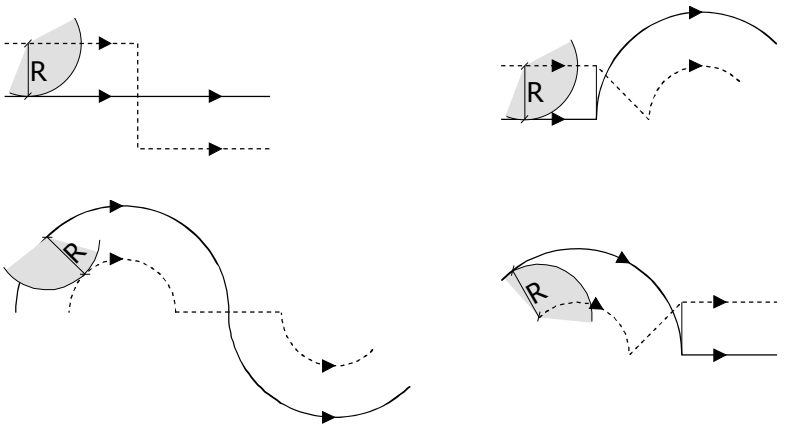
The last movement before the change and the first one after the change are compensated independently from one another. To change the type of compensation, the different cases are solved according to these criteria.

- The compensated paths cut each other. The programmed paths are compensated each on its corresponding side. The side change takes place in the intersection point between both paths.



The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

- The compensated paths do not cut each other. The CNC adds another section between the two paths, perpendicular to the first path at its end point and perpendicular to the second path at its starting point. Both points are located at a distance equal to the tool radius, as measured from the programmed path.

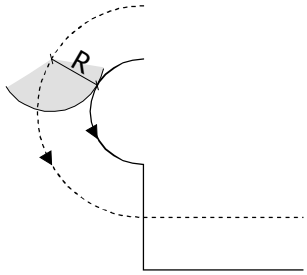


The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

- Back-and-forth path along the same way.

The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

- Intermediate path is as long as the laser radius.



The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

13.5.4 Canceling radius compensation.

After executing the function G40, the CNC deactivates the radius compensation during the next movement on the work plane, which must be a linear movement. The way in which this radius compensation is canceled depends on the G138/G139 cancellation strategy and the type of G136/G137 transition selected.

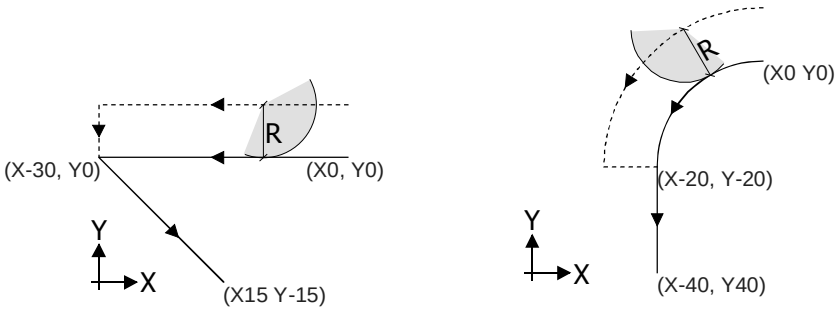
- G139/G136 Laser movement to the endpoint, contouring the corner along a circular path.
- G139/G137 Laser movement to the endpoint, contouring the corner along linear paths.
- G138 Laser movement directly to the endpoint. Regardless of the type of transition (G136/G137) programmed.

The following tables show the different possibilities of canceling tool radius compensation depending on the selected functions.

Ending the radius compensation when there is no programmed movement on the plane.

After canceling the radius compensation, it may occur that the plane axes are not involved in the first motion block, for example, because they have not been programmed, the same point has been programmed or a null incremental move has been programmed. In this case, the CNC cancels the compensation at the laser position as follows.

Depending on the last movement made on the plane, the tool moves to the endpoint (uncompensated) of the programmed path.



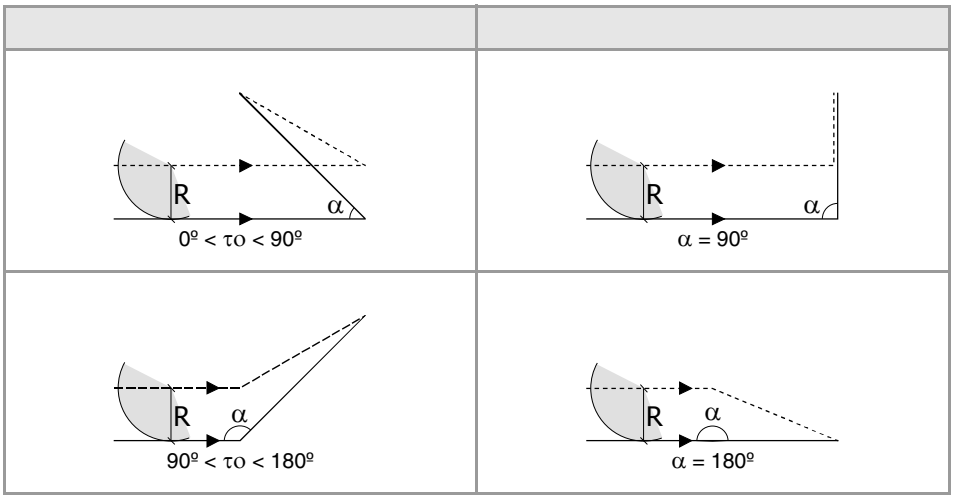

```
…
G90
G01 X-30
G01 G40 X-30 Z10
G01 X15 Y-15
…
```

```
…
G90
G03 X-20 Y-20 I0 J-20
G91 G40 Y0
G01 X-20
…
```

The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

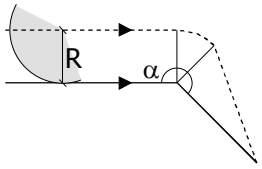
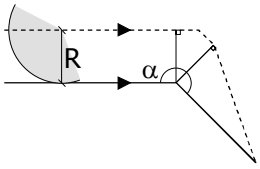
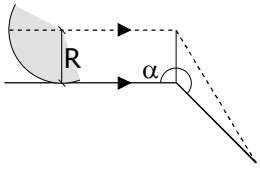
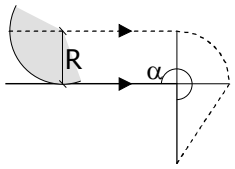
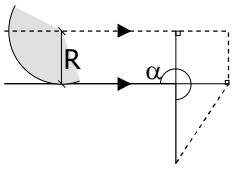
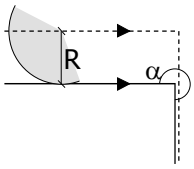
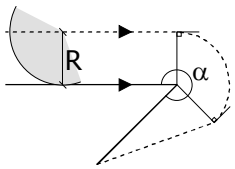
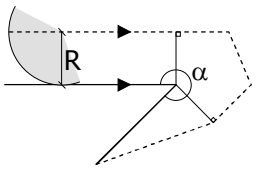
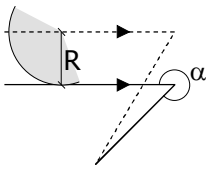
Ending radius compensation; straight path - straight path.

When the angle between paths is less than or equal to 180º, the way in which the compensated paths are joined is independent of the functions G136/G137 and G138/G139 selected.



The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

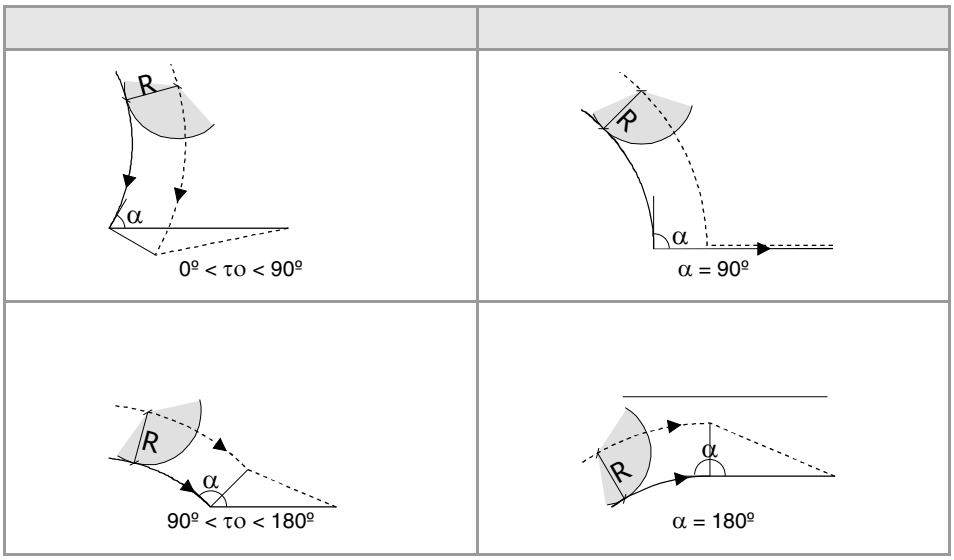
When the angle between the paths is greater than 180º, the way in which the compensated paths are joined depends on the functions G136/G137 and G138/G139 selected.

G139/G136	G139/G137	G138
		
		
		

The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

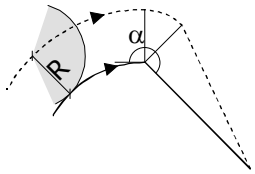
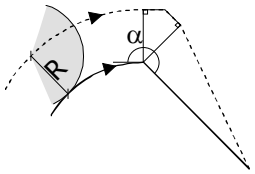
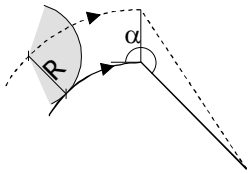
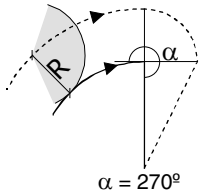
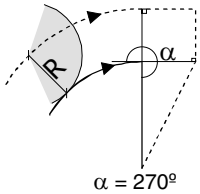
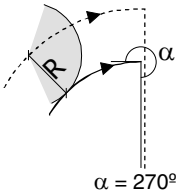
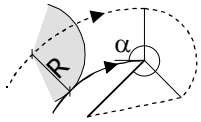
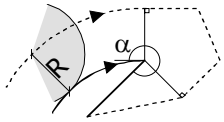
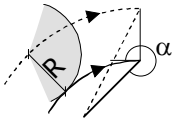
Ending radius compensation; arc - straight path.

When the angle between the tangent of the circular path and the straight path is less than or equal to 180º, the way in which the compensated paths are joined is independent of the G136/G137 and G138/G139 functions selected.



The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

When the angle between the tangent of the circular path and the straight path is greater than 180º, the way in which the compensated paths are joined depends on the G136/G137 and G138/G139 functions selected.

G139/G136	G139/G137	G138
 180º < τo < 270º	 180º < τo < 270º	 180º < τo < 270º
 α = 270º	 α = 270º	 α = 270º
 270º < τo < 360º	 270º < τo < 360º	 270º < τo < 360º

The solid line represents the programmed path, and the dotted line shows the path of the center of the laser.

13.6 Variables.

(V.)[ch].G.TOR

Variable that can be read and written via program, PLC(asynchronous writing) and interface.
The variable returns the execution value; reading it interrupts block preparation.

Laser radio.

Syntax.

·ch· Channel number.

V.[2].G.TOR	Channel ·2·. Laser radio.

(V.)[ch].G.TOI

Variable that can be read and written via program, PLC(asynchronous writing) and interface.
The variable returns the execution value; reading it interrupts block preparation.

Radius adjustment.

Syntax.

·ch· Channel number.

V.[2].G.TOI	Channel ·2·. Radius adjustment.

13.

RADIUS COMPENSATION. Variables.

FAGOR

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

INITIAL AND FINAL SUBROUTINES. 14

These subroutines perform the initial and final operations of the laser and are executed from the part program using instructions #INITIALSUB and #FINALSUB.

Initial and final subroutines.

The subroutines supplied by Fagor provide basic laser management, so they should be configured appropriately by the OEM. The OEM is responsible for ensuring that the subroutine complies with all safety aspects relating to handling the laser.

The subroutines associated with #INITIALSUB and #FINALSUB are defined in the machine parameters INITIALSUB (by default, Initialsub.fst) and FINALSUB (by default, Finalsub.fst). These parameters allow the OEM to associate different subroutines to the instructions.

These subroutines are saved in the folder ..\MTB\Sub. Since it is an OEM subroutine with en .fst extension, with the CNC in USER mode, the CNC loads it into the RAM when starting up the application. With the CNC in SETUP mode (tuning), the CNC loads this subroutine into the RAM the first time it executes it inside the program.

When updating the version, only the subroutines supplied by Fagor will be updated when choosing the third installation level "rename previous version and install it completely".

In the first two installation options, the CNC copies the subroutines to ..\MTB\sub, but only if they do not already exist. In the third installation option, "rename previous version and install it completely", the CNC always updates the subroutines.

14.1 Executing the initial subroutine

The subroutine #INITIALSUB executes the initial subroutine, where the initial laser operations are defined. It is advisable to program this instruction at the start of the program.

Programming.

Program the instruction alone in the block.

Programming format.

The programming format is:

#INITIALSUB

```
#INITIALSUB
```

14.2 Executing the final subroutine.

The subroutine #FINALSUB executes the final subroutine, where the final laser operations are defined. It is advisable to program this instruction at the end of the program.

Programming.

Program the instruction alone in the block.

Programming format.

The programming format is:

#FINALSUB

```
#FINALSUB
```

CHANGING THE STATUS OF THE PLC MARKS.

15

Do not alter the PLC marks on the inputs and outputs with the instruction #PLC unless you know what the effects will be. Only the OEM should use this instruction to change the status of the PLC marks for the digital inputs and outputs. Changing the status of these elements influences the operation of the PLC, and therefore how the CNC and the machine behaves.

If the #PLC instruction activates a logical digital output, the PLC will activate the corresponding physical output.

15.1 Changing the status of the PLC marks.

The instruction #PLC makes it possible to change the status of the PLC marks and the logical digital inputs and outputs from the part program or subroutine. Block preparation is not interrupted during the status change.

Programming.

A total of 20 PLC marks can be defined in each instruction, including both digital inputs and digital outputs.

Programming format.

The programming format is the following; the arguments appear between curly brackets.

```
#PLC [{plc_mark}={status}, I{input}={status}, O{output}={status}]
```

{plc_mark}	PLC mark. Units: A valid PLC mark.
I{input}	Digital input. Units: From 1 to 1024.
O{output}	Digital output. Units: From 1 to 1024.
{status}	Status of the PLC mark. Units: 0 = Deactivate; 1 = Activate.

```
#PLC [CUTTING=0, I200=1, I100=0]
    (Deactivate the CUTTING mark).
    (Activate the I200 input).
    (Deactivate the I100 input).
#PLC [PIERCING=1, O56=0, O12=1]
    (Activate the PIERCING mark).
    (Deactivate the O56 output).
    (Activate the O12 output).
```

PLC marks.

This instruction makes it possible to change any PLC mark; it does not allow the logs to be changed.

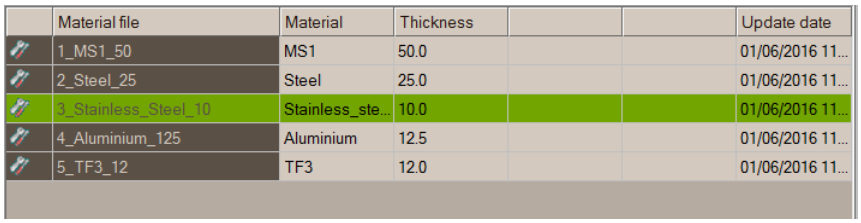
Considerations.

It is recommended to use this instruction in the subroutines associated with piercing and cutting (by default, Piercing.fst / Cuttingon.fst / Cuttingoff.fst), to indicate to the PLC when these operations start (value 1) and end (value 0), by modifying the status of the PIERCING and CUTTING marks, respectively.

SELECT A MATERIAL FROM THE TECHNOLOGICAL TABLE. 16

16.1 Select a material.

The instruction #MATERIAL allows the activation of a material from the technological table, which in turn will be associated with a cutting and a piercing table.



	Material file	Material	Thickness			Update date
	1_MS1_50	MS1	50.0			01/06/2016 11...
	2_Steel_25	Steel	25.0			01/06/2016 11...
	3_Stainless_Steel_10	Stainless_ste...	10.0			01/06/2016 11...
	4_Aluminium_125	Aluminium	12.5			01/06/2016 11...
	5_TF3_12	TF3	12.0			01/06/2016 11...

Programming.

When specifying this instruction, define the material file exactly as it is written on the list of materials (technological table), without the extension xml.

Programming format.

The programming format is the following; the arguments appear between curly brackets.

```
#MATERIAL [{material_file}]
```

{material_file}	Material file specified in the technological tables (without the xml extension).

```
#MATERIAL [STAINLESS_STEEL_2.5MM]
```

Material name.

The name can have 50 characters and cannot contain any special characters or blank spaces; it is not case sensitive.

If the name defined in the instruction does not exist on the table, the CNC will display the corresponding error. To avoid writing errors, the CNC allows the user to copy and take the name from the table.

Considerations.

The CNC allows a material file to be activated directly from the technological tables. Use the mouse to double click on a material or position the cursor over a material and click on [ENTER] to access its piercing and cutting tables. Use the softkey menu to activate the table.

16.

SELECT A MATERIAL FROM THE TECHNOLOGICAL TABLE.

Select a material.

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

PIERCING (INITIAL PIERCING).

17

The piercing operation is the initial penetration process into the sheet metal at the point where a profile cutting process defined by G1, G2 and G3 blocks will begin. The piercing process is performed with machining conditions defined by the piercing type selected on the technological tables.

Subroutine associated with the piercing.

The piercing command is associated with a subroutine (Piercing.fst), to be executed by the CNC before the command. The subroutine supplied by Fagor provides basic management of the piercing process, so it should be configured appropriately by the OEM.
The OEM is responsible for ensuring that the subroutine has all the safety aspects related to handling the laser.

The subroutine associated with the piercing is defined in the machine parameter PIERCING (by default, Piercing.fst). This parameter allows the OEM to associate a different subroutine from the piercing process.

When activating a piercing process, the CNC executes the following subroutine.

Subroutine.	Properties.
Piercing.fst	The CNC executes this subroutine each time a piercing process is activated (#PIERCING).

This subroutine is saved in the folder ..\MTB\Sub. Since it is an OEM subroutine with en .fst extension, with the CNC in USER mode, the CNC loads it into the RAM when starting up the application. With the CNC in SETUP mode (tuning), the CNC loads this subroutine into the RAM the first time it executes it inside the program.

In the two first installation options, the CNC copies the subroutines to ..\MTB\sub, if they do not already exist. In the third installation option, "rename previous version and install it completely", the CNC always updates the subroutines.

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

After running M02, M30 or after a reset, the CNC maintains the selected piercing type and its related variables.

17.1 Activating the piercing operation.

The instruction #PIERCING executes its associated subroutine (by default, Piercing.fst) and activates the piercing operation. After executing this instruction, the laser pierces the sheet with the machining conditions defined by the piercing type selected on the technological tables.

Programming.

The piercing type may be defined when programming this instruction. Optionally, local parameters and variables can be started in the subroutine Piercing.fst.

Programming format.

The programming format is the following; the arguments appear between curly brackets and the optional ones between angle brackets.

```
#PIERCING
#PIERCING [T{type}]
#PIERCING [<T{type}><,P0={value} ... P25={value}> <,V.P.var_name={value}>
<,V.S.var_name={value}>]
```

T{type}	Optional (by default, the last programmed value or 1 if there are none). Piercing type, defined on the active technological table. Units: From 1 to 10.
P0={value} ... P25={value}	Optional. Start the local parameters of the subroutine.
V.P.var_name={value}	Optional. User local variables.
V.S.var_name={value}	Optional. User global variables.

```
#PIERCING
#PIERCING [T4]
#PIERCING [T4, P1=12, P20=45, V.P.POWER=100]
#PIERCING [P0=1, P1=12]
```

Piercing type.

The piercing parameters are defined on the technological tables, where up to 10 different piercing types can be defined for each type of material. The programming of the piercing type is optional; if it is not programmed, the CNC assumes the last programmed value. If no value is programmed, the first on the technological table is activated (T1). If programmed with a 0 value, the CNC will display the corresponding error message.

Starting local parameters in the subroutine Piercing.fst.

This instruction allows the starting of local parameters in the subroutine Piercing.fst. The parameters may be defined in two ways: using the parameter numbers P0-P25 or the letters A-Z (except "Ñ" and "Ç"), "A" for P0 and "Z" for P25. Both ways of setting local parameters are equivalent and may be combined in the same block. The parameter P19 corresponds to the letter T (piercing type).

The CNC has the following variables to consult the local parameters used in calling the subroutine.

Variable.	Meaning.
(V.)C.PCALLP_A ... (V.)C.PCALLP_Z	Parameter programmed in the call for a subroutine or in instructions #CUTTING and #PIERCING. Units: 0 = No; 1 = Yes.
(V.)G.CALLP	Local parameters programmed in the call for a subroutine or in #CUTTING and #PIERCING instructions. Units: Bits; the bit with the least weight (bit 0) corresponds to parameter P0, bit 1 to parameter P1, and so on, successively. Each bit indicates whether the parameter is programmed (bit=1) or not (bit=0).

Starting variables in the subroutine Piercing.fst.

This instruction allows the user to call the subroutine Piercing.fst using the user variables as call parameters, The remaining variables, including those on the technological tables, cannot be used as call parameters.

Variable.	Meaning.
(V.)P.name	User local variables. These variables maintain their value in local and global subroutines called upon from the program.
(V.)S.name	User global variables. These variables maintain their value between programs and after a reset.

If no parameter or variable is programmed, the subroutine record the value programmed in the previous call. Since they are global parameters, the CNC does not manage call nesting with these variables.

```
#PIERCING [T1 V.P.POWER=100]
    The user variable maintains its value in later calls.
```

Considerations.

If after executing this instruction there is no active material, the CNC will display the corresponding error.

17.2 Subroutine associated with the piercing.

The instruction #PIERCING has an associated subroutine (by default, Piercing.fst), defined by the OEM (machine parameter PIERCING).

Start variables and execute the subroutine Piercing.fst.

Each time the CNC executes the #PIERCING instruction, it updates the piercing variables ((V.)TTPIR.name) with the values that correspond to the selected type, as indicated in the technological tables. After updating the variables, the CNC executes the Piercing.fst subroutine.

The PLC PIERCING mark.

The OEM must manage this mark from the subroutine associated with the piercing (by default, Piercing.fst). At the start of the subroutine (or when the OEM deems it necessary), the OEM must activate the PIERCING mark (assigning it the value 1) to indicate to the PLC that it must start the piercing operation. At the end of the subroutine (or when the OEM deems it necessary), the OEM must deactivate this mark (assigning it the value 0) to indicate to the CNC that it has finished the piercing operation.

- The instruction #PLC makes it possible to manage a PLC mark from the part or subroutine program, without interrupting the block preparation.
- This mark shows the state of the variable V.PLC.PIERCING. The OEM can also use this variable in the subroutine associated with the piercing (by default, Piercing.fst) to modify the status of this mark. Entering this variable interrupts the block preparation.

17.3 Variables associated with the piercing operation.

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W). The reading of these variables does not stop the preparation of blocks, except as otherwise indicated.

Variable.	PRG	Meaning.
(V.)[ch].G.PIERCINGTYPE	R	Piercing type.
(V.)PLC.PIERCING	R/W(*)	There is an ongoing piercing operation. Units: 0 = No; 1 = Yes.
(V.)C.PCALLP_A ... (V.)C.PCALLP_Z	R	Parameter programmed in the call for a subroutine or in instructions #CUTTING and #PIERCING. Units: 0 = No; 1 = Yes.
(V.)G.CALLP	R	Local parameters programmed in the call for a subroutine or in #CUTTING and #PIERCING instructions. Units: Bits; the bit with the least weight (bit 0) corresponds to parameter P0, bit 1 to parameter P1, and so on, successively. Each bit indicates whether the parameter is programmed (bit=1) or not (bit=0).
(V.)P.var_name	R/W	User local variables. These variables maintain their value in local and global subroutines called upon from the program.
(V.)S.var_name	R/W	User global variables. These variables maintain their value between programs and after a reset.

(*) The variable returns the execution value; reading it interrupts block preparation.

Syntax of the variables.

·ch· Channel number.

V.[2].G.PIERCINGTYPE	Piercing type. Channel ·2·.
V.PLC.PIERCING	
V.C.PCALLP_A	Parameter "A".
V.C.PCALLP_F	"F" parameter.
V.G.CALLP	
V.P.POWER	Local user variables.
V.S.POWER	Global user variables.

17.

PIERCING (INITIAL PIERCING).

Variables associated with the piercing operation.

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

CUTTING (SHEET CUTTING FOLLOWING A PROFILE).

18

The cutting operation is the sheet cutting process following a profile defined by blocks G1, G2 and G3, with the machining conditions defined by the cutting type selected on the technological tables. Normally, the cutting process is preceded by an initial boring or piercing process. On the technological tables, each cutting type is associated with a color for graphic representation.

Subroutines associated with cutting.

The cutting command is associated with two subroutines (Cuttingon.fst and Cuttingoff.fst) that the CNC executes before and after the command. The subroutines supplied by Fagor provide basic management of the cutting process, so they should be configured appropriately by the OEM.

The OEM is responsible for ensuring that the subroutine has all the safety aspects related to handling the laser.

The subroutines associated with cutting are defined in the machine parameters CUTTINGON (by default, Cuttingon.fst) and CUTTINGOFF (by default, Cuttingoff.fst). These parameters allow the OEM to associate different subroutines to the cutting process.

When activating or canceling a piercing process, the CNC executes the following subroutines.

Subroutine.	Properties.
Cuttingon.fst	The CNC executes this subroutine each time a cutting process is activated (#CUTTING ON).
Cuttingoff.fst	The CNC executes this subroutine each time a cutting process is canceled (#CUTTING OFF).

These subroutines are saved in the folder ..\MTB\Sub. Since they are OEM subroutines with a .fst extension, with the CNC in USER mode, the CNC loads these subroutines into the RAM when starting up the application. Being the CNC in SETUP mode, the CNC loads these subroutines into RAM memory the first time it executes them inside the program.

In the first two installation options, the CNC copies the subroutine to ..\MTB\sub, but only if it does not already exist. In the third installation option, "rename previous version and install it completely", the CNC always updates the subroutines.

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

After executing M02, M30 or after a reset, the CNC maintains the selected cutting type and its associated variables.

18.1 Activating the cutting operation.

The instruction #CUTTING ON allows the activation of the cutting operation. After executing this instruction, the laser cuts the sheet following the profile defined by blocks G1, G2 and G3, with the machining conditions defined by the cutting type selected on the technological tables.

Programming.

The cutting type can be programmed when programming this instruction. Optionally, local parameters and variables can be started in the subroutine Cuttingon.fst.

Programming format.

The programming format is the following; the arguments appear between curly brackets and the optional ones between angle brackets.

```
#CUTTING ON
#CUTTING ON [T{type}]
#CUTTING ON [<T{type}><,P0={value} ... P25={value}> <,V.P.var_name={value}>
<,V.S.var_name={value}>]
```

T{type}	Optional (by default, the last programmed value or 1 if there are none). Cutting type, defined on the active technological table. Units: From 1 to 10.
P0={value} ... P25={value}	Optional. Start the local parameters of the subroutine.
V.P.var_name={value}	Optional. User local variables.
V.S.var_name={value}	Optional. User global variables.

```
#CUTTING ON
#CUTTING ON [T4]
#CUTTING ON [T4, P1=12, P20=45, V.P.POWER=100]
#CUTTING ON [P0=1, P1=12]
```

Cutting type.

The cutting parameters are defined on the technological tables, where up to 10 different cutting types can be defined for each type of material. The programming of the cutting type is optional; if it is not programmed, the CNC assumes the last programmed value. If no value is programmed, the first on the technological table is activated (T1). If programmed with a 0 value, the CNC will display the corresponding error message.

Start local parameters in the subroutine Cuttingon.fst.

This instruction allows the starting of local parameters in the subroutine Cuttingon.fst. The parameters may be defined in two ways: using the parameter numbers P0-P25 or the letters A-Z (except "Ñ" and "Ç"), "A" for P0 and "Z" for P25. Both ways of setting local parameters are equivalent and may be combined in the same block. The parameter P19 corresponds to the letter T (cutting type).

The CNC has the following variables to consult the local parameters used in calling the subroutine.

Variable.	Meaning.
(V.)C.PCALLP_A ... (V.)C.PCALLP_Z	Parameter programmed in the call for a subroutine or in instructions #CUTTING and #PIERCING. Units: 0 = No; 1 = Yes.
(V.)G.CALLP	Local parameters programmed in the call for a subroutine or in #CUTTING and #PIERCING instructions. Units: Bits; the bit with the least weight (bit 0) corresponds to parameter P0, bit 1 to parameter P1, and so on, successively. Each bit indicates whether the parameter is programmed (bit=1) or not (bit=0).

Start variables in the subroutine Cuttingon.fst.

This instruction allows the user to call the subroutine Cuttingon.fst using the user variables as call parameters, The remaining variables, including those on the technological tables, cannot be used as call parameters.

Variable.	Meaning.
(V.)P.name	User local variables. These variables maintain their value in local and global subroutines called upon from the program.
(V.)S.name	User global variables. These variables maintain their value between programs and after a reset.

If no parameter or variable is programmed, the subroutine record the value programmed in the previous call. Since they are global parameters, the CNC does not manage call nesting with these variables.

```
#CUTTING ON [T1, V.P.POWER=100]
```
The user variable maintains its value in later calls.

Considerations.

If after executing this instruction there is no active material, the CNC will display the corresponding error.

18.2 Canceling the cutting operation.

The #CUTTING OFF instruction deactivates cutting. Cutting is also deactivated after a reset or an M30.

Programming.

Program the instruction alone in the block.

Programming format.

The programming format is:

#CUTTING OFF

```
#CUTTING OFF
```

18.3 Subroutines associated with cutting.

The instruction #CUTTING ON and #CUTTING OFF have an associated subroutine (by default, Cuttingon.fst / Cuttingoff.fst), defined by OEM (machine parameters CUTTINGON / CUTTINGOFF).

Start the variables and execute the subroutines.

Each time the CNC executes the #CUTTING ON instruction, it updates the cutting variables ((V.)TTCUT.name) with the values corresponding to the selected type, as indicated in the technological tables. After updating the variables, the CNC executes the Cuttingon.fst subroutine

Each time the CNC executes the instruction #CUTTING OFF, it executes the subroutine Cuttingoff.fst.

The PLC CUTTING mark.

The OEM must manage this mark from the subroutines associated with cutting (by default, Cuttingon.fst / Cuttingoff.fst). At the start of the subroutine Cuttingon.fst (or when the OEM deems it necessary), the OEM must activate the CUTTING mark (assigning it a value of 1) to indicate to the PLC that it should start the cutting operation. At the end of the Cuttingoff.fst subroutine (or when the OEM deems it necessary), the OEM must deactivate this mark (assigning it a value of 0) to indicate to the CNC that the cutting operation has finished.

- The instruction #PLC makes it possible to manage a PLC mark from the part or subroutine program, without interrupting the block preparation.
- This mark shows the state of the variable V.PLC.CUTTING. The OEM can also use this variable in the subroutines associated with cutting (by default, Cuttingon.fst / Cuttingoff.fst) to modify the status of this mark. Entering this variable interrupts the block preparation.

18.4 Variables associated with the cutting operation.

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W). The reading of these variables does not stop the preparation of blocks, except as otherwise indicated.

Variable.	PRG	Meaning.
(V.)[ch].G.CUTTINGTYPE	R	Cutting type.
(V.)PLC.CUTTING	R/W(*)	There is an ongoing cutting operation. Units: 0 = No; 1 = Yes.
(V.)C.PCALLP_A ... (V.)C.PCALLP_Z	R	Parameter programmed in the call for a subroutine or in instructions #CUTTING and #PIERCING. Units: 0 = No; 1 = Yes.
(V.)G.CALLP	R	Local parameters programmed in the call for a subroutine or in #CUTTING and #PIERCING instructions. Units: Bits; the bit with the least weight (bit 0) corresponds to parameter P0, bit 1 to parameter P1, and so on, successively. Each bit indicates whether the parameter is programmed (bit=1) or not (bit=0).
(V.)P.var_name	R/W	User local variables. These variables maintain their value in local and global subroutines called upon from the program.
(V.)S.var_name	R/W	User global variables. These variables maintain their value between programs and after a reset.

(*) The variable returns the execution value; reading it interrupts block preparation.

Syntax of the variables.

·ch· Channel number.

V.[2].G.CUTTINGTYPE	Cutting type. Channel ·2·.
V.PLC.CUTTING	
V.C.PCALLP_A	Parameter "A".
V.C.PCALLP_F	"F" parameter.
V.G.CALLP	
V.P.POWER	Local user variables.
V.S.POWER	Global user variables.

18.

CUTTING (SHEET CUTTING FOLLOWING A PROFILE).

Variables associated with the cutting operation.

FAGOR

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

SYNCHRONIZED SWITCHING (GRID PATTERN CUTTING)

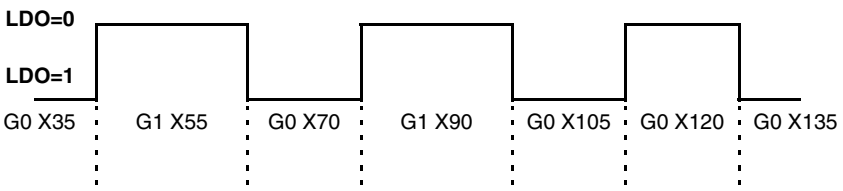
19

Synchronized switching refers to the process to control a local digital output of the CNC depending on the type of movement programmed on the axes. This feature is valid for all kinds of corner G7, G60, G5, G61, G50 or HSC that is programmed.

- The selected digital output is activated when switching from G0 to G1/G2/G3.
- The selected digital output is canceled when switching from G1/G2/G3 to G0.

This way, since it is possible to make the signal status depend on the type of tool path, this feature may be used for various applications such as laser cutting systems or others that require a digital output signal synchronized with the type of tool path.

The following diagram shows the state of the local digital output (LDO) depending on programmed G0-to-G1 transitions or vice versa. The digital output associated with synchronized switching is defined in machine parameter SWTOUTPUT.



The CNC does not allow working with synchronized switching (#SWTOUT) if the gap control (#GAPCTRL) or the leapfrog (#LEAP) are active. While leapfrog is active, if the leap movements on XY are in G0, the CNC will deactivate the synchronized switching.

19.1 Activate synchronized switching.

The instruction #SWTOUT ON may be used to activate synchronized switching. After executing this instruction, a G0-to-G1/G2/G3 transition activates the associated digital output and a G1/G2/G3-to-G0 transition cancels it. Once the digital output is activated, it stays active until a transition to G0 occurs or M30, reset is executed or synchronized switching is canceld (#SWTOUT OFF) PLC signals STOP and _FEEDHOL also cancel the digital output; when these signals disappear, if the digital output was active before, it activates it again.

Programming.

When defining this instruction, it is optionally possible to define an offset (in time or distance) to anticipate or delay the activation or cancellation of the digital output.

Programming format.

The programming format is the following; the arguments appear between curly brackets and the optional ones between angle brackets.

```
#SWTOUT ON
[<TON={time}>,<TOF={time}>,<PON={distance}>,<POF={distance}>, <G0/G1>]
```

TON={time} TON{time}	Optional (by default, 0). Time shift to anticipate or delay the activation of the digital output. Units: Milliseconds.
TOF={time} TOF{time}	Optional (by default, 0). Time shift to anticipate or delay the deactivation of the digital output. Units: Milliseconds.
PON={distance} PON{distance}	Optional (by default, 0). Distance shift to anticipate or delay the activation of the digital output. Units: mm or inches.
POF={distance} POF{distance}	Optional (by default, 0). Distance shift to anticipate or delay the deactivation of the digital output. Units: mm or inches.
G0/G1	Optional (by default, the last programmed value or the machine parameter SWTG0FEED if there is none). Type of feedrate for sections in G0.

```
#SWTOUT ON
#SWTOUT ON [TON=50, TOF=40, G0]
#SWTOUT ON [TON50, TOF40]
#SWTOUT ON [PON=0.3]
```

Offsets (time or distance) to anticipate or delay the activation or deactivation of the digital output.

Parameters TON, TOF, PON and POF are optional; it is possible to program all of them, some or none of them and in any order. A positive value of these parameters anticipates the activation or deactivation of the digital output and a negative value delays it. The "=" sign may be left out.

On CNC power-up, the offset wears are set to zero. After power-up, the values programmed for the offsets (using the instruction #SWTOUT or the variables) are maintained even after an error, a reset or an M30.

Type of feedrate for sections in G0.

This command establishes whether the sections in G0 are executed at a fast feedrate (parameter G00FEED) or if the feedrate is kept constant; i.e., the feedrate of the previous block.

Value.	Meaning.
G0	Movements in G0 are carried out at a rapid feedrate (parameter G00FEED).
G1	The cut is made at a constant feedrate (with the feedrate of the previous G1/G2/G3), without accelerating in the sections in G0. This option makes it possible to prevent possible problems with machine frequency excitation, thus producing better quality precision and surface cuts.

The programming of this command is optional; if not programmed, the CNC assumes the last programmed value or the machine parameter value SWTG0FEED if there is no previous value. If according to the parameter or the command (#SWTOUT) the G0s are executed as G1 (constant feedrate), G0 is displayed in the history.

When it is necessary to anticipate or delay the activation or deactivation of the digital output.

Depending on the feedrate, the cycle time, the type of servo system used, etc. it might be necessary to program the offset values TON, TOF, PON, POF so the switch-on/off of the output can take place at the desired point.

For example, when observing that the system used in the application is switching on too late, either set a positive TON value to anticipate the activation by that amount of time or a positive PON value to anticipate the activation by that distance. If on the other hand, the system used in the application is switching on too early, either set a negative TON value to delay the activation by that amount of time or a negative PON value to delay the activation by that distance. Proceed the same way for switch-off, but using offsets TOF and POF.

19.2 Cancel synchronized switching.

The instruction #SWTOUT OFF cancels synchronized switching. Synchronized switching is also canceled after a reset or an M30.

To ensure that the switching is deactivated correctly, the last block before the deactivation instruction (#SWTOUT OFF) must be a movement in G0.

Programming.

Program the instruction alone in the block.

Programming format.

The programming format is:

#SWTOUT OFF

```
#SWTOUT OFF
```

19.3 Variables related to synchronized switching.

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W). Reading these variables interrupts block preparation.

Variable.	PRG	Meaning.
(V.)[ch].G.TON	R/W	Time shift to anticipate or delay the activation of the digital output. Units: Milliseconds.
(V.)[ch].G.TOF	R/W	Time shift to anticipate or delay the deactivation of the digital output. Units: Milliseconds.
(V.)[ch].G.PON	R/W	Distance shift to anticipate or delay the activation of the digital output. Units: mm or inches.
(V.)[ch].G.POF	R/W	Distance shift to anticipate or delay the deactivation of the digital output. Units: mm or inches.
(V.)[ch].G.LASEROTMON	R	Time remaining (0.5 µs cycles) to activate the laser output. Units: 0.5 µs cycles.
(V.)[ch].G.LASEROTMOFF	R	Time remaining (0.5 µs cycles) to deactivate the laser output. Units: 0.5 µs cycles.

Syntax of the variables.

·ch· Channel number.

V.2.G.TON	Channel ·2·.
V.[2].G.LASEROTMON	Channel ·2·.

The value of these variables is equivalent to the one programmed in parameters TON, TOF, PON and POF of the instruction #SWTOUT. When executing the instruction without parameters and the variables have a value assigned to it, the CNC assumes these latter values as active. Likewise, when programming the parameters of the instruction, the variables assume these values as their own.

To modify the offset values without interrupting block preparation, change these values from the PLC or use the parameters of the instruction #SWTOUT. If these variables are changed from the PLC, the new values are assumed when executing the instruction #SWTOUT. If the instruction is active when changing the values from the PLC, it assumes the new values.

19.

SYNCHRONIZED SWITCHING (GRID PATTERN CUTTING)

Variables related to synchronized switching.

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

PWM (PULSE-WIDTH MODULATION).

20

Pulse width modulation, known as PWM, may be used to control a duty cycle of the Laser signal and therefore modify the power of the Laser. One of the applications of the PWM is using the Laser like a drill by applying and removing power to it very fast.

Activating the PWM via the PLC

The PWM may be activated from the PLC (PWMON mark) and from the part-program. Activating the PWM via PLC has priority over activating it from the CNC.

PWM status from the CNC.	PWM status from the PLC.	PWM status.
Off.	Off.	PWM off.
Active.	Off.	PWM active from the CNC.
Off.	Active.	PWM active from the PLC.
Active.	Active.	PWM active from the PLC.

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

The behavior of the PWM after executing M02 or M30 or after a reset depends on the machine manufacturer's definitions (general parameter PWMCANCEL).

20.1 Activate the PWM.

The #PWMOUT ON instruction may be used to activate the PWM. This function interrupts block preparation. The PWM may be activated from any channel, but only the channel that activates it will have control over it. When trying to activate the PWM from a channel when it is already active, the CNC will issue the corresponding error message.

The modifications to the PWM both via program or via PLC are updated (refreshed) without waiting for the PWM cycle in progress to be completed and as continuously as possible with respect to the previous conditions; in other words, it does not wait for the default signal to set to zero or to one in each change.

Programming.

When defining this instruction, there is an option to set the frequency and the percentage of the duty cycle.

Programming format.

The programming format is the following; the arguments appear between curly brackets and the optional ones between angle brackets.

```
#PWMOUT ON [<FREQ/F{freq}>, <DUTY/D{duty}, <PWMBTIME{time}>,
<PWMBEND{status}>]
```

FREQ{freq} F{freq}	Optional (by default, the last programmed value or 0 if there are none). PWM frequency. Units: Between 2 and 5000 Hz.
DUTY{duty} D{duty}	Optional (by default, the last programmed value or 50% if there are none). Percentage of duty cycle. Units: Between 0 and 100%.
PWMBTIME{time}	Optional (by default, 0). Duration of the PWM in burst mode. Units: Milliseconds.
PWMBEND{status}	Optional (by default, 0). PWM status after completing the burst mode. Units: 0 = Inactive; 1 = Active.

```
#PWMOUT ON [FREQ 200, DUTY 50]
#PWMOUT ON [F200, D50]
#PWMOUT ON [FREQ 200, DUTY 25, PWMBTIME 50, PWMBEND 1]
```

PWM frequency.

The frequency is programmed using the FREQ command (or simply F) and may be a value between 2 Hz and 5000 Hz. This command is optional; if not programmed, the CNC assumes the last programmed value or 0 if there is no previous value.

Percentage of duty cycle.

The percentage of duty cycle is programmed using the DUTY program (or simply D) and may have a value between 0.1% and 100%. This command is optional; if not programmed, the CNC assumes the last programmed value or 50 if there is no previous value.

Burst mode.

The burst mode consists in activating the PWM for a set time period (PWMBTIME) and, when done, the PWM output should stay at a set level (PWMBEND).

The PWMBTIME command sets the time the PWM stays active (in ms rounded up to loop units). The PWMBEND command indicates the status in which the PWM stays after the time period set by PWMBTIME; if set to "1", the PWM stays active, but if set to "0" or not programmed, the PWM is turned off.

Remarks.

Program simulation.

The CNC does not activate the PWM in any simulation mode.

Tool inspection.

During tool inspection, the CNC does not deactivate the PWM The OEM may have programmed the PLC to deactivate the PWM during tool inspection and resume the PWM at the end of the inspection.

Starting variables.

On power-up and when canceling the PWM, all the variables and marks of the CNC and PLC recover their initial values.

Execution channels.

When there is a CNC error in the channel that has the PWM active (or in any channel if it is managed via PLC), the CNC behavior depends on the machine manufacturer's definitions (general parameter PWMCANCEL). Once the PWM is deactivated for some error, that PWM will not be reactivated when the error disappears; therefore, they have to be reactivated either by program or via PLC.

PWMCANCEL	Meaning.
Yes.	The CNC cancels the PWM.
No.	The CNC keeps the status of the output, i.e., it does not cancel the output if it is active.

20.2 Cancel the PWM.

The #PWMOUT OFF instruction may be used to deactivate the PWM. On power-up and when canceling the PWM, all the variables and marks of the CNC and PLC recover their initial values.

Programming.

Program the instruction alone in the block.

Programming format.

The programming format is:

#PWMOUT OFF

```
#PWMOUT OFF
```

20.3 Variables related to the PWM.

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W). Reading these variables does not interrupt block preparation. These variables only work when the Laser is active via CNC.

Variable.	PRG	Meaning.
(V.)G.PWMON	R	PWM status, when it has been activated via CNC. Units: 0 = Inactive; 1 = Active.
(V.)G.PWMFREQ	R	PWM frequency, when the PWM has been activated via CNC. Units: Herz.
(V.)G.PWMDUTY	R	PWM duty cycle, when the PWM has been activated via CNC. Units: Percentage. The reading from the PLC will be given in tenths (x10); i.e. if the parameter value is ·10·, the PLC reading will return a value of ·100·.
(V.)G.PWMBTIME	R	Duration of the PWM in burst mode. Units: Milliseconds.
(V.)G.PWMBEND	R	PWM status after completing the burst mode. Units: 0 = Inactive; 1 = Active.

Syntax of the variables.

```
V.G.PWMON
V.G.PWMFREQ
V.G.PWMDUTY
V.G.PWMBTIME
V.G.PWMBEND
```

POWER CONTROL BASED ON FEED.

21

The CNC permits the control of the power and/or the duty, based on the feed on the path of the X Y and Z axes. The same programming command allows power control using the analog output associated with the spindle or the PWM duty, where both options can be activated simultaneously. The CNC uses the real or theoretical feed rate, as specified by the OEM (general machine parameter PWRCTRLACT). If RTCP is active, the CNC will consider the movement of the nozzle tip.

To control the PWM duty, the "PWM" software option is required.

Activating power control via PLC

The PLC can activate the power control by executing the corresponding CNC block with the CNCEX command. The instruction executed from the PLC updates the instruction programmed from the CNC and vice-versa.

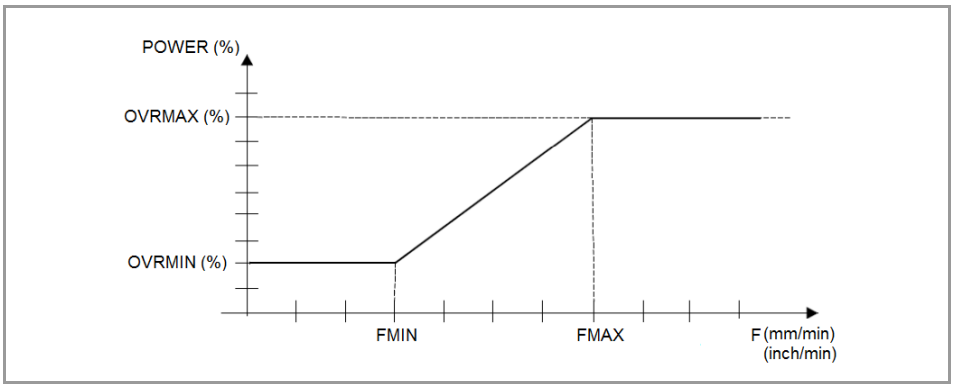
The PLC can enable and disable the power control with the ENABLEPWROUT mark. This mark is active by default. If the PLC deactivates this mark, the CNC disable the power control, setting the maximum defined in the instruction #PWRCTRL ON.

The PLC can enable and disable the duty control with the ENABLEPWRDUTY mark. This mark is active by default. If the PLC deactivates this mark, the CNC disable the power control, setting the maximum defined in the instruction #PWRCTRL ON.

21.1 Activating power control.

The #PWRCTRL ON instruction permits the power and/or duty control to be activated (both options can be active at the same time). This instruction sets the minimum (OVRMIN) and maximum (OVRMAX) power/duty percentages to be applied to the minimum (FMIN) and maximum (FMAX) feeds between which the power/duty is proportional to the feed.

- If the feed is below the minimum value set, the CNC applies the minimum power/duty percentage.
- If the feed is between the minimum and maximum values set (both included), the power/duty percentage is proportional to the feed.
- If the feed is above the maximum value set, the CNC applies the maximum power/duty percentage.



Programming.

When defining this instruction, define the output type (analog or duty), the minimum and maximum percentages, and the minimum and maximum feeds.

Programming format.

The programming format is the following; the arguments appear between curly brackets and the optional ones, between angle brackets.

```
#PWRCTRL ON [<DUTY/D>/<OUT/O,> <OVRMIN{%},> <OVRMAX{%},>
<FMIN{feed},> < FMAX{feed}>]
```

DUTY	Optional. PWM duty control.
D	This command is optional only if there is a type of #PWRCTRL previously activated (OUT/O or DUTY/D); if both or none are active, the programming is required.
OUT	Optional. Power control.
O	This command is optional only if there is a type of #PWRCTRL previously activated (OUT/O or DUTY/D); if both or none are active, the programming is required.
OVRMIN{%}	This is optional if there is an active #PWRCTRL value (by default, the last value programmed). Minimum power (if OUT) or duty (if DUTY) percentage. Units: Percentage.
OVRMAX{%}	This is optional if there is an active #PWRCTRL value (by default, the last value programmed). Maximum power (if OUT) or duty (if DUTY) percentage. Units: Percentage.
FMIN{feed}	This is optional if there is an active #PWRCTRL value (by default, the last value programmed). Feed below which the CNC applies the OVRMIN percentage. Units: In millimeters/minute or inches/minute.
FMAX{feed}	This is optional if there is an active #PWRCTRL value (by default, the last value programmed). Feed above which the CNC applies the OVRMAX percentage. Units: In millimeters/minute or inches/minute.

```
#PWRCTRL ON [DUTY, OVRMIN 50, OVRMAX 100, FMIN 230, FMAX 570]
#PWRCTRL ON [O, OVRMIN 80, OVRMAX 110, FMIN 100, FMAX 1000]
#PWRCTRL ON [D, OVRMIN V.TTCUT.PWROVRMIN, OVRMAX V.TTCUT.PWROVRMAX,
FMIN V.TTCUT.PWRFMIN, FMAX V.TTCUT.PWRFMAX]
```
(The instruction takes the values defined on the active technological table).

Power control via an analog output.

OUT command (O, in simplified form). This command is optional only if there is a type of #PWRCTRL previously activated (OUT/O or DUTY/D); if both or none are active, the programming is required.

When programming OUT, the parameters OVRMIN and OVRMAX set the programmed power percentage to be applied to the defined ramp. This power is programmed using command "S" (in watts) and can be modified with the spindle override.

The CNC defines the ramp while taking into account the programmed power. If the spindle override is increased or decreased, the change will affect the resulting power, but always within the limits set by OVRMIN and OVRMAX. The CNC does not take into account the values defined in the spindle parameters LOSPDLIM and UPSPDLIM.

PWM duty control.

DUTY command (D, in simplified form). This command is optional only if there is a type of #PWRCTRL previously activated (OUT/O or DUTY/D); if both or none are active, the programming is required.

When programming DUTY, the parameters OVRMIN and OVRMAX set the programmed duty percentage to be applied to the defined ramp. If the spindle override changes, it will affect the output power, and only the limits set in the spindle parameters LOSPDLIM and UPSPDLIM will be considered.

Feedrate.

The CNC uses the real or theoretical feed rate, as specified by the OEM (general machine parameter PWRCTRLACT).

Considerations.

Execution channels.

The power control may be activated from any channel, but only the channel that activates it will have control over it. When trying to activate the power control from a channel when it is already active in another channel, the CNC will issue the corresponding error message. The CNC allows the activation of a #PWRCTRL applied to OUT with another applied to DUTY in different channels.

The variables and block preparation.

The variables assume the values programmed in the instruction #PWRCTRL ON during block preparation. If the PLC modifies these variables during the execution of a program, the CNC maintains the values calculated during block preparation.

The execution and preparation should be synchronized to assume the changes in the PLC. If the machine parameter PWRCTRLCANCEL=No, the CNC performs the synchronization at the start of the execution, after a reset and when entering the tool inspection.

Spindle speed override.

When #PWRCTRL ON [OUT], the spindle override affects the active power, but the latter will never be lower than OVRMIN or greater than OVRMAX; that is, the power is maintained within the OVRMIN - OVRMAX range without taking into account the spindle parameters LOSPDLIM and UPSPDLIM.

When #PWRCTRL ON [DUTY], the spindle override affects the active power, within the limits set for the spindle parameters LOSPDLIM and UPSPDLIM.

Signals _STOP and _FEEDHOL of the PLC.

The signals _STOP y _FEEDHOL of the PLC do not affect the power control. If the PLC stops the axes, the CNC continues to take the minimum power set in #PWRCTRL until the PLC manages it.

Tool inspection.

The tool inspection does not deactivate the power control. The OEM may have programmed the PLC to cancel the power control when entering inspection and activate it when resuming the program at the end of the inspection.

Program simulation.

The CNC does not activate the power control in any simulation mode.

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

The instruction #PWRCTRL is modal. The CNC cancels the power control on power-up. The way the power control will behave after executing an M02 or M30 or after a reset depends on the setting of machine parameter PWRCTRLCANCEL.

Value.	Meaning.
Yes.	The CNC cancels the power control. A channel error cancels the power control. If an error disables the power control, it will not be re-enabled when the error disappears. It should be programmed again via the program or PLC.
No	The CNC does not cancel the power control. A channel error does not cancel the power control; this will maintain its previous status. At the beginning of the execution, after a reset and upon accessing the tool inspection, the CNC synchronizes the preparation of blocks with the execution.

21.2 Canceling the power control.

The instruction #PWRCTRL OFF deactivates both power controls (analog (OUT) and duty (DUTY)) and sets the power at the value defined by OVRMAX. If, in this status, a new power is programmed, the CNC will not take into account the OVRMAX limit, but rather the values defined in the spindle parameters LOSPDLIM and UPSPDLIM.

The way the power control will behave after executing an M02 or M30 or after a reset depends on the setting of machine parameter PWRCTRLCANCEL.

Programming.

When defining this instruction, it is possible to select the output type (analog or duty); if no output is selected, the CNC will deactivate both of them.

Programming format.

The programming format is the following; the optional arguments appear between angle brackets.

```
#PWRCTRL OFF
#PWRCTRL OFF [<DUTY/D>/<OUT/O>]
```

DUTY	Optional. PWM duty control.
D	This command is only optional if there is a type of #PWRCTRL active.
OUT	Optional. Power control.
O	This command is only optional if there is a type of #PWRCTRL active.

```
#PWRCTRL OFF [DUTY]
#PWRCTRL OFF [D]
#PWRCTRL OFF [OUT]
#PWRCTRL OFF [O]
#PWRCTRL OFF
```

Power or duty control.

The OUT (or simplified to O) and DUTY (simplified to D) commands are only optional if there is a type of #PWRCTRL active; if both are active and no command is programmed, the CNC deactivates both.

21.3 Variables associated with the power control.

Power control via an analog output.

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W). Reading these variables does not interrupt block preparation.

Variable.	PRG	Meaning.
(V.)G.PWROUTON	R	Power control status. This variable is also affected by the mark ENABLEPWROUT of the PLC. Units: 0 = Inactive; 1 = Active.
(V.)G.PWROUTOVRMIN	R	Minimum percentage to be applied. Units: Percentage.
(V.)G.PWROUTOVRMAX	R	Maximum percentage to be applied. Units: Percentage.
(V.)G.PWROUTFMIN	R	Feed below which the CNC applies the OVRMIN percentage to the power. Units: In millimeters/minute or inches/minute.
(V.)G.PWROUTFMAX	R	Feed above which the CNC applies the OVRMIN percentage to the power. Units: In millimeters/minute or inches/minute.

These variables are activated when the #PWRCTRL command is programmed from the program. When the power control is activated from the PLC (CNCEX command), these variables do not update their value (except (V.)G.PWROUTON).

These variables can be read and written from the PLC (except (V.)G.PWROUTON). If the PLC modifies these variables during the execution of a program, the CNC maintains the values calculated during block preparation.

Syntax of the variables.

```
V.G.PWROUTON
V.G.PWROUTOVRMIN
V.G.PWROUTOVRMAX
V.G.PWROUTFMIN
V.G.PWROUTFMAX
```

Power control via the PWM duty.

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W). Reading these variables does not interrupt block preparation.

Variable.	PRG	Meaning.
(V.)G.PWRDUTYON	R	Duty control status. This variable is also affected by the mark ENABLEPWRDUTY of the PLC. Units: 0 = Inactive; 1 = Active.
(V.)G.PWRDUTYOVRMIN	R	Minimum percentage to be applied. Units: Percentage.
(V.)G.PWRDUTYOVRMAX	R	Maximum percentage to be applied. Units: Percentage.
(V.)G.PWRDUTYFMIN	R	Feed below which the CNC applies the OVRMIN percentage to the power. Units: In millimeters/minute or inches/minute.
(V.)G.PWRDUTYFMAX	R	Feed above which the CNC applies the OVRMIN percentage to the power. Units: In millimeters/minute or inches/minute.

These variables are activated when the #PWRCTRL command is programmed from the program. When the power control is activated from the PLC (CNCEX command), these variables do not update their value (except (V.)G.PWRDUTYON).

These variables can be read and written from the PLC (except (V.)G.PWRDUTYON). If the PLC modifies these variables during the execution of a program, the CNC maintains the values calculated during block preparation.

Syntax of the variables.

V.G.PWRDUTYON
V.G.PWRDUTYOVRMIN
V.G.PWRDUTYOVRMAX
V.G.PWRDUTYFMIN
V.G.PWRDUTYFMAX

21.

POWER CONTROL BASED ON FEED.

Variables associated with the power control.

FAGOR AUTOMATION

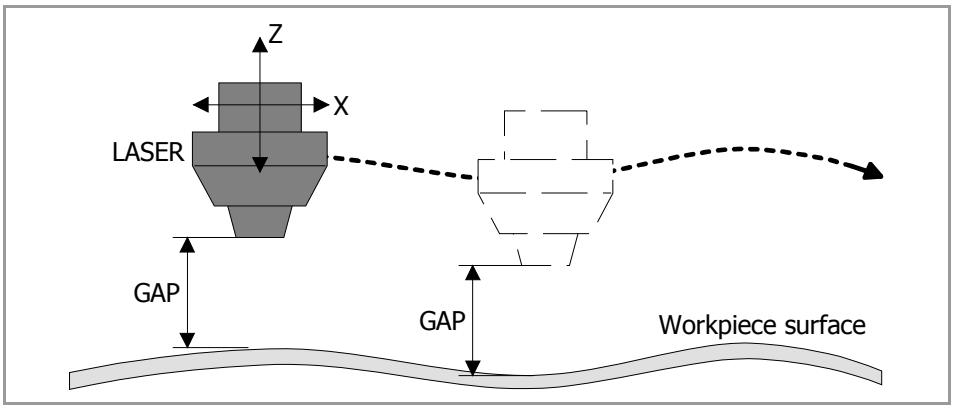
CNC 8060 L
CNC 8070 L

(REF: 1707)

GAP CONTROL.

22

Gap control makes it possible to maintain a set distance between the laser nozzle and the surface of the sheet. This distance is calculated by a sensor connected to the CNC, so that it offsets the sensor variations on the distance programmed with additional movements in the axis programmed for the gap.



The CNC allows only one gap control. When trying to activate the gap control from a channel when it is already active in another channel, the CNC will issue the corresponding error message.

For the gap control, the software option "GAP CONTROL" should be available.

The CNC does not permit working with gap control (#GAPCTRL) and synchronized switching (#SWTOUT) at the same time.

Activating gap control via PLC.

The PLC can activate the gap control by executing the corresponding CNC block with the CNCEX command. The instruction executed from the PLC updates the instruction programmed from the CNC and vice-versa.

22.1 Activating gap control.

The #GAPCTRL ON instruction may be used to activate the gap control. The CNC allows a second gap control to be programmed without the need to deactivate the first one.

After executing this instruction, the axis associated with the sensor moves in programmed feed (if there is no programmed feed, the feed defined in MAXFEED), until detecting the part with the sensor or the programmed coordinate (POS). The CNC considers that the approach block is complete when the sensor reaches the gap with the tolerance defined in the GAPTOL machine parameter. Once the part has been detected, the CNC will maintain the defined gap.

During the approach movement, the CNC is able to stop the movement (signals _STOP o _FEEDHOL). In this case, the axis will remain stationary and without gap control until the movement is resumed.

Programming.

When defining this instruction, the axis to be controlled and the gap to be maintained must be defined. As an option, the approach coordinate may be defined.

Programming format.

The programming format is the following; the arguments appear between curly brackets and the optional ones, between angle brackets.

```
#GAPCTRL ON [<X..C> / <X..C{pos}> <, GAP{gap}>]
```

<X..C>	Optional (by default , the last programmed value). Axis associated with the sensor.
<X..C{pos}>	Optional (by default , the last programmed value). Axis associated with the sensor and approach coordinate. Units: mm or inches.
<GAP{gap}>	Optional (by default , the last programmed value). Distance (gap) to be kept from the surface. Units: mm or inches.

```
#GAPCTRL ON [Z, GAP 1]
#GAPCTRL ON [Z-11, GAP 2]
#GAPCTRL ON [GAP 5]
#GAPCTRL ON [Z-10]
#GAPCTRL ON
```

Axis associated with the sensor and approach coordinate.

When programming the instruction for the first time, the axis programming is required, but afterward it is optional. The approach coordinate is optional can be absolute or incremental, depending on whether the function G90 (absolute) or G91 (incremental) is active.

- If the axis is not programmed, the CNC assumes the last value programmed.

  ```
  #GAPCTRL ON [GAP 5]
  ```

- If only the axis is programmed (without the coordinate), the nozzle lowers until it meets the sheet metal. If the nozzle does not encounter the sheet metal, the movement continues until it reaches the software limit. The feedrate will be that for which the axis can stop at the distance defined by the difference between the GAPDISTLIMIT parameter minus the programmed gap.

  ```
  #GAPCTRL ON [Z, GAP 5]
  ```

- If the axis and the coordinate are programmed, the nozzle lowers until it meets the sheet metal. If the nozzle reaches the programmed coordinate without detecting the sheet metal, the CNC will display the corresponding error.

  ```
  #GAPCTRL ON [Z10, GAP 5]
  ```

Distance to be maintained from the sheet surface.

When programming the instruction for the first time, the gap programming is required. Programming after that is optional. If not programmed, the CNC assumes the last value programmed.

Considerations.

Execution channels.

The gap control may be activated from any channel, but only the channel that activates it will have control over it. When trying to activate the gap control from a channel when it is already active in another channel, the CNC will issue the corresponding error message.

Behavior in case of error.

If there is an error in a channel where gap control is active, the CNC will deactivate the gap control. When the gap control is deactivated by any error, it will not be reactivated when the error disappears.

Program simulation.

The CNC does not activate the gap control in any simulation mode.

Signals _STOP and _FEEDHOL of the PLC.

The signals _STOP and _FEEDHOL of the PLC do not deactivate the gap control, except in the approach movement. In this case, the axis will remain stationary and without gap control until the movement is resumed.

Tool inspection.

The tool inspection does not deactivate the gap control, except when attempting to move the axis associated with the sensor; in this case, the CNC reactivates the gap control when replacing the axis.

Movements in G0 and blocks without movement.

The movements in G0 and blocks without movement (M, G, etc.) do not deactivate the gap control.

Jog, MDI/MDA and reference search.

If the gap control is active, the axis associated with the sensor cannot be moved in jog or MDI/MDA and home searches cannot be performed; in these cases, the CNC will show the corresponding warning message.

Retrace.

During retrace, the CNC maintains the gap control until it finds its activation block (#GAPCTRL ON).

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

After executing an M02 or M30, and after an emergency or a reset, the CNC deactivates the gap control. On power-up and when canceling the gap control, all variables and marks of the PLC recover their initial values.

22.2 Canceling gap control.

The #GAPCTRL OFF instruction disables the gap control. On power-up and when canceling the gap control, all variables and marks of the PLC recover their initial values. It is not necessary to deactivate one gap control before activating another.

Programming.

Program the instruction alone in the block.

Programming format.

The programming format is:

#GAPCTRL OFF

```
#GAPCTRL OFF
```

22.3 Variables associated with the gap control.

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W). The reading of these variables does not stop the preparation of blocks, except as otherwise indicated.

Variable.	PRG	Meaning.
(V.)G.GAPCTRLON	R(*)	Gap control status. Units: 0 = Inactive; 1 = Active.
(V.)[ch].G.GAPAXISPOS	R	Programmed approach coordinate. Units: mm or inches.
(V.)[ch].G.GAPPRG	R	Programmed gap (in the instruction #GAPCTRL or #LEAP). Units: mm or inches.
(V.)G.GAP	R(*)	Actual gap value, detected by the sensor. Units: mm or inches.
(V.)[ch].A.GAPCOMP.xn	R(*)	Compensation introduced by gap control into the axis associated with the sensor to maintain the gap. This variable should be used only in the sensor adjustment process. Units: mm or inches.

(*) The variable returns the execution value; reading it interrupts block preparation..

Syntax of the variables.

·ch· Channel number.

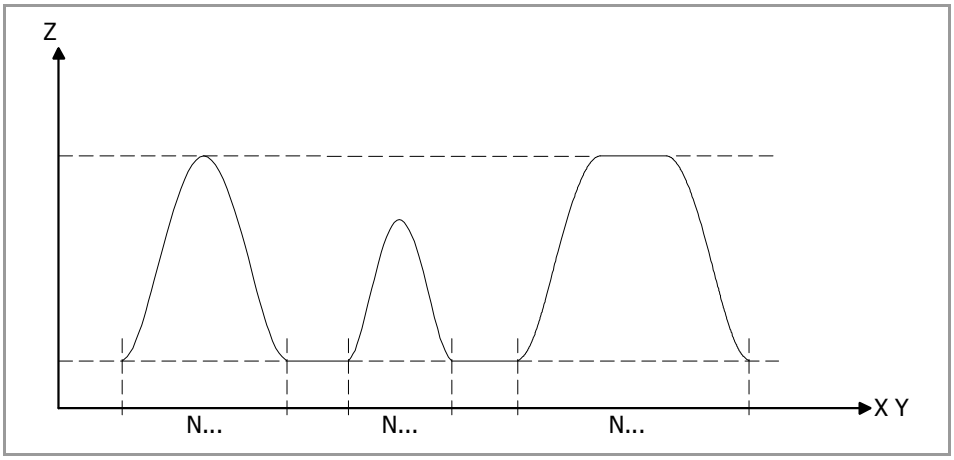
·xn· Name, logic number or index of the axis.

```
V.G.GAPCTRLON
V.[2].G.GAPAXISPOS          Channel ·2·.
V.[2].G.GAPPRG              Channel ·2·.
V.G.GAP
V.[2].A.GAPCOMP.xn          Channel ·2·.
```

LEAPFROG.

23

With the leapfrog, during positioning in the XY plane between cutoff operations, the CNC raises the Z axis (longitudinal axis) to place it away from the sheet, thus avoiding collisions due to undulations on the sheet, chips or cut parts protruding from the sheet. The CNC executes the Z movement as fast as possible (respecting the acceleration and jerk of the Z axis), but without affecting the dynamic of the movements on XY. The CNC executes the lowering and raising movement of the axis Z on a single block on XY.



The CNC does not permit operating with leapfrog (#LEAP) and synchronized switching (#SWTOUT) at the same time. While leapfrog is active, if the leap movements on XY are in G0, the CNC will deactivate the synchronized switching.

23.1 Activating the leapfrog.

The instruction #LEAP may be used to activate the PWM. The CNC allows the activation of one leapfrog per channel. The leap is programmed inside the XY motion block and the leap takes place as the block moves.

Programming.

When defining this instruction, the leap coordinates and the maximum permitted height should be defined. Once a #LEAP has been programmed with its associated parameters, they are maintained when the instruction #LEAP is programmed without any parameters and followed by the X Y coordinates

Programming format.

The programming format is the following; the arguments appear between curly brackets and the optional ones between angle brackets.

```
#LEAP [<G90/G91>, <POS{pos}>, <POSLIMIT{pos}>, GAP{pos}, <TIME{time}>]
<X{pos} Y{pos}>
```

G90/G91	Optional (by default, G91). Type of coordinates for the leap coordinate, either on the longitudinal axis, absolute (if G90) or incremental (if G91); this does not affect the movements on X or Y.
POS{pos}	Optional (by default, coordinate at the beginning of the leap). The meaning of this command depends on whether or not the gap control is active before the leap. • If the gap control is not active, the command POS indicates the position of the longitudinal axis (normally the Z axis) at the end of the leap. • If the gap control is active, the axis drops until it reaches the gap above the sheet; if it reaches the defined POS position without detecting the sensor signal, the CNC will display the corresponding error. Units: mm or inches.
POSLIMIT{pos}	Optional (by default , the last programmed value). In the case of G90, the maximum coordinate to be reached by the longitudinal axis (usually Z) during the leap. If G91, maximum leap height from the real starting coordinates. Units: mm or inches.
GAP{pos}	Optional (by default , the last programmed value). Distance (gap) to maintain after leap, when gap control is active (#GAPCTRL). Units: mm or inches.
TIME{time}	Optional (by default, 0). Dwell, moving up the longitudinal axis, before commencing the movement of the XY axes. Units: Seconds.
X{pos} , Y{pos}	Optional. Last point of the leap on the plane. Units: mm or inches.

```
#LEAP [G90, POS10, POSLIMIT50, TIME2] X20 Y30
    (Leap coordinates in absolute coordinates).
    (The final leap coordinate is Z10).
    (The maximum leap coordinate is Z50.)
    (Dwell, moving along the longitudinal axis, before commencing the movement along the XY axes).
#LEAP [G91, POS10, POSLIMIT50] X20 Y30
    (Leap coordinates in incremental coordinates).
    (The final leap coordinate is 10 mm higher than the initial coordinate.)
    (The maximum height of the leap is 50 mm).
#LEAP [POSLIMIT50] X20 Y30
    (Leap coordinates in incremental coordinates).
    (The initial and final leap coordinates on the in Z-axis are equal).
#LEAP [G90, POS10, POSLIMIT50]
    (The CNC memorizes the data but does not skip).
#LEAP X20 Y30
    (The CNC executes the skip with the last values programmed).
```

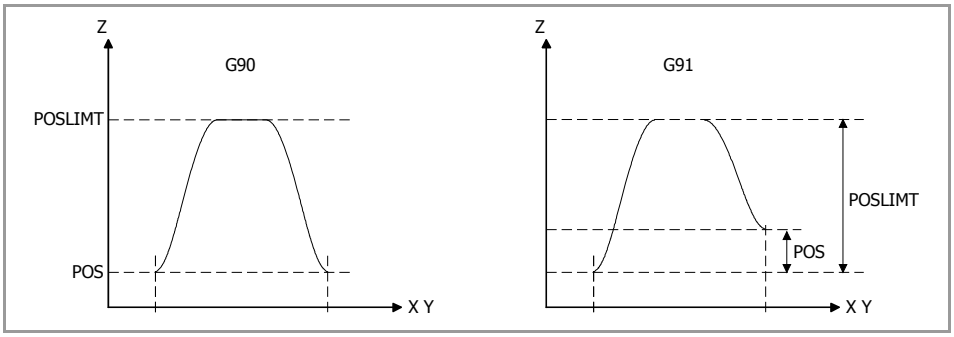
Type of coordinate for the leap coordinates.

The programming of G90/G91 in the instruction only affects the longitudinal axis (usually Z), in this block (POS and POSLIMIT commands). It does not affect movements on X or Y.

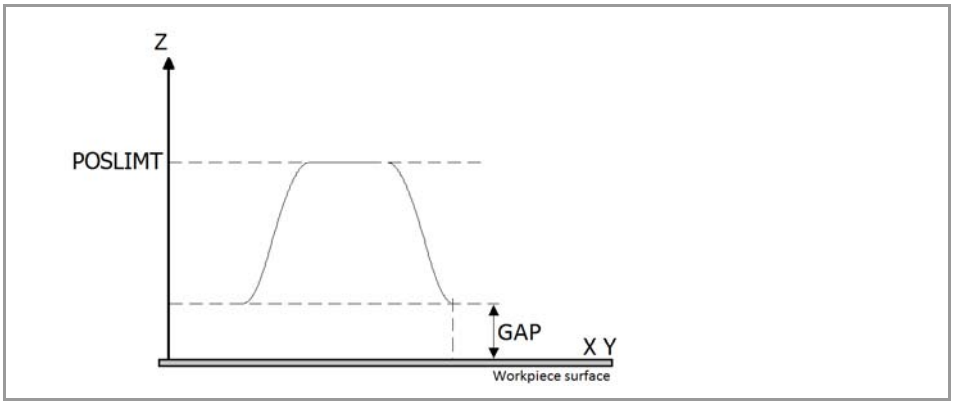
Longitudinal axis position (usually Z) at the end of the leap.

The meaning of this POS command depends on whether or not the gap control (#GAPCTRL ON) is active before the leap.

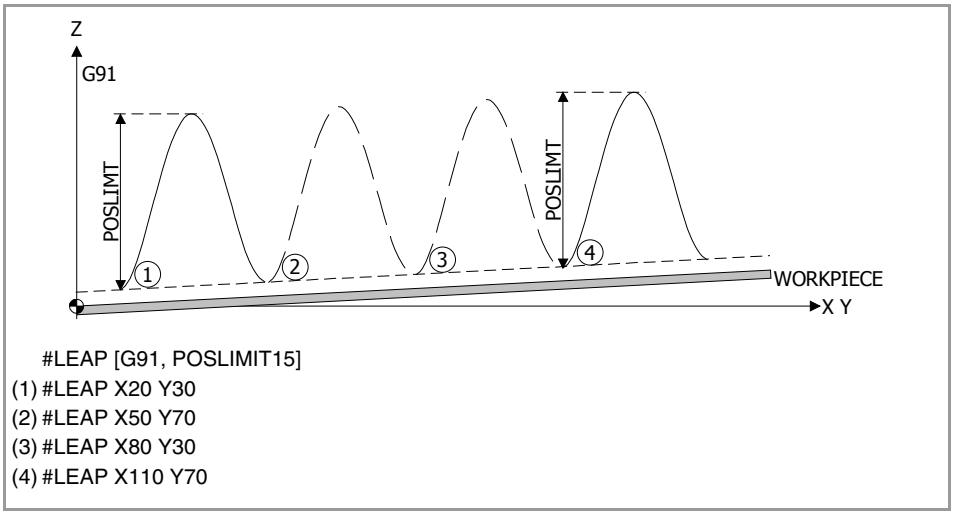
- If the gap control is not active, the command POS indicates the position of the longitudinal axis (normally the Z axis) at the end of the leap.



- If the gap control is active, the axis drops until it reaches the gap above the sheet; if it reaches the defined POS position without detecting the sensor signal, the CNC will display the corresponding error.



If the command #LEAP is in incremental coordinates (G91), the CNC calculates the POS and POSLIMIT coordinates from the real starting position.



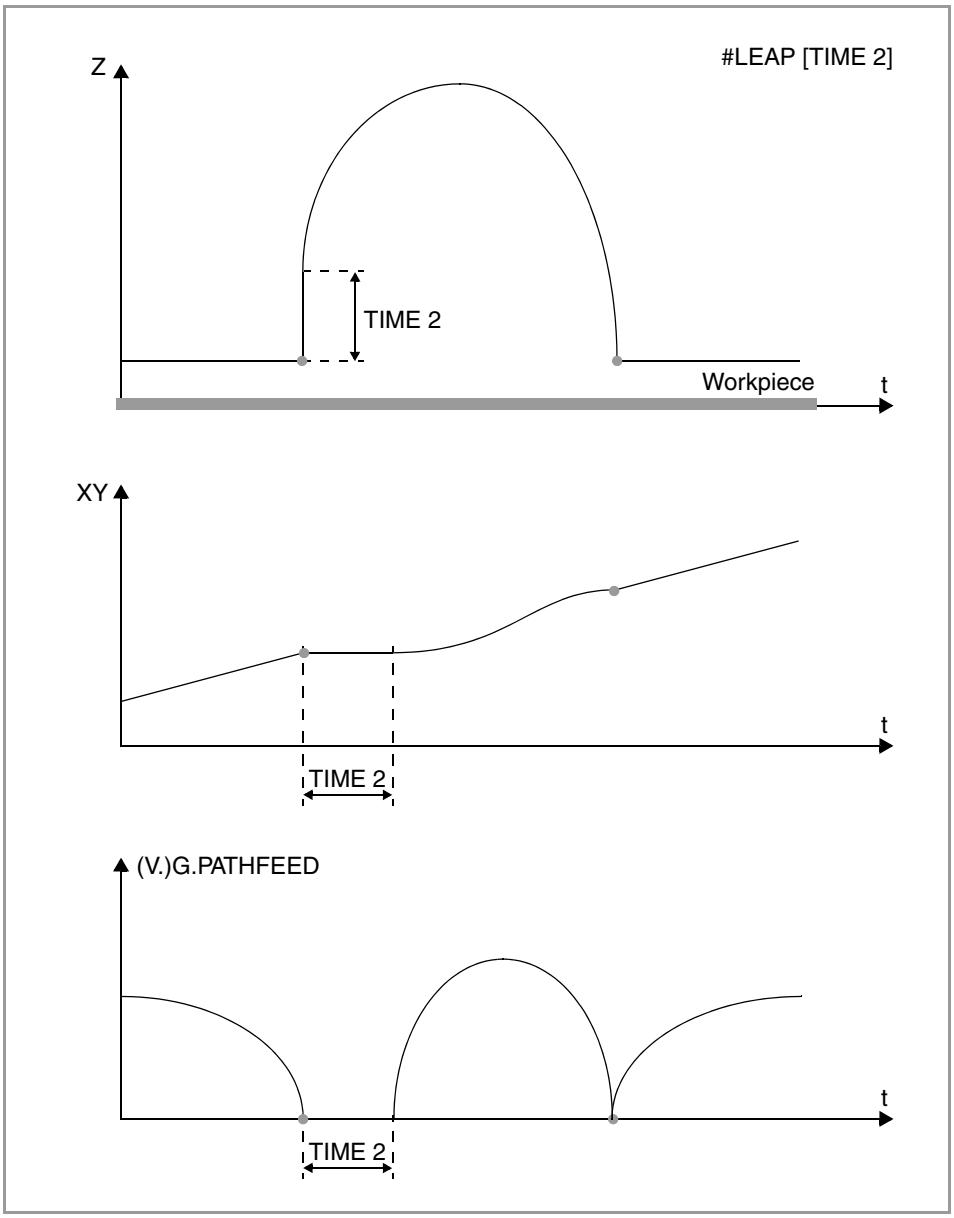
#LEAP [G91, POSLIMIT15]
(1) #LEAP X20 Y30
(2) #LEAP X50 Y70
(3) #LEAP X80 Y30
(4) #LEAP X110 Y70

Maximum coordinate to be reached by the longitudinal axis (usually Z) during the leap.

When programming the instruction #LEAP for the first time, the POSLIMIT programming is required. Due to the leap longitude and the Z-axis dynamics, the axis may not reach this coordinate.

Dwell, moving up the longitudinal axis, before commencing the movement of the XY axes.

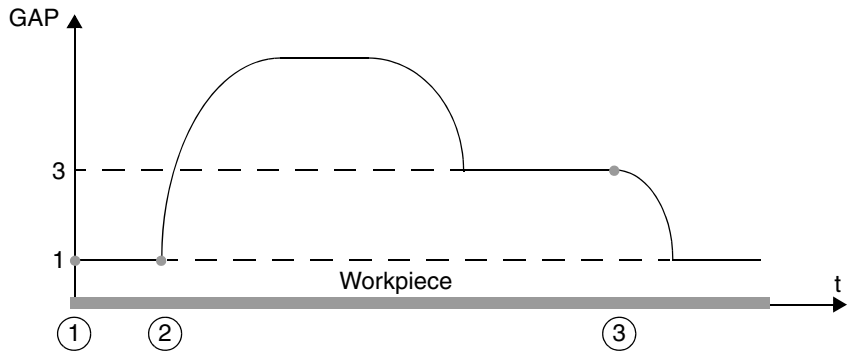
The TIME command indicates the dwell before initiating the movement along the axes of the plane. This dwell provides a minimum retraction of the Z axis to ensure that the leap attains a sufficient Z coordinate. During this time, the longitudinal axis retracts from the sheet; after this time has elapsed, the plane axes start to move along with the Z axis, if it has not yet achieved the programmed coordinate.



Distance (gap) to maintain after leap, when gap control is active (#GAPCTRL).

This parameter must only be programmed while the gap control is active (#GAPCTRL ON). A leap with gap control is slower than without gap control. Likewise, a leap with gap control can mean that lowering is slower than raising.

- If the gap control is active (#GAPCTRL ON), the "GAP" command defines the distance to be maintained after the leap; if this command is not programmed, the CNC will use the active gap. At the end of the leap, the nozzle drops until it sits over the defined gap.
  - If the final position has not been programmed (POS command), the nozzle will lower until the signal is received and position itself over the sheet at a GAP distance.
  - If the final position has been programmed (POS command) and the nozzle reaches this position without detecting the gap sensor signal, the CNC will display the corresponding error.



1 #GAPCTRL ON [GAP 1]
2 #LEAP [GAP 3]
3 #GAPCTRL ON [GAP 1]

Last point of the leap on the plane.

The coordinates of the axes on the X Y plane should go beyond the instruction #LEAP. The CNC does not allow the programming of the longitudinal axis (usually Z) in these axes. If the end point is not programmed, the CNC simply memorizes the skip data. This makes it possible to later program only the end point of the skip and carry out the skip with the saved data.

```
#LEAP [G90, POS10, POSLIMIT50]
    (The CNC memorizes the data but does not skip).
    .
#LEAP X20 Y30
    (The CNC executes the skip with the last values programmed).
    .
#LEAP X120 Y30
    (The CNC executes the skip with the last values programmed).
```

Considerations.

Signals _STOP and _FEEDHOL of the PLC.

The signals _STOP and _FEEDHOL of the PLC also affect the Z-axis when it is synchronized with the movement on XY.

Z-axis feed rate and override.

The programmed feed in the XY block defines the leap profile in the Z-axis. The override also affects the Z-axis when it is synchronized with the movement on XY.

Tool inspection.

The CNC allows the inspection during the leap, but only for movements in manual mode and for execution of functions M, H, F y S; it does not allow access to the MDI/MDA mode.

Travel limits.

The maximum rising coordinate of the longitudinal axis (usually the Z-axis) will be limited by the software limit; if the axis attempts to exceed it, the CNC will show the corresponding error.

Program simulation.

In any simulation level, the CNC will only activate the #LEAP when the movement involves all three X Y Z axes.

Synchronized switching (#SWTOUT).

The leapfrog is compatible with the synchronized switching (#SWTOUT). During the leap, if the movements on XY are in G0, the CNC will deactivate the synchronized switching. When completing the leap, if the following block is not in G0, the CNC will reactivate the synchronized switching through a synchronization.

Retrace.

During the retrace, the axis will be retraced only until the final point of the leap, i.e., there are no leaps during the retrace.

Function properties.

The instruction #LEAP is not modal.

23.2 Variables associated with the leapfrog.

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W). Reading these variables does not interrupt block preparation.

Variable.	PRG	Meaning.
(V.)[ch].G.LEAPON	R	Leapfrog status. Units: 0 = Inactive; 1 = Active.
(V.)[ch].G.LEAPG91	R	Type of coordinates for the leap coordinate, absolute (if G90) or incremental (if G91). Units: 0 = G90; 1 = G91.
(V.)[ch].G.LEAPPOS	R	Longitudinal axis position (usually Z) at the end of the leap. Units: mm or inches.
(V.)[ch].G.LEAPPOSLIMIT	R	Maximum coordinate to be reached by the longitudinal axis (usually Z) during the leap. Units: mm or inches.
(V.)[ch].G.LEAPTIME	R	Dwell before the leap. Units: Seconds.
(V.)[ch].G.GAPPRG	R	Programmed gap (in the instruction #GAPCTRL or #LEAP). Units: mm or inches.

Syntax of the variables.

·ch· Channel number.

V.[2].G.LEAPON	Channel ·2·.
V.[2].G.LEAPG91	Channel ·2·.
V.[2].G.LEAPPOS	Channel ·2·.
V.[2].G.LEAPPOSLIMIT	Channel ·2·.

23.

LEAPFROG.

Variables associated with the leapfrog.

FAGOR AUTOMATION

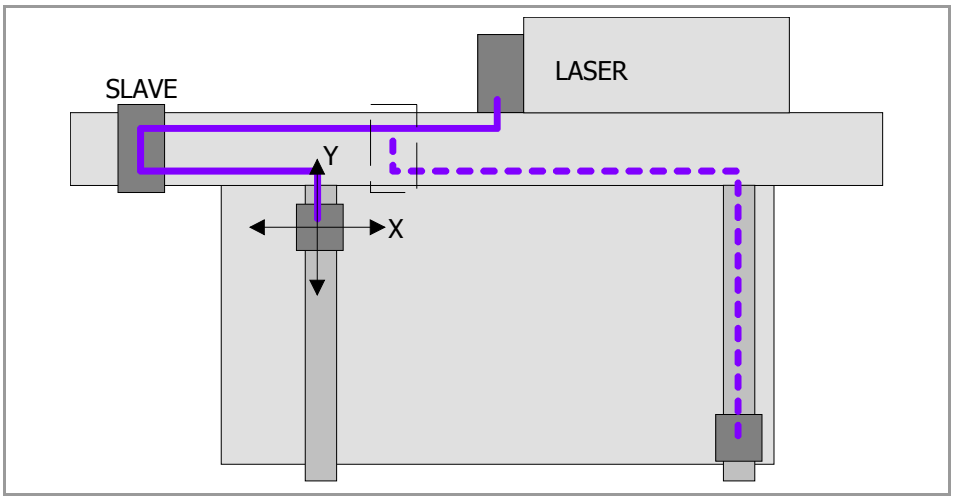
CNC 8060 L
CNC 8070 L

(REF: 1707)

CO2 LASER PATH DISPERSION COMPENSATION.

24

The CNC permits compensation for dispersion on the CO2 laser path or CBDS (Constant Beam Delivery System). To do that, the CNC generates an independent movement in the slave axis, which follows the main XY axes, so that the total laser path is constant, thus avoiding dispersion variations in the cutoff point on the sheet.



The compensation requires the "Synchronism and cams" software option.

Management via PLC.

The PLC can activate and cancel the dispersion compensation by executing the corresponding CNC block with the CNCEX command.

24.1 Activating dispersion compensation.

The command #FOLLOW ON activates the dispersion compensation. This instruction synchronizes an axis (slave axis) with the value of its V.A.FLWMASTER.xn. This variable takes into account the displacement of the three X Y Z axes and calculates the displacement of the slave axis to keep the laser path constant.

The instruction #TFOLLOW is not permitted as the coordinates types (theoretical or actual) are defined by the variable V.A.FLWMASTER.xn. In this case, theoretical coordinates.

The CNC allows the definition of #FOLLOW and axes in the CNC, but only the axis defined in the machine parameters compensate the displacement of the three X Y Z axes For the remaining axes, the value of the variable should be written from the PLC or the part program.

Once the #FOLLOW ON has been programmed, it will only be deactivated using #FOLLOW OFF.

Programming.

When defining this instruction, define the master element as MASTER, the slave axis, the transmission ratio and, optionally, the synchronization type.

Programming format.

The programming format is the following; the arguments appear between curly brackets and the optional ones, between angle brackets.

```
#FOLLOW ON [MASTER, {slave}, N{ratio}, D{ratio}, POS]
```

MASTER	A command to indicate that the slave axis should follow its variable V.A.FLWMASTER.xn instead of a concrete master axis.
{slave}	Name of the slave axis.
N{ratio}	Numerator of the gear ratio.
D{ratio}	Denominator of the gear ratio.
POS	Command to execute a position synchronization.

```
#FOLLOW ON [MASTER, U, N1, D2, POS]
```
The slave axis follows its variable V.A.FLWMASTER.U, synchronized in position (command POS), where the N/Do ratio is defined as ½, as the laser path in axis U is dual.

Synchronizing the slave axis to the variable V.A.FLWMASTER.xn.

Through the command MASTER, the slave axis defined in #FOLLOW is synchronized to its variable V.A.FLWMASTER.xn. The value of the variable is calculated by the CNC or the PLC, depending on how it has been configured by the OEM.

- One of the LASERFOLLOW1, LASERFOLLOW2 or LASERFOLLOW3 parameters has a value other than 0. In this case, the CNC automatically updates its variable V.A.FLWMASTER.xn, taking into account the movement of the plane axes as follows:

  V.A.FLWMASTER.xn = V.MPG.LASERFOLLOWOFFSET +
  V.MPG.LASERFOLLOW1 * V.A.TPOS.X +
  V.MPG.LASERFOLLOW2 * V.A.TPOS.Y +
  V.MPG.LASERFOLLOW3 * V.A.TPOS.Z

- The value of all three of the LASERFOLLOW1, LASERFOLLOW2 and LASERFOLLOW3 parameters is 0. The OEM (through the PLC) or the user (from the part program) should enter the value of the variable V.A.FLWMASTER.xn.

Considerations.

Programming.

- If a FOLLOW instruction has been programmed via the command MASTER to an axis by the machine parameter, and a follow-up of that same axis is programmed by CNC or PLC, the CNC will display the corresponding error message.
- If the slave axis is programmed while #FOLLOW is active, the CNC will display the corresponding error message.

Signals _STOP and _FEEDHOL of the PLC.

The signals _STOP and _FEEDHOL do not deactivate the #FOLLOW.

Tool inspection.

The tool inspection does not deactivate the #FOLLOW. The OEM may have programmed the PLC to cancel the #FOLLOW when entering inspection and activate it when resuming the program at the end of the inspection.

Program simulation.

In the program simulation, the CNC will not activate #FOLLOW. In the simulated execution, the CNC will only activate #FOLLOW if there are movements on the axes involved.

Leapfrog.

The leapfrog will not deactivate the #FOLLOW.

Machine reference zero (home) search.

- It is advisable to activate the compensation (#FOLLOW ON) at power-on and after a machine reference search.
- After the machine reference search of the XYZ axes, when activating the synchronization, the axis automatically displaces the corresponding coordinates, although no movement has been programmed.
- The synchronization cannot be active during the machine reference search. It is advisable to cancel the synchronization (#FOLLOW OFF) at the start of the search subroutine.

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

The instruction #FOLLOW is modal. This function is neither affected by functions M02 and M30 nor by a reset of the CNC. The CNC will deactivate the #FOLLOW after an emergency or error that causes an emergency. When reassembling then machine, the CNC reactivates the synchronization with the first reset; if you do not wish to reactivate it, cancel it from the PLC (command FOLLOW OFF).

24.2 Canceling dispersion compensation.

The instruction #FOLLOW OFF deactivates the synchronization of the slave axis and, thus, the compensation.

Programming.

Program the instruction alone in the block. When defining this instruction, the axes to be canceled should be defined.

Programming format.

The programming format is the following; the arguments appear between curly brackets.

```
#FOLLOW OFF [{slave}]
```

{slave} Name of the slave axis.

```
#FOLLOW OFF [U]
```

24.3 Associated variables.

The following variables may be accessed via part-program or via MDI/MDA mode. Each of them indicates whether it may be read (R) or written (W). Reading these variables interrupts block preparation.

Variable.	PRG	Meaning.
(V.)[ch].A.FLWMASTER.xn	R/W	Variable that acts as a master in the synchronization. The CNC synchronizes the slave axis with this variable. Units: mm or inches.

Syntax of the variables.

·ch· Channel number.

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

```
V.A.FLWMASTER.Z          Z axis.
V.A.FLWMASTER.4          Axis with logic number ·4·.
V.[2].A.FLWMASTER.1      Axis with index ·1· in the channel ·2·.
```

Remarks.

When the CNC uses this variable to compensate the laser dispersion, the CNC updates it in each position loop.

If the CNC uses this variable for other applications, by entering its valor from the PLC, the update will be defined by the PLC times. If you wish to set the update for each loop, program the PLC to enter each loop or use a periodic period subroutine equal to the cycle time of the CNC (parameter LOOPTIME). In these cases, after reaching the slave axis, the initial synchronization attempts to follow the variable values in each loop. The axis follows the variable without any filtering process, so if these values are not filtered when the PLC enters them, the slave axis may move abruptly.

PROGRAMMING EXAMPLE.

25

A CNC program is formed by a set of blocks. Each block contains all functions or commands required to execute an operation.

#MATERIAL [Steel_25]		Selection of material.
#INITIALSUB		Executing the initial subroutine
F30000		Feedrate.
X213 Y24		Positioning.
#PIERCING	PROFILE 1	Piercing operation (executing the associated subroutine). First, the CNC updates the piercing variables with the values of the technological parameters of the piercing type selected (T1 to T10); frequency, duty, power, etc. After that, the CNC executes the associated subroutine.
#CUTTING ON [T5]		Starting the cutting operation (associated subroutine). First, the CNC updates the cutting variables with the values of the technological parameters of the cutting type selected (T1 to T10); frequency, duty, power, etc. After that, the CNC executes the associated subroutine.
G41		Tool radius compensation.
G01 X-42.059 Y2.625		Profile.
.		
.		
#CUTTING OFF		Ending the cutting operation (associated subroutine). The CNC executes the associated subroutine.
G40		
#LEAP [POSLIMIT 100] X35 Y37		Positioning the laser in the following piercing.
#PIERCING	PROFILE 2	
#CUTTING ON [T1]		
G41		
G02 X-0.2 Y-0.2 I-0.2 J0		
.		
.		
#CUTTING OFF		
G40		
#LEAP X-79.9 Y16.8		
#PIERCING	PROFILE 3	
#CUTTING ON [T10]		
G41		
G02 X-0.2 Y-0.2 I-0.2 J0		
.		
.		
#CUTTING OFF		
G40		
#FINALSUB		Executing the final subroutine.
M30		End of program.

25.

PROGRAMMING EXAMPLE.

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

CNC VARIABLES.

26

26.1 Understanding how variables work.

Accessing the variables.

The internal CNC variables may be accessed from the part program, MDI/MDA, PLC and from any application or external interface (for example FGUIM). Each variable must indicate whether it can only be read or read and written.

Access the variables from a part-program. Accessing during execution or during block preparation.

The CNC reads several blocks ahead of the one being executed in order to calculate in advance the path to follow. This prior reading is known as "block preparation".

The CNC checks certain variables during block preparation and others during execution. The variables that use the execution value interrupt block preparation temporarily and the CNC resumes it when it is done reading/writing the variable. Accessing the variables from the PLC or from an external interface never interrupts block preparation.

Be careful with the variables that interrupt block preparation because when they are inserted between machining blocks with compensation may cause undesired profiles. Interrupting block preparation may result in compensated paths different from the one programmed, undesired joints when working with very short moves, etc.

In any case, it is possible to use the #FLUSH instruction to force the evaluation of a variable when it is being executed. This instruction interrupts block preparation in advance, executes the last prepared blocs, synchronizes the preparation and execution of blocks and then goes on with the execution of the program and block preparation.

Accessing the variables. from the PLC. Synchronous or asynchronous access.

PLC access to the variable, both for reading and writing, may be either synchronous or asynchronous. A synchronous access is resolved immediately whereas an asynchronous access takes several PLC cycles to resolve.

The tool variables will be read asynchronously when the tool is neither the active one nor in the magazine. The tool variables will be written asynchronously whether the tool is the active one or not.

Example of how to access asynchronous variables.
Reading of the radius value of offset ·1· of tool ·9· when it is not in the tool magazine.

```
<condition> AND NOT M11 = CNCRD (TM.TORT.[9][1], R11, M11)
```

The PLC activates mark M11 when the operation begins and keeps it active until the operation ends.

```
DFD M11 AND CPS R11 EQ 3 = ···
```

It waits for the consultation to end before evaluating the data.

Example of how to access synchronous variables.

Reading the actual (real) feedrate.

```
<condition> = CNCRD (G.FREAL, R12, M12)
```

The PLC activates mark M12 when the operation begins and keeps it active until the operation ends.

```
CPS R12 GT 2000 = ···
```

There is no need to wait for consulting the data because the synchronous variables are resolved immediately.

It resets the clock enabled by the PLC with the value contained in register R13.

```
<condition> = CNCWR (R13, PLC.TIMER, M13)
```

Accessing the variables. from the PLC. Accessing numeric variables

When the PLC accesses numeric variables that can have decimals, the values will be given in PLC units.

FAGOR
FAGOR AUTOMATION
CNC 8060 L
CNC 8070 L
(REF: 1707)

26.1.1 Accessing numeric variables from the PLC.

When the PLC accesses numeric variables that can have decimals, the values will be given as follows.

- The coordinates will be given in ten-thousandths if they are in mm or hundred-thousandths if they are in inches.

Units.	Reading from the PLC.
1 millimeter.	10000.
1 inch.	100000.
1 degree.	10000.

- The feedrate of the axes is given in tenth-thousandths if mm or in hundred-thousandths if inches.

Units.	Reading from the PLC.
1 millimeter/minute.	10000.
1 inch/minute.	100000.
1 degree/minute.	10000.

- The spindle speed will be given in ten-thousandths.

Units.	Reading from the PLC.
G97. 1 rpm.	10000.
G96. 1 meter/minute.	10000.
G96. 1 foot/minute.	10000.
G192. 1 rpm.	10000.
M19. 1 degree/minute.	10000.

- The percentages will be given with the real value, in tenths or in hundredths depending on the variable. If not indicated otherwise, the PLC will read the actual value. If not so, it will indicate if the variable will be read in tenths (x10) or in hundredths (x100).

Units.	Reading from the PLC.
1 %.	1.
1 % (x10).	10.
1 % (x100).	100.

- Time will be given in thousandths.

Units.	Reading from the PLC.
1 second.	1000.

- Voltage will given as follows. The variables associated with the machine parameter table return the actual value (in millivolts). For the rest of the variables (in volts), the reading will appear in ten-thousandths.

Units.	Reading from the PLC.
1 volt.	10000.

- The pressures will be expressed in thousandths.

Units.	Reading from the PLC.
1 bar.	1000.
1 psi	1000.

- The power will be expressed in ten thousandths.

Units.	Reading from the PLC.
1 watt.	10000.

26.2 Variables in a single-channel system.

The generic mnemonic associated with the variables is written as follows.

(V.){prefix}.{variable}
(V.){prefijo}.{variable}.{eje/cabezal}

The –V.– indicator.

Programming the -V.- indicator depends on where the variable is used. To access the variables from the part-program or in MDI/MDA mode, the mnemonic begins with the indicator -V.- Accessing the variables from the PLC or from an external interface requires NOT using the -V.- indicator.

All mnemonics in this manual show this indicator as (V.), indicating that it must only be programmed when necessary.

Mnemoni.	Part-program. MDI/MDA mode.	PLC. External interface.
(V.)MPG.NAXIS	V.MPG.NAXIS	MPG.NAXIS

Prefixes of the variables.

The prefix must always be programmed. Prefixes make it possible to easily identify the group the variable belongs to.

Prefix.	Meaning.
A	Axis and/or spindle variables.
C	Canned cycle or subroutine calling parameters.
E	Interface related variables.
G	General variables.
MPA	Variables related to axis and/or spindle machine parameters.
MPG	Variables related to general machine parameters.
MPK	Variables related to kinematic machine parameters.
MPM	Variables related to machine parameters for M functions.
MPMAN	Variables related to machine parameters for JOG mode.
MTB	Variables related to OEM machine parameters.
P	User local variables.
PLC	PLC related variables.
S	User global variables.
SP	Spindle related variables.
TM	Variables related to tools or tool magazines.

Axis and spindle variables.

Axis and spindle variables are identified with the prefix –A.–. When these variables refer to a spindle, they may also be accessed with the prefix –SP.–.

(V.)A.{variable}.{axis/spindle}
(V.)SP.{variable}.{spindle}
(V.)SP.{variable}

The variables of the axis machine parameters (–MPA.– prefix) can also be accessed using the –SP– prefix when referring to a spindle.

(V.)MPA.{variable}.{axis/spindle}
(V.)SP.{variable}.{spindle}
(V.)SP.{variable}

Identifying the axes and the spindles in the variables.

In these variables one must indicate which axis or spindle they refer to. The axis may be referred to by its name or logic number; the spindle may be referred to by its name, logic number or index in the spindle system.

In these variables one must indicate which axis or spindle they refer to. In variables with the prefix –A.– and –MPA.–, the axes and the spindles are identified with their name or logic number. In variables with the prefix –SP.–, the spindles are identified with their name or spindle index. If no spindle is selected in the variables with -SP.- prefix, the variable refers to the master spindle.

Mnemoni.	Meaning when the variable is executed by the part-program, the MDI/MDA mode or the PLC.
V.MPA.variable.Z V.A.variable.Z	Z axis.
V.MPA.variable.S V.A.variable.S V.SP.variable.S	Spindle S.
V.MPA.variable.4 V.A.variable.4	Axis or spindle with logic number ·4·.
V.SP.variable.2	Spindle with index ·2· in the system.
V.SP.variable	Master spindle.

Mnemoni.	Meaning when the variable is executed by an external interface.
V.MPA.variable.Z V.A.variable.Z	Z axis.
V.MPA.variable.S V.A.variable.S V.SP.variable.S	Spindle S.
V.MPA.variable.4 V.A.variable.4	Axis with logic number ·4·.
V.SP.variable.2	Spindle with index ·2· in the system.
V.SP.variable	Master spindle.

The logic number of the axes is determined by the order in which they have been defined in the machine parameter table (AXISNAME). The first axis of the table will be logic axis -1- and so on.

The logic number of the spindles is determined by the order in which they have been defined in the machine parameter table (NAXIS + SPDLNAME). The logic numbering of the spindles continues from the last logic axis; hence, in a 5-axis system, the first spindle of the table will be logic spindle ·6· and so on.

The index of a spindle in the system is determined by the order in which they have been defined in the machine parameter table (SPDLNAME). The index of the first spindle of the table will be ·1· and so on.

AXISNAME	SPDLNAME	Logic order.	Index of the spindle in the system.
AXISNAME 1		Logic number 1.	
AXISNAME 2		Logic number 2.	
AXISNAME 3		Logic number 3.	
AXISNAME 4		Logic number 4.	
AXISNAME 5		Logic number 5.	
	SPDLNAME 1	Logic number 6.	Index 1.
	SPDLNAME 2	Logic number 7.	Index 2.

Variables of the master spindle.

In a multi-spindle system, the master spindle is the main spindle, the one receiving the commands when no specific spindle is mentioned. In a single-spindle channel, that will be its master spindle.

The variables of the master spindle are identified with the prefix –SP.– but without indicating the spindle. These variables may be used to access the data of the master spindle without knowing its name or logic number. These variables are meant for displaying data and programming cycles.

26.3 Variables in a multi-channel system.

The generic mnemonic associated with the variables is written as follows.

(V.)[channel].{prefix}.{variable}
(V.)[canal].{prefix}.{variable}.{axis/spindle}

The –V.– indicator.

Programming the -V.- indicator depends on where the variable is used. To access the variables from the part-program or in MDI/MDA mode, the mnemonic begins with the indicator -V.- Accessing the variables from the PLC or from an external interface requires NOT using the -V.- indicator.

All mnemonics in this manual show this indicator as (V.), indicating that it must only be programmed when necessary.

Mnemoni.	Part-program. MDI/MDA mode.	PLC. External interface.
(V.)[2].MPG.NAXIS	V.[2].MPG.NAXIS	[2].MPG.NAXIS

Programming the channel.

Programming the channel makes it possible to access the variables of the channel itself or another channel from a channel. The first channel is identified with the number 1, "0" is not a valid number.

Programming the channel number is optional; if not programmed, it works as follows depending on who executes the variable. The following table does not apply to the axis and spindle variables.

Where they are executed.	Meaning when no channel has been programmed.
Part-program. MDI/MDA mode.	Channel that is executing the variable.
PLC	First channel or main channel.
External interface.	Active channel.

Prefixes of the variables.

The prefix must always be programmed. Prefixes make it possible to easily identify the group the variable belongs to.

Prefix.	Meaning.
A	Axis and/or spindle variables.
C	Canned cycle or subroutine calling parameters.
E	Interface related variables.
G	General variables.
MPA	Variables related to axis and/or spindle machine parameters.
MPG	Variables related to general machine parameters.
MPK	Variables related to kinematic machine parameters.
MPM	Variables related to machine parameters for M functions.
MPMAN	Variables related to machine parameters for JOG mode.
MTB	Variables related to OEM machine parameters.
P	User local variables.
PLC	PLC related variables.

Prefix.	Meaning.
S	User global variables.
SP	Spindle related variables.
TM	Variables related to tools or tool magazines.

Axis and spindle variables.

Axis and spindle variables are identified with the prefix –A.–. When these variables refer to a spindle, they may also be accessed with the prefix –SP.–.

(V.)[channel].A.{variable}.{axis/spindle}
(V.)[channel].SP.{variable}.{spindle}
(V.)[channel].SP.{variable}

The variables of the axis machine parameters (–MPA.– prefix) can also be accessed using the –SP– prefix when referring to a spindle.

(V.)[channel].MPA.{variable}.{axis/spindle}
(V.)[channel].SP.{variable}.{spindle}
(V.)[channel].SP.{variable}

Identifying the axes and the spindles in the variables.

In these variables one must indicate which axis or spindle they refer to. In variables with the prefix –A.– and –MPA.–, the axes and the spindles are identified with their name, logic number or index in the channel. In variables with the prefix –SP.–, the spindles are identified with their name, index in the channel or spindle index. If no spindle is selected in the variables with -SP.- prefix, the variable refers to the master spindle.

Mnemoni.	Meaning when the variable is executed by the part-program, the MDI/MDA mode or the PLC.
V.MPA.variable.Z V.A.variable.Z	Z axis.
V.MPA.variable.S V.A.variable.S V.SP.variable.S	Spindle S.
V.MPA.variable.4 V.A.variable.4	Axis or spindle with logic number ·4·.
V.[2].MPA.variable.4 V.[2].A.variable.4	Axis with index ·4· in the channel ·2·.
V.SP.variable.2	Spindle with index ·2· in the system.
V.[2].SP.variable.1	Spindle with index ·1· in the channel ·2·.
V.SP.variable	Master spindle of the channel. If the variable is executed by the PLC, spindle master of the first channel.
V.[2].SP.variable	Master spindle of channel ·2·.

Mnemoni.	Meaning when the variable is executed by an external interface.
V.MPA.variable.Z V.A.variable.Z	Z axis.
V.MPA.variable.S V.A.variable.S V.SP.variable.S	Spindle S.
V.MPA.variable.4 V.A.variable.4	Axis with logic number ·4·.
V.[2].MPA.variable.4 V.[2].A.variable.4	Axis with index ·4· in the channel ·2·.
V.SP.variable.2	Spindle with index ·2· in the active channel.
V.[2].SP.variable.1	Spindle with index ·1· in the channel ·2·.
V.SP.variable	Master spindle of the active channel.
V.[2].SP.variable	Master spindle of channel ·2·.

When referring to the axis or spindle by its name, programming the channel they are in is not a determining factor; thus, programming them in this case is irrelevant. When programming the channel, if the axis or spindle is not in it, its programming is ignored.

The logic number of the axes is determined by the order in which they have been defined in the machine parameter table (AXISNAME). The first axis of the table will be logic axis -1- and so on.

The logic number of the spindles is determined by the order in which they have been defined in the machine parameter table (NAXIS + SPDLNAME). The logic numbering of the spindles continues from the last logic axis; hence, in a 5-axis system, the first spindle of the table will be logic spindle ·6· and so on.

The index of a spindle in the system is determined by the order in which they have been defined in the machine parameter table (SPDLNAME). The index of the first spindle of the table will be ·1· and so on.

AXISNAME	SPDLNAME	Logic order.	Index of the spindle in the system.
AXISNAME 1		Logic number 1.	
AXISNAME 2		Logic number 2.	
AXISNAME 3		Logic number 3.	
AXISNAME 4		Logic number 4.	
AXISNAME 5		Logic number 5.	
	SPDLNAME 1	Logic number 6.	Index 1.
	SPDLNAME 2	Logic number 7.	Index 2.

The index of an axis in the channel is determined by the order in which they have been defined in the machine parameter table (CHAXISNAME). The index of the first axis of the table will be ·1· and so on.

The index of a spindle in the channel is determined by the order in which they have been defined in the machine parameter table (CHSPDLNAME). The index of the first spindle of the table will be ·1· and so on.

CHAXISNAME CHSPDLNAME	Index of the axis in the channel.	Index of the spindle in the channel.
CHAXISNAME 1	Index 1.	
CHAXISNAME 2	Index 2.	
CHAXISNAME 3	Index 3.	
CHSPDLNAME 1		Index 1.
CHSPDLNAME 2		Index 2.

Variables of the master spindle.

In a multi-spindle system, the master spindle is the main spindle of the channel, the one receiving the commands when no specific spindle is mentioned. Each channel has a master spindle. In a single-spindle channel, that will always be the master spindle.

The variables of the master spindle are identified with the prefix –SP.– but without indicating the spindle. These variables may be used to access the data of the master spindle without knowing its name or logic number. These variables are meant for displaying data and programming cycles.

Programming the channel number is optional; if not programmed, it works as follows depending on who executes the variable.

Where they are executed.	Meaning when no channel has been programmed.
Part-program. MDI/MDA mode.	Channel that is executing the variable.
PLC	First channel or main channel.
External interface.	Active channel.

26.4 Variables associated with machine parameters.

26.4.1 General machine parameters.

DEFAULT CONDITIONS (SYTEM).

(V.)MPG.PRESSURE

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

Default pressure units.

V.MPG.PRESSURE

Values of the variable.

Value.	Meaning.
0	Bar.
1	Psi.

SYNCHRONIZED SWITCHING.

(V.)MPG.SWTOUTPUT

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

Local digital output associated with synchronized switching.

V.MPG.SWTOUTPUT

(V.)MPG.SWTDELAY

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

Delay of the device associated with synchronized switching.

V.MPG.SWTDELAY

(V.)MPG.SWTG0FEED

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

Observe the G00 feedrate during synchronized switching.

V.MPG.SWTG0FEED

Values of the variable.

Value.	Meaning.
0	No. The cut is made at a constant feedrate (with the feedrate of the previous G1/G2/G3), without accelerating in the G0 sections.
1	Yes. Movements in G0 are carried out at a rapid feedrate (parameter G00FEED).

Remarks.

The behavior can be modified from the part program with the instruction #SWTOUT.

PWM (PULSE-WIDTH MODULATION).

(V.)MPG.PWMOUTPUT

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

Local digital output associated with the PWM.

V.MPG.PWMOUTPUT

Values of the variable.

Value.	Meaning.
0	PWM missing.
1	Local output 1 (pin LI/O1).
2	Local output 2 (pin LI/O2).

(V.)MPG.PWMCANCEL

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

Cancel the PWM after an M30 or a reset.

V.MPG.PWMCANCEL

Values of the variable.

Value.	Meaning.
0	No.
1	Yes.

POWER CONTROL.

(V.)MPG.PWRCTRLACT

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

Managing power based on the actual feed.

V.MPG.PWRCTRLACT

Values of the variable.

Value.	Meaning.
0	No. The CNC manages the power based on the theoretical feed.
1	Yes. The CNC manages the power based on the actual feed.

(V.)MPG.PWRCTRLCANCEL

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

Canceling power control with reset/M2/M30.

V.MPG.PWRCTRLCANCEL

Values of the variable.

Value.	Meaning.
0	No.
1	Yes.

GAP CONTROL.

(V.)MPG.GAPANAINTYPE

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

Type of analog input connected to the sensor.

V.MPG.GAPANAINTYPE

Values of the variable.

Value.	Meaning.
0	Remote CAN.
2	Drive.

(V.)MPG.GAPANAINID

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

Number of the analog input connected to the sensor.

V.MPG.GAPANAINID

Values of the variable.

Value.	Meaning.
1 - 60	The analog input is in the remote modules. The variable returns the number of the analog output.
101 -132 201 - 232	The analog input is in a Sercos drive. The first digit indicates the analog output to be used (1 or 2) and the other two digits indicate the logic address of the drive (1 through 32).

(V.)MPG.GAPDISTLIMIT

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

Distance corresponding to the limit analog signal of the sensor.

V.MPG.GAPDISTLIMIT

(V.)MPG.GAPVOLTLIMIT

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

Voltage (in millivolts) corresponding to the limit path of the sensor (parameter GAPDISTLIMIT).

V.MPG.GAPVOLTLIMIT

(V.)MPG.GAPSENSOROFFSET

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

Offset (in millivolts) to apply to the sensor from the CNC.

V.MPG.GAPSENSOROFFSET

(V.)MPG.GAPSENSORCH

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

Change the sensor signal sign.

V.MPG.GAPSENSORCH

Values of the variable.

Value.	Meaning.
0	No.
1	Yes.

(V.)MPG.GAPGAIN

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

Proportional gain applied to the signal sensor, in position.

V.MPG.GAPGAIN

(V.)MPG.GAPAPPROACHDYN

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

Dynamic response during the final part of the movement approaching the plate.

V.MPG.GAPAPPROACHDYN

(V.)MPG.ORDER

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

Sensor filter. The higher the order number, the greater the slope of attenuation. An order 2 is recommended.

V.MPG.ORDER

(V.)MPG.TYPE

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.
Units: Value listed.
Sensor filter. Filter type.

Values of the variable.

Value.	Meaning.
1	Low Passing.

(V.)MPG.FREQUENCY

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

Sensor filter. Filter cutoff frequency. A value of 30 Hz is recommended.

V.MPG.FREQUENCY

(V.)MPG.GAPERRORCANCEL

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

Canceling the gap error outside the range defined by GAPMIN/GAPMAX.

V.MPG.GAPERRORCANCEL

Values of the variable.

Value.	Meaning.
0	No.
1	Yes.

(V.)MPG.GAPMIN

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

Minimum sensor limit.

V.MPG.GAPMIN

(V.)MPG.GAPMAX

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

Maximum sensor limit.

V.MPG.GAPMAX

(V.)MPG.GAPTOLCANCEL

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

Canceling the gap error outside the range defined by GAPTOL.

V.MPG.GAPTOLCANCEL

Values of the variable.

Value.	Meaning.
0	No.
1	Yes.

(V.)MPG.GAPTOL

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

Error tolerance margin.

V.MPG.GAPTOL

(V.)MPG.GAPCOLLISIONMODE

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

CNC behavior in the event of a sensor collision.

V.MPG.GAPCOLLISIONMODE

Values of the variable.

Value.	Meaning.
0	Not displaying an error.
1	Displaying an error.

(V.)MPG.GAPTONEG

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

Movement block in #GAPCTRL towards descending coordinate.

V.MPG.GAPTONEG

Values of the variable.

Value.	Meaning.
0	No. The positive direction of the Z axis is down and the negative direction is up (inverted Z axis). The axis moves in a positive direction (down) to reach the gap.
1	Yes. The positive direction of the Z axis is up and the negative is down (under normal conditions). The axis moves in the negative direction (down) to reach the gap.

LEAPFROG.

(V.)MPG.LEAPDYNOVR

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

Dynamic response of the leap during the leapfrog.

V.MPG.LEAPDYNOVR

COMPENSATING THE DISPERSION BY THE CO2 LASER PATH.

(V.)MPG.LASERFOLLOWAXIS

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

Logic number of the slave axis to which the #FOLLOW is applied.

V.MPG.LASERFOLLOWAXIS

(V.)MPG.LASERFOLLOWOFFSET

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

Source offset of the X Y Z axes regarding the slave axis.

V.MPG.LASERFOLLOWOFFSET

(V.)MPG.LASERFOLLOW1

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

Effect of the first of the X Y Z axes on the slave axis.

V.MPG.LASERFOLLOW1

Values of the variable.

Value.	Meaning.
-1	Negative.
0	Not affected.
1	Positive.

(V.)MPG.LASERFOLLOW2

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

Effect of the second of the X Y Z axes on the slave axis.

V.MPG.LASERFOLLOW2

Values of the variable.

Value.	Meaning.
-1	Negative.
0	Not affected.
1	Positive.

(V.)MPG.LASERFOLLOW3

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

Effect of the third of the X Y Z axes on the slave axis.

V.MPG.LASERFOLLOW3

Values of the variable.

Value.	Meaning.
-1	Negative.
0	Not affected.
1	Positive.

26.4.2 General machine parameters. Execution channels.

CONFIGURATION OF THE HSC MODE (CHANNEL).

(V.)[ch].MPG.CURVFREQRES
Possible values: From 0 to 500.0000 Hz
Default value: 0.
Associated variable: (V.)[ch].MPG.CURVFREQRES

Channel [ch]. First resonance frequency of the machine in the arcs.

Syntax.

·ch· Channel number.

V.[2].MPG.CURVFREQRES	Channel ·2·.

DEFAULT CONDITIONS (CHANNEL).

(V.)[ch].MPG.G00COMP
Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.
Units: Value listed.

Channel [ch]. Apply tool radius compensation in G00.

Syntax.

·ch· Channel number.

V.[2].MPG.G00COMP	Channel ·2·.

Values of the variable.

Value.	Meaning.
0	No.
1	Yes.

DEFINITION OF THE SUBROUTINES.

(V.)[ch].MPG.INITIALSUB
Variable to be read via interface.
Units: Text.

Channel [ch]. Subroutine associated with the command #INITIALSUB.

Syntax.

·ch· Channel number.

[2].MPG.INTIALSUB	Channel ·2·.

(V.)[ch].MPG.PIERCING
Variable to be read via interface.
Units: Text.

Channel [ch]. Subroutine associated with the command #PIERCING.

Syntax.

·ch· Channel number.

[2].MPG.PIERCING	Channel ·2·.

(V.)[ch].MPG.CUTTINGON

Variable to be read via interface.
Units: Text.

Channel [ch]. Subroutine associated with the command #CUTTING ON.

Syntax.

·ch· Channel number.

[2].MPG.CUTTINGON	Channel ·2·.

(V.)[ch].MPG.CUTTINGOFF

Variable to be read via interface.
Units: Text.

Channel [ch]. Subroutine associated with the command #CUTTING OFF.

Syntax.

·ch· Channel number.

[2].MPG.CUTTINGOFF	Channel ·2·.

(V.)[ch].MPG.FINALSUB

Variable to be read via interface.
Units: Text.

Channel [ch]. Subroutine associated with the command #FINALSUB.

Syntax.

·ch· Channel number.

[2].MPG.FINALSUB	Channel ·2·.

26.5 Variables associated with the logic signals of the PLC.

26.5.1 PLC consulting logic signals; laser.

PIERCING ENABLED.

(V.)PLC.PIERCING

Variable that can be read and written via program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: Value listed.

Piercing enabled. The status of this variable is displayed in the PIERCING mark of the PLC.

V.PLC.PIERCING

Values of the variable.

Value.	Meaning.
0	Not active.
1	Active.

Remarks.

The OEM can use this variable in the subroutine associated with piercing (by default, Piercing.fst) to change the status of the PIERCING mark and indicate to the PLC when to start (value 1) and end (valor 0) the piercing operation. Entering this variable interrupts the block preparation.

The instruction #PLC also enables a PLC mark to be managed from the part program or subroutine without interrupting the block preparation.

ACTIVE CUTTING.

(V.)PLC.CUTTING

Variable that can be read and written via program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: Value listed.

Active cutting. The status of this variable is displayed in the CUTTING mark of the PLC.

V.PLC.CUTTING

Values of the variable.

Value.	Meaning.
0	Not active.
1	Active.

Remarks.

The OEM can use this variable in the subroutines associated with cutting (by default, Cutting.fst / Cuttingoff.fst) to change the status of the CUTTING mark and indicate to the PLC when to start (value 1) and end (valor 0) the cutting operation. Entering this variable interrupts the block preparation.

The instruction #PLC also enables a PLC mark to be managed from the part program or subroutine without interrupting the block preparation.

TECHNOLOGY TABLES.

(V.)PLC.COMVARACT

The CNC activates this mark when the user validates the common parameters of the active piercing or cutting table. The PLC should disable this mark when management with these variables is complete.

V.PLC.COMVARACT

(V.)PLC.CUTVARACT

Variable that can be read and written via program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: - .

The CNC activates this mark when the user validates the cutting parameters of the active table. The PLC should disable this mark when management with these variables is complete.

V.PLC.CUTVARACT

Remarks.

When the subroutine Cuttingon.fst changes the cutting type (#CUTTING ON), the CNC does not activate this mark, as the synchronization with the PLC is implemented through M functions.

(V.)PLC.PIRVARACT

Variable that can be read and written via program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: - .

The CNC activates this mark when the user validates the piercing parameters of the active table. The PLC should disable this mark when management with these variables is complete.

V.PLC.PIRVARACT

Remarks.

When the subroutine Piercing.fst changes the piercing type (#PIERCING), the CNC does not activate this mark, as the synchronization with the PLC is implemented through M functions.

GAP CONTROL.

(V.)PLC.INPOSGAP

Variable that can be read and written via program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: - .

The CNC activates this mark if the gap is within the range defined by the parameters GAPMIN-GAPMAX.

V.PLC.INPOSGAP

(V.)PLC.INTOL

Variable that can be read and written via program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: - .

The CNC activates this mark if the gap is within the tolerance limit defined by the parameter GAPTOL regarding the gap value programmed.

```
V.PLC.INTOL
```

LEAPFROG.

(V.)PLC.INPOSLIMIT

Variable that can be read and written via program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Units: - .

The CNC activates this mark (changes from 0 to 1) when the leap programmed with a #LEAP reaches the highest point (POSLIMIT command). The CNC deactivates this mark (changes from 1 to 0) when the leap starts to lower from this point.

```
V.PLC.INPOSLIMIT
```

Remarks.

If the leap does not reach the highest point due to lack of space, the CNC will not activate this mark. In the event of any error, the CNC will deactivate this mark.

26.5.2 PLC modifiable logic signals; laser.

LASER STATUS.

(V.)PLC.LASERON

Variable that can be read and written via program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: - .

The PLC activates this mark to indicate that the laser is on.

(V.)PLC.SHUTTERON

Variable that can be read and written via program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: - .

The PLC activates this mark to indicate that the laser source shutter is open.

(V.)PLC.LASERREADY

Variable that can be read and written via program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: - .

The PLC activates this mark to indicate that the laser is ready.

PWM ACTIVE FROM THE PLC.

(V.)PLC.PWMON

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: - .

This mark may be used to activate and deactivate the PWM via PLC. Activating the PWM via PLC has priority over activating it from the CNC.

V.PLC.PWMON

(V.)PLC.PWMFREQ

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: Herz.

PWM frequency, when the PWM has been activated via PLC. This variable only works when the Laser has been activated via PLC.

V.PLC.PWMFREQ

Remarks.

The changes from the variable are updated (refreshed) in the next loop; the intermediate times between loops will be ignored.

(V.)PLC.PWMDUTY

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: Percentage.

PWM duty cycle, when the PWM has been activated via PLC. This variable only works when the Laser has been activated via PLC.

V.PLC.PWMDUTY

Remarks.

The changes from the variable are updated (refreshed) in the next loop; the intermediate times between loops will be ignored.

The reading from the CNC will be divided by 10; i.e. for the value of ·1·, the CNC reading will return a value of ·0.1·.

POWER CONTROL.

(V.)PLC.ENABLEPWROUT

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: - .

This mark is active by default. This mark is associated with the power control via an analog output associated with the spindle (#PWRCTRL ON [OUT]). The PLC deactivates this mark to disable the active power control. The PLC activates this mark to enable the active power control in the CNC; if no power control is active, this mark does nothing.

V.PLC.ENABLEPWROUT

(V.)PLC.ENABLEPWRDUTY.

Variable that can be read and written from the interface and read from the program.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: - .

This mark is active by default. This mark is associated with the power control via a PWM duty (#PWRCTRL ON [DUTY]). The PLC deactivates this mark to disable the active power control. The PLC activates this mark to enable the active power control in the CNC; if no power control is active, this mark does nothing.

V.PLC.ENABLEPWRDUTY

GAP CONTROL.

(V.)PLC.ENABLEGAP

Variable that can be read and written via program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: - .

This mark is active by default. The PLC deactivates this mark to disable the active gap control. The PLC activates this mark to enable the active gap control in the CNC; if no gap control is active, this mark does nothing.

V.PLC.ENABLEGAP

(V.)PLC.GAPCOLLISION

Variable that can be read and written via program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: - .

The PLC activates this mark to indicate that the collision sensor has been activated. This mark is not kept in memory. With this mark active, the CNC behavior depends on the machine parameter GAPCOLLISION.

V.PLC.GAPCOLLISION

LEAPFROG.

(V.)PLC.ENABLELEAP

Variable that can be read and written via program and interface.
The variable returns the execution value; reading it interrupts block preparation.
Report variable (to be used from the scripts).
Units: - .

The PLC deactivates this mark to disable the active leapfrog. The PLC activates this mark to enable the active leapfrog in the CNC; if no gap control is active, this mark does nothing.

V.PLC.ENABLELEAP

26.6 Variables associated with the active material.

(V.)[ch].G.ACTIVEMATERIAL

Variable to be read via interface.
Units: Text.

Name of the active technological table.

Syntax.

·ch· Channel number.

V.[2].G.ACTIVEMATERIAL	Channel ·2·.

26.7 Variables associated with the active cutting.

(V.)[ch].G.CUTTINGTYPE

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

Cutting type.

Syntax.

·ch· Channel number.

V.[2].G.CUTTINGTYPE	Channel ·2·.

Remarks.

- If there is no active material, the variable will return a ·0· value.
- After powering up the CNC, if there is any active material, the variable will return a 1 value.
- After a reset or M30, the variable maintains its value.

26.8 Variables associated with the active piercing.

(V.)[ch].G.PIERCINGTYPE

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

Piercing type.

Syntax.

·ch· Channel number.

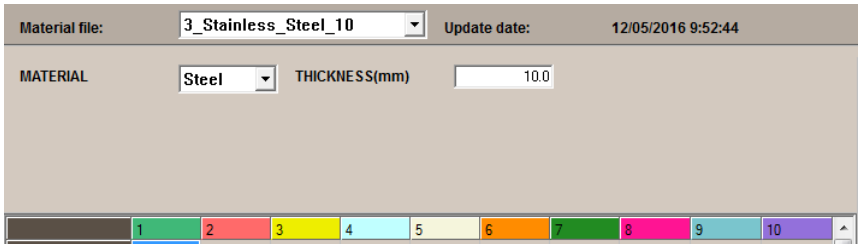
V.[2].G.PIERCINGTYPE	Channel ·2·.

Remarks.

- If there is no active material, the variable will return a ·0· value.
- After powering up the CNC, if there is any active material, the variable will return a 1 value.
- After a reset or M30, the variable maintains its value.

26.9 Variables associated with the technological tables.

26.9.1 Common parameters.



(V.)TT.name

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

Common parameter defined by the OEM. Replace "name" with the parameter name.

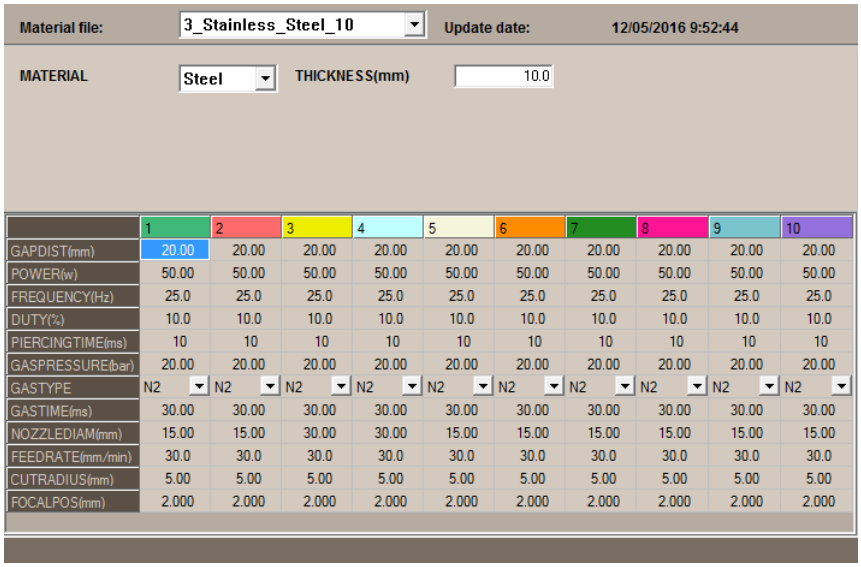
(V.)TT.THICKNESS

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

Sheet metal thickness.

```
V.TT.THICKNESS
```

26.9.2 Piercing technological table.



	1	2	3	4	5	6	7	8	9	10
GAPDIST(mm)	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00
POWER(w)	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00
FREQUENCY(Hz)	25.0	25.0	25.0	25.0	25.0	25.0	25.0	25.0	25.0	25.0
DUTY(%)	10.0	10.0	10.0	10.0	10.0	10.0	10.0	10.0	10.0	10.0
PIERCINGTIME(ms)	10	10	10	10	10	10	10	10	10	10
GASPRESSURE(bar)	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00
GASTYPE	N2	N2	N2	N2	N2	N2	N2	N2	N2	N2
GASTIME(ms)	30.00	30.00	30.00	30.00	30.00	30.00	30.00	30.00	30.00	30.00
NOZZLEDIAM(mm)	15.00	15.00	30.00	30.00	15.00	15.00	15.00	15.00	15.00	15.00
FEEDRATE(mm/min)	30.0	30.0	30.0	30.0	30.0	30.0	30.0	30.0	30.0	30.0
CUTRADIUS(mm)	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00
FOCALPOS(mm)	2.000	2.000	2.000	2.000	2.000	2.000	2.000	2.000	2.000	2.000

(V.)TTPIR.name

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

Technological parameters for the piercing.

- Parameters set by Fagor. Replace "name" with the parameter name.
- Parameter set by the OEM or the user. Replace "name" with the name of the variable associated with the parameter, set by the OEM.

(V.)TTPIR.GAPDIST

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

Part gap or distance.

```
V.TTPIR.GAPDIST
```

(V.)TTPIR.POWER

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

Power.

```
V.TTPIR.POWER
```

(V.)TTPIR.FREQUENCY

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

Frequency.

```
V.TTPIR.FREQUENCY
```

(V.)TTPIR.DUTY

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

Duty cycle.

V.TTPIR.DUTY

(V.)TTPIR.PIERCINGTIME

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

Piercing duration.

V.TTPIR.PIERCINGTIME

(V.)TTPIR.GASPRESSURE

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

Gas pressure.

V.TTPIR.GASPRESSURE

(V.)TTPIR.GASTYPE

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

Type of gas.

V.TTPIR.GASTYPE

Values of the variable.

Value.	Meaning.
10	N2.
20	O2.
30	Air.

(V.)TTPIR.GASTIME

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

Dwell time after activating the gas.

V.TTPIR.GASTIME

(V.)TTPIR.NOZZLEDIAM

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

Nozzle diameter.

V.TTPIR.NOZZLEDIAM

(V.)TTPIR.FEEDRATE

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.
Units: In millimeters/minute or inches/minute.

Work feedrate.

V.TTPIR.FEEDRATE

(V.)TTPIR.CUTRADIUS

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

Laser radio.

V.TTPIR.CUTRADIUS

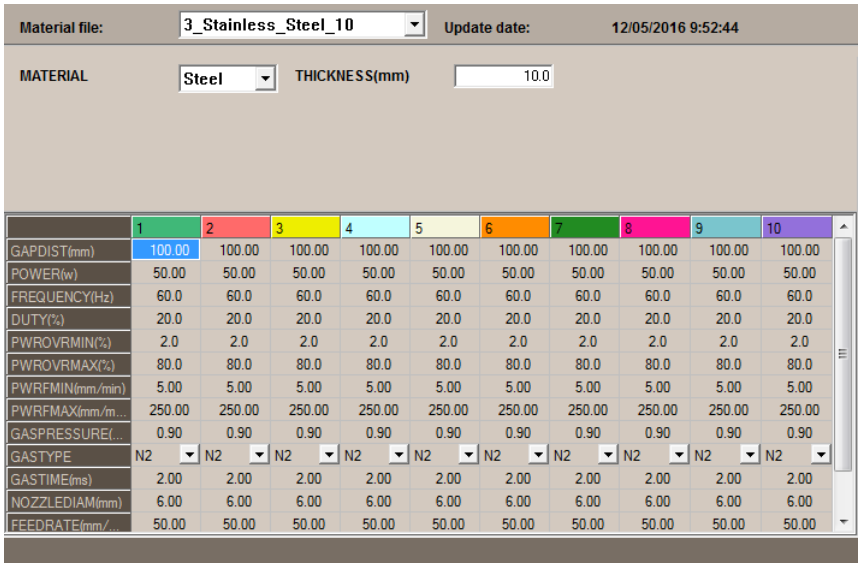
(V.)TTPIR.FOCALPOS

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

Laser focus position regarding the sheet surface. With a positive value, the focus is above the sheet; with a negative value, the focus is under or inside the sheet.

V.TTPIR.FOCALPOS

26.9.3 Cutting technological table.



	1	2	3	4	5	6	7	8	9	10
GAPDIST(mm)	100.00	100.00	100.00	100.00	100.00	100.00	100.00	100.00	100.00	100.00
POWER(w)	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00
FREQUENCY(Hz)	60.0	60.0	60.0	60.0	60.0	60.0	60.0	60.0	60.0	60.0
DUTY(%)	20.0	20.0	20.0	20.0	20.0	20.0	20.0	20.0	20.0	20.0
PWROVRMIN(%)	2.0	2.0	2.0	2.0	2.0	2.0	2.0	2.0	2.0	2.0
PWROVRMAX(%)	80.0	80.0	80.0	80.0	80.0	80.0	80.0	80.0	80.0	80.0
PWRFMIN(mm/min)	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00	5.00
PWRFMAX(mm/m...	250.00	250.00	250.00	250.00	250.00	250.00	250.00	250.00	250.00	250.00
GASPRESSURE(...	0.90	0.90	0.90	0.90	0.90	0.90	0.90	0.90	0.90	0.90
GASTYPE	N2	N2	N2	N2	N2	N2	N2	N2	N2	N2
GASTIME(ms)	2.00	2.00	2.00	2.00	2.00	2.00	2.00	2.00	2.00	2.00
NOZZLEDIAM(mm)	6.00	6.00	6.00	6.00	6.00	6.00	6.00	6.00	6.00	6.00
FEEDRATE(mm/...	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00	50.00

(V.)TTCUT.name

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

Technological parameters for the cutting.

- Parameters set by Fagor. Replace "name" with the parameter name.
- Parameter set by the OEM or the user. Replace "name" with the name of the variable associated with the parameter, set by the OEM.

(V.)TTCUT.GAPDIST

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

Part gap or distance.

```
V.TTCUT.GAPDIST
```

(V.)TTCUT.POWER

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

Power.

```
V.TTCUT.POWER
```

(V.)TTCUT.FREQUENCY

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

Frequency.

```
V.TTCUT.FREQUENCY
```

(V.)TTCUT.DUTY

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

Duty cycle.

V.TTCUT.DUTY

(V.)TTCUT.PWROVRMIN

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

Minimum power percentage for power control. Power percentage for a feed greater than or equal to the minimum (PWRFMIN).

V.TTCUT.PWOVRMIN

(V.)TTCUT.PWROVRMAX

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

Maximum power percentage for power control. Power percentage for a feed greater than or equal to the maximum (PWRFMAX).

V.TTCUT.PWOVRMAX

(V.)TTCUT.PWRFMIN

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.
Units: In millimeters/minute or inches/minute.

Minimum feed for power control. Feed below which the CNC applies the minimum power percentage (PWROVRMIN). Between the minimum and maximum feeds, power is proportional to the feed.

V.TTCUT.PWRFMIN

(V.)TTCUT.PWRFMAX

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.
Units: In millimeters/minute or inches/minute.

Maximum feed for power control. Feed above which the CNC applies the maximum power percentage (PWROVRMAX). Between the minimum and maximum feeds, power is proportional to the feed.

V.TTCUT.PWRFMAX

(V.)TTCUT.GASPRESSURE

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

Gas pressure.

V.TTCUT.GASPRESSURE

(V.)TTCUT.GASTYPE

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

Type of gas.

V.TTCUT.GASTYPE

Values of the variable.

Value.	Meaning.
10	N2.
20	O2.
30	Air.

(V.)TTCUT.GASTIME

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

Dwell time after activating the gas.

V.TTCUT.GASTIME

(V.)TTCUT.NOZZLEDIAM

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

Nozzle diameter.

V.TTCUT.NOZZLEDIAM

(V.)TTCUT.FEEDRATE

Variable that can only be read from the program, PLC and interface.
The variable returns the value of block preparation.
Units: In millimeters/minute or inches/minute.

Work feedrate.

V.TTCUT.FEEDRATE

(V.)TTCUT.CUTRADIUS

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

Laser radio.

V.TTCUT.CUTRADIUS

(V.)TTCUT.FOCALPOS

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

Laser focus position regarding the sheet surface. With a positive value, the focus is above the sheet; with a negative value, the focus is under or inside the sheet.

V.TTCUT.FOCALPOS

26.10 Variables related to power.

POWER PROGRAMMING.

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

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

Channel [ch]. Actual power.

Syntax.

·ch· Channel number.

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

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

POWER IN G97.

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

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

Channel [ch]. Active power in G97.

Syntax.

·ch· Channel number.

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

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

Remarks.

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

Defined power.	(V.)PLC.S.sn	(V.)A.PRGS.sn	(V.)A.SPEED.sn
By program; S5000. By PLC; none.	0	5000	5000
By program; S5000. By PLC; S9000.	9000	5000	9000
By program; S5000. By PLC; S3000.	3000	5000	3000

Defined power.	(V.)PLC.S.sn	(V.)A.PRGS.sn	(V.)A.SPEED.sn
By program; S5000. By PLC; none. By MDI; S8000.	0	8000	8000
By program; S5000. By PLC; S9000. By MDI; S8000.	9000	8000	9000
By program; S5000. By PLC; S3000. By MDI; S8000.	3000	8000	3000

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

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

Channel [ch]. Active power by PLC for G97.

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

Syntax.

·ch· Channel number.

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

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

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

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

Channel [ch]. Active power by program for G97.

With G97 active, programming in MDI mode of the new power updates the value of this variable.

Syntax.

·ch· Channel number.

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

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

POWER LIMIT.

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

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

Channel [ch]. Active power limit.

Syntax.

·ch· Channel number.

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

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

Remarks.

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

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

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

Channel [ch]. Active power limit by PLC.

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

Syntax.

·ch· Channel number.

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

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

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

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

Channel [ch]. Active power limit by program.

Syntax.

·ch· Channel number.

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

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

POWER PERCENTAGE (OVERRIDE).

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

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

Channel [ch]. Active power percentage.

Syntax.

·ch· Channel number.

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

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

Remarks.

The power speed (override) percentage may be set by program, by PLC or from the control panel; the one set by program has the highest priority and the one selected from the control panel has the lowest.

(V.)A.PRGSSO.sn (V.)SP.PRGSSO.sn	(V.)PLC.SSO.sn	(V.)A.CNCSSO.sn (V.)SP.CNCSSO.sn	(V.)A.SSO.sn (V.)SP.SSO.sn
0	0	100 %	100 %
0	80 %	100 %	80 %
110 %	80%	100 %	110 %
70 %	80 %	100 %	70 %
70 %	0	100 %	70 %

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

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

Channel [ch]. Power percentage by PLC.

The percentage set by PLC has higher priority than the one set from the control panel, but lower priority than the one set by program.. To cancel the value set by PLC, set the variable to ·0·.

Syntax.

·ch· Channel number.

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

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

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

Variable that can be read and written from the program and read from the PLC and from the interface.
The variable returns the value of block preparation.
Units: Percentage.

Channel [ch]. Power percentage by program.

The percentage set by program has higher priority than the one set by PLC or from the control panel. To cancel the value set by program, set the variable to ·0·.

Syntax.

·ch· Channel number.

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

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

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

Variable that can be read and written (asynchronous writing) from the interface and read from the program and from the PLC.
The variable returns the execution value; reading it interrupts block preparation.
Units: Percentage.

Channel [ch]. Power percentage on the control panel.

The percentage set from the control panel has lower priority than the one set by PLC or by program.

Syntax.

·ch· Channel number.

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

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

26.11 Variables related to the PWM.

(V.)G.PWMON

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

PWM status, when it has been activated via CNC.

Syntax.

V.G.PWMON

Values of the variable.

Value.	Meaning.
0	PWM off.
1	PWM active.

Remarks.

Activating the Laser via PLC has priority over activating it from the CNC.

PWM status from the CNC.	PWM status from the PLC.	PWM status.
Off.	Off.	PWM off.
Active.	Off.	PWM active from the CNC.
Off.	Active.	PWM active from the PLC.
Active.	Active.	PWM active from the PLC.

(V.)G.PWMFREQ

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

PWM frequency, when the PWM has been activated via CNC. Variable associated with the command PWMFREQ of the instruction #PWMOUT. This variable only works when the Laser is active via CNC.

Syntax.

V.G.PWMFREQ

(V.)G.PWMDUTY

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

PWM duty cycle, when the PWM has been activated via CNC. Variable associated with the command PWMFREQ of the instruction #PWMOUT. This variable only works when the Laser is active via CNC.

Syntax.

V.G.PWMDUTY

Remarks.

The reading from the PLC will be given in tenths (x10); i.e. for a value ·0.1·, the PLC reading will return a value of ·1·. For a value of ·100·, the reading from the PLC will return a value of ·1000·.

(V.)G.PWMBTIME

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

Amount of time (in ms rounded up to loop units) that PWM stays active in burst mode. Variable associated with the command PWMBTIME of the instruction #PWMOUT.

V.G.PWMBTIME

(V.)G.PWMBEND

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

PWM status after completing the burst mode. Variable associated with the command PWMBEND of the instruction #PWMOUT.

V.G.PWMBEND

Values of the variable.

Value.	Meaning.
0	PWM off.
1	PWM active.

26.12 Variables associated with the power control.

POWER CONTROL VIA AN ANALOG OUTPUT.

(V.)G.PWROUTON

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

Power control status (#PWRCTRL ON [OUT]). This variable is also affected by the mark ENABLEPWROUT of the PLC.

V.G.PWROUTON

Values of the variable.

Value.	Meaning.
0	Not active.
1	Active.

(V.)G.PWROUTOVRMIN

Variable that can be read and written from the PLC and read from the program and from the interface.
The variable returns the value of block preparation.
Units: Percentage.

Minimum percentage to be applied. Variable associated with the command OVRMIN of the instruction #PWRCTRL ON [OUT].

V.G.PWROUTOVRMIN

Remarks.

- The variable assumes the values programmed in the instruction #PWRCTRL during block preparation. If the PLC modifies these variables during the execution of a program, the CNC maintains the values calculated during block preparation.
- This variable is activated when programming the command #PWRCTRL from the program. When the power control is activated from the PLC (CNCEX command), this variable does not update its value.

(V.)G.PWROUTOVRMAX

Variable that can be read and written from the PLC and read from the program and from the interface.
The variable returns the value of block preparation.
Units: Percentage.

Maximum percentage to be applied. Variable associated with the command OVRMAX of the instruction #PWRCTRL ON [OUT].

V.G.PWROUTOVRMAX

Remarks.

- The variable assumes the values programmed in the instruction #PWRCTRL during block preparation. If the PLC modifies these variables during the execution of a program, the CNC maintains the values calculated during block preparation.
- This variable is activated when programming the command #PWRCTRL from the program. When the power control is activated from the PLC (CNCEX command), this variable does not update its value.

(V.)G.PWROUTFMIN

Variable that can be read and written from the PLC and read from the program and from the interface.
The variable returns the value of block preparation.
Units: In millimeters/minute or inches/minute.

Feed below which the CNC applies the OVRMIN percentage to the power. Variable associated with the command FMIN of the instruction #PWRCTRL ON [OUT].

V.G.PWROUTFMIN

Remarks.

- The variable assumes the values programmed in the instruction #PWRCTRL during block preparation. If the PLC modifies these variables during the execution of a program, the CNC maintains the values calculated during block preparation.
- This variable is activated when programming the command #PWRCTRL from the program. When the power control is activated from the PLC (CNCEX command), this variable does not update its value.

(V.)G.PWROUTFMAX

Variable that can be read and written from the PLC and read from the program and from the interface.
The variable returns the value of block preparation.
Units: In millimeters/minute or inches/minute.

Feed above which the CNC applies the OVRMIN percentage to the power. Variable associated with the command FMAX of the instruction #PWRCTRL ON [OUT].

V.G.PWROUTFMAX

Remarks.

- The variable assumes the values programmed in the instruction #PWRCTRL during block preparation. If the PLC modifies these variables during the execution of a program, the CNC maintains the values calculated during block preparation.
- This variable is activated when programming the command #PWRCTRL from the program. When the power control is activated from the PLC (CNCEX command), this variable does not update its value.

POWER CONTROL VIA THE PWM DUTY CYCLE.

(V.)G.PWRDUTYON

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

Power control status (#PWRCTRL ON [DUTY]). This variable is also affected by the mark ENABLEPWRDUTY of the PLC.

V.G.PWRDUTYON

Values of the variable.

Value.	Meaning.
0	Not active.
1	Active.

Remarks.

The variable assumes the values programmed in the instruction #PWRCTRL during block preparation. If the PLC modifies these variables during the execution of a program, the CNC maintains the values calculated during block preparation.

(V.)G.PWRDUTYOVRMIN

Variable that can be read and written from the PLC and read from the program and from the interface.
The variable returns the value of block preparation.
Units: Percentage.

Minimum percentage to be applied. Variable associated with the command OVRMIN of the instruction #PWRCTRL ON [DUTY].

```
V.G.PWRDUTYOVRMIN
```

Remarks.

- The variable assumes the values programmed in the instruction #PWRCTRL during block preparation. If the PLC modifies these variables during the execution of a program, the CNC maintains the values calculated during block preparation.
- This variable is activated when programming the command #PWRCTRL from the program. When the power control is activated from the PLC (CNCEX command), this variable does not update its value.

(V.)G.PWRDUTYOVRMAX

Variable that can be read and written from the PLC and read from the program and from the interface.
The variable returns the value of block preparation.
Units: Percentage.

Maximum percentage to be applied. Variable associated with the command OVRMAX of the instruction #PWRCTRL ON [DUTY].

```
V.G.PWRDUTYOVRMAX
```

Remarks.

- The variable assumes the values programmed in the instruction #PWRCTRL during block preparation. If the PLC modifies these variables during the execution of a program, the CNC maintains the values calculated during block preparation.
- This variable is activated when programming the command #PWRCTRL from the program. When the power control is activated from the PLC (CNCEX command), this variable does not update its value.

(V.)G.PWRDUTYFMIN

Variable that can be read and written from the PLC and read from the program and from the interface.
The variable returns the value of block preparation.
Units: In millimeters/minute or inches/minute.

Feed below which the CNC applies the OVRMIN percentage to the power. Variable associated with the command FMIN of the instruction #PWRCTRL ON [DUTY].

```
V.G.PWRDUTYFMIN
```

Remarks.

- The variable assumes the values programmed in the instruction #PWRCTRL during block preparation. If the PLC modifies these variables during the execution of a program, the CNC maintains the values calculated during block preparation.
- This variable is activated when programming the command #PWRCTRL from the program. When the power control is activated from the PLC (CNCEX command), this variable does not update its value.

(V.)G.PWRDUTYFMAX

Variable that can be read and written from the PLC and read from the program and from the interface.
The variable returns the value of block preparation.
Units: In millimeters/minute or inches/minute.

Feed above which the CNC applies the OVRMIN percentage to the power. Variable associated with the command FMAX of the instruction #PWRCTRL ON [DUTY].

```
V.G.PWRDUTYFMAX
```

Remarks.

- The variable assumes the values programmed in the instruction #PWRCTRL during block preparation. If the PLC modifies these variables during the execution of a program, the CNC maintains the values calculated during block preparation.
- This variable is activated when programming the command #PWRCTRL from the program. When the power control is activated from the PLC (CNCEX command), this variable does not update its value.

FAGOR
FAGOR AUTOMATION
CNC 8060 L
CNC 8070 L
(REF: 1707)

26.13 Variables associated with the leapfrog.

(V.)[ch].G.LEAPON

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

Channel [ch]. Leapfrog status.

Syntax.

·ch· Channel number.

V.[2].G.LEAPON	Channel ·2·.

Values of the variable.

Value.	Meaning.
0	Inactive.
1	Active.

Remarks.

The variable assumes value 1 only during the movement associated with the block.

(V.)[ch].G.LEAPG91

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

Channel [ch]. Type of coordinates for the leap coordinate, absolute (if G90) or incremental (if G91).

Syntax.

·ch· Channel number.

V.[2].G.LEAPG91	Channel ·2·.

Values of the variable.

Value.	Meaning.
0	G90.
1	G91.

(V.)[ch].G.LEAPPOS

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

Channel [ch]. Longitudinal axis position (usually Z) at the end of the leap.

Syntax.

·ch· Channel number.

V.[2].G.LEAPPOS	Channel ·2·.

(V.)[ch].G.LEAPPOSLIMIT

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

Channel [ch]. Maximum coordinate to be reached by the longitudinal axis (usually Z) during the leap.

Syntax.

·ch· Channel number.

V.[2].G.LEAPPOSLIMIT	Channel ·2·.

(V.)[ch].G.LEAPTIME

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

Channel [ch]. Dwell before the leap.

Syntax.

·ch· Channel number.

V.[2].G.LEAPTIME	Channel ·2·.

26.14 Variables related to synchronized switching.

TIME SHIFTS TO ANTICIPATE THE ACTIVATION OR DEACTIVATION OF THE DIGITAL OUTPUT.

The value of these variables is equivalent to the one programmed in the parameters of the instruction #SWTOUT. When executing the instruction without parameters and the variables have a value assigned to it, the CNC assumes these latter values as active. Likewise, when programming the parameters of the instruction, the variables assume these values as their own.

On CNC power-up, the offset wears are set to zero. After power-up, the values programmed for the offsets (using the instruction #SWTOUT or the variables) are maintained even after an error, a reset or an M30.

These variables interrupt block preparation. To modify the offset values without interrupting block preparation, change these values from the PLC or use the parameters of the instruction #SWTOUT. If these variables are changed from the PLC, the new values are assumed when executing the instruction #SWTOUT. If the instruction is active when changing the values from the PLC, it assumes the new values.

(V.)[ch].G.TON

Variable that can be read and written via program, PLC and interface.
The variable returns the execution value; reading/writing it interrupts block preparation.
Units: Milliseconds.

Time shift (milliseconds) to anticipate the activation of the digital output. Variable associated with the command TON of the instruction #SWTOUT.

Syntax.

·ch· Channel number.

V.[2].G.TON	Channel ·2·.

(V.)[ch].G.TOF

Variable that can be read and written via program, PLC and interface.
The variable returns the execution value; reading/writing it interrupts block preparation.
Units: Milliseconds.

Time shift (milliseconds) to anticipate the deactivation of the digital output. Variable associated with the command TOF of the instruction #SWTOUT.

Syntax.

·ch· Channel number.

V.[2].G.TOF	Channel ·2·.

(V.)[ch].G.PON

Variable that can be read and written via program, PLC and interface.
The variable returns the execution value; reading/writing it interrupts block preparation.
Units: mm or inches.

Distance shift (offset, mm(inches) to anticipate the activation of the digital output. Variable associated with the command PON of the instruction #SWTOUT.

Syntax.

·ch· Channel number.

V.[2].G.PON	Channel ·2·.

(V.)[ch].G.POF

Variable that can be read and written via program, PLC and interface.
The variable returns the execution value; reading/writing it interrupts block preparation.
Units: mm or inches.

Distance shift (offset, mm(inches) to anticipate the deactivation of the digital output. Variable associated with the command POF of the instruction #SWTOUT.

Syntax.

·ch· Channel number.

V.[2].G.POF	Channel ·2·.

TIME REMAINING TO ACTIVATE OR DEACTIVATE THE LASER OUTPUT.

(V.)[ch].G.LASEROTMON

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading/writing it interrupts block preparation.
Units: 0.5 µs cycles.

Time remaining (0.5 µs cycles) to activate the laser output.

Syntax.

·ch· Channel number.

V.[2].G.LASEROTMON	Channel ·2·.

(V.)[ch].G.LASEROTMOFF

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading/writing it interrupts block preparation.
Units: 0.5 µs cycles.

Time remaining (0.5 µs cycles) to deactivate the laser output.

Syntax.

·ch· Channel number.

V.[2].G.LASEROTMOFF	Channel ·2·.

26.15 Variables associated with the compensation of the CO2 laser dispersion.

(V.)[ch].A.FLWMASTER.xn

Variable that can be read and written via program, PLC and interface.
The variable returns the execution value; reading it interrupts block preparation.
Units: - .

Generic variable for an axis follow-up.

Syntax.

·ch· Channel number.

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

V.A.FLWMASTER.Z	Z axis.
V.A.FLWMASTER.4	Axis with logic number ·4·.
V.[2].A.FLWMASTER.1	Axis with index ·1· in the channel ·2·.

Remarks.

When the CNC uses this variable to compensate the laser dispersion, the CNC updates it in each position loop.

If the CNC uses this variable for other applications, by entering its valor from the PLC, the update will be defined by the PLC times. If you wish to set the update for each loop, program the PLC to enter each loop or use a periodic period subroutine equal to the cycle time of the CNC (parameter LOOPTIME). In these cases, after reaching the slave axis, the initial synchronization attempts to follow the variable values in each loop. The axis follows the variable without any filtering process, so if these values are not filtered when the PLC enters them, the slave axis may move abruptly.

26.16 Variables associated with the gap control.

(V.)G.GAPCTRLON

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

Gap control status. This variable is also affected by the mark ENABLEGAP of the PLC.

```
V.G.GAPCTRLON
```

Values of the variable.

Value.	Meaning.
0	Not active.
1	Active.

Remarks.

During tool inspection with the active gap control, when moving the axis associated with the sensor (MDI or manual mode), the CNC deactivates it (the variable assumes value 0). The CNC reactivates the gap control (the variable assumes value 1) when completing inspection, after completing repositioning of the axis associated with the sensor.

During the leapfrog, the CNC deactivates the gap control and reactivates it after the leap.

(V.)[ch].G.GAPAXISPOS

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

Approach coordinate. Variable associated with the command X..C of the instruction #GAPCTRL.

Syntax.

·ch· Channel number.

```
V.[2].G.GAPAXISPOS          Channel ·2·.
```

(V.)[ch].G.GAPPRG

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

Distance (gap) to be kept from the surface. Variable associated with the command GAP of the instruction #GAPCTRL.

Syntax.

·ch· Channel number.

```
V.[2].G.GAPPRG              Channel ·2·.
```

(V.)G.GAP

Variable that can only be read from the program, PLC and interface.
The variable returns the execution value; reading/writing it interrupts block preparation.
Units: mm or inches.

Actual gap value, detected by the sensor.

```
V.G.GAP
```

(V.)[ch].A.GAPCOMP.xn

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

Value that the gap compensation is adding to the axis. Compensation (offset) applied to the axis associated with the sensor to control the GAPCTRL.

Syntax.

·ch· Channel number.

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

V.A.GAPCOMP.Z	Z axis.
V.A.GAPCOMP.4	Axis with logic number ·4·.
V.[2].A.GAPCOMP.1	Axis with index ·1· in the channel ·2·.

26.17 Variables associated with the subroutine call parameters.

(V.)C.PCALLP_A
...
(V.)C.PCALLP_Z

Variable that can be read from the program.
The variable returns the value of block preparation.
Units: Value listed.

Parameter programmed in the call for a subroutine or in instructions #CUTTING and #PIERCING. This variable is valid for OEM subroutines (G180-G189 and G380-G399) and for subroutines called upon using #PCALL or #MCALL.

Syntax.

V.C.PCALLP_A	Parameter "A".
V.C.PCALLP_F	"F" parameter.

Values of the variable.

Value.	Meaning.
0	It has not been programmed.
1	It has been programmed.

Remarks.

Call to a subroutine.	Reading the variable.
#PCALL sub.nc A12.56 D3	V.C.PCALLP_A = 1 V.C.PCALLP_D = 1
G180 A12 B34.5667	V.C.PCALLP_A = 1 V.C.PCALLP_B = 1
#CUTTING [T3]	V.C.PCALLP_T = 1

(V.)G.CALLP

Variable that can be read from the program.
The variable returns the value of block preparation.
Units: - .

Local parameters programmed in the call for a subroutine or in #CUTTING and #PIERCING instructions.

Syntax.

V.G.CALLP

Values of the variable.

The variable returns a value coded in bits, in which each bit represents a parameter; the bit with the lowest weight (bit 0) corresponds to parameter P0, bit 1, to parameter P1, and so on. Each bit indicates whether the parameter is programmed (bit=1) or not (bit=0).

26.18 Variables defined by the user.

(V.)P.var_name

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

User local variables.

Syntax.

·var_name· Name of the variable.

V.P.myvar	Variable with the name of "myvar".

Remarks.

These variables maintain their value in local and global subroutines called upon from the program.

(V.)S.var_name

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

User global variables.

Syntax.

·var_name· Name of the variable.

V.S.myvar	Variable with the name of "myvar".

Remarks.

These variables maintain their value between programs and after a reset. Variables are deleted when the CNC is turned off and they can also be deleted from the part-program using the #DELETE instruction.

```
#DELETE V.S.myvar1 V.S.myvar2
```

26.

CNC VARIABLES.

Variables defined by the user.

FAGOR

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

User notes:

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

User notes:

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

User notes:

FAGOR

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)

User notes:

FAGOR AUTOMATION

CNC 8060 L
CNC 8070 L

(REF: 1707)