

CNCCelite

8058/8060
8065/8070

Operating manual.

Ref: 2508

FAGOR
AUTOMATION



TRANSLATION OF THE ORIGINAL MANUAL

This manual is a translation of the original manual. This manual, as well as the documents derived from it, have been drafted in Spanish. In the event of any contradictions between the document in Spanish and its translations, the wording in the Spanish version shall prevail. The original manual will be labeled with the text "ORIGINAL MANUAL".

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 the following 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.



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 subject to changes due to technical modifications. Fagor Automation reserves the right to change 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, hence no absolute match is guaranteed. However, the contents of this document are regularly checked and updated implementing the necessary corrections in a later 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 this manual.....	11
About the product.....	13
EC Declaration of Conformity, warranty conditions and quality certificates.....	19
Safety conditions.....	21
Returning conditions.....	25
CNC maintenance.....	27
New Features.....	29

CHAPTER 1 DESCRIPTION OF THE KEYS

1.1 Keyboard.....	31
1.1.1 Fagor logo key (only HORIZONTAL KEYB 2.0 + TOUCHPAD models).....	33
1.1.2 Numeric keypad (only HORIZONTAL KEYB 2.0 + TOUCHPAD models).....	33
1.1.3 Touchpad (only HORIZONTAL KEYB 2.0 + TOUCHPAD models).....	34
1.2 Operator panel.....	35
1.3 Keyboard shortcuts.....	36

CHAPTER 2 GENERAL CONCEPTS

2.1 Turning the CNC on and off.....	39
2.1.1 Emergency shutdown with UPS.....	41
2.2 Work modes and software protection at the CNC.....	42
2.3 Ethernet.....	45
2.4 USB ports.....	45
2.5 Disco CFast (Q7-A platform).....	46

CHAPTER 3 HOW TO OPERATE THE CNC

3.1 General description of the interface.....	47
3.2 Configuring the softkey menus.....	49
3.2.1 Expanded softkey menus.....	49
3.2.2 Horizontal menu with standard or drop-down soft keys.....	49
3.3 General CNC-status bar.....	50
3.4 Operating modes.....	53
3.4.1 Description of the various operating modes.....	54
3.5 Execution channels.....	56
3.6 Task window.....	58
3.7 Channel synchronization window.....	59
3.8 CNC warning and errors window.....	60
3.8.1 Warnings.....	60
3.8.2 Errors.....	60
3.8.3 Fatal error.....	61
3.8.4 Error 3753. Absolute encoder. Monitor the coordinate difference during startup (parameter MAXDIFREF).....	61
3.9 PLC messages.....	62
3.10 PLC errors.....	63
3.11 File selection window.....	64
3.12 Mini-numeric keypad.....	66
3.13 Calculator.....	67
3.13.1 Softkey menu.....	68
3.13.2 Defining expressions.....	70
3.14 Dialog boxes.....	71

CHAPTER 4 AUTOMATIC MODE

4.1 Interface description.....	73
4.1.1 Description of a typical screen of this work mode.....	73
4.1.2 Softkey menus.....	76
4.2 Display the status of the program or of the active subroutines.....	77
4.3 Program simulation and execution.....	78
4.3.1 Select program.....	78
4.3.2 Select the first and last blocks of the execution.....	79
4.3.3 Execute a program.....	81
4.3.4 Mode change protection for an interrupted program.....	82
4.3.5 Resume the execution of a program from the block where it was canceled.....	83



CNCelite
8058 8060
8065 8070

REF: 2508

4.3.6	Cancel the execution and resume from another block while keeping the history.	84
4.3.7	Simulated execution of a program.	86
4.3.8	Execute a program (retrace).	89
4.3.9	Executing a program in 8055 language.	92
4.4	Executing program blocks separately.	93
4.5	Tool inspection.	94
4.5.1	Tool inspection (execution in retrace mode, independent interpolator or rigid tapping).	97
4.6	Block search.	99
4.6.1	Treatment of functions M, H, F, S.	101
4.7	Show/hide the dynamic override of the HSC.	102
4.8	FFC (Fagor Feed Control)	103
4.9	Display the status of the DMC (Dynamic Machining Control).	104
4.9.1	DMC status and progress.	104
4.9.2	Learning phase.	105
4.9.3	Deactivating the DMC.	106

CHAPTER 5**MANUAL (JOG) MODE**

5.1	Interface description.	107
5.1.1	Description of a typical screen of this work mode.	107
5.1.2	Softkey menus.	109
5.2	Operations with the axes.	110
5.2.1	Home search.	110
5.2.2	Jog	111
5.2.3	Jogging the axes with handwheels	113
5.2.4	Moving an axis to a particular position (coordinate).	115
5.2.5	Coordinate preset.	116
5.3	Spindle control.	117
5.4	Tool selection and tool change	118
5.5	Setting the feedrate and spindle speed.	119
5.6	Setting and activating the zero offsets and the fixture offsets.	120

CHAPTER 6**MANUAL (JOG) MODE. TOOL CALIBRATION**

6.1	Manual calibration. Calibration without a probe	123
6.2	Semi-automatic calibration. Calibration with a probe	127
6.3	Automatic calibration with a probe and a canned cycle	130
6.3.1	Mill or lathe model ("trihedron" geometrical configuration)	130
6.3.2	Lathe model ("plane" geometrical configuration)	133

CHAPTER 7**JOG MODE. PART CENTERING (MILL MODEL)**

7.1	How to define the data.	137
7.2	Data programming.	138
7.3	Basic operation.	142

CHAPTER 8**EDISIMU MODE (EDITING AND SIMULATION)**

8.1	Interface description.	145
8.1.1	Softkey menus.	147
8.2	Program editing and simulation	148
8.2.1	Program editing.	148
8.2.2	Editing a program in the 8055 CNC language	149
8.2.3	Translating a program in 8055 language.	151
8.2.4	Program simulation	153
8.2.5	Simulation errors	155
8.3	Editing window.	156
8.3.1	Softkey and hotkey menus.	158
8.3.2	Contextual programming assistance.	159
8.3.3	Help for programming subroutines.	160
8.3.4	Syntax errors when editing	161
8.4	Working in the editing window.	162
8.4.1	Select a program.	162
8.4.2	Operations with blocks. Cut and paste.	162
8.4.3	Find a line or a text in the program.	163
8.4.4	Undo and redo.	163
8.4.5	Operations with files.	164
8.4.6	Customizing the editor (general options).	165
8.4.7	Customizing the editor (appearance).	169
8.4.8	Customizing the editor (syntax highlighting).	169
8.4.9	TEACH-IN	170
8.4.10	Import DXF files	171
8.4.11	Import files from the 8055 (PIM and PIT).	174

8.4.12	Profile editor.....	175
8.4.13	Canned cycle editor.....	175
8.4.14	Inclined planes (·M· model).....	176
8.4.15	Geometric-help editor.....	177
8.5	Graphic assistance for the program editor.....	181
8.6	Graphics window.....	182
8.7	Program window.....	183
8.8	Working in the program window.....	184
8.8.1	Select the first and last blocks of the execution.....	184
8.8.2	Simulate program blocks separately.....	186
8.8.3	Display the status of the program or of the active subroutines.....	187
8.9	Statistics window.....	188
8.9.1	Time estimates.....	189

CHAPTER 9 FMC (FAGOR MACHINING CALCULATOR).

9.1	FMC calculator.....	193
9.2	Materials table.....	194
9.3	Operations table.....	195
9.4	Working with the FMC.....	196
9.5	Backup/Restore. Data backup copy.....	196

CHAPTER 10 FCAS (FAGOR COLLISION AVOIDANCE SYSTEM).

10.1	FCAS status.....	198
10.2	FCAS operation.....	199

CHAPTER 11 PROFILE EDITOR

11.1	Interface description.....	201
11.1.1	How to use the profile editor.....	203
11.2	Define a new profile, enlarge an existing one or import one from a file.....	204
11.2.1	Define any profile using straight and curved sections.....	205
11.2.2	Define a circular profile.....	207
11.2.3	Define a rectangular profile.....	208
11.2.4	Enlarge a profile.....	208
11.2.5	Import a profile from a DXF file.....	209
11.2.6	Editing a profile in ProGTL3 language.....	212
11.3	Modify a profile and insert corners.....	213
11.4	Configuring the profile editor. Displayed area.....	215
11.5	Configuring the profile editor. Define the work plane.....	215
11.6	End the session at the editor.....	215
11.7	Examples of how to define profiles.....	216
11.7.1	Profile editor. Example 1 (milling).....	216
11.7.2	Profile editor. Example 2 (milling).....	217
11.7.3	Profile editor. Example 3 (milling).....	219
11.7.4	Profile editor. Example 4 (lathe).....	220

CHAPTER 12 GRAPHIC ENVIRONMENT (MILL MODEL)

12.1	Description of the graphic environment.....	222
12.1.1	Softkey menus.....	225
12.2	Type of graphics.....	226
12.3	Zoom.....	227
12.4	Dimensions.....	228
12.5	Point of view.....	228
12.6	Measurement.....	229
12.7	Clear screen.....	230
12.8	Colors.....	230
12.9	Options.....	231
12.10	Real coordinates.....	232
12.11	Real coordinates with enlarged error.....	232
12.12	Simulation speed.....	232

CHAPTER 13 GRAPHICS HD (·M· MODEL).

13.1	Graphic environment.....	233
13.1.1	HD graphics in automatic mode.....	234
13.1.2	HD graphics in edisimu mode.....	235
13.1.3	Horizontal softkey menu.....	236
13.2	Move, rotate and zoom in or out on the graphic.....	237
13.3	Select the type of view.....	238
13.4	Configuring the graphics window (properties of each window).....	239
13.5	Configuring and activating the sections.....	240

13.6	Configure the colors for the tool path and solid.	242
13.7	General configuration of the graphics.	243
13.8	Configuration. Cancel the graphics.	244
13.9	Configuration. Load machine.	244
13.10	Actions. Move the active sections.	245
13.11	Actions. Print the graphic.	246
13.12	Delete the graphic.	246
13.13	Editing, displaying and hiding parts.	247
13.14	Automatic dimensions.	248
13.15	Saving part / Loading part.	248
13.16	Measure the part.	249
13.17	Viewing the tool paths and the solid.	250
13.18	Simulation speed (edisimu mode only).	251

CHAPTER 14 GRAPHIC ENVIRONMENT (LATHE MODEL)

14.1	Description of the graphic environment.	254
14.1.1	Softkey menus.	257
14.2	Type of graphics.	258
14.3	Zoom.	259
14.4	Dimensions.	260
14.5	Measurement.	260
14.6	Clear screen.	261
14.7	Colors.	261
14.8	Options.	262
14.9	Real coordinates.	263
14.10	Real coordinates with enlarged error.	263
14.11	Simulation speed.	263

CHAPTER 15 GRAPHICS HD (-T- MODEL).

15.1	Graphic environment.	265
15.1.1	HD graphics in automatic mode.	266
15.1.2	HD graphics in edisimu mode.	267
15.1.3	Horizontal softkey menu.	268
15.2	Move, rotate and zoom in or out on the graphic.	269
15.3	Select the type of view.	270
15.4	Configuring the graphics window (properties of each window).	271
15.5	Configuring and activating the sections.	272
15.6	Configure the colors for the tool path and solid.	273
15.7	General configuration of the graphics.	274
15.8	Configuration. Cancel the graphics.	275
15.9	Configuration. Load machine.	275
15.10	Actions. Move the active sections.	276
15.11	Actions. Print the graphic.	277
15.12	Delete the graphic.	277
15.13	Editing, displaying and hiding parts.	278
15.14	Automatic dimensions.	279
15.15	Saving part / Loading part.	279
15.16	Measure the part.	280
15.17	Viewing the tool paths and the solid.	281
15.18	Simulation speed (edisimu mode only).	282

CHAPTER 16 MDI/MDA MODE

16.1	Interface description.	283
16.1.1	Softkey menus.	284
16.2	Edit and execute individual blocks.	285
16.3	Block history.	286

CHAPTER 17 USER TABLES

17.1	User table presentation.	287
17.1.1	Softkey menus.	288
17.2	Zero offset tables.	289
17.3	Fixture table.	291
17.4	Arithmetic parameter tables.	292
17.5	Active offsets table.	294
17.6	Operations with tables.	295
17.6.1	Data editing.	295
17.6.2	Save and recall tables.	296
17.6.3	Find text.	297

CHAPTER 18 TOOL AND MAGAZINE TABLE

18.1	Presentation of the tool tables and magazine tables.	299
18.1.1	Softkey menus.	300
18.1.2	Search for a text in the tables.	301
18.1.3	Save and load the tables.	302
18.1.4	Printing the tables.	304
18.2	Tool table.	305
18.3	Tool table (full mode).	307
18.3.1	Vertical softkey menu.	307
18.3.2	The tool list.	308
18.3.3	Description of the tool data.	309
18.4	Tool table (simple mode).	318
18.4.1	Vertical softkey menu.	318
18.4.2	Configuring the tool data display.	318
18.4.3	Data of the M tools (standard page - screen-).	319
18.4.4	Data of the T tools (standard page - screen-).	320
18.4.5	Data of the M/T tools (offset page - screen-).	321
18.4.6	Editing the tool table.	321
18.5	Operations with the tool table (full mode).	322
18.5.1	Editing the tool table.	322
18.6	Active-tools table.	323
18.6.1	Softkey menus.	324
18.6.2	Changing the tool of the spindle.	324
18.7	Table for the status of the tool change process.	325
18.8	Magazine table.	326
18.8.1	Softkey menus.	327
18.8.2	List of magazine positions.	328
18.8.3	Magazine information.	329
18.9	Operations with the magazine table.	331
18.9.1	Loading / unloading tools to / from the magazine.	331
18.9.2	Load / unload a tool to / from the tool changer arm.	333

CHAPTER 19 UTILITIES MODE

19.1	Interface description.	335
19.1.1	Softkey menus.	337
19.2	Set how to display the list of programs.	339
19.3	Select files and create folders.	340
19.4	Find in files.	341
19.5	Protection passwords.	342
19.6	Data safety backup. Backup - Restore.	344
19.7	Encrypting files.	346

CHAPTER 20 PLC

20.1	Appearance of the PLC mode.	349
20.1.1	Icon description (vertical softkeys).	350
20.2	"Programs" service.	351
20.2.1	Softkey menus. PLC project.	352
20.2.2	Encrypting/decrypting from the PLC.	353
20.2.3	Softkey menus. Files of the PLC project.	354
20.3	Program editing.	355
20.4	Editing in C language or mnemonic language.	357
20.4.1	Softkey "Analyze".	358
20.4.2	Softkey "File".	358
20.4.3	Softkey "Undo".	358
20.4.4	Softkey "Operations with blocks".	359
20.4.5	Softkey "Find/Replace".	360
20.4.6	Softkey "Customize".	361
20.5	Editing in contact (ladder) language (softkeys).	362
20.5.1	Softkey "Analyze".	363
20.5.2	Softkey "File".	363
20.5.3	"Edit" softkey.	364
20.5.4	Softkey "View".	366
20.5.5	Softkey "Marks".	366
20.5.6	Softkey "Find".	367
20.5.7	Softkey "Customize".	367
20.6	Program monitoring.	368
20.6.1	Softkey menus. Monitoring in C language or mnemonic language.	369
20.6.2	Softkey menus. Monitoring in contact (ladder) language.	370
20.7	"Commands" service.	372
20.7.1	Softkey menus. Options of the "Commands" service.	372
20.8	"Outputs" service.	373
20.8.1	Softkey menus. Options of the "Outputs" service.	374

20.9	"Logic analyzer" service.....	375
20.9.1	Editing logic analyzer data	376
20.9.2	Save, load and reset the analyzer configuration	378
20.9.3	Execute and analyze trace.....	379
20.9.4	Customize the appearance of the logic analyzer	379
20.10	"Monitoring" service	380
20.10.1	Description of resource tables	381
20.10.2	Definition of the table resources	382
20.10.3	Options of the "Monitoring" service (softkeys)	383
20.11	"Cross references" service.....	384
20.11.1	Softkey menus. Options of the "Cross reference" service	385
20.12	"Statistics" service.....	386
20.12.1	Softkey menus. Options of the "Statistics" service.....	387
20.13	"Messages" service.....	388
20.13.1	Softkey menus. Options of the "Messages" service.....	389
20.13.2	Editing the message and error table	390
20.13.3	Displaying PLC messages	391
20.13.4	Displaying PLC errors	392
20.13.5	Grouping the additional information text files in a single file.	393
20.13.6	Save, load and print a message and error table	394

CHAPTER 21**MACHINE PARAMETERS.**

21.1	Interface description.	396
21.1.1	Connection and machine parameter table.	396
21.1.2	M function customization table.	397
21.1.3	Kinematics configuration table.	398
21.1.4	OEM parameter table.	399
21.1.5	Horizontal softkey menu.	400
21.1.6	Vertical softkey menu.	400
21.2	Operations with tables.	401
21.2.1	Navigating the tables.	401
21.2.2	Machine parameter editing.	402
21.2.3	Import and export cross compensation and ballscrew data (position and error).	404
21.2.4	Validating machine parameters.	405
21.2.5	Saving and retrieving machine parameters from a file.	406
21.2.6	Print a table.....	407
21.2.7	Initialize a table.	407
21.2.8	Search for text or a value in the active table.	407

CHAPTER 22**SETUP ASSISTANCE**

22.1	Oscilloscope.	410
22.1.1	Interface description.....	410
22.1.2	Softkey menus	412
22.1.3	Configuration screen.....	414
22.1.4	Configure and execute the oscilloscope function.....	415
22.1.5	Machine parameter editing.	416
22.2	Bode diagram.	418
22.2.1	Interface description.....	418
22.2.2	Softkey menus	419
22.2.3	Machine parameter editing.	422
22.2.4	Configuration screen.....	423
22.3	Circularity (roundness) test.....	426
22.3.1	Interface description.....	426
22.3.2	Softkey menus	428
22.3.3	Configuring and executing the circularity (roundness) test.....	429
22.3.4	Configure the graphic environment.....	430
22.3.5	Define and execute the movement subroutine	431
22.3.6	Data capture for the graphic	432
22.3.7	Adjustment of the machine parameters involved.....	433
22.3.8	Validate the changes and save the configuration used	434
22.3.9	Machine parameters that may be modified.....	435

CHAPTER 23**DIAGNOSIS**

23.1	Appearance of the diagnosis mode.	437
23.1.1	Softkey menus	438
23.2	Configuration diagnosis	439
23.2.1	System diagnosis.....	439
23.2.2	Hard disk monitoring via S.M.A.R.T. data.....	439
23.2.3	Software diagnosis.....	440
23.2.4	Hardware diagnosis.....	441
23.2.5	CAN bus diagnostics.....	442

23.2.6	Sercos bus diagnostics.....	443
23.2.7	EtherCAT bus diagnostics.....	444
23.3	Report generation.....	445
23.4	View the history of errors and warnings issued by the CNC.....	446
23.5	Generate the Fagor file for error diagnosis.....	446
23.6	Access particular utilities of the operative system and other applications.....	446
23.7	User log.....	447
23.7.1	OEM configuration of the user log.....	447
23.7.2	Maximum size of the userLog1.txt file.....	448
23.8	Hardware registration using QR codes.....	449
23.9	Operating Terms.....	450
23.9.1	Generate the "Operating Terms" codes (for OEM only).....	451
23.9.2	Enter the CNC code to activate or cancel the temporary user license.....	453

CHAPTER 24

APPS.

24.1	DiskMonitor application. Changing the work mode and device registration.....	456
24.2	Network Settings. Configuring network properties.....	458
24.3	System. Set the date and time.....	459
24.4	APPS. Send emails.....	460
24.4.1	Configure and activate the application.....	462
24.5	Configuring the brightness and contrast of the monitors.....	464
24.6	Program Optimizer. Preprocessing of CAM programs.....	464

CHAPTER 25

KINEMATICS CALIBRATION.

25.1	Kinematics calibration (Kinematic offsets).....	467
25.1.1	Page 1. Cycle data.....	467
25.1.2	Page 2. Calibration progress.....	470
25.1.3	Page 3. Calibration result.....	471
25.1.4	Page 4. Display of calculated data.....	472
25.1.5	How to calibrate a kinematics.....	473
25.1.6	File with the results of the calibration.....	474
25.2	Rotational kinematics calibration.....	476
25.2.1	Page 1. Cycle data.....	476
25.2.2	Page 2. Display of the progress of the rotational calibration.....	479
25.2.3	Page 3. Display and activate the different offset tables calculated.....	480



CNCelite
8058 8060
8065 8070

REF: 2508

ABOUT THIS MANUAL.

Title.	Operating manual.
Models.	CNCelite 8058 8060 8065 8070
Type of documentation.	End user manual. This manual describes the CNC interface and the various working modes, as well as how to operate each one.
Remarks.	Always use the manual reference associated with the software version or a later manual reference. You can download the latest manual reference from the download section on our website. Limitations. The availability of some of the features described in this manual are dependent on the acquired software options. Moreover, the machine manufacturer (OEM) customizes the CNC performance of each machine using the machine parameters and the PLC. Because of this, the manual may describe features that are not available for the CNC or the machine. Consult the machine manufacturer for the available features.
Electronic document.	man_elite_58_60_65_70_opt.pdf. Manual available from the download section of our website.
Language.	English [EN]. Refer to our website, download area, the languages available for each manual.
Date of publication.	August, 2025
Manual reference	Ref: 2508
Associated version.	v2.50.05
Responsibility exemption.	The information described in this manual may be subject to changes due to technical modifications. Fagor Automation reserves the right to change the contents of this manual without prior notice.
Trademarks.	This manual may contain third-party trademarks or trade names, however, they do not have them associated ® or ™ symbols. 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.
Website / Email.	http://www.fagorautomation.com Email: contact@fagorautomation.es



CNCelite
8058 8060
8065 8070

REF: 2508



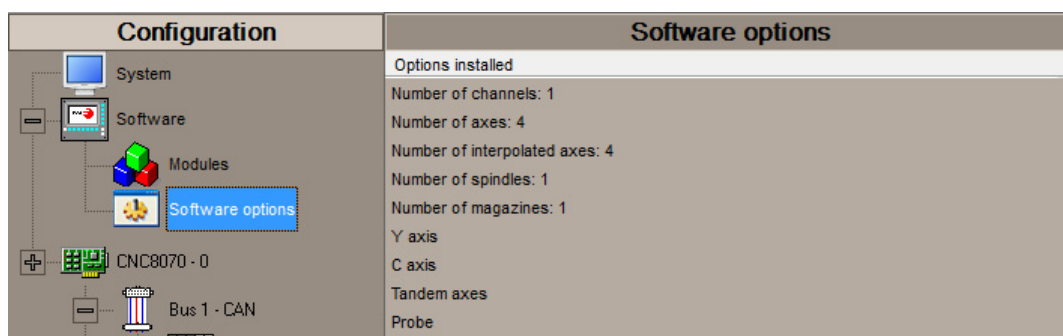
CNCelite
8058 8060
8065 8070

REF: 2508

About the product.

SOFTWARE OPTIONS.

Some of the features described in this manual are dependent on the acquired software options. The active software options for the CNC can be consulted in the diagnostics mode (accessible from the task window by pressing [CTRL] [A]), under software options. Consult Fagor Automation regarding the software options available for your model.



Software option	Description.
SOFT ADDIT AXES	Option to add axes to the default configuration.
SOFT ADDIT SPINDLES	Option to add spindles to the default configuration.
SOFT ADDIT TOOL MAGAZ	Option to add magazines to the default configuration.
SOFT ADDIT CHANNELS	Option to add channels to the default configuration.
SOFT 4 AXES INTERPOLATION LIMIT	Limited to 4 interpolated axes.
SOFT DIGITAL SERCOS	Option for a Sercos digital bus.
SOFT THIRD PARTY DRIVES	Option to use EtherCAT third party drives.
SOFT THIRD PARTY I/Os	Option to use third party I/O modules.
THIRD PARTY FEEDBACK	Option to use third-party encoders (BiSS and EnDat protocol). Encoders associated with simulated axes do not require this option.
SOFT OPEN SYSTEM	Option for open systems. The CNC is a closed system that offers all the features needed to machine parts. Nevertheless, at times there are some customers who use third-party applications to take measurements, perform statistics or other tasks apart from machining a part. This feature must be active when installing this type of application, even if they are Office files. Once the application has been installed, it is recommended to close the CNC in order to prevent the operators from installing other kinds of applications that could slow the system down and affect the machining operations.

Software option	Description.
SOFT i4.0 CONNECTIVITY PACK	Options for Industry 4.0 connectivity. This option provides various data exchange standards (for example, OPC UA), which allows the CNC (and therefore the machine tool) to be integrated into a data acquisition network or into a MES or SCADA system.
SOFT EDIT/SIMUL	Option to enable edisimu mode (edition and simulation) on the CNC, which can edit, modify and simulate part programs.
SOFT DUAL-PURPOSE (M-T)	Option to enable the dual-purpose machine, which allows milling and turning cycles. On Y-axis lathes, this option allows for pockets, bosses and even irregular pockets with islands to be made during milling cycles. On a C-axis mill, this option allows turning cycles to be used.
SOFT TOOL RADIUS COMP	Option to enable radius compensation. This compensation programs the contour to be machined based on part dimensions without taking into account the dimensions of the tool that will be used later on. This avoids having to calculate and define the tool paths based on the tool radius.
SOFT PROFILE EDITOR	Option to enable the profile editor in edisimu mode and in the cycle editor. This editor can graphically, and in a guided way, define rectangular, circular profiles or any profile made up of straight and circular sections and it can also import dxf files. After defining the profile, the CNC generates the required blocks and add them to the program.
SOFT HD GRAPHICS In a multi-channel system, this feature requires the MP-PLUS (83700201) processor.	High definition solid 3D graphics for the execution and simulation of part-programs and canned cycles of the editor. During machining, the HD graphics display, in real time, the tool removing the material from the part, allowing the condition of the part to be seen at all times. These graphics are required for the collision control (FCAS).
SOFT IIP CONVERSATIONAL	The IIP (Interactive Icon-based Pages) mode, or conversational mode, works with the CNC in a graphical and guided way based on predefined cycles. There is no need to work with part programs, have any previous programming knowledge or be familiar with Fagor CNCs. Working in conversational mode is easier than in ISO mode, as it ensures proper data entry and minimizes the number of operations to be defined.
SOFT RTCP This feature requires the MP-PLUS (83700201) processor.	Option to enable dynamic RTCP (Rotating Tool Center Point) required to machine with 4, 5 and 6 axis kinematics; for example, angular and orthogonal spindles, tilting tables, etc. The RTCP orientation of the tool may be changed without modifying the position occupied by the tool tip on the part.
SOFT C AXIS	Option to enable C-axis kinematics and associated canned cycles. The machine parameters of each axis or spindle indicate whether it can operate as a C axis or not. For this reason, it is not necessary to add specific axes to the configuration.
SOFT Y AXIS	Option to enable lathe Y-axis kinematics and associated canned cycles.
SOFT TANDEM AXES	Option to enable tandem axle control. A tandem axis consists of two motors mechanically coupled to each other forming a single transmission system (axis or spindle). A tandem axis helps provide the necessary torque to move an axis when a single motor is not capable of supplying enough torque to do it. When activating this feature, it should be kept in mind that for each tandem axis of the machine, another axis must be added to the entire configuration. For example, on a large 3-axis lathe (X Z and tailstock), if the tailstock is a tandem axis, the final purchase order for the machine must indicate 4 axes.

Software option	Description.
SOFT SYNCHRONISM	Option to enable the synchronization of paired axes and spindles, in speed or position, and through a given ratio.
SOFT KINEMATIC CALIBRATION	Option to enable tool calibration. For the first time, this kinematics calibration allows for the kinematics offsets to be calculated using various approximate data and, also, from time to time to correct any possible deviations caused by day-to-day machining operations.
SOFT 60 HSSA I MACHINING SYSTEM	Option to enable the HSSA-I (High Speed Surface Accuracy) algorithm for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved.
SOFT HSSA II MACHINING SYSTEM	Option to enable the HSSA-II (High Speed Surface Accuracy) algorithm for high speed machining (HSC). This new HSSA algorithm allows for high speed machining optimization, where higher cutting speeds, smoother contours, a better surface finishing and greater precision are achieved. The HSSA-II algorithm has the following advantages compared to the HSSA-I algorithm. • Advanced algorithm for point preprocessing in real time. • Extended curvature algorithm with dynamic limitations. Improved acceleration and jerk control. • Greater number of pre-processed points. • Filters to smooth out the dynamic machine behavior.
SOFT TANGENTIAL CONTROL	Option to enable tangential control. "Tangential Control" maintains a rotary axis always in the same orientation with respect to the programmed tool path. The machining path is defined on the axes of the active plane and the CNC maintains the orientation of the rotary axis along the entire tool path.
SOFT PROBE	Option to enable functions G100, G103 and G104 (for probe movements) and probe canned cycles (which help to measure part surfaces and to calibrate tools). For the laser model, it only activates the non-cycle function G100. The CNC may have two probes; usually a tabletop probe to calibrate tools and a measuring probe to measure the part.
SOFT FVC STANDARD SOFT FVC UP TO 10m3 SOFT FVC MORE TO 10m3	Options to enable volumetric compensation. The precision of the parts is limited by the machine manufacturing tolerances, wear, the effect of temperature, etc., especially on 5-axis machines. Volumetric compensation corrects these geometric errors to a larger extent, thus improving the precision of the positioning. The volume to be compensated is defined by a point cloud and for each point the error to be corrected is measured. When mapping the total work volume of the machine, the CNC knows the exact position of the tool at all times. There are 3 options, which depend on the size of the machine. • FVC STANDARD: Compensation for 15625 points (maximum 1000 points per axis). Quick calibration (time), but less precise than the other two, but sufficient for the desired tolerances. • FVC UP TO 10m3: Volume compensation up to 10 m³. More accurate than FVC STANDARD, but requires a more accurate calibration using a Tracer or Tracker laser). • FVC MORE TO 10m3: Volume compensation greater than 10 m³. More accurate than FVC STANDARD, but requires a more accurate calibration using a Tracer or Tracker laser.

Software option	Description.
SOFT CONV USER CYCLES	Option to enable user conversational cycles. The user and the OEM can add their own canned cycles (user cycles) using the FGUIM application that comes installed on the CNC. The application offers a guided way to define a new component and its softkey menu without having to be familiar with script languages. User cycles work in a similar way as Fagor canned cycles.
SOFT PROGTL3	Option to enable the ProGTL3 programming language (ISO language extension), allowing profiles to be programmed using a geometric language and without the need to use an external CAD system. This language can program lines and circles where the end point is defined as the intersection of 2 other sections, pockets, ruled surfaces, etc.
SOFT PPTRANS	Option to enable the program translator, which can convert programs written in other languages to Fagor ISO code.
SOFT DMC	Option to enable the DMC (Dynamic Machining Control). DMC adapts the feedrate during machining to maintain the cutting power as close as possible to ideal machining conditions.
SOFT FMC	Option to enable the FMC (Fagor Machining Calculator). The FMC application consists of a database of materials to be machined and machining operations, with an interface to choose suitable cutting conditions for these operations.
SOFT FFC	Option to enable the FFC (Fagor Feed Control). During the execution of a canned cycle of the editor, the FFC function makes it possible to replace the feedrate and speed programmed in the cycle with the active values of the execution, which are acted upon by the feed override and speed override.
SOFT 60/65/70 OPERATING TERMS	Option to enable a temporary user license for the CNC, which is valid until the date set by the OEM. While the license is valid, the CNC will be fully operational (according to the purchased software options).
SOFT FCAS	Option to enable the FCAS (Fagor Collision Avoidance System). The FCAS option, within the system limitations, monitors the automatic, MDI/MDA, manual and tool inspection movements in real time, so as to avoid collisions between the tool and the machine. The FCAS option requires that the HD graphics to be active and that there is a defined a model configuration of the machine adjusted to reality (.xca file), which includes all its moving parts.
SOFT GENERATE ISO CODE	ISO generation converts canned cycles, calls to subroutines, loops, etc. into their equivalent ISO code (G, F, S, etc functions), so the user can modify it and adapt it to his needs (eliminate unwanted movements, etc.). The CNC generates the new ISO code while simulating the program, either from the DISIMU mode or from the conversational mode.
SOFT PWM CONTROL	Option to enable PWM (Pulse - Width Modulation) control on laser machines. This feature is essential for cutting very thick sheets, where the CNC must create a series of PWM pulses to control laser power when drilling the initial point. This function is only available for Sercos bus control systems and must also use one of the two fast digital outputs available from the central unit.
SOFT GAP CONTROL	Option to enable gap control, which makes it possible to set a fixed distance between the laser nozzle and the sheet surface with the use of a sensor. The CNC compensates the difference between the distance measured by the sensor and the programmed distance with additional movements on the axis programmed for the gap.

Software option	Description.
SOFT MANUAL NESTING	Option to enable nesting in the automatic option. Nesting consists of creating a pattern on the sheet material using previously defined figures (in dxf, dwg or parametric files), so as to use most of the sheet as possible. Once the pattern has been defined, the CNC creates a program. During manual nesting, the operator distributes the parts on top of the sheet material.
SOFT AUTO NESTING	Option to enable nesting in the automatic option. Nesting consists of creating a pattern on the sheet material using previously defined figures (in dxf, dwg or parametric files), so as to use most of the sheet as possible. Once the pattern has been defined, the CNC creates a program. During automatic nesting, the application distributes the figures on the sheet material and optimizes the spaces.
SOFT DRILL CYCL OL	Option to enable ISO drilling cycles (G80, G81, G82, G83).



CNCelite
8058 8060
8065 8070

REF: 2508

EC DECLARATION OF CONFORMITY, WARRANTY CONDITIONS AND QUALITY CERTIFICATES.

EC-DECLARATION OF CONFORMITY

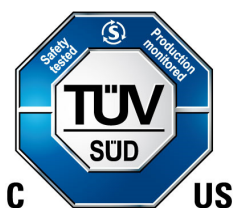


The declaration of conformity is available from the downloads section of the Fagor Automation corporate website.

<https://www.fagorautomation.com/en/downloads/>

Type of file: Declaration of conformity.

NRTL CERTIFICATE FOR USA AND CANADA



The quality certificates are available from the company label on the Fagor Automation corporate website.

<https://www.fagorautomation.com/en/sections/quality/>

WARRANTY TERMS

The sales and warranty conditions are available from the downloads section of the Fagor Automation corporate website.

<https://www.fagorautomation.com/en/downloads/>

Type of file: General sales - warranty conditions.



CNCelite
8058 8060
8065 8070

REF: 2508



CNCelite
8058 8060
8065 8070

REF: 2508

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 or material damage originated from not complying with these basic safety rules.



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 access the interior 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 REPAIRS

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 access the interior 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 HARM

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 electric shocks.

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. Also, before connecting the inputs and outputs of this product, make sure that the ground connection has been done.

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, harm or damages, do not work in explosive environments.



CNCelite
8058 8060
8065 8070

REF: 2508

PRECAUTIONS AGAINST DAMAGE TO THE PRODUCT

Work 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 this unit in the proper place.	It is recommended, whenever possible, to install the CNC away from coolants, chemical product, blows, etc. that could damage it. This unit meets 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 mains as the unit. - Nearby portable transmitters (radio-telephones, Ham radio transmitters). - Nearby radio / TC transmitters. - Nearby arc welding machines. - Nearby high voltage lines.
Enclosures.	It is up to the manufacturer to guarantee that the enclosure where the unit has been installed meets all the relevant directives of the European Union.
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.
Connecting the power supply to ground.	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.
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 in the manual.



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



Warning or caution 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).



CNCelite
8058 8060
8065 8070

REF: 2508

RETURNING CONDITIONS.

Pack it in its original package along with its original packaging material. If you do not have the original packaging material, pack it as follows:

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



CNCelite
8058 8060
8065 8070

REF: 2508



CNCelite
8058 8060
8065 8070

REF: 2508

CNC MAINTENANCE.

CLEANING

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

To clean the operator panel and the monitor, a smooth cloth should be used which has been dipped into de-ionized water and /or non abrasive dish-washer soap (liquid, never powder) or 75° alcohol. Never use air compressed at high pressure to clean the unit because it could cause the accumulation of electrostatic charges that could result in electrostatic shocks.

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, benzole, 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 access the interior of this unit.



CNCelite
8058 8060
8065 8070

REF: 2508

NEW FEATURES.

Manual reference: Ref: 2508
Date of publication: August, 2025
Associated software: v2.50.05

Below is a list of the features added in this software version and the manuals that describe them.

List of features.	Manual
New software option "THIRD PARTY FEEDBACK". - Option to use third-party encoders (BiSS and EnDat protocol). - Encoders associated with simulated axes do not require this option.	
Simulator. - CNC simulator with support for 64-bit Elkhart Lake. - Single setup for free simulator, paid simulator and CNC. The same setup works for CNC (if the Hardware ID exists), paid simulator (if the license file exists) or free simulator (if there is no Hardware ID or license file).	
The PLC comes with the _EMERGEN signal at 0, so that a restore can be done as soon as you receive the CNC without having to modify the PLC.	
PLC Offset. Apply the offset active on the virtual axis to the trihedron axes.	[INST]
Configuring the SSI protocol. - Configuration of start of reading on an SSI encoder. - Machine parameters: SSIGAPCLK	[INST]
Multi-axis: Verification of loop directions. - To facilitate setup, the CNC allows you to close the position loop with the first feedback (SPEEDFBID) and view the second feedback count (POSITIONFBID) with the A.POS2NC.xn variable. This is controlled using the FBMIXTIME=-1 parameter. - Machine parameter: FBMIXTIME	[INST]
The number of global user variables (V.S.) increases from 118 to 250.	[PRG] [VAR]
Change of permissions of the following variables. Variable that can be read and written via program and interface. - Variable: (V.)MPCMP.POSERROR[point].table Error to be compensated at each point, for positive displacements. - Variable: (V.)MPCMP.NEGERROR[point].table Error to be compensated at each point, for negative displacements.	[VAR]
The Operating Terms page shows the Hardware ID of the CNC.	

[CYC-M] Machining canned cycles (-M- model).

[EASY] Easyplane.

[ERR] Solución de errores.

[INST] Installation manual.

[PRG] Programming manual.

[PROGTL3] Lenguaje ProGTL3.

[VAR] CNC variables.



CNCelite
8058 8060
8065 8070

REF: 2508

DESCRIPTION OF THE KEYS

1

1.1 Keyboard.

Function keys / Help key.

Key.	Function.
	Keys F1 through F12 select the options of the softkey menus.

Key.	Function.
	Display of CNC help and error messages.

Auxiliary key / Calculator (not available on all models).

Key.	Function.
	Change the status of an icon. In the MC/TC mode, it toggles between the standard and the auxiliary screens.

Key.	Function.
	Calculator.

Browsing keys.

Key.	Function.
	Main menu.
	It is used to switch between the different windows of the screen.
	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.

Key.	Function.
	OEM-configurable key that can carry out one of the following actions. • Open the pages in active mode; use [SHIFT] to reverse the sequence. • Enter the CNC channels. • Displays the available pages and channels in the softkey menus.

Work modes.

Key.	Function.
	Automatic mode.
	Manual mode.
	EDISIMU mode.
	MDI/MDA mode.

Key.	Function.
	User tables.
	Tool and magazine table.
	Utilities mode.
	OEM-configurable key that can carry out one of the following actions. • Enter the work modes. • 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.

Key.	Function.
   	The arrow keys move the cursor one position to the left, right, up or down.
 	The home and end keys move the cursor the beginning or end of the line.

Key.	Function.
 	The previous-page or next-page keys show the previous or next page at the part-program or PLC program editor.
	The tab key moves the cursor to the next field of the active menu.

Editing keys.

Key.	Function.
	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.
	It toggles between uppercase and lowercase letters.
	Hold this key down and key in the corresponding ASCII code.
 	Delete.
	Insert or overwrite.

Key.	Function.
	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.
	Key to validate commands, data and program blocks of the editor.
 	Delete.
	Escape key, to cancel the current action without assuming the changes.

1.

DESCRIPTION OF THE KEYS
Keyboard.

1.1.1 Fagor logo key (only HORIZONTAL KEYB 2.0 + TOUCHPAD models)

Keys.	Meaning.
 + [1] .. [0]	Keys [F1] to [F10].
 + [-]	Key [F11].
 + [=]	Key [F12].
 + [S]	Cancel or activate the keystroke sound.
 + [T]	Cancel or activate the touchpad (touch panel).
 + [C]	Change the function of the axis keys on the numeric keypad; write the name of axis1 to axis6 / always write X, Y, Z, A, B, C.
 + [D]	Minimize everything / Show the last active application.
 + [P]	Print screen; can be combined with [SHIFT] and with [ALT].
 + [E]	Open the Windows Explorer.

1.1.2 Numeric keypad (only HORIZONTAL KEYB 2.0 + TOUCHPAD models).

Numeric keypad optimized to operate with the CNC (keys for axes, feedrate, speed, etc). The second function of the keys is available with a long press of the key; the [SHIFT] key is not necessary. The behavior of the three axis keys can be modified with the [FAGOR]+[C] keys.



Option 1.

The keys write the name of the first six axes of the channel.

Option 2.

The keys always write the characters X Y Z A B C.

1.1.3 Touchpad (only HORIZONTAL KEYB 2.0 + TOUCHPAD models).

The actions of touching with one or two fingers work in reverse if the mouse is configured for left-handed people. The touchpad can be disabled or activated with the [FAGOR]+[T] key.

1.

DESCRIPTION OF THE KEYS
Keyboard.

Primary button.



Tap lightly with one finger to simulate a mouse click.

Double click.



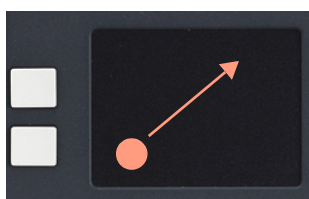
Tap lightly twice with one finger to simulate the double click of a mouse.

Secondary button.



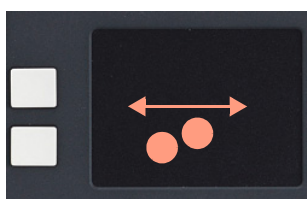
Tap lightly with two fingers to simulate the right click of a mouse.

Move the cursor.



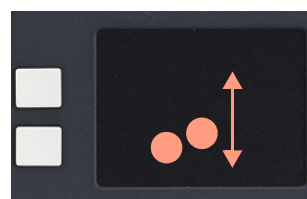
Move a finger to move the cursor.

Move screen (pan).



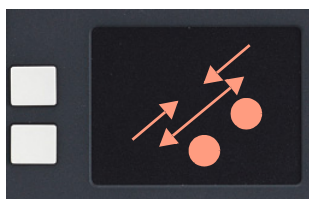
Move two fingers to the left and right to move horizontally (pan).

Move screen (scroll).



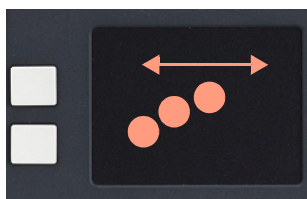
Move two fingers up and down to move vertically (scroll).

Zoom.



Pinch or separate two fingers to zoom in or out.

Turn the page.



Drag three fingers horizontally to move forward or back a page.

Minimize the window.



Drag three fingers downward to minimize the active window.

Start menu.



Drag three fingers upward to access the start menu.

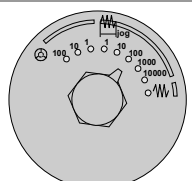
1.2 Operator panel.

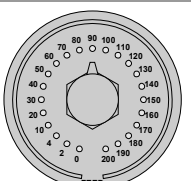
Jog keyboard.

Key.	Function.
 	Keys to select axes and jog them in the positive direction.
 	Keys to select axes and jog them in the negative direction.

Key.	Function.
   	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.

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

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

Execution keys.

Key.	Function.
	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 key. It initializes the system setting the initial conditions as defined by machine parameters.

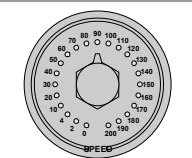
Key.	Function.
	Single block execution mode. When selecting the "single block" execution mode, the program simulation will be interrupted at the end of each block.
	Machine reference zero (home) search.

Spindle control.

Key.	Function.
	Start the spindle clockwise.
	Stop the spindle.
	Start the spindle counterclockwise.

Key.	Function.
 	Percentage variation of the spindle speed.
	Spindle orientation.
	Constant Surface Speed (for lathe model only).

Speed selector (not available on all models)

Selector.	Function.
	Selector of percentage of spindle speed override 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.).

1.3 Keyboard shortcuts.

1.

DESCRIPTION OF THE KEYS

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]	Tool and magazine table. 
[CTRL] + [F12]	Utilities mode. 
[CTRL] + [K]	Calculator.

Browsing keys.	
[CTRL]+[F1]	Previous menu. 
[CTRL]+[F2]	Switch window. 
[CTRL]+[F3]	Switch screens. 
[ALT]+[B]	Two-color key. 



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.	

DESCRIPTION OF THE KEYS

Keyboard shortcuts.

1.

1.

DESCRIPTION OF THE KEYS

Keyboard shortcuts.



CNCelite
8058 8060
8065 8070

REF: 2508

2.1 Turning the CNC on and off

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 (automatic or jog) selected by the machine manufacturer.

Messages and options during the start-up.

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

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

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

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

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

- Setup mode.

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

- User mode.

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

Turning the CNC off.



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

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

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

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

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

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

- Setup mode.

If the CNC is started up in setup mode, it offers the option to save a backup copy. Once setup is completed, it is recommended 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.

GENERAL CONCEPTS
Turning the CNC on and off

2.1.1 Emergency shutdown with UPS.

The central unit is powered by an external DC power supply (24 V DC). Optionally, it is possible to connect an external UPS to help the equipment to shut down correctly in the event of a power failure. When a power failure occurs (24 V DC voltage drop), and the CNC has a UPS, it responds as follows:

The supply is interrupted for less than 2 seconds.

The CNC 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.

The supply is interrupted for more than 2 seconds.

The CNC displays the corresponding error and starts the automatic shutdown sequence.

- 1 The CNC stops the running program.
- 2 Shutdown of the CNC application.
- 3 Shutdown of the system.
- 4 Battery disconnection.

2.

GENERAL CONCEPTS

Turning the CNC on and off

2.2 Work modes and software protection at the CNC.

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

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

Icon.	Work mode.
	Administrator mode.
	Setup mode.
	User mode.

Changing work mode (diskMonitor window).

The DiskMonitor application toggles between the CNC work and protection mode (administrator, setup and user modes). This process implies having to restart the unit and, if the manufacturer has defined it this way, it will also be necessary to enter the corresponding password. See ["24.1 DiskMonitor application. Changing the work mode and device registration."](#) on page 456.

Protecting or unprotecting folders (diskMonitor application).

The DiskMonitor application allows you to protect or unprotect any system folder or file that is not essential for the proper operation of the CNC. See ["24.1 DiskMonitor application. Changing the work mode and device registration."](#) on page 456.

Permanently registering devices in setup and user modes (diskMonitor application).

When devices are connected physically to the device (for example, a pendrive) in setup or user modes, the device identifies and loads the drivers during each startup. The equipment does not register these devices permanently because the disk is protected. To force the registration of these devices you must use the DiskMonitor application. See ["24.1 DiskMonitor application. Changing the work mode and device registration."](#) on page 456.

2.

GENERAL CONCEPTS
Work modes and software protection at the CNC.

Administrator 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.



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

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 work mode, it will request the access password.

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 the following image on the desk, with yellow background, indicating the active work 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 work mode, it will request the access password.

2.

GENERAL CONCEPTS

Work modes and software protection at the CNC.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

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.

Restrictions for working in Setup mode and switching to User mode.

The purpose of the following restrictions is for the OEM to deliver his machines with the CNC in protected work 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 (work 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.

GENERAL CONCEPTS
Work modes and software protection at the CNC.



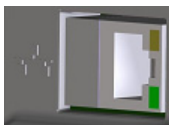
CNCelite
8058 8060
8065 8070

REF: 2508

2.3 Ethernet.



Do not handle the connectors while the CNC is turned on. Before doing this, make sure that the CNC is unplugged from the power outlet.



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.

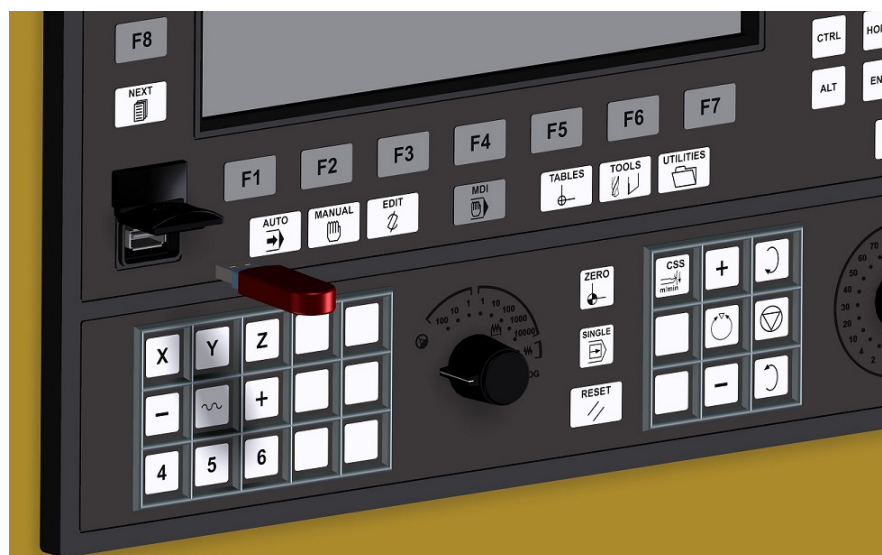
2.4 USB ports..



Do not run part programs from a USB port. Fagor Automation recommends using the USB port only to exchanging data, such for programs, reports, etc. If you require more storage space, then use a CFast disk.



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



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

- Pendrive.
- Keyboard.
- Mouse.

2.

GENERAL CONCEPTS
Ethernet.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

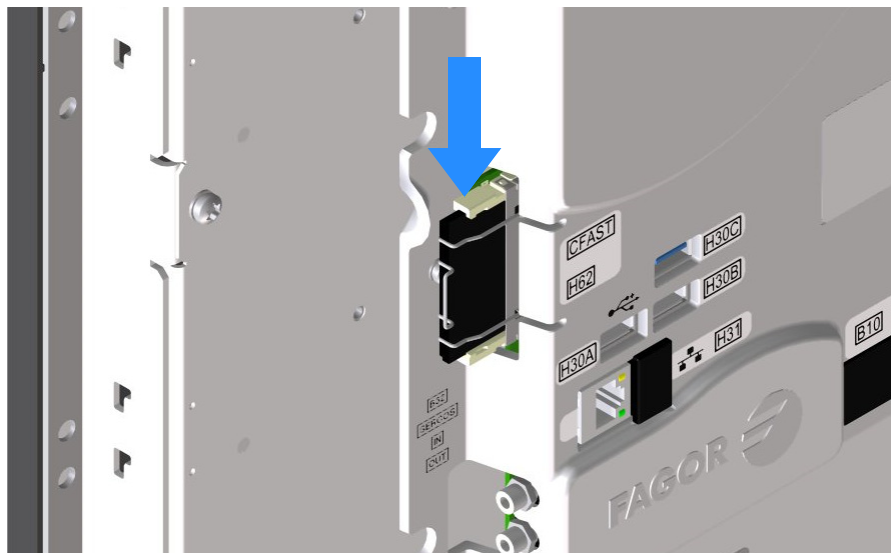
2.5 Disco CFast (Q7-A platform).



Do not handle the compact flash (including extracting it or inserting it) with the CNC powered on. Before handling the compact flash, make sure that the CNC is unplugged from the power outlet.



To help increase storage space, Fagor Automation lists several CFast cards in its catalog; when using a third-party CFast, always use an industrial grade CFast SLC as these support temperatures from between -40°C and +85°C (-40 °F and 185 °F) and can last five years with constant day-to-day writing. Fagor Automation shall not be held responsible for any problems caused by using lower-quality CFast cards.



Q7-A platform (8065).

Additional storage space for user programs.

2.

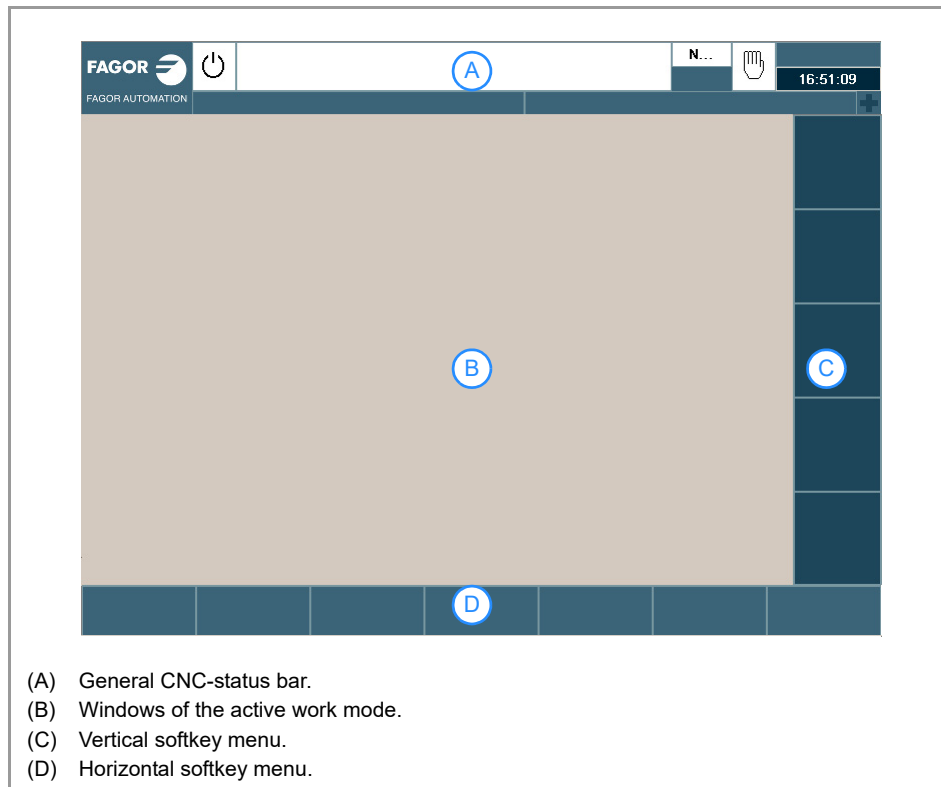
GENERAL CONCEPTS

Disco CFast (Q7-A platform).

HOW TO OPERATE THE CNC

3

3.1 General description of the interface.



General CNC-status bar.

The top bar shows information about the programme running, active operating mode, PLC messages, etc. See ["3.3 General CNC-status bar."](#) on page 50.

Windows of the active work mode.

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

Softkey menus.

The softkeys are the option menus offered in each operating mode, to perform actions, configure the mode, etc. The standard Fagor configuration shows 7 horizontal and 5 vertical softkeys. The horizontal menu options are selected using [F1] to [F7], and the vertical ones using [F8] to [F12]. See ["3.2.1 Expanded softkey menus."](#) on page 49.



CNCelite
8058 8060
8065 8070

REF: 2508

The vertical menu can be displayed on the left or right side of the interface, depending on the OEM configuration (VMENU parameter). The OEM can also configure the way the softkey menu (SFTYPE parameter) is used, either through menus and sub-menus (there are different softkey levels within a work mode) or through "drop-down" menus (there is only 1 softkey level, no sub-menus). See ["3.2.2 Horizontal menu with standard or drop-down soft keys."](#) on page 49.

3.

HOW TO OPERATE THE CNC

General description of the interface.



CNCelite
8058 8060
8065 8070

REF: 2508

3.2 Configuring the softkey menus.

3.2.1 Expanded softkey menus.

The number of softkeys can be configured by the OEM, who can expand each menu up to 12 softkeys, for example, for monitors larger than 15". The softkeys are activated using the keys [F1] through [F12]. In the expanded menus, the keys [F1] to [F7] activate the first seven soft keys of the horizontal menu (starting from the left) and the keys [F8] to [F12] activate the first five soft keys of the vertical menu (starting from the bottom).

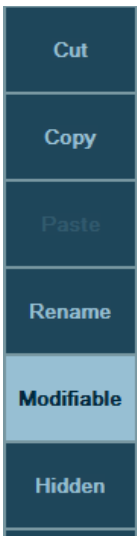
Displaying the dropdown menus.

Depending on the configuration (resolution and number of softkeys), if the softkeys are too small to display their image then only their text will be displayed.

Vertical menu with 5 softkeys.



Vertical menu with 8 softkeys.



3.2.2 Horizontal menu with standard or drop-down soft keys.

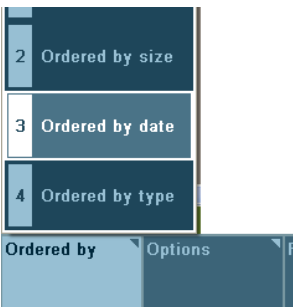
The OEM configures the functions of the horizontal softkey menu (SFTYPE parameter) in one of the following ways.

Standard softkey menu.

Softkey tree based on menus and submenus. The [F1] to [F7] keys access the submenus and also select the menu softkeys. To return to the main menu, press the [BACK] key (previous menu).

Pop-up softkey menu.

Softkey tree based on popup menus so there is only one softkey level.



The softkey menu expands and shrinks with keys [F1] through [F7]. The arrow keys move the focus along the menu. The menu softkeys are selected by pressing the [ENTER] key and the softkey number on the numeric keyboard.

3.

HOW TO OPERATE THE CNC
Configuring the softkey menus.

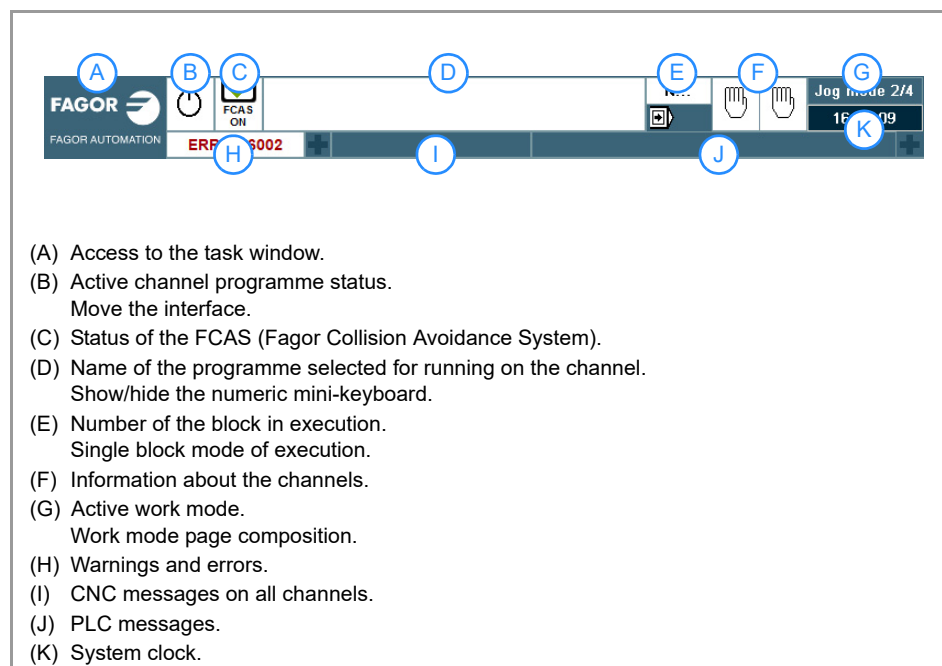


CNCelite
8058 8060
8065 8070

REF: 2508

3.3 General CNC-status bar.

The status bar of the CNC (top of the screen) shows the following information. The colour of the bar will vary depending on the status of the programme being run.



Color.	Meaning.
White	Programmed stopped.
Green.	Program in execution.
Dark green.	Program interrupted.
Red.	Program in error.

Access to the 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 ["3.6 Task window."](#) on page 58.

Active channel programme status.

Icon showing the status of the program of the active channel:

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.

Move the interface while dragging it with the mouse.

For monitors larger than 15", the CNC window may only cover a portion of the screen, thereby leaving room for a keyboard or virtual jog panel. In this case, the user can move the interface by using the mouse to click on the program status icon for the active channel, found on the top status bar. Double click on this icon and the interface is positioned in the upper left corner (POSX=0 POSY=0).

Status of the FCAS (Fagor Collision Avoidance System).

Icon showing FCAS (Fagor Collision Avoidance System) status: The CNC will display this icon when the software option associated with the FCAS is available. See ["10. FCAS \(Fagor Collision Avoidance System\)." on page 197.](#)

Icon.	Meaning.
	Active feature.
	Feature is disabled; the machine configuration is not loaded (xca file), the HD graphics are not selected or the PLC has disabled this feature.
 	Blinking icon. - The axes are close to the collision zone. - CPU overload during collision detection calculations.

Name of the programme selected for running on the channel.

The CNC allows selecting and executing a different program in each channel. The bar shows the name of the programme selected in the active channel.

Show/hide the numeric mini-keyboard.

See ["3.12 Mini-numeric keypad." on page 66.](#)

Number of the block being executed and block by block execution mode.

Number of the block (not line) being executed, if it has been programmed. The bottom icon indicates that the Single-block execution mode is active.

Information about the channels.

Number of channels available and active channel (indicated in blue). The icon indicates the operating mode of each channel. Click with the mouse or press on a touch-screen to enter the desired mode of channel. Clicking on the active channel icon has the same effect as pressing the [ESC] key. The key combination [ALT]+[S] displays the channel synchronisation window. See ["3.5 Execution channels." on page 56.](#) See ["3.7 Channel synchronization window." on page 59.](#)

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

3.

HOW TO OPERATE THE CNC

General CNC-status bar.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Active work mode (automatic, manual, etc.) and page composition.

Active work mode (automatic, manual, etc.) selected screen number and total number of screens available. By clicking on the active work mode, the CNC shows the list of available pages and which ones are visible.

Warnings and errors.

The warning and error windows can be hidden and displayed using the key sequence [ALT]+[W], by double-clicking on them or by clicking on the "+" symbol. 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. See ["3.8 CNC warning and errors window."](#) on page 60.

CNC messages on all channels.

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

PLC messages.

When the PLC triggers a message, this area displays the message number and its associated text. If there is more than one active message, the bar will show the highest priority message (the one with the lowest number) and the "+" symbol next to it. To display the list of active messages, press the key combination [CTRL]+[M] or click on the PLC message line. See ["3.9 PLC messages."](#) on page 62.

System clock.

The system clock shows the time of the operating system. The operating system clock can be adjusted using the application available in diagnostic mode. See ["24.3 System. Set the date and time."](#) on page 459.

3.

HOW TO OPERATE THE CNC
General CNC-status bar.

3.4 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 not 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.
	Automatic mode.
	Manual mode.
	EDISIMU mode (editing and simulation).
	MDI/MDA mode.
	User tables (zero offsets, fixtures and arithmetic parameters).
	Tool and magazine table.
	Utilities mode.
	Configurable mode. OEM-configurable key that can access to one of the following operating modes. • Machine parameters. • PLC. • Diagnosis mode. • Setup assistance.

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.



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

3.4.1 Description of the various operating modes

The CNC offers the following operating modes.

Operation mode.	Operations allowed.
Automatic mode	• Display the data regarding axis position, "M" and "G" function history, active tool, spindle speed, axis feedrate, etc. • Execute a part-program in "single block" or "automatic" mode. • Set the execution conditions (first and last blocks) before executing the program. • Searching a block. Recover the history of a program up to a particular block, with the option to change the active F, S, M and H functions and resume program execution from that block. • Show a graphic representation of the program being executed. • Perform a tool inspection while executing the program.
Jog mode	• Display the data regarding axis position, "M" and "G" function history, active tool, spindle speed, axis feedrate, etc. • Home the axes (Machine reference zero search). • Jog the axes with the handwheels or with the JOG keys. • Move an axis to a coordinate after selecting the target point. • Preset a coordinate. • Act upon the master spindle using the Jog keyboard. • Change tools. • Activate external devices of the machine using the keys of the upper side of the operator panel. The external devices associated with each key must be defined by the machine manufacturer. • Calibrate a tool in jog mode (without probe), in semi-automatic mode (when using a table-top probe) or using the tool calibration cycle (also when using a table-top probe). • Part centering (mill model). • Turn the CNC off from the softkey menu.
MDI/MDA mode	• Edit and execute individual program blocks. • Save the set of blocks executed as an independent program.
EDISIMU mode (editing and simulation)	• Edit, modify or view the contents of a part-program. • Simulate the execution of a program displaying a graphic representation of the program being simulated. • Estimate the total program execution time and the machining time for each tool. • Syntax test of the program.
User tables	• Edit and modify the tables for zero offsets, fixtures and arithmetic parameters. • Save the content of a table. • Recover the content of a table. • Print the content of a table.
Tool table Magazine Table	• Editing and modifying the tool table • Editing and modifying the tool magazine table. Display and manage the tool arrangement in the magazines and in the tool changing arms (if any). • Display the information about the tool change process; the operation carried out when executing an M06, manager status, change status (in execution or at rest), magazine involved in the change (if the change is taking place) and whether the change process is or not in an error state. • Load and unload a tool from the magazines through the spindle, using a routine. • Saving the table contents. • Recalling the table contents. • Printing the table contents.

3.

HOW TO OPERATE THE CNC

Operating modes



CNCelite
8058 8060
8065 8070

REF: 2508

Operation mode.	Operations allowed.
Utilities mode	• See the files stored at the CNC, in a peripheral device or at another CNC (or PC) connected via Ethernet. • Create new folders to save files. • Select a file group and carry out operations such as copy, rename or delete files. • Change file attributes. • Do a file search based on a text already defined in them. • Set a password to restrict the access to the customizing tool FGUIM, to machine parameters and to the PLC. • Make or restore a backup copy of the CNC data.
PLC	In this operating mode, the PLC may be accessed to check its operation or the status of the different PLC variables. It is also possible to edit and analyze the PLC program as well as the message and error files of the PLC.
Machine parameters	• Edit and modify the machine parameter tables. • Validate the machine parameter tables. • Saving and restoring the table contents. • Printing the table contents. The machine parameter tables may be edited in this mode or be copied from a floppy disk or other peripheral connected to the CNC via Ethernet.
Tuning mode (Setup assistance)	The set of utilities of the setup assistance is meant to speed up and simplify the machine setup procedure. The setup assistance offers the following utilities. • The oscilloscope. • The circularity (roundness) test. • The bode diagram. It must be borne in mind that these utilities are assistance tools that only show the system's response to the various adjustments; it is always up to the technician to choose the best adjustment.
DDSSetup mode	• See the list of devices connected to the servo bus. • Change the access level for the parameters of Sercos drives. • Edit the parameters and variables of Sercos drives. • Edit the parameters of Mechatrolink servos. • Display the list of active errors at the drive. • Monitor in real time the value of the drive variables. • View the status of the operation being carried out at the drive and the status of the digital inputs and outputs of the drives. • Configure and start up the internal command generator of the Sercos drives.
Diagnosis mode	• Test the hardware and software of the PC that the CNC is based upon. • Show system information. • Show information about the modules that make up the CNC software. • Show information about the elements connected to the CNC through the CAN bus, Sercos or Mechatrolink. The diagnosis is a testing and displaying tool; it cannot be used to modify the displayed values.

3.

HOW TO OPERATE THE CNC
Operating modes

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

3.5 Execution channels.

The CNC can have a single execution channel (single channel system) or several (multi-channel system). Each channel is a different work environment that can act upon a part of or the whole system. Each channel can execute a different program, be in a different work mode and have its own data. If necessary, the channels can communicate and synchronize with each other and carry out actions that are coordinated with each other. They can also share information through variables and arithmetic parameters.

The use of channels is intended for machines such as two-head lathes, where each channel would have one of the heads and two axes; machines with feeders, where the machine and the feeder would be different channels; etc.

The axes and spindles of a channel.

A channel can have a group of axes and spindles that act independently or in parallel with the rest of the channels. It is also possible to configure a channel without either axes or heads initially, and add them later from a part programme or from the MDI/MDA mode. An axis or head cannot be on two channels at the same time.

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

The active channel.

The channels are managed from the interface like with a channel selector switch. There is an active channel at all times and it is the one being displayed. When changing channels, it displays it and it becomes the active channel.

Changing channels.

The way to access the different channels is managed with the [NEXT] key. This key may be configured either to access the channels sequentially or to show the list of available channels on the softkey menu. It is also possible to change channels by clicking on the icons of the status bar.

It cannot change channels.

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

Key set up to show the system menu.

The system menu shows, on one of the softkey menus, the list of available channels. Press the appropriate soft key to go to the desired channel. Depending on how the system menu is configured, the system menu will be disabled in one of the following ways.

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

Group of channels.

Two or more channels may be configured forming a group that will have the following characteristics. The groups of channels are defined via machine parameters.

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

3.

HOW TO OPERATE THE CNC
Execution channels.



CNCelite
8058 8060
8065 8070

REF: 2508

Work modes and channels.

Some operating modes are specific for each channel whereas others are common to all the channels (unique for the whole CNC). The specific operating modes for each channel act independently from the rest of the channels; for example, the automatic mode can execute a different program in each channel. The common operating modes act on the whole CNC, affect all the channels; for example, the PLC is common to the whole CNC.

Operation mode.	Relationship with the channel.
Automatic mode	The automatic mode is specific for each channel.
Jog mode	The jog mode is specific for each channel.
MDI/MDA mode	The MDI mode is specific for each channel.
EDISIMU mode (editing and simulation)	The EDISIMU mode is specific for each channel.
User tables	These tables are common to all the channels.
Tool table Magazine Table	These tables are common to all the channels.
Utilities mode	The utilities mode is common to all the channels.
PLC	The PLC mode is common to all the channels.
Machine parameters	The machine parameters are common to all the channels.
Tuning mode (Setup assistance).	The tuning mode is common to all the channels.

Operation of the operator panel.

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

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

3.

HOW TO OPERATE THE CNC

Execution channels.

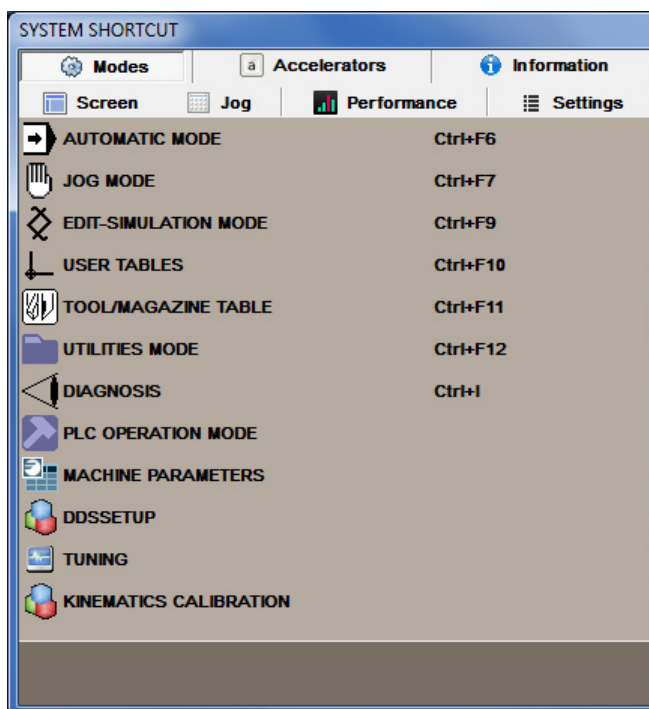
FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

3.6 Task window.

The task window provides easy access to different CNC hotkeys, functions and work modes. 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. Double click to select any option in the window.



Tab.	Contents.
Modes.	Accessing work modes.
Accelerators.	Access to some CNC functions; conversational mode, dynamic bar, etc.
Information.	CNC information panels; messages, errors, etc.
Screens.	Options for browsing the CNC.
Jog.	Virtual keyboard and control panel options (only recommended for a simulator CNC).
Performance.	Information about the hard disk, memory and CPU use.
Settings.	Configuration of some control functions; graphics, virtual keyboard, etc.

Moving around in the task window

Besides the mouse, the following keys may be used.

Key.	Meaning.
[↑][↓]	Move the cursor up and down.
[CTRL]+[TAB]	Change tab.
[ENTER]	Entering the selected option.

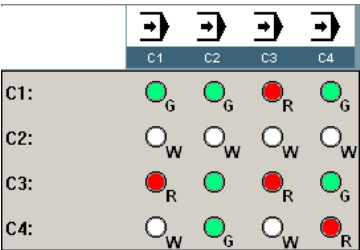
3.

HOW TO OPERATE THE CNC
Task window.

3.7 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.



Channel 1 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-).

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.

3.

HOW TO OPERATE THE CNC

Channel synchronization window.

3.8 CNC warning and errors window.

When a warning or error occurs, the CNC displays a window with the error category, its number and its associated text. Warnings and errors can also be triggered from within the part programme (#WARNING, #WARNINGSTOP and #ERROR) and appear in the same type of window. There are three categories of errors; warning, error and fatal error.

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.

Show/hide warnings and errors.

The warning and error windows can be hidden and displayed using the key sequence [ALT]+[W], by double-clicking on them or by clicking on the "+" symbol. 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].



3.8.1 Warnings.

The CNC displays the warnings in a green window. The system warnings are just notifications, they do not interrupt the execution of the part-programme and they may be removed 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.

3.8.2 Errors.

The CNC displays the errors in a red window. 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.

3.

HOW TO OPERATE THE CNC
CNC warning and errors window.

3.8.3 Fatal error.

The CNC displays the fatal errors in a purple window. They are errors that force the operator to turn the CNC off. If the error persists, contact the Service Department at Fagor Automation.

3.8.4 Error 3753. Absolute encoder. Monitor the coordinate difference during startup (parameter MAXDIFREF).

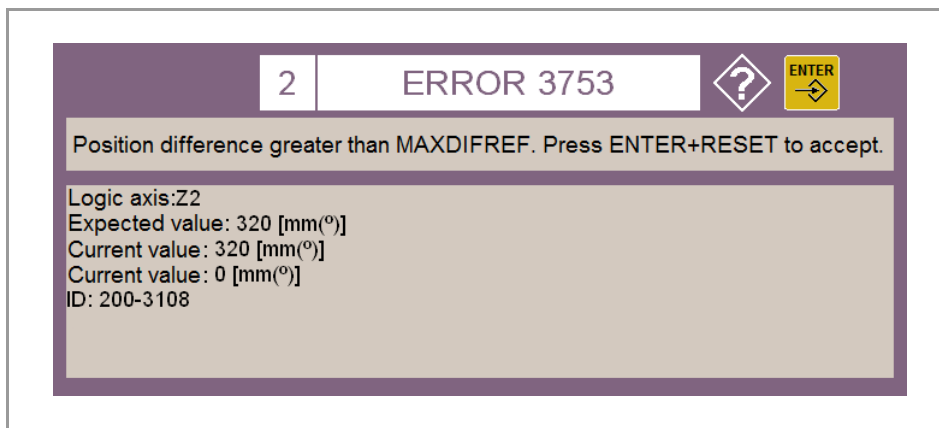
When the axis has an absolute encoder, during startup the CNC takes into account the number of encoder overshoots to calculate the corresponding startup coordinates. Also, for rotary axes, the CNC inserts the coordinates in the module and within the limits.

The CNC compares the position calculated for startup with that saved during shutdown, and if there is any difference this may be because the axis had been moved after the CNC had shutdown or due to a absolute feedback failure. The parameter MAXDIFREF indicates the maximum allowable difference between the coordinate saved by the CNC and that read by the encoder at startup.

- If the difference is less than the MAXDIFREF, the CNC starts up with the new position calculated by the encoder and sets the mark PLC REFPOIN = 1, indicating that it is not necessary to home the axis.
- If the difference is greater than the MAXDIFREF, the CNC will start up using the position saved by the CNC at shutdown and display error 3753. The CNC will show this error in a purple window, being the first one on the list, so that the user is aware of the situation and determine the validity of the coordinates.

The coordinate is valid if the position of the machine coincides with the “current value” field of the error. The user must clear the error by pressing [ENTER]+[RESET]; pressing only [RESET] will not clear the error.

The coordinate is valid if the position of the machine coincides with the “expected value” field. The user must clear the error by pressing [ENTER]+[RESET] and execute a G174 in MDI with the expected value. Afterwards, the CNC must be started-up to assume the coordinate correctly.

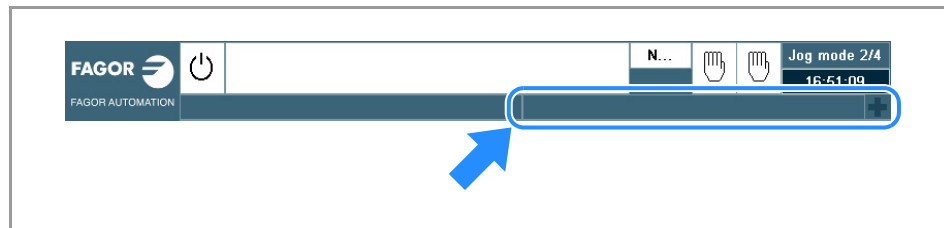


3.

HOW TO OPERATE THE CNC
CNC warning and errors window.

3.9 PLC messages.

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 is more than one active message, the bar will show the highest priority message (the one with the lowest number) and the "+" symbol next to it. 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 information 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.

3.10 PLC errors.

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.
	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.

3.

HOW TO OPERATE THE CNC
PLC errors.

FAGOR
AUTOMATION

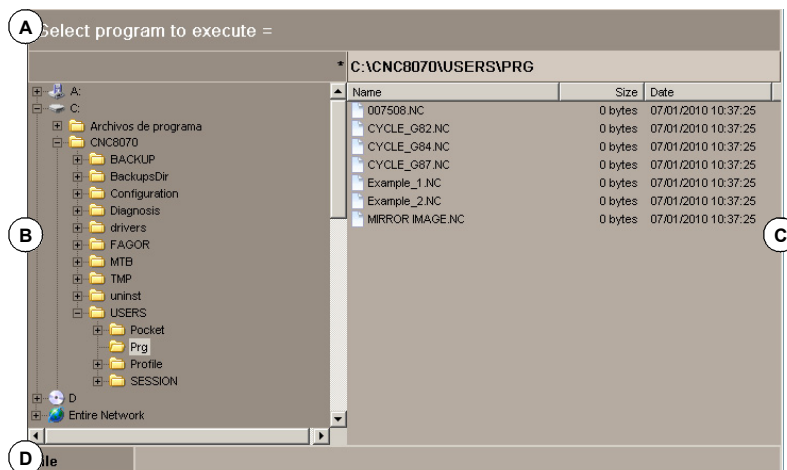
CNCelite
8058 8060
8065 8070

REF: 2508

3.11 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 Files contained in the selected folder.
- D Area to define the name of the new files. A file can also be selected when knowing its name.

Move around in the window

Besides the mouse, the following keys may be used.

Key.	Meaning.
	It switches the window focus.
	With the focus on a folder, it closes and opens the submenu of folders.
	To move the focus through the elements of the window.
	To move the focus page by page.
	To move the focus to the beginning or end of the list.
	Opens the selected file.
	To cancel the selection and close the program listing.

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.

3.

HOW TO OPERATE THE CNC
File selection 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.

Search a 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.

It may be searched using either the softkey menu or the following keys. Depending on how the search is carried out, the focus goes to the last file found, whose name also appears at the top of the window. To end the search, press [ESC].

Key.	Meaning.
	Search for the next program (in descending order).
	Search for the previous program (in ascending order).
	End the search and close the dialog box.

When defining a search, the softkey menu shows the "Next" (up) and "Previous" (down) options.

3.

HOW TO OPERATE THE CNC

File selection window

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

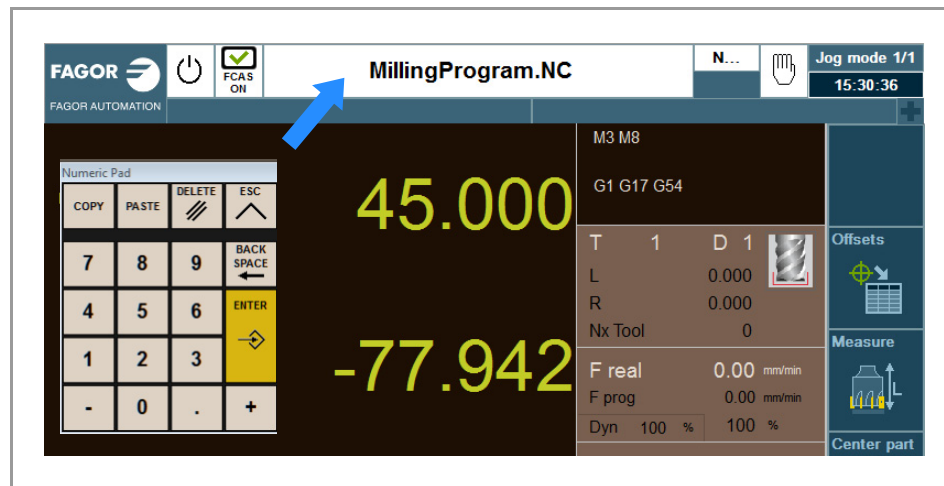
REF: 2508

3.12 Mini-numeric keypad.



In versions prior to v6.21.40, clicking on the top center of the status bar (name of the selected program) displays the main page of the CNC (same as the [Main-Menu] key).

By clicking on the top center of the status bar (name of the selected program), the CNC displays a mini numeric keypad to make it easier to enter data by using the touch screen. Clicking again in this area hides the mini keyboard. The mini keyboard can be dragged anywhere around the screen.



3.

HOW TO OPERATE THE CNC
Mini-numeric keypad.



CNCelite
8058 8060
8065 8070

REF: 2508

3.13 Calculator.

The calculator allows you to perform arithmetic, binary and trigonometric calculations and to convert units. The calculator can be accessed in any operating mode through the task window, with the key combination [CTRL]+[K] or with the [CALC] key. The [INS] key allows you to insert the result in the editable element that prompts the calculator. An editable element is any element that may have a focus or cursor, such as the program editor, tables, editable data, etc. Press [ESC] to close the calculator.

The calculator looks like this.

(A) Results window.

(B) Editing window.

(C) Explorer window.

(D) Insert the last result in the current expression.

Results window.

This window shows the final result of the expression. This value may be recovered with the "Acc" button so it can be used in later calculations.

Editing window.

This window displays 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.

Explorer window.

This window shows the result of the expression as it is 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:

Result.	Meaning.
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.

3.

HOW TO OPERATE THE CNC
Calculator.



CNCelite
8058 8060
8065 8070

REF: 2508

3.13.1 Softkey menu.

The horizontal softkey menu shows the operations available; arithmetic, trigonometric, etc. The vertical menu shows the units selected and available.

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

Arithmetic operations.

Softkey.	Operation.	Example.
+	Add.	$x + y$
-	Subtract. Change sign.	$x - y$ $-x$
*	Multiply.	$x * y$
/	Divide.	x / y
%	Percentage.	$x\% y$
^	Power.	x^y
!	Factorial.	$x!$

Binary operations.

Softkey.	Operation.	Example.
AND	Binary AND.	$x \text{ AND } y$
OR	Binary OR.	$x \text{ OR } y$
XOR	Exclusive OR.	$x \text{ XOR } y$
NOT	Complement.	NOT 101

Trigonometric operations.

Softkey.	Operation.	Example.
SINE X	Sine.	$\sin x$
COSINE X	Cosine.	$\cos x$
TAN X	Tangent.	$\tan x$
INV SINE X	Arc-sine.	$\text{InvSin } x$
INV COSINE X	Arc-cosine.	$\text{InvCos } x$
INV TAN X	Arc-tangent.	$\text{InvTan } x$

Conversion functions.

Softkey.	Operation.	Example.
MM -> INCHES	Millimeters to inches.	$\text{MMToI } x$
INCHES ->MM	Inches to millimeters.	$\text{IToMM } x$
ABS	Absolute value.	$\text{Abs } x$
INT	Integer.	$\text{Int } x$

Constants.

Softkey.	Operation.	Example.
PI	Pi value.	PI
MM -> INCHES	Value of the ratio between millimetres and inches.	MMPERI
INCHES -> MM	Value of the ratio between inches and millimetres.	IPERMM

3.

HOW TO OPERATE THE CNC
Calculator.

Extended functions.

Softkey.	Operation.	Example.
SQRT X	Square root.	Sqrt x
LN X	Neperian logarithm.	Ln x
LOG X	Decimal logarithm.	Log x
e ^ X	"e" function.	InvLn x
10 ^ X	Decimal exponent.	InvLog x
Integral FX	It calculates the integral.	N=100:A=1:B=5:Integral(x+2)
Zero FX	Function zero.	N=100:E=1e-10:A=5:Zero(x^2)

- Integral FX function

Integral of 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, there will be 100 intervals. If "N" is greater than 500000, the intervals will be 500000.

- ZERO function

Value that makes the function 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, 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.

Minimise.

Minimise the calculator.

Work units.

The result of the operations may be given in various units. The units can be changed using the following softkeys. It highlights the units currently selected.

Softkeys.	Meaning.
	Hexadecimal (\$), decimal (D) or binary (B) units.
	Units in degrees or radians.

3.

HOW TO OPERATE THE CNC
Calculator.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

3.13.2 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 "N",) may be used to store numeric values and use them later.

Assigning a value to a variable: $A = 34.234$

Reference to a variable: $\text{Sin}(A/2)$

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

$A=34.234:\text{Sin}(A/2)$ is the same as $\text{Sin}((A=34.234)/2)$

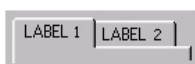


The values of the calculator variables "A" - "Z" are independent from the values of local parameters "A" - "Z" (also called P0 through P25).



3.14 Dialog boxes

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



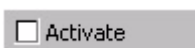
- Selection panels.

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



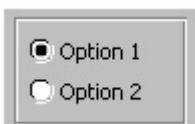
- Drop list.

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



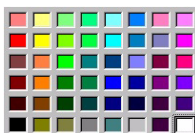
- Selection box.

It activates an option.



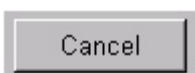
- Option selection.

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.

3.

HOW TO OPERATE THE CNC

Dialog boxes



CNCelite
8058 8060
8065 8070

REF: 2508

AUTOMATIC MODE

4

4.1 Interface description.

4.1.1 Description of a typical screen of this work mode.

The automatic mode has several pages, with different information in each one of them. The visibility of these pages is configurable from the status bar. A typical screen of the automatic mode can show the following information:



- A Information on the pages.
- B Program window.
- C Position of the axes.
- D Active "M" functions.
- E Active "G" functions and high-level commands.
- F Tool information.
- G "F" axis feedrate.
- H "S" spindle speed.
- I Execution time "Cy Time".
- J Number of parts made "Part Ct".
- K CNC messages.



CNCelite
8058 8060
8065 8070

Information on the pages.

Active operating mode, selected page number and number of pages visible. Clicking on the mouse, the CNC shows the list of pages available and also allows configuration of those that are hidden or visible.

REF: 2508

Program window.

The "Display" softkey enables selection of the information that is shown in this window; program blocks, active subroutines or DMC status.

Program blocks; program stopped.

- Cursor for selecting blocks. The cursor can be moved with the [▲][▼] keys.
- Initial and final block of the execution (if selected).
- The bottom shows the line number being executed (Line) and, if necessary, the line number where the execution was canceled (Break Line).

Program blocks; program; program being executed.

- The cursor shows the block being executed.
- The bottom part shows the line number being executed (Line).
- 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.

Active subroutines.

Information relating to the status of the subroutines, canned cycles, repetition blocks and loops. See ["4.2 Display the status of the program or of the active subroutines."](#) on page 77.

DMC status.

Condition and progress of the DMC function. See ["4.9 Display the status of the DMC \(Dynamic Machining Control\)."](#) on page 104.

Position of the axes.

In the screen configuration supplied by Fagor, the data displayed will be different on each screen. Displaying the following information is the most typical.

- The programmed coordinate, in other words, the target position.
- The current position of the axes referred to part zero or machine reference zero (home) being this position referred to the tip of the tool or to its base.
- Distance left (to go) for the axis to reach the coordinate.
- The following error (axis lag).
- The axis name in reverse video indicates that the axis is moving.
- If the name of the axis appears in red, it means that the PLC is inhibiting the movement of the axes (INHIBIT mark active).

Tool information.

This information depends on the CNC model; milling machine or lathe.

- Milling model.
 - Number of the active tool "T".
 - Active "D" tool offset.
 - Icon indicating the tool type.
 - Length of the active tool "L".
 - Radius of the active tool "R".
 - Number of the next tool "Nx Tool".
- Lathe model.
 - Number of the active tool "T".
 - Active "D" tool offset.
 - Icon indicating the tool type.
 - Offsets (dimensions) of the tool on each axis.

4.

AUTOMATIC MODE
Interface description.



CNCelite
8058 8060
8065 8070

REF: 2508

“F” axis feedrate.

- “F real” real feedrate.
- “F prog” programmed feedrate. When setting a new feedrate in the MDI/MDA mode, it will become the new feedrate for the automatic mode.
- Feedrate percentage active on the CNC (jog panel, program or PLC).
- Dynamic percentage “Dyn” machining.

“S” spindle speed.

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

- Real speed “S real” (“S1 real”, “S2 real”, etc).
- Icon for turning direction.
- Programmed speed “S prog” (“S1 prog”, “S2 prog”, etc).
- Active gear set.
- Active speed percentage for the CNC (JOG panel, program or PLC).
- Spindle position “S pos” (“S1 pos”, “S2 pos”, etc).
- Following error of the “S fwe” spindle (“S1 fwe”, “S2 fwe”, etc.).

If the text “Sreal” appears in red, it means that the PLC is inhibiting the movement of the spindle (INHIBIT mark active).

4.

AUTOMATIC MODE
Interface description.

4.1.2 Softkey menus.

Horizontal softkey menu.

When selecting the program screen, the horizontal softkey menu shows the following options.

Softkey.	Description.
Block selection.	Select the first and last block of the execution either manually or through a block search.
Simulated execution.	Show the available simulated execution modes.
Display.	Toggle between the various screens of the automatic mode and display information related to the status of the subroutines, canned cycles, repetition blocks and loops.
MDI.	Access the MDI/MDA mode.
Exblk.	Execute only the block selected with the cursor.
Graphics.	Show the graphics screen.
Single block.	Activate the single block execution mode.

Vertical softkey menu.

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

Softkey.	Description.
	Select a program for execution.
	Begin tool inspection. Tool inspection is only available when program execution is interrupted.
	End simulated execution and start executing the program
	Select the program that is being edited.
	Activating the Fagor Feed Control (FFC) function. The softkey is only active during the execution of the canned cycles of the editor.
	DMC options.
	FCAS not active.
	FCAS active.

4.

AUTOMATIC MODE
Interface description.

FAGOR
AUTOMATION



CNCelite
8058 8060
8065 8070

REF: 2508

4.2 Display the status of the program or of the active subroutines.

From the horizontal softkey menu, it is possible to toggle between the display of program blocks and the display of information related to the status of the subroutines, canned cycles, repetition blocks and loops.

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

Description of the information displayed.

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

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

The window shows the following information.

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

4.

AUTOMATIC MODE

Display the status of the program or of the active subroutines.

4.3 Program simulation and execution.

4.3.1 Select program.

The CNC allows selecting and executing a different program in each channel. Each channel executes the program selected in it. To select a program, press one of the following softkeys of the vertical menu.

Softkey.	Meaning.
	This softkey shows the list of available programs. See "3.11 File selection window" on page 64.
	This softkey selects directly the program being edited in the same channel.

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

4.

AUTOMATIC MODE

Program simulation and execution.

4.3.2 Select the first and last blocks of the execution.

After selecting the "block selection" softkey, the horizontal softkey menu shows the following options.

Softkey.	Description.
Set beginning	First block for execution or for manual block search.
Stop condition.	Set the condition to end the execution or the manual block search.
Automatic block search.	Do an automatic block search.
Manual block search.	Do a manual block search.
Find text.	Find text

Set beginning.

This option sets as starting block for execution the block selected with the cursor. When not setting the starting block, the program will begin executing from the first block.

The first block may be selected using the cursor or the "Find text" option of the softkey menu. The selected block stays active until canceled (selecting another block or selecting the same one again) or executing the program.

Stop condition.

This option sets the block where the execution of the program or subroutine will be interrupted. After executing that block, the execution may be resumed with the [START] key or canceled with the [RESET] key. If the last block is not set, the program execution will end executing one of the end-of-program functions "M02" or "M30".

Select subroutine.

This option selects the stop condition in a global subroutine which has been called upon from the program. When selecting this option, the CNC shows a list of the available subroutines. After selecting the desired subroutine, it will appear in the program window.

Set stop block.

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

The last block may be selected using the cursor or the "Find text" option of the softkey menu. The selected block stays active until canceled (selecting another block or selecting the same one again) or executing the program.

Number of times.

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

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

4.

AUTOMATIC MODE

Program simulation and execution.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

Automatic block search.

This option recovers the program history up to the block where the previous execution was canceled. The CNC remembers the block where interruption was canceled, thus not being necessary to set the stop block. If after the block search, the execution is resumed from that point, it will do it in the same conditions as if it were executed from the beginning. See ["4.6 Block search."](#) on page 99.

When this option is active, the softkeys that define the execution conditions are applied to the block search.

Set beginning.

This option sets as starting block for block search the block selected with the cursor. If the first block is not defined, the block search starts at the first block of the program.

Manual block search.

This option recovers the program history up to a particular block of the program or of the subroutine, set by the operator. If after the block search, the execution is resumed from that point, it will do it in the same conditions as if it were executed from the beginning. See ["4.6 Block search."](#) on page 99.

When this option is active, the softkeys that define the execution conditions are applied to the block search.

Set beginning.

This option sets as starting block for block search the block selected with the cursor. If the first block is not defined, the block search starts at the first block of the program.

Stop condition.

This option sets the block of the program or of a subroutine where the block search will be interrupted.

- Select subroutine. This option selects the stop condition in a global subroutine which has been called upon from the program.
- Set stop block. This option sets as block search interruption block the block selected with the cursor.
- Number of times. This option sets as stop condition, that the block selected as the last block has been executed a specific number of times.

If a block containing a multiple machining cycle (·M· model) has been selected as a stop block, the block search will end just before the beginning of the nth modal cycle repeated in the multiple machining.

Find text.

This option shows a dialog box for placing the cursor on a particular line of the program or search for text or a character string in the program.

Go to line.

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

Find text

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

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

4.

AUTOMATIC MODE
Program simulation and execution.



CNCelite
8058 8060
8065 8070

REF: 2508

4.3.3 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. See ["4.3.2 Select the first and last blocks of the execution."](#) on page 79.

Start executing

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



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



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

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

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

Interrupting the execution

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

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

Stop the execution

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



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

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 rapid traverse to programmed movements (except to G00 movements and to threading operations). 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.

AUTOMATIC MODE

Program simulation and execution.

FAGOR
AUTOMATION

CNCelite

8058 8060

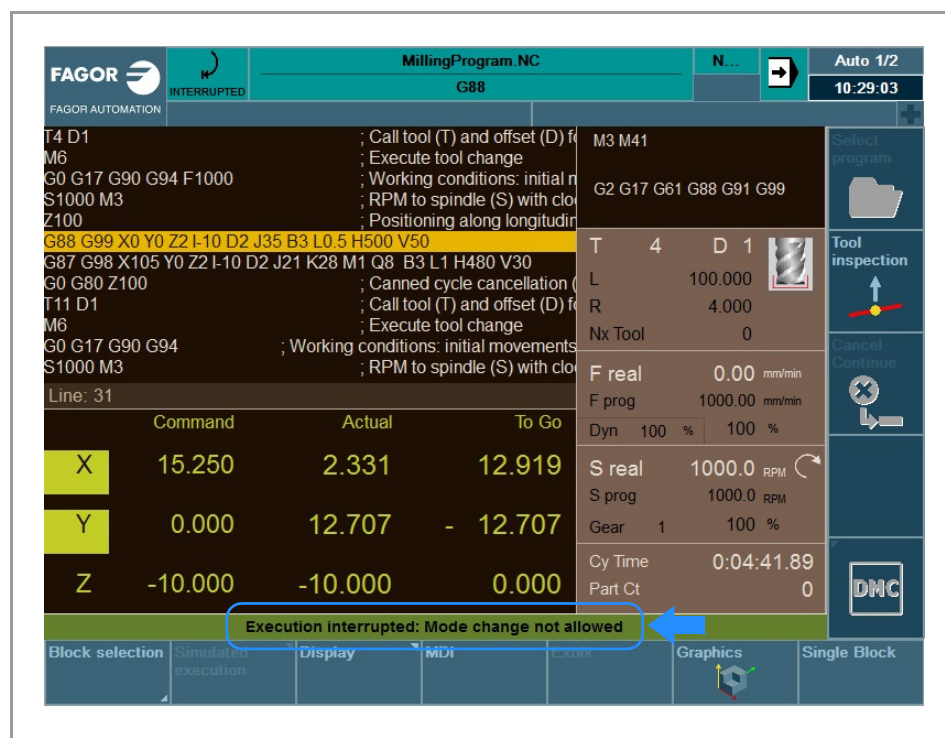
8065 8070

REF: 2508

4.3.4 Mode change protection for an interrupted program.

Accessing some work modes with an interrupted program means having to cancel the execution of the program. To avoid unwanted cancellations (and having to perform a block search, etc. to resume execution), the CNC will not allow these modes to be changed for an interrupted program. The user may choose one of the following options:

- [START] The CNC resumes program execution.
- [RESET] The CNC cancels the execution of the program and allows the mode to be changed.



4.

AUTOMATIC MODE
Program simulation and execution.

4.3.5 Resume the execution of a program from the block where it was canceled.

After cancelling the execution of a program (error, reset, etc.) we usually want to resume the execution from a particular block, usually a few blocks ahead the interruption block. To do that, when cancelling the execution, the CNC saves and shows on the screen the block number where the execution was cancelled (INT=#####).

The way to proceed, once the execution is cancelled, is as follows.

- 1 In automatic mode, using the "EXBLK" softkey, execute the header blocks of the program that set the machining conditions. See ["4.4 Executing program blocks separately."](#) on page 93. The MDI/MDA mode allows changing the machining conditions at any time.
- 2 Once the machining conditions have been set, cancel the independent execution of blocks ("EXBLK" softkey) and press the block selecting softkey to find and set the starting block.
- 3 Select the text or line searching option. When the execution has been cancelled with reset or some error, the "Line" section of this window shows the line number where the interruption took place. Press OK to place the cursor on that program line.
- 4 If necessary, move the cursor to select another line. The "Set beginning" option sets as starting block for execution the block selected with the cursor.
- 5 Press [START] to begin executing from the selected block maintaining the history active at the time.



Through block search, the CNC offers, , the most complete way to resume program execution after interrupting (cancelling) the execution. Block search may be used to restore the program history up to a particular block, reposition axes, go into tool inspection, etc. See ["4.6 Block search."](#) on page 99.

4.

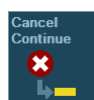
AUTOMATIC MODE
Program simulation and execution.

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

The CNC allows canceling the execution of an interrupted program (M0 or [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 The vertical menu displays the following softkey when the program is interrupted. When this softkey is pressed, the CNC cancels the execution and stores the history. The message bar indicates that a program block may be selected to resume execution.

- 2 The CNC allows the axes to be moved manually (jog or handwheels) and blocks to be executed from the MDI/MDA mode (positioning axes, modifying machining conditions, etc.).
- 3 Select the block to resume execution and press [ENTER]. The block may be before or after the interrupted block.
- 4 Once the starting 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.
- The execution cannot be canceled inside a canned cycle, both ISO and that of the cycle editor.
- 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.

4.

AUTOMATIC MODE

Program simulation and execution.

FAGOR
AUTOMATION



CNCelite
8058 8060
8065 8070

REF: 2508

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.

- 1 Interrupt the program.
- 2 Make the changes in the [EDISIMU] mode.
- 3 Perform a program cancel-continue, to accept the changes.
- 4 Resume execution.

4.

AUTOMATIC MODE

Program simulation and execution.



CNCelite
8058 8060
8065 8070

REF: 2508

4.3.7 Simulated execution of a program.

With simulated execution, it is possible to simulate a program, interrupt it at a point and start execution from that point on. Depending on the type of simulation selected, it can involve movement of axes, spindle, etc. After interrupting the simulation and before starting the execution, the CNC allows changing the conditions of the program via MDI/MDA, moving the axes and acting upon the spindle from the jog keyboard and it also offers the possibility to reposition the axes and the spindles.

General operation.

Start the simulation of the program

- 1 On the horizontal softkey menu, select the desired type of simulation.
 - Theoretical travel.
 - G M S T functions.
 - Rapid.
 - G functions.
 - Main plane.
 - Rapid [S=0].
- 2 If necessary, set the desired simulation conditions (first and last block) See ["4.3.2 Select the first and last blocks of the execution."](#) on page 79.
- 3 Press the [START] key to start the simulation. The program may be simulated in "single block" or "continuous" mode; the mode may be selected even while simulating the program.

End simulation and start executing the program.

- 1 Press the [STOP] of the operator panel to interrupt the simulation. Once the program is interrupted, simulation may be resumed with the [START] key or switch to execution mode from the vertical softkey menu.

Softkey.	Description.
	End simulated execution and start executing the program

- 2 When switching to execution mode (after pressing the softkey), the CNC goes into tool inspection mode. In this mode, it is possible to reposition the axes, access the MDI/MDA mode to change the conditions of the program, etc. See ["4.5 Tool inspection."](#) on page 94.
- 3 To complete the tool inspection and before starting the execution of the program, the spindle turning direction must be restored and the axes 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.

- 4 Press the [START] key to start the execution.

4.

AUTOMATIC MODE
Program simulation and execution.

Types of simulation.

The simulation options are available on the horizontal softkey menu. By default, the menu only offers two of the options. To enable all the options, it is necessary to expand the softkey and access the configurator (last softkey).

	Graphic representation	Movement of the axes	Spindle control	Sends M-H-S-T to the PLC
Theoretical travel.	Programmed	No	No	No
G functions.	Tool center	No	No	No
G M S T functions.	Tool center	No	No	Yes
Main plane.	Tool center	Yes	Yes	No
Rapid.	Tool center	Yes	Yes	Yes
Rapid [S=0].	Tool center	Yes	No	Yes

Theoretical travel.

- The simulation ignores tool radius compensation (functions G41 and G42), hence drawing the programmed tool path.
- The simulation does not send M H S T functions to the PLC.
- The simulation does not move the axes of the machine nor starts the spindle.
- The simulation takes into account the dwells programmed with G4.
- The simulation takes into account the program stops programmed with M00 and M01.

G functions.

- The simulation ignores tool radius compensation (functions G41 and G42), hence drawing the tool center path.
- The simulation does not send M H S T functions to the PLC.
- The simulation does not move the axes of the machine nor starts the spindle.
- The simulation takes into account the dwells programmed with G4.
- The simulation takes into account the program stops programmed with M00 and M01.

G M S T functions.

- The simulation ignores tool radius compensation (functions G41 and G42), hence drawing the tool center path.
- The simulation sends M H S T functions to the PLC.
- The simulation does not move the axes of the machine nor starts the spindle.
- The simulation takes into account the dwells programmed with G4.
- The simulation takes into account the program stops programmed with M00 and M01.

Main plane.

- The simulation ignores tool radius compensation (functions G41 and G42), hence drawing the tool center path.
- The simulation does not send M H S T functions to the PLC.
- The simulation only executes the movement of the axes that make up the main plane. The axes move at the maximum feedrate allowed regardless of the programmed "F" value. It is possible to change that feedrate with the feedrate override switch.
- The simulation starts the spindle if it has been programmed.
- The simulation ignores the dwells programmed with G4.
- The simulation takes into account the program stops programmed with M00 and M01.

Rapid.

- The simulation ignores tool radius compensation (functions G41 and G42), hence drawing the tool center path.
- The simulation sends M H S T functions to the PLC.
- The simulation moves the axes of the machine. The axes move at the maximum feedrate allowed regardless of the programmed "F" value. It is possible to change that feedrate with the feedrate override switch.

4.

AUTOMATIC MODE

Program simulation and execution.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

4.

AUTOMATIC MODE

Program simulation and execution.

- The simulation starts the spindle if it has been programmed.
- The simulation ignores the dwells programmed with G4.
- The simulation takes into account the program stops programmed with M00 and M01.

Rapid [S=0].

- The simulation ignores tool radius compensation (functions G41 and G42), hence drawing the tool center path.
- The simulation sends M H S T functions to the PLC.
- The simulation moves the axes of the machine. The axes move at the maximum feedrate allowed regardless of the programmed "F" value. It is possible to change that feedrate with the feedrate override switch.
- The simulation does not start the spindle, except when the spindle works in closed loop M19.
- The simulation ignores the dwells programmed with G4.
- The simulation takes into account the program stops programmed with M00 and M01.

4.3.8 Execute a program (retrace).

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

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

Turn the retrace function on.

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

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

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

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

When is the retrace function canceled.

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

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

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

Resume the execution after the retrace function has been canceled.

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

Influence of certain functions while executing in retrace mode.

Machining conditions.

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

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

4.

AUTOMATIC MODE

Program simulation and execution.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

Tool and offset.

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

•M• functions.

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

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

Functions that cancel the retrace function.

Definition.	Functions and instructions.
Gear change.	M41 M42 M43 M44
Spindle orientation.	M19
Effective spindle stop. If the spindle was stopped, this function does not cancel the retrace function.	M5
Change of work plane.	G17 G18 G19 G20
Changing zero offset and fixture offsets, etc. Functions G53-G59, G159 only cancel the retrace function if they cancel an active zero offset.	G92 G158 G53-G59 G159
Probing.	G100 G101 G102 #SELECT PROBE.
Activate, cancel or modify constant surface speed.	G96 G97
"C" axis. The instruction #CAX OFF only cancels the retrace function if it cancels an active #CAX.	#CAX ON #CAS OFF #FACE ON #FACE OFF #CYL ON #CYL OFF
Tapping.	G33 G63.
Changes in feed-forward or in AC-forward.	G134 G135
Home search.	G74
Changing the parameter set.	G112
Changing the software limits.	G198 G199
Synchronizing.	#SYNC
High speed machining.	#HSC ON #HSC OFF
Incline axis. Freezing (suspending) an incline axis does not cancel the retrace function.	#ANGAX ON #ANGAX OFF
Slaving axes.	#LINK #UNLINK
Axis parking.	#PARK #UNPARK
Tangential control.	G45 G145 #TANGCTRL ON #TANGCTRL OFF #TANGCTRL SUSP #TANGCTRL RESUME
Axis swapping.	#SET AX #CALL AX #PUT AX #RENAME AX

4.

AUTOMATIC MODE
Program simulation and execution.



CNCelite
8058 8060
8065 8070

REF: 2508

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

4.

AUTOMATIC MODE

Program simulation and execution.



CNCelite
8058 8060
8065 8070

REF: 2508

4.3.9 Executing a program in 8055 language.

A program edited in 8055 language may be executed at the CNC in two ways.

- Keep the program extension (.pim or .pit) and enable at the editor the compatibility with 8055 programs. Before executing or simulation the program, the CNC translates (converts) it. The CNC always simulates and executes the converted program, which is also the one shown on the screen during the execution.
- Change the program extension (.pim or .pit) or disable at the editor the compatibility with 8055 programs. In either case, the CNC tries to execute the program as if it were in its own language. The CNC will interpret the programmed canned cycles (PCALL instructions), but it will issue an error message if there is any instruction in 8055 language.

The conversion of the part-program.

The CNC translates the program only converted one; the first time it simulates the program or the first time that it selects the program in automatic mode. The CNC keeps both programs; the one written in 8055 format (the one edited) and its equivalent translated. If when selecting the program, a file with extension .pit or .pim is selected, the CNC opens the converted file and displays it on the screen.

The program, translated from automatic mode is saved with the same name but with the extension *m55* (milling program) or *t55* (lathe program), in the folder `..\Users\Prg\PRG_8055_TO_8070`

When modifying the program edited in the 8055 CNC language, the CNC converts it again. When modifying the converted program, the CNC does not update the one written in 8055 CNC language.

If any error occurs during execution, the CNC will display the block that caused it. The block will be in the language of the CNC, but it will be easily identified due to the translation format.

Considerations for program execution.

- The tool offsets programmed in the 8055 language can be translated as D1 when the translator has been configured accordingly.
- At the 8055, 3D pockets are defined in several files (plane profile and depth profiles). Only import into the CNC the file generated by the cycle editor which already contains the geometry of all the profiles. The name of the file will be 995###.pim, where ### is the value of the CAJ.3D field on the editing page of the cycle.
- Some cycles of the 8055 are not available at the CNC, such as the Y-axis cycles (profiles, rectangular pockets, circular pockets and 2D pockets in the YZ and XY planes) and the rectangular, circular and 2D pockets in the ZC and XC planes. The CNC will also execute these cycles even if they are not available at the cycle editor.
- To change a program in 8055 language by adding blocks in the CNC's own language, you need to change the extension of the file (.pim or .pit) to prevent the translator (converter) from issuing an error.
- The CNC accepts numerical block labels (N9999) in 8055 format, programmed in RPT and GOTO instructions, but it can only find them if there are in blocks after the calling instruction. If the block is in a previous line, it must have the format of the CNC label adding the ":" character.

4.

AUTOMATIC MODE
Program simulation and execution.

4.4 Executing program blocks separately.

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

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

4.

AUTOMATIC MODE
Executing program blocks separately.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

4.5 Tool inspection.



There are some exceptions or particular cases where this tool inspection cannot be accessed. In these cases, the CNC enables a particular tool inspection as explained in the next section.

- The CNC is executing the program in retrace mode.
- The independent interpolator is involved in the movement of an axis (#MOVE, #FOLLOW, #CAM, etc).
- A rigid tapping is active (the rest of the threading do not admit any type of inspection.).

When the execution is interrupted, tool inspection allows jogging the axes, start and stop the spindle, execute blocks in MDI/MDA mode, etc. When tool inspection is done, it allows repositioning the axes to the interruption point or to the starting point of the interrupted block and resume 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, start and stop the spindle, execute blocks in MDI/MDA mode, etc.
- 3 Reposition the axes and restore the spindle turning direction.
- 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.
- Act upon the master spindle of the channel from the operator panel.
- Execute blocks in MDI/MDA mode.

Once tool inspection is completed and before resuming the execution of the program, the spindle turning direction must be restored and the axes 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, spindle speed, 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 cycles or 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

4.

AUTOMATIC MODE
Tool inspection.

FAGOR
AUTOMATION



CNCelite
8058 8060
8065 8070

REF: 2508

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 and the spindle.

To complete the tool inspection and before resuming the execution of the program, the spindle turning direction must be restored and the axes 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 status of the master spindle has changed during the inspection, the softkeys will show the M3, M4, M5 or M19 function to restore.



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.

Repositioning the master spindle.

The spindle turning status may be restored either together with the repositioning of the axes or separately. The same vertical softkeys will show the M3, M4, M5 or M19 to be restored. If the spindle was interrupted in a positioning with M19, repositioning will complete that positioning. Once the spindle has recovered its status, it will no longer be available.

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.

4.

AUTOMATIC MODE

Tool inspection.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

Manual tool calibration.

Tool inspection also allows accessing manual tool calibration. Once the tool has been calibrated, exit tool inspection, the CNC requests repositioning the axes before resuming program execution. The softkey to access tool calibration will be different depending on the software installed (lathe model or mill model). See ["6.1 Manual calibration. Calibration without a probe"](#) on page 123.

Softkey.	Type of calibration.
	Tool calibration in a mill model.
	Tool calibration in a lathe model.

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.

4.5.1 Tool inspection (execution in retrace mode, independent interpolator or rigid tapping).

In the following cases, the CNC uses a particular case of tool inspection that also allows moving the axes, acting upon the spindle and changing the feedrate, spindle speed, executing M functions, etc. using the MHFS mode.

- The CNC is executing the program in retrace mode.
- The independent interpolator is involved in the movement of an axis (#MOVE, #FOLLOW, #CAM, etc).
- A rigid tapping is active (the rest of the threading do not admit any type of inspection.).



When in one of the previous cases, the execution is interrupted, this tool inspection allows the following operations:

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

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

Rigid tapping and tool inspection mode

When interrupting the rigid tapping and accessing the tool inspection mode, it is possible to jog the axes, in jog or with handwheel. When the axis is involved in the tapping, the interpolated spindle will also move; the spindle with which the threading is performed. If several axes are involved in the rigid tapping, when moving one of the axes all the axes involved in the thread will also move with it.

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

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

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

Repositioning the axes and the spindle.



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

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

Repositioning the axes.

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

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

4.

AUTOMATIC MODE

Tool inspection.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Restoring the spindle turning direction.

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

Changing the machining conditions.

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

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

4.**AUTOMATIC MODE**

Tool inspection.

4.6 Block search.

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

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

Optimize block search execution time.

The block search is faster if the program is structured with the machining conditions at the beginning and the rest of the program is movement blocks (G0, G1, G2, G3) together with functions F, S, M and tool changes.

Block search and canned cycles.

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

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

Types of block search.

The CNC has two search modes, automatic and manual. See ["4.3.2 Select the first and last blocks of the execution."](#) on page 79.

Automatic block search.

The automatic block search may be used to recover the program history up to the block where the previous execution was canceled. The CNC remembers the block where interruption was canceled, thus not being necessary to set the stop block.

Manual block search.

The manual block search may be used to recover the program history up to a particular block of the program or of the subroutine, set by the operator. In this search, it is possible to set as an ending condition to repeat the stop block a particular number of times, for example multiple machining cycles, loops, etc.

Executing block search.

To recover the program history, proceed as follows:

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

In the automatic block search, there is no need to select the stop block; by default, the CNC runs the search up to the block where the program was interrupted.

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

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

The CNC reads the program up to the stop block, activates the "G" functions that it reads, sets the feedrate and spindle speed conditions and calculates the position where the

4.

AUTOMATIC MODE

Block search.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

axes should be. The M functions are sent depending on how the machine is configured; they may be sent while reading the program or at the end of the program.

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

Repositioning the axes.

Once the block search is finished, the CNC will show the axes that are out of position. The axes may be repositioned individually or several at the same time in one of the following ways:

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

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

Changing the machining conditions.

After positioning the axes and before resuming execution, tool inspection may be accessed to change the machining conditions. In tool inspection, it is possible to change the feedrate and the spindle speed, execute blocks in MDI/MDA mode as well as activate M and H functions.

4.

AUTOMATIC MODE

Block search.



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

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

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

Machine parameters. M function table.

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

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

In this case, the functions are sent out to the PLC during block search as they are being read. Once block search is finished and after positioning the axes, tool inspection may be accessed to change the machining conditions.

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

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

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

This window will only show one of the functions M03/M04/M05/M19 and M41/M42/M43/M44. The rest of the M functions such as M0, M1, M2, M6, M8, M9, M30 are not shown because they are not modal.

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

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

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

4.

AUTOMATIC MODE

Block search.

4.7 Show/hide the dynamic override of the HSC.

The [CTRL][H] hotkey shows/hides at the bottom of the screen, the dynamic override of the HSC. 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.

Y	0.000	0.000	0.000	S prog	0.0	RPM
				Gear	1	100 %
Z	-75.000	-75.000	0.000	Cy Time	0:00:00.00	
				Part Ct	0	
DYNAMICS %				30.0000	100.0000	
					200.0000	

4.

AUTOMATIC MODE

Show/hide the dynamic override of the HSC.



4.8 FFC (Fagor Feed Control)

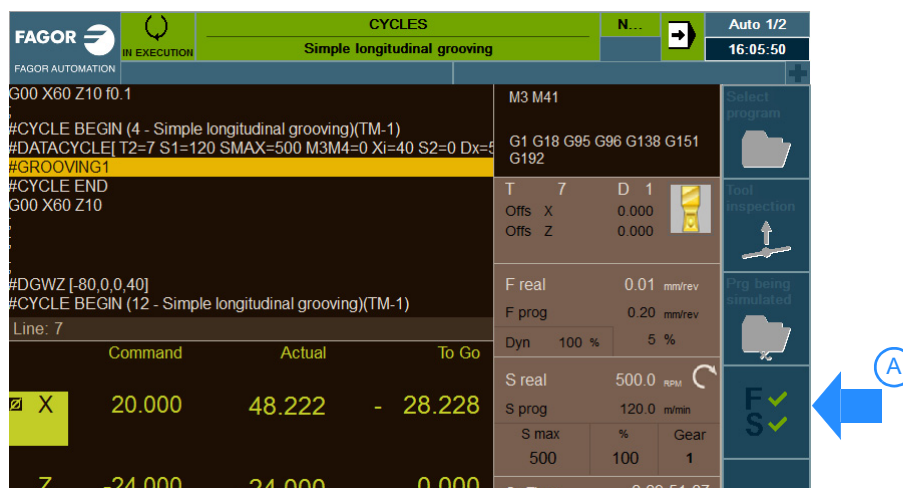
During the execution of a canned cycle of the editor, the Fagor Feed Control (FFC) function makes it possible to replace the feedrate and speed programmed in the cycle with the active values of the execution, which are acted upon by the feed override and speed override. During the machining of the cycle, the user can adjust the feedrate and the speed from the jog panel, using the feed and speed override switches. When the user sees that the feedrate and speed values are correct, the new values can be saved in the cycle by simply pressing a softkey. In the following cycle machinings, the CNC will use the new values saved. The new values are saved in the program at the end of the execution (M30, reset or error).

Softkey.	Description.
	Activating the Fagor Feed Control (FFC) function. The softkey is only active during the execution of the canned cycles of the editor.

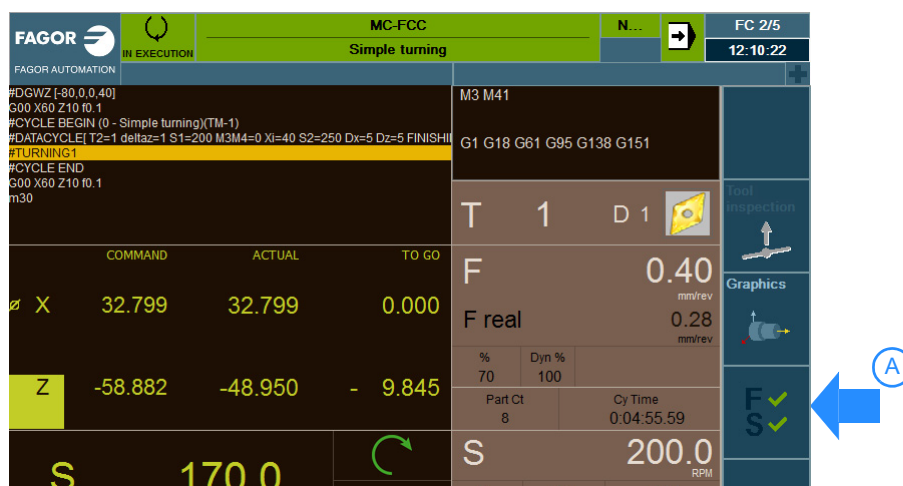
The FFC option is available for the roughing and finishing operations of the editor machining cycles; it is not available for the probe cycles or the ISO cycles of the machining. This option is also available independently during the execution of a cycle.

The FFC option modifies the feedrate and the speed; it does not change the penetration feedrate "Fz".

Standard operating mode (non-conversational).



Standard operating mode (conversational).



(A) Softkey to activate the Fagor Feed Control (FFC).

4.

AUTOMATIC MODE
FFC (Fagor Feed Control)

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

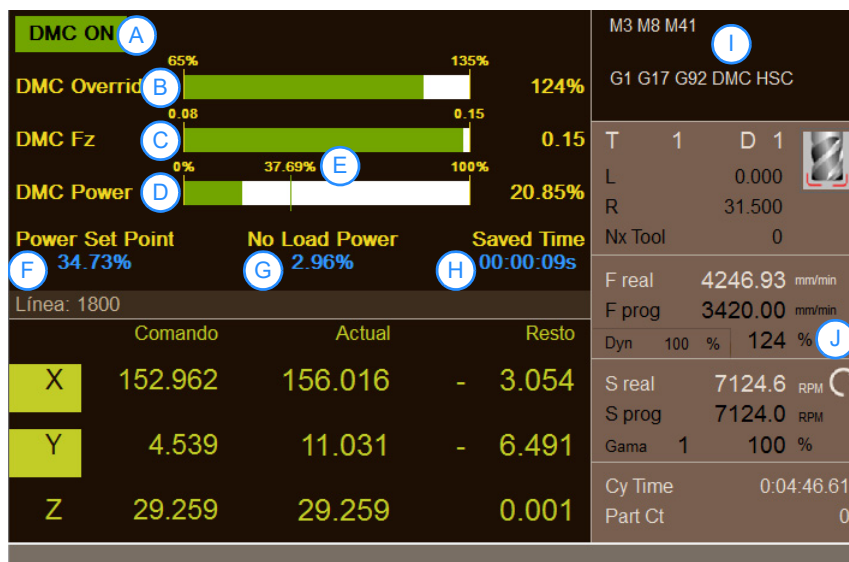
REF: 2508

4.9 Display the status of the DMC (Dynamic Machining Control).

4.9.1 DMC status and progress.

During the execution of a program while DMC is active, the automatic mode can display the status and progress of this function; to do this, on the "Display" softkey of the horizontal menu, select the option "DMC". To return to the standard screen in automatic mode, select the "Standard" option on the same softkey. The data displayed on the DMC page are the same as on the standard page, except for those for DMC itself (in the upper left corner).

Automatic mode. DMC status and progress.

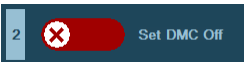


- A DMC status; ON/OFF.
- B DMC override. The consumption bar limits will be those defined in the machine parameters. The override applied by DMC is shown on the right.
- C Feed per tooth. The consumption bar limits are the values programmed in the instruction #DMC ON (FZMIN and FZMAX commands); if they have not been programmed, the CNC does not display this bar.
- D Power consumed by the spindle (defined as the percentage of rated power).
- E Power consumed by the spindle under ideal conditions; "target power" + "no-load power".
- F Target power or ideal cutting power (defined as a percentage of rated power); learned power or power programmed in the instruction #DMC ON (PWRSP command).
- G No-load power (defined as a percentage of the rated power).
- H Time saved by the DMC action.
- I When DMC is active, the program history will display "DMC" and will not change, even though the CNC performs incompatible functions that temporarily deactivate DMC.
- J Final override (DMC override + switch override).

Vertical softkey menu.

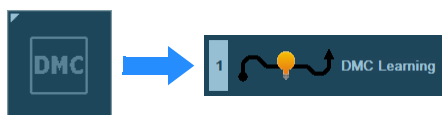
Softkey.	Meaning.
	DMC options (dropdown menu). - Repeat the DMC learning phase. - Deactivating the DMC.

DMC options.

Softkey.	Meaning.
	This softkey allows the DMC learning phase to be repeated at any time, as long as the DMC is active.
	This softkey allows the DMC to be deactivated.
	This softkey allows the DMC to be reactivated.

4.9.2 Learning phase.

Every time #DMC ON is programmed without target power (PWRSP command), DMC determines it through a learning phase that it initiates automatically. Once said value is obtained, the normal DMC operation will begin.



The learning phase can be repeated at any time, while the DMC is active, by pressing the "DMC learning" softkey in the automatic mode. After pressing the softkey, while the DMC is active, the next entry into the part will begin a learning phase, whether or not the target power had been programmed while activating the DMC.

With the axes in movement, the learning phase begins once DMC detects the entry into the part. DMC waits for the feedrate to reach the programmed value, and during the movement of the axes, it calculates the target power ("power consumed" – "no-load power"). The learning phase has a one minute duration, starting from when the tool enters the part at a distance equal to the radius. If the tool exits the part, it ceases counting the time until the tool re-enters the piece.

4.

AUTOMATIC MODE

Display the status of the DMC (Dynamic Machining Control).

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

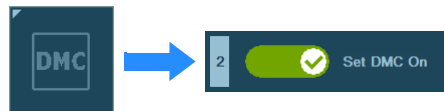
REF: 2508

4.9.3 Deactivating the DMC.

The softkey menu allows the DMC to be deactivated.



The DMC can be activated from the same softkey menu.



4.

AUTOMATIC MODE

Display the status of the DMC (Dynamic Machining Control).



MANUAL (JOG) MODE

5

When accessing the manual mode from the automatic mode (or viceversa), the CNC keeps the machining conditions selected in the last mode.

5.1 Interface description.

5.1.1 Description of a typical screen of this work mode.

The manual mode has several pages, with different information in each one of them. The visibility of these pages is configurable from the status bar. A typical screen of the jog mode can show the following information:



- A Information on the pages.
- B Position of the axes.
- C Active "M" functions.
- D Active "G" functions and high-level commands.
- E Tool information.
- F "F" axis feedrate.
- G "S" spindle speed.
- H Manual movement type (continuous jog, incremental jog or handwheels).
- I Reserved.

Information on the pages of this mode.

Active operating mode, selected page number and number of pages visible. Clicking on the mouse, the CNC shows the list of pages available and also allows configuration of those that are hidden or visible.

5.

MANUAL (JOG) MODE
Interface description.**Position of the axes.**

In the screen configuration supplied by Fagor, the data displayed will be different on each screen. The most common thing is to show the current position of the axes referred to part zero in large characters, and the amount of following error (axis lag) in small characters. Another more detailed option shows the coordinates of the tool tip and those of the tool base, being both referred to part zero and machine zero.

- The axis name in reverse video indicates that the axis is selected from the jog keypad.
- If the name of the axis appears in red, it means that the PLC is inhibiting the movement of the axes (INHIBIT mark active).
-  In handwheel mode, this symbol next to an axis indicates that the axis has an individual handwheel associated with it.

Tool information.

This information depends on the CNC model; milling machine or lathe.

- Milling model.
 - Number of the active tool "T".
 - Active "D" tool offset.
 - Icon indicating the tool type.
 - Length of the active tool "L".
 - Radius of the active tool "R".
 - Number of the next tool "Nx Tool".
- Lathe model.
 - Number of the active tool "T".
 - Active "D" tool offset.
 - Icon indicating the tool type.
 - Offsets (dimensions) of the tool on each axis.

"F" axis feedrate.

- "F real" real feedrate.
- "F prog" programmed feedrate.
- Feedrate percentage active on the CNC (jog panel, program or PLC).
- Dynamic percentage "Dyn" machining.

The text "F real" appears in red indicating that the PLC is inhibiting the movement of the axes (FEEDHOL mark active).

"S" spindle speed.

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

- Real speed "S real" ("S1 real", "S2 real", etc).
- Icon for turning direction.
- Programmed speed "S prog" ("S1 prog", "S2 prog", etc).
- Active gear set.
- Active speed percentage for the CNC (JOG panel, program or PLC).
- Spindle position "S pos" ("S1 pos", "S2 pos", etc).
- Following error of the "S fwe" spindle ("S1 fwe", "S2 fwe", etc.).

If the text "Sreal" appears in red, it means that the PLC is inhibiting the movement of the spindle (INHIBIT mark active).

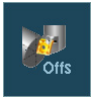
5.1.2 Softkey menus.

Horizontal softkey menu.

Softkey.	Description.
Home search.	Select the axes for a home search.
Preset coordinates.	Select the axes to preset a coordinate or move it to a coordinate.
MDI.	Access the MDI/MDA mode.
Turn the CNC off. 	Start the CNC shut-down sequence, it is the same as pressing the key combination [ALT][F4].
Display.	Toggle between the different screens of the jog mode.

Vertical softkey menu.

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

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.
	Tool calibration (·M· model). See chapter "6 Manual (jog) mode. Tool calibration".
	Tool calibration (·T· model). See chapter "6 Manual (jog) mode. Tool calibration".
	Part centering (·M· model).

5.

MANUAL (JOG) MODE
Interface description.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

5.2 Operations with the axes.

5.2.1 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.

Home search via keyboard.

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

To select the numbered axes (e.g. "X1"), select any axis and then move the selection until positioning on the desired one. The focus moves with the [↑][↓] 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.

Home search via softkey menu.

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

- 1 On the home search softkey of the horizontal menu, select the axis to be homed. The CNC will frame that axis and will display the "1" symbol in the numeric area indicating that a home search will take place.
- 2 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.

Home search via keyboard.



- 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.

Home search via softkey menu.

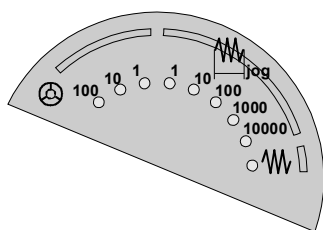
- 1 On the home search softkey of the horizontal menu, select the "All" option. The CNC will display a dialog box requesting confirmation to execute the home search.
- 2 Press [START] to go ahead with the home search or [ESC] to cancel the operation.

5.

MANUAL (JOG) MODE
Operations with the axes.

5.2.2 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

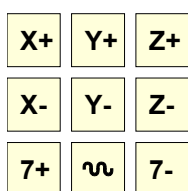

Incremental jog


Handwheels

The jog keyboard and the feedrate selector

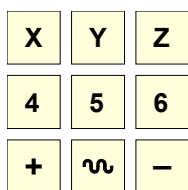
JOG keypad.

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.

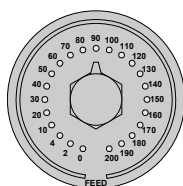
To jog an axis requires activating both the axis key and the moving direction. There are two options, depending on how the jog keyboard has been configured.

- 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.

5.

MANUAL (JOG) MODE
Operations with the axes.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

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.

MANUAL (JOG) MODE

Operations with the axes.



5.2.3 Jogging the axes with handwheels

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

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

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

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

General handwheel

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

- If there are several axes selected in handwheel mode, the general handwheel will move all of them.
- If an axis has been selected which 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.

5.

MANUAL (JOG) MODE

Operations with the axes.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

5.

MANUAL (JOG) MODE

Operations with the axes.

Selecting an axis from the automatic mode

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

Individual handwheel

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



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

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



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

Feed handwheel.

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



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



5.2.4 Moving an axis to a particular position (coordinate).



As of version v2.21.01, the CNC uses the Sub_ManMove.nc subroutine to execute this type of movement. This subroutine can, for example, execute the movements with a different feedrate than the one selected on the CNC.

Move an axis to a particular position (coordinate) via keyboard.

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

Move an axis to a particular position (coordinate) via softkey menu.

- 1 On the axis preset softkey of the horizontal menu, select an axis. The CNC will highlight that axis to indicate that it is selected.
- 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.

MANUAL (JOG) MODE
Operations with the axes.

5.2.5 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".

Preset a position (coordinate) via keyboard.

- 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 moves with the [↑][↓] keys.
- 2 Key in the desired preset value.
- 3 Press [ENTER] to preset the entered value or [ESC] to cancel the operation.

Preset a position (coordinate) via softkey menu.

- 1 On the axis preset softkey of the horizontal menu, select an axis. The CNC will highlight that axis to indicate that it is selected.
- 2 Key in the desired preset value.
- 3 Press [ENTER] to preset the entered value or [ESC] to cancel the operation.

5.

MANUAL (JOG) MODE

Operations with the axes.

5.3 Spindle control.

Displaying the data of several spindles.

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

Spindle control

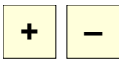
The spindle may be controlled manually using the following keys of the operator panel. The keys always refer to the master spindle of the active channel.

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

Key.	Meaning.
	Start the spindle clockwise (same as M03 function) at the active speed. The CNC shows the M03 function in the program history.
	Start the spindle counterclockwise (same as M04 function) at the active speed. The CNC shows the M04 function in the program history.
	Stop the spindle (same as M05 function). The CNC shows the M05 function in the program history.
	Orient the spindle (same as M19 function). The CNC shows the M19 function in the program history.

Vary the speed override from the operator panel.

With the operator panel, it is possible to change the percentage of spindle speed using a jog keyboard or a switch (depending on model).

Key.	Meaning.
	Increases or decreases the percentage of spindle speed. The maximum and minimum values as well as the incremental step are set by the OEM, the typical values being a variation between 50% and 120% with a 5% step.
	It sets the percentage of turning speed to be applied. The maximum and minimum values are set by the OEM, the typical values being a variation between 50% and 120%.

5.

MANUAL (JOG) MODE
Spindle control.



CNCelite
8058 8060
8065 8070

REF: 2508

5.4 Tool selection and tool change

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

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

5.**MANUAL (JOG) MODE**

Tool selection and tool change



CNCelite
8058 8060
8065 8070

REF: 2508

5.5 Setting the feedrate and spindle speed.

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.

Setting a new spindle speed.

The spindle speed set in the jog mode is applied to the spindle displayed at the time. If there are several spindles in the channel, the rest of the spindles may be displayed by pressing the [S] key. The spindle speed set in jog mode is maintained when switching to automatic mode and vice versa.

Follow these steps to set a new spindle speed.

- 1 Press [S] at the alphanumeric keyboard until selecting the desired spindle. When pressing this key for the first time, the CNC will highlight the relevant data indicating that it is selected.
- 2 Enter the new spindle speed.
- 3 Press [START] to assume the entered value or [ESC] to cancel the operation.

5.**MANUAL (JOG) MODE**

Setting the feedrate and spindle speed.

5.6 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 (JOG) MODE

Setting and activating the zero offsets and the fixture offsets.



MANUAL (JOG) MODE. TOOL CALIBRATION

6

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

Softkey.	Type of calibration.
	Tool calibration in a mill model.
	Tool calibration in a lathe model.

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

Types of calibration

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



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

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

Manual calibration. Calibration without a probe.

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

Semi-automatic calibration. Calibration with a probe.

This calibration mode is available when using a table-top probe. The positioning movements are carried out manually and the CNC executes the probing movements.

Automatic calibration. Calibration with a probe and a canned cycle.

This calibration mode is available when using a table-top probe. The CNC executes all the movements using the calibration canned cycle #PROBE.



CNCelite
8058 8060
8065 8070

REF: 2508

Probe selection

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

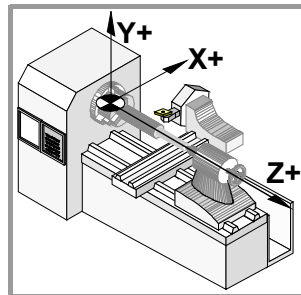
```
#SELECT PROBE [1]
  Selects the first probe.
#SELECT PROBE [2]
  Selects the second probe.
```

6.

MANUAL (JOG) MODE. TOOL CALIBRATION

Geometrical configuration of the axes on a lathe: "plane" or "trihedron".

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

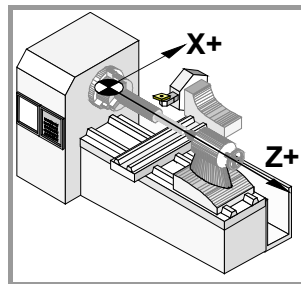


Geometrical configuration of "trihedron" type axes.

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

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

With this configuration, the planes behave in the same way as on a milling machine except that the usual work plane will be G18 (if it has been configured like that).



Geometrical configuration of "plane" type axes.

It is the typical configuration of a lathe.

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

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

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

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

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

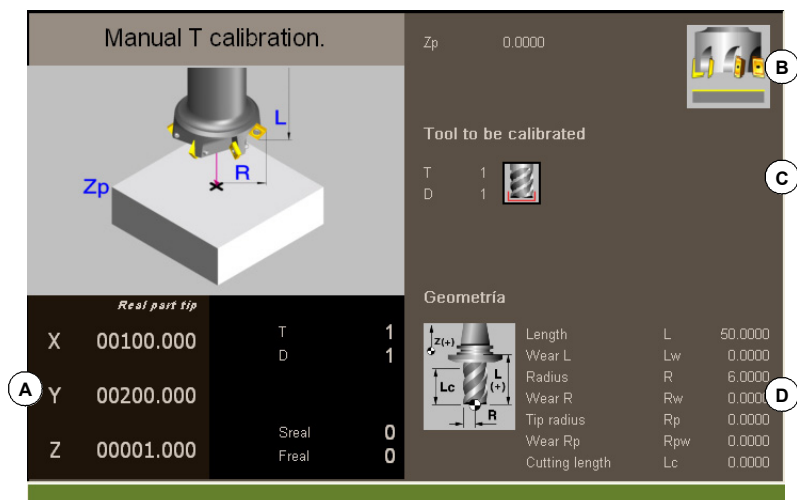
When using milling tools on a lathe, the longitudinal compensation axis may be changed by means of the `#TOOL AX` instruction or the G20 function.



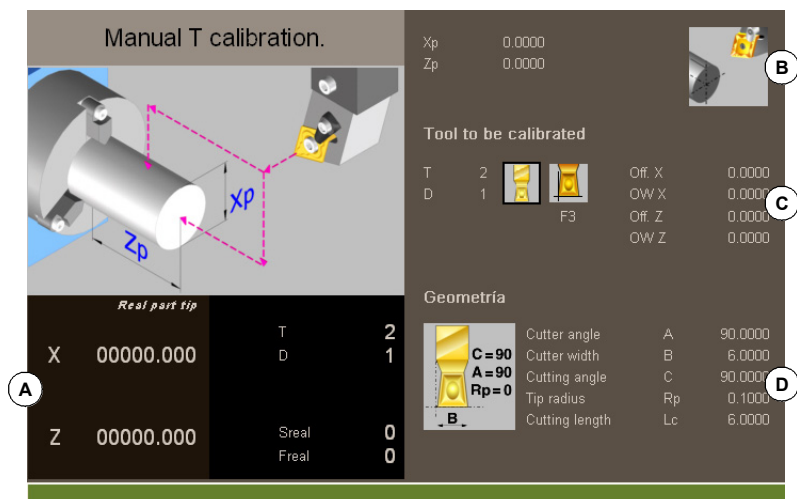
6.1 Manual calibration. Calibration without a probe

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

- Tool calibration window (·M· model).



- Tool calibration window (·T· model).



- A Machine data. Position of the axes, tool and active tool offset, real spindle speed and real feedrate of the axes.
- B Data of the part used for calibration and drawing showing that calibration is possible. If the window does not show this drawing, some of the data is missing.
- C Necessary data for calibration.
- D Tool data.

Tool calibration

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

Selecting a tool

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

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

6.

MANUAL (JOG) MODE. TOOL CALIBRATION
Manual calibration. Calibration without a probe

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

6.

MANUAL (JOG) MODE. TOOL CALIBRATION
Manual calibration. Calibration without a probe**Calibration in a mill model**

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

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

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

Tool calibration in a lathe model (Configuration of plane type axes).

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

Tool calibration in a lathe model (Configuration of trihedron type axes).

- For the lathe tools, it calibrates the tool offsets on each axis. When validating the calibration in one of the offsets, the wear of that offset is reset to zero.
- There are two options for the milling tools and may be selected with the following icons.



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



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

Validating the calibration.

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

Softkey.	Description.
	Validating the length calibration of a milling tool.
	Validating the offsets of a milling tool.
	Validating the offsets of a lathe tool.

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

Definition of data

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

For a lathe tool.

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

Data	Meaning
Zp Xp	Dimensions of the reference part being used in the calibration. These coordinates are referred to the main axes of the tool.
T	Tool to be calibrated.

Data	Meaning
D	Tool offset to be calibrated.
Off Y Off X	Tool offsets on each axis.
Lw	Offset wear on each axis.

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

For a milling tool.

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

Tool calibration steps

To calibrate the length, follow these steps.

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

To calibrate a tool, it must be the active tool. When selecting a tool and pressing [ENTER], the CNC only shows the data for that tool. Press [CYCLE START] for the CNC to make the tool change so it becomes the active tool. See *"Selecting a tool"* on page 127.
- 3 Calibrate the tool. Approach the tool manually until touching the part and then validate the calibration using the softkey menu.

After validating the calibration, it updates the values and initializes the wear value to zero. Then, the new values are saved in the tool table.
- 4 Press [START] for the CNC to assume the new values of the offset.

To calibrate another tool, repeat steps 2 and 3.

Considerations for the offsets and their wear.

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

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

Sign criteria for the offsets and their wear.

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

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

6.

MANUAL (JOG) MODE. TOOL CALIBRATION
Manual calibration. Calibration without a probe

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Entering wear values incrementally or absolutely.

In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute. See *"Select the type of wear values to enter, incremental or absolute."* on page 315.

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

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

6.**MANUAL (JOG) MODE. TOOL CALIBRATION**

Manual calibration. Calibration without a probe



CNCelite
8058 8060
8065 8070

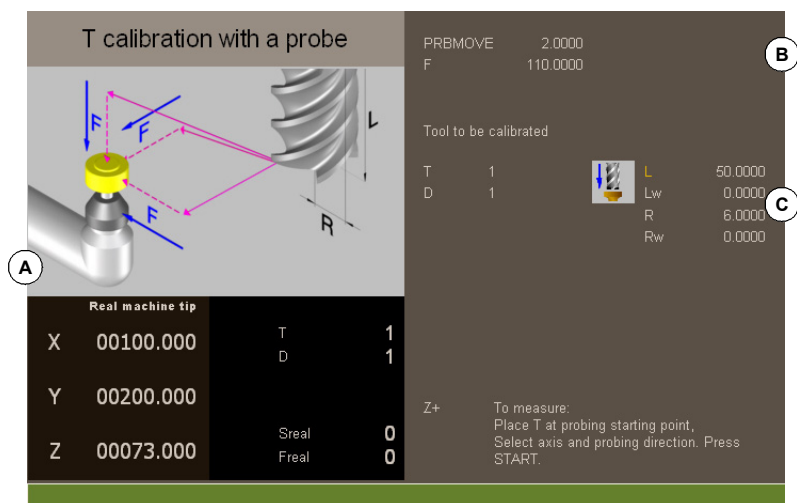
REF: 2508

6.2 Semi-automatic calibration. Calibration with a probe

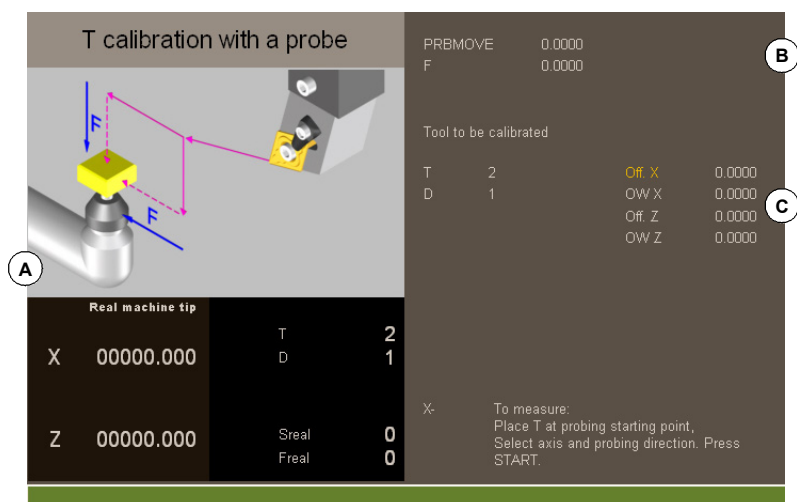
This option is only available when using a tabletop probe installed on the machine.

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

- Tool calibration window (·M· model).



- Tool calibration window (·T· model).



A Machine data. Position of the axes, tool and active tool offset, real spindle speed and real feedrate of the axes.

B Data of probing movement.

C Necessary data for calibration.

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

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

Tool calibration

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

Selecting a tool

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

6.

MANUAL (JOG) MODE. TOOL CALIBRATION
Semi-automatic calibration. Calibration with a probe

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

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

Calibration in a mill model

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

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

Calibration in a lathe model

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

Validating the calibration

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

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

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

Definition of data

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

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

Tool calibration steps

To calibrate the tool, follow these steps:

- 1 Define the probing distance and feedrate. If the feedrate is not defined, the probing movement will be made at the feedrate set by the OEM.
- 2 Select the tool and the offset to be calibrated. After the selection, the CNC shows the dimensions defined in the tool table for that offset.

To calibrate a tool, it must be the active tool. When selecting a tool and pressing [ENTER], the CNC only shows the data for that tool. Press [CYCLE START] for the CNC to make the tool change so it becomes the active tool. See *"Tool calibration"* on page 127.

- 3 Manually approach the tool to the probe until it is placed on the path that will be used for probing. To calibrate the radius with a cylindrical probe, the path must coincide with the probe's center point; if not, the radius will be calculated wrong.

6.

MANUAL (JOG) MODE. TOOL CALIBRATION
Semi-automatic calibration. Calibration with a probe

- 4 Calibrate the tool. Select the axis and the probing direction on the softkey menu and press [START].

The probe moves in parallel to the axis and in the selected direction until touching the probe. It updates the measured value and resets the wear value to zero. The data is stored in the tool table.

- 5 Press [START] again for the CNC to assume the new values of the offset. For the new values to be assumed, press [START] while the bottom message is displayed; otherwise, it executes the probing movement again.

Considerations for the offsets and their wear.

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

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

Sign criteria for the offsets and their wear.

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

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

Entering wear values incrementally or absolutely.

In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute. See *"Select the type of wear values to enter, incremental or absolute."* on page 315.

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

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

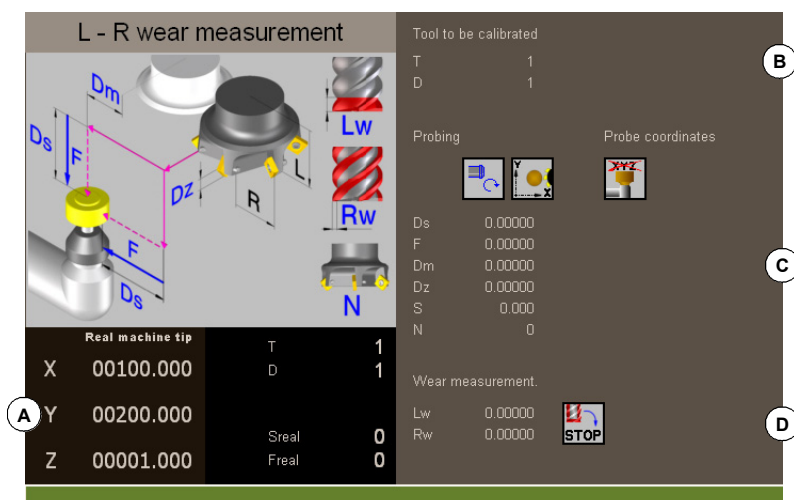
6.

MANUAL (JOG) MODE. TOOL CALIBRATION
Semi-automatic calibration. Calibration with a probe

6.3 Automatic calibration with a probe and a canned cycle

6.3.1 Mill or lathe model ("trihedron" geometrical configuration)

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



- A Machine data. Position of the axes, tool and active tool offset, real spindle speed and real feedrate of the axes.
- B Tool to be calibrated.
- C Data for probe calibration and position.
- D Data for tool wear measurement.

Tool calibration

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

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

Selecting a tool

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

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

Select another position for the probe.



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

Tool calibration

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



- Calibrate the offsets and resets the wears to zero.



- Calibrate the length and radius and measure the wears.

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

6.

MANUAL (JOG) MODE. TOOL CALIBRATION
Automatic calibration with a probe and a canned cycle

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Definition of data

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

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

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

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

To calibrate the offsets of a milling or lathe tool.

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

6.

MANUAL (JOG) MODE. TOOL CALIBRATION

Automatic calibration with a probe and a canned cycle

FAGOR
AUTOMATION 
CNCelite
8058 8060
8065 8070

REF: 2508

Tool calibration steps

To calibrate the length, follow these steps.

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

6.

MANUAL (JOG) MODE. TOOL CALIBRATION

Automatic calibration with a probe and a canned cycle

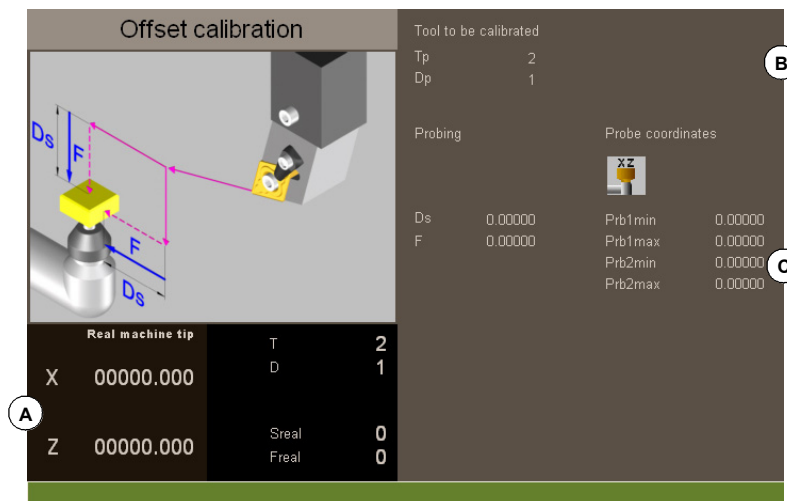


CNCelite
8058 8060
8065 8070

REF: 2508

6.3.2 Lathe model ("plane" geometrical configuration)

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



A Machine data. Position of the axes, tool and active tool offset, real spindle speed and real feedrate of the axes.

B Tool to be calibrated.

C Data for probe calibration and position.

Tool calibration

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

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

Selecting a tool

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

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

Select another position for the probe.



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

Tool calibration

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

6.

MANUAL (JOG) MODE. TOOL CALIBRATION

Automatic calibration with a probe and a canned cycle

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Definition of data

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

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

Tool calibration steps

To calibrate the length, follow these steps.

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

6.

MANUAL (JOG) MODE. TOOL CALIBRATION

Automatic calibration with a probe and a canned cycle



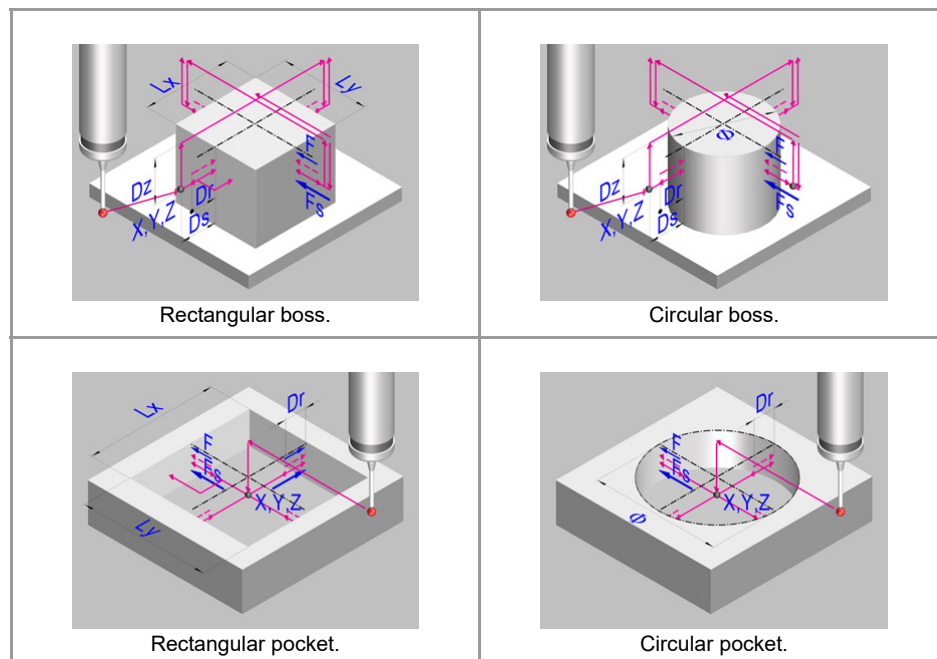
JOG MODE. PART CENTERING (MILL MODEL)

7

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



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



Requirements prior to the calibration.

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

Requirements prior to the calibration.

To center the part, follow these steps.

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

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Data returned by the cycle after the measurement.

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

P296	Inclination angle between the part and the abscissa axis (rectangular part).
P297	Part surface coordinate.
P298	Part center coordinate along the abscissa axis.
P299	Part center coordinate along the ordinate axis.

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

7.

JOG MODE. PART CENTERING (MILL MODEL)

7.1 How to define the data.

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

The direct access keys correspond to the name of the parameters; [F] for forward movements, [T] for tools, etc. Each time the same key is pressed, the next value of the same type is selected.

Manual data entry.

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

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

Leaving some data undefined.

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

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

Defining data using arithmetic parameters..

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

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

Teach-in mode for data entry.

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

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

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

7.

JOG MODE. PART CENTERING (MILL MODEL)

How to define the data.

7.2 Data programming.

Probe data.

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

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

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

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

Probing movement.

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

Xm Probe position along the abscissa axis

Ym Probe position along the ordinate axis.

Zm Probe position along the axis perpendicular to the plane.

The part centering cycle is executed in the active plane.

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

This parameter indicates the type of part/pocket to be centered.



Rectangular part centering.



Circular part centering.



Rectangular pocket centering.



Circular pocket centering.

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

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



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



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



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



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

·icon· **Center the part/pocket on one or two axes with one or two probings on the first side.**

This parameter indicates on how many axes the part is centered and the number of probing movements to carry out on the first side.



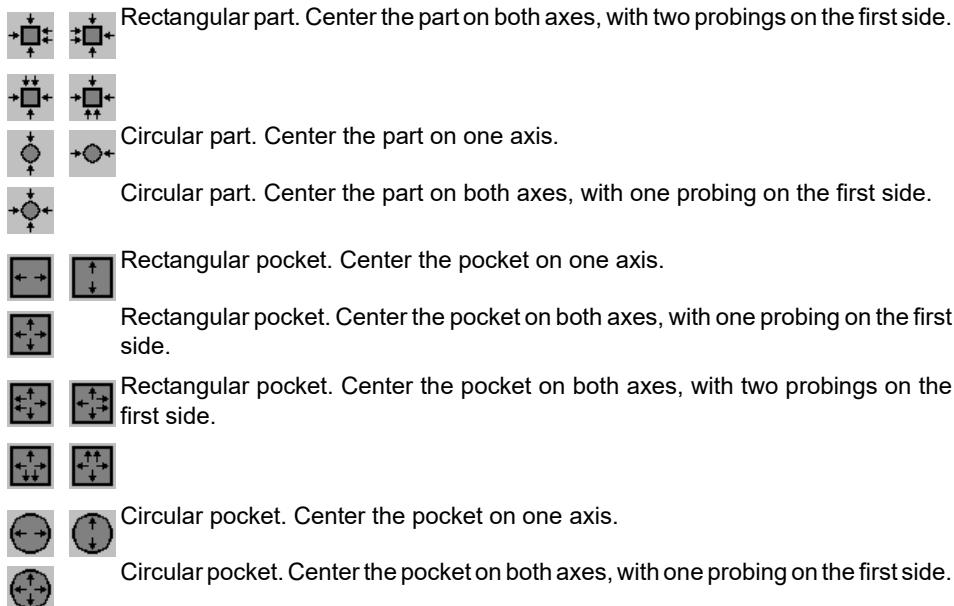
Rectangular part. Center the part on one axis.



Rectangular part. Center the part on both axes, with one probing on the first side.

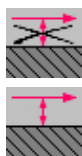
7.

JOG MODE. PART CENTERING (MILL MODEL)
Data programming.



·icon· **Surface coordinate measuring.**

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

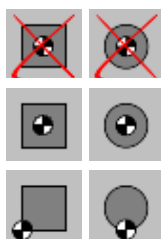


The cycle does not measure the surface coordinate.

The cycle measures the surface coordinate.

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

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



Do not preset the part zero.

Preset the part zero at the center of the part.

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

·icon· **Pattern rotation.**

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



Do not rotate the coordinates (pattern).

Rotate the coordinates (pattern).

·Lx Ly Ø· **Part dimensions.**

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

Lx Rectangular part. Part length along the abscissa axis.

Ly Rectangular part. Part length along the ordinate axis.

Ø Circular part. Part diameter.

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

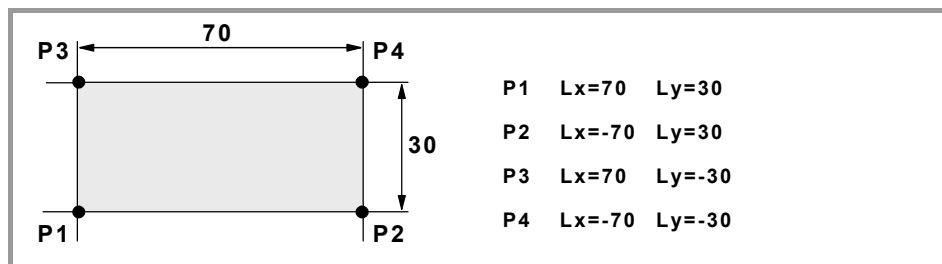
7.

JOG MODE. PART CENTERING (MILL MODEL)
Data programming.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508



·Ds· **Safety distance.**

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

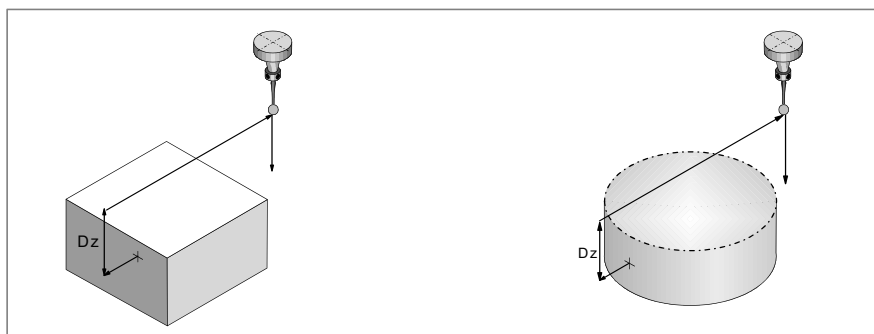
Distance with respect to the point to measure, to which the probe approaches before making the probing movement.

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

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

·Dz· **Safety distance along Z.**

Distance for the probe to go up from the probing coordinate (position) for its movements over the part. This parameter must have enough value to prevent the probe from colliding when moving over the part.



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

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

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

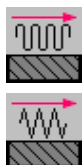
·F· **Probing feedrate.**

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

This parameter sets the feedrate for the second probing movement.

·icon· **Feedrate for approach movements.**

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



The movements are carried out in rapid.

The movements are carried out at work feedrate.

Coordinate preset value.

·Px Py Pz·Coordinate preset value on each axis.

Px	Preset value on the abscissa axis.
Py	Preset value on the ordinate axis.
Pz	Preset value on the axis perpendicular to the plane.

These parameters are only valid when presetting the part zero and any this point may be assigned any value.

Save the result of the measurement.

·icon· Save the result of the measurement in a zero offset G159.

This parameter indicates whether or not the result of the measurement will be saved in a zero offset G159. Regardless of the selected option, the cycle always saves the result of the measurements in the corresponding arithmetic parameters.



Save the result of the measurement in a zero offset G159.



Do not save the result of the measurement in a zero offset G159.

·OffsetG159· Zero offset number

Number of the zero offset into which to save the result of a measurement. If it is set to 0, the cycle does not save any data into the zero offset.

Programming of M functions.

·M before· M functions to be executed before the cycle.

The cycle allows executing up to 4 M functions before the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

·M after· M functions to be executed after the cycle.

The cycle allows executing up to 4 M functions after the cycle. To execute only some of them, define them first and leave the rest unprogrammed.

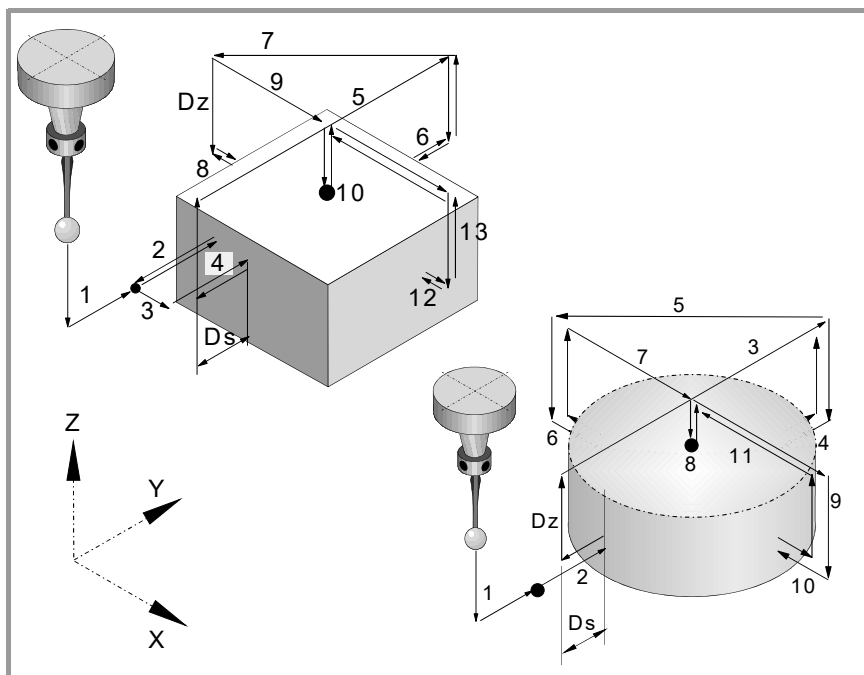


We recommend using these functions, for example, to manage wireless probes. Wireless probes are not always active, they have to be turned on before using the probing cycles and turned off afterwards. For this type of probes, set an M function to turn the probe on and another one to turn it off. Having the probe turn on/off programmed with M functions inside the cycle avoids executing the cycle without having the probe active or leaving the probe always active after executing the cycle.

7.

JOG MODE. PART CENTERING (MILL MODEL)
Data programming.

7.3 Basic operation.



- 1 The cycle executes the "M-before" functions.
- 2 Approach movement (if any of parameters $\cdot XYZ \cdot$ has been defined).
 Movement of the probe at the selected feedrate from the cycle calling point to the first approach point defined by parameters $\cdot XYZ \cdot$.
 The approaching movement is made in two stages. It first moves in the work plane and then along the axis perpendicular to the plane.
- 3 Probing of the first point
 The cycle makes a first probing movement, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.
 First probing movement. Probe movement along the selected axis at feedrate $\cdot Fs \cdot$ until the probe signal is received.
 Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in $\cdot Dr \cdot$.
 Second probe. Probing movement at feedrate $\cdot F \cdot$ until the probe signal is received.
 Withdrawal movement. Rapid probe movement (G00) from the probing point to the starting point.
- 4 If the part is rectangular and probing on two points has been programmed, repeat the probing in a different point of the surface. The cycle uses this second probing to calculate the inclination of the part.
 Approach movement. Movement parallel to the probed side to touch a different point of the same side, located at 1/4 the length programmed in $\cdot Ly \cdot$.
 Probing movement. Probing movement at feedrate $\cdot F \cdot$ until the probe signal is received.
 Withdrawal movement. The probe returns to the safety position located at a $\cdot Ds \cdot$ distance from the part.
- 5 Approach movement to the second probing point.
 Movement of the probe from the first approach point to the second located in front of the first one.
 The perpendicular axis moves the $\cdot Dz \cdot$ distance up in rapid (G00). Then the probe moves over the part, at the selected feedrate, to the next approach point. The probe moves a $\cdot Dz \cdot$ distance back down at feedrate $\cdot Fs \cdot$.
 For the movement, the cycle takes into consideration the dimensions of the part and, if it is rectangular, its inclination angle. If in this last movement, the probe touches the part, the CNC shows an error at the cycle parameter value.

7.

JOG MODE. PART CENTERING (MILL MODEL)

Basic operation.

6 Probing of the second point.

It is done like the previous one. The probe returns to the safety position located at a $\cdot D_s \cdot$ distance from the part.

7 If part centering on two axes has been programmed, approach movement to the third probing point.

Movement of the probe from the second approach point to the center of the part and then to the third point.

8 If part centering on two axes has been programmed, probing of the third point.

It is done like the previous one.

9 Approach movement to the top surface (only if it has been programmed).

When selecting to measure the surface, the probe moves the $\cdot D_z \cdot$ distance up in rapid (G00) and the probe moves to the center of the part.

10 Measuring the top surface of the boss or the bottom of the pocket (only if it has been programmed).

The probe makes a first probing move, a withdrawal movement and a second probing move. After the second probing move, the probe withdraws to the approach point.

First probing movement. Probing movement, at feedrate $\cdot F_s \cdot$, until touching the part surface.

Withdrawal movement. Return of the probe in rapid (G00) the distance indicated in $\cdot D_r \cdot$.

Second probe. Probing movement at feedrate $\cdot F \cdot$ until the probe signal is received.

Withdrawal movement. Rapid probe movement (G00) from the probing point to the safety position.

11 If part centering on two axes has been programmed, approach movement to the fourth probing point.

Movement of the probe from the third approach point to the fourth.

12 If part centering on two axes has been programmed, probing of the four point.

It is done like the previous one.

13 Probe positioning at the calculated center.

This movement is carried out in two stages. It first moves along the perpendicular axis in rapid (G00) and then in the plane.

14 If programmed, the cycle saves the result of the measurement in the selected G159 zero offset and in the corresponding arithmetic parameters.

15 The cycle executes the "M-after" functions.

7.

JOG MODE. PART CENTERING (MILL MODEL)

Basic operation.

7.

JOG MODE. PART CENTERING (MILL MODEL)

Basic operation.



CNCelite
8058 8060
8065 8070

REF: 2508

EDISIMU MODE (EDITING AND SIMULATION)

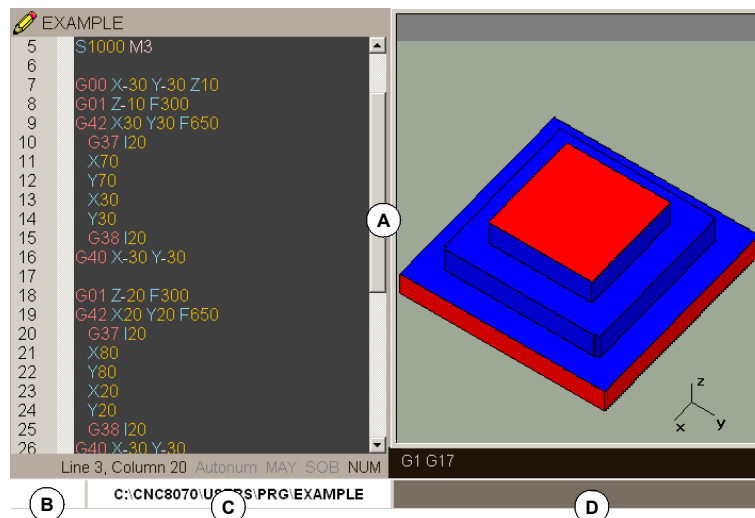
8

Program editor, Unicode format.

The editor of the CNC admits Unicode characters. When saving a program, the editor respects its original format, ANSI or Unicode, except when editing a Unicode character, in which case the CNC will always save the program in Unicode format.

8.1 Interface description.

On a typical screen for this work mode, the information is laid out as follows.



- A Windows of the EDISIMU mode. Every screen may consist of one or more windows.
- B Status of the program selected in this work mode or channel number when using them. In any case, the background color will be different depending on the status of the program being simulated.

Ready	Background color: White.
In simulation	Background color: Green.
Interrupted	Background color: Dark green.
In error	Background color: Red.
- C Program name and location.
- D CNC messages.



CNCelite
8058 8060
8065 8070

REF: 2508

Window description

As mentioned earlier, each screen may consist of one or many of the following windows (later sections of this chapter describe each one of them in greater detail):



When the screen consists of several windows, the softkey menu will show the options of the active window. To switch windows and access the desired softkey menu, press the [FOCUS] key.

- **Edit window:** For editing new programs or modify the existing ones. Editing is possible using a profile editor, a conversational canned cycle editor or using the TEACH-IN feature.
- **Graphic window:** This window shows a graphic representation of the program during the simulation. It also allows taking measurements on the graphics.
- **Program window:** For selecting the starting and ending conditions of the simulation.
- **Statistics window:** For estimating the machining time for each tool and the total program execution time.
- **Cycle editor.** The cycle editor makes it easier to edit machining and probing cycles.
- **Profile editor.**
- **Editor for inclined planes.**
- **Geometric help editor.**

8.

EDISIMU MODE (EDITING AND SIMULATION)

Interface description.



8.1.1 Softkey menus.

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.
	START (simulation) Starts program simulation or resume it if it was interrupted. During simulation, the graphic window will show the graphic representation of the program being simulated.
	STOP (simulation) Interrupt program simulation. Simulation will resume by pressing the START icon.
	RESET (simulation) Cancel program simulation. If an error occurs during simulation, reset eliminates the error status and returns the simulation mode to its initial conditions.
	Change the channel being displayed for editing and simulation. It does not affect the active channel at the CNC. (This icon will only be available when using channels).
	Select the "single block" or "continuous" mode; either one may be selected even while executing a program. When "single block" mode is active (the icon will appear pressed), program simulation will be interrupted at the end of each block. When the "automatic" mode is active, the simulation will take place until the end of the program or up to the block selected as end of simulation.
	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.
	Offer an estimate of the total execution time at 100% of the programmed feedrate. The result will appear at the statistics window. (This icon is only available when the statistics window is displayed).
	Configuring simulation options. • Activate and cancel tool radius compensation. • Conditional stop. • Activate and deactivate the software travel limits. • Block skip condition. • Cancel the channel synchronization in the simulation.

8.

EDISIMU MODE (EDITING AND SIMULATION)

Interface description.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

8.2 Program editing and simulation

8.2.1 Program editing

The process of editing or modifying a program is the following:

- 1 Open the program at the editor or create a new program.
- 2 Use the cursor to select the block where the new blocks will be inserted. Edit the program blocks using the editing modes available.
- 3 Save the program so it can be modified or executed later on. This operation may be performed automatically as the program is being edited or if the editor has been personalized.

Available editing modes

Any of the various editing modes available may be used while editing a program. The available editing modes are:

CNC language

It is edited block by block and each one may be written in ISO language or in High level language. When editing high level commands and depending on the type of command, the editor offers a list of available commands. See ["8.3.2 Contextual programming assistance."](#) on page 159.

8055 CNC language

Part-programs may be edited in the CNC's own language and in the 8055 language. The 8055 CNC programming language is enabled at the part-program editor using the "Customize" softkey of the horizontal menu. Within this option, activate the softkey of the 8055 editor. See ["8.2.2 Editing a program in the 8055 CNC language"](#) on page 149.

Canned cycle editor

With this editor, it is possible to define machining and probing canned cycles quickly and easily. When done editing the cycle, the CNC generates the necessary blocks and it will add them to the program inserting them after the block indicated by the cursor. This mode offers the following advantages:

- There is no need to know the canned cycle parameters.
- The CNC only lets entering the data being shown, thus avoiding any data entry errors when defining the cycles.
- The programmer is assisted at all times with help messages.

Profile editor

With this editor it is possible to edit new profiles quickly and easily. The editor shows a graphic representation of the profile being defined. After defining the profile data, the CNC generates the necessary blocks and it will add them to the program inserting them after the block indicated by the cursor.

Inclined planes (·M· model only)

Help to the user for programming inclined planes using the instructions #CS and #ACS. Using the "Insert" softkey, the block corresponding to the programmed instruction is inserted after the block where the cursor is.

Geometric help

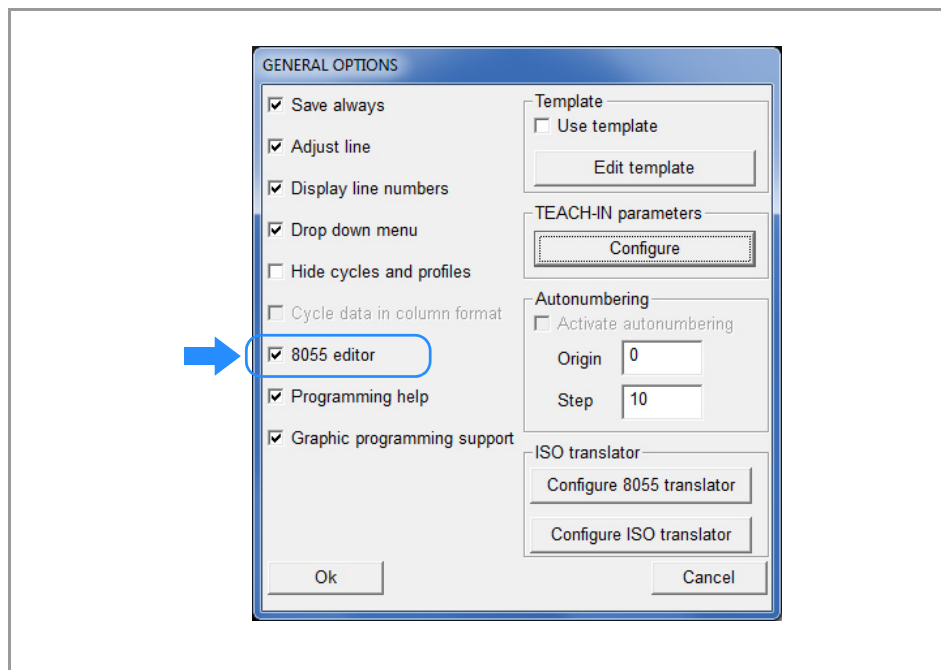
Assistance to the user for programming geometric help (scaling factor, corner rounding, etc.) using G72, G73 instructions, etc. Using the "Insert" softkey, the block corresponding to the programmed instruction is inserted after the block where the cursor is.

TEACH-IN

It is basically the same as editing in CNC language, except when it comes to programming coordinates. This option displays the coordinates of each axis and lets enter them directly into the block indicated by the cursor.

8.2.2 Editing a program in the 8055 CNC language

Part-programs may be edited in the CNC's own language and in the 8055 language. The 8055 CNC programming language is enabled at the part-program editor using the "Customize" softkey of the horizontal menu. Within this option, activate the softkey of the 8055 editor. When this option is deactivated, the CNC always works with its own language.



Even when this option is active, programs may also be edited in the CNC language. The CNC assumes one or the other language depending on the name assigned to the part-program. Once a programming language has been selected, it does not admit instructions of the other language; i. e. Both languages cannot be combined in the same part-program.

Even when this option is active, it is possible to edit and execute part-programs at the CNC that have been written in the 8055 CNC language. The CNC does not allow executing instructions of the 8055 CNC from the MDI mode.

Editing a program in the 8055 CNC language

Having set the editor to accept the syntax of the 8055 CNC, the CNC assumes that this language will be used in the following cases. In the rest of the cases, the CNC interprets that the 8055 CNC language will not be used and assumes its own syntax.

- When the program name has the extension *pit* (lathe program) or *pim* (mill program). These are the native extensions of the 8055 CNC.
- When the program name is a number with no more than 6 digits (999999) like the program names at the 8055 CNC. In these cases, the CNC automatically adds the extension *pim* (for a mill) or *pit* (for a lathe).

Once the program has been opened, edit it using the 8055 CNC language; the CNC will not admit instructions of the CNC.

Program editing limitations

It is not possible to program calls to subroutines that are contained in other programs.

The syntax check is not available for programs written in 8055 CNC language. The softkey to run the syntax check of the program will be disabled.

8.

EDISIMU MODE (EDITING AND SIMULATION)

Program editing and simulation

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Executing and simulating a program edited in the 8055 CNC language

To execute or simulate a program written in the 8055 CNC language (those with extensions *pim* or *pit*), the CNC converts it into its own language. The CNC always opens, simulates and executes the converted program, which is also the one shown on the screen during the execution.

If any error occurs during execution, the CNC will display the block that caused it. The block will be in the language of the CNC, but it will be easily identified due to the translation format.

The conversion of the part-program.

The program is only converted one; the first time that is simulated or the first time that the program is selected in automatic mode. When modifying the program edited in the 8055 CNC language, the CNC converts it again. When modifying the converted program, the CNC does not update the one written in 8055 CNC language.

The CNC keeps both programs; the one written in 8055 format and its equivalent translated. The converted (translated) program is saved with the same name but with the extension *m55* (milling program) or *t55* (lathe program), in the folder selected by the user (by default ..\Users\Prg\PRG_8055_TO_8070) See ["ISO Translator. Configuring the 8055 translator."](#) on page 167.

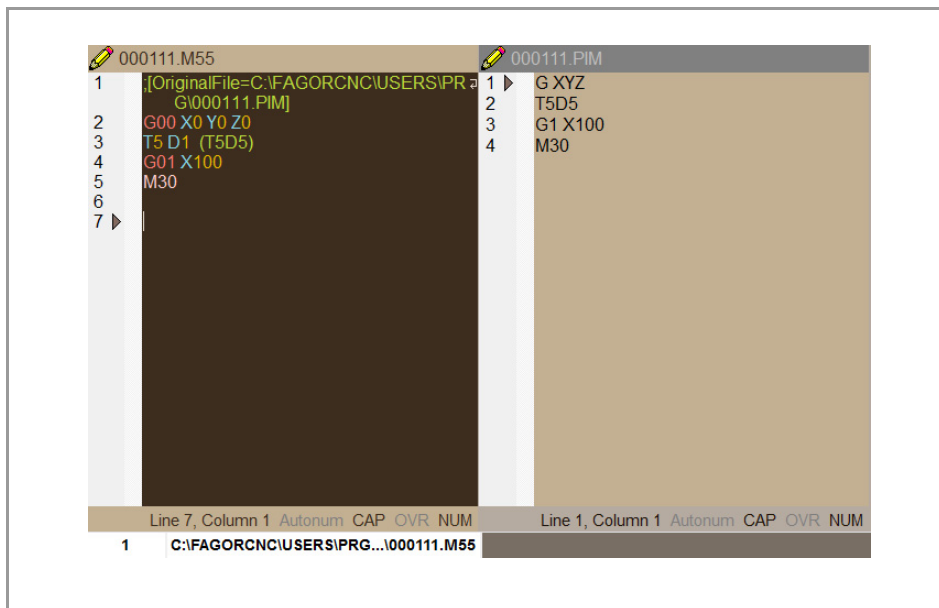
8.

EDISIMU MODE (EDITING AND SIMULATION)

Program editing and simulation

8.2.3 Translating a program in 8055 language.

Page 5 of the edisimu mode can help translate programs written in the 8055 language (pit or pim extension). Open the file in 8055 format in the right window and press the "Translate" softkey; the left window displays the translated file.



The CNC keeps both programs; the one written in 8055 format and its equivalent translated. The converted (translated) program is saved with the same name but with the extension *m55* (milling program) or *t55* (lathe program), in the folder selected by the user (by default ..\Users\Prg\PRG_8055_TO_8070) Deleting a translated program (pim/pit) also deletes the original (m55/t55).

If the syntax of the converted block is considerably different from the original, the CNC adds the original block as comment so as to identify it easily.

8055 language.	Converted (translated) program
G XYZ	G0 X0 Y0 Z0 (G XYZ)
T1 D2 M6	T1 D1 M06 (T1 D2 M6)
G01 G05 G90 F1000	G01 G05 G90 F1000
X10 Y23 Z33	X10 Y23 Z33
G75 X100	G100 X100 (G75 X100)
M30	M30

When the cannot translate a block because it has a function without equivalent, it will show the message "function without translation". The syntax analyzer will show this message when the index of a machine parameter is indicated parametrically.

Configure the 8055 translator options.

See ["ISO Translator. Configuring the 8055 translator."](#) on page 167.

Program translation in 8055 format containing conversational cycles.

The CNC translates the conversational cycles and displays them in their own language. These cycles can be opened and modified in the editor by pressing the [RECALL] key. After editing, the CNC can insert the cycle inside another program.

Cycles that contain profiles.

If the 8055 cycles contain profiles, and these profiles are in the same folder as the program to be translated, the profiles will be translated to the folder ..\Users\Profile, where the rest of the profiles are (for example, those of the profile editor). To prevent overwriting the profiles of this folder, different names should be assigned to profiles so that they are different to each other.

8.

EDISIMU MODE (EDITING AND SIMULATION)
Program editing and simulation

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

It is recommended to activate the option "Request confirmation before replacing files" in the translation options, so that the translator requests confirmation before replacing an existing program or profile.

Translation of conversational cycles in 8055 format.

When pressing [RECALL] for a cycle in language 8055, the CNC will translate it and display it in the cycle editor. After editing, the CNC can insert the cycle inside another program.

8.**EDISIMU MODE (EDITING AND SIMULATION)**

Program editing and simulation

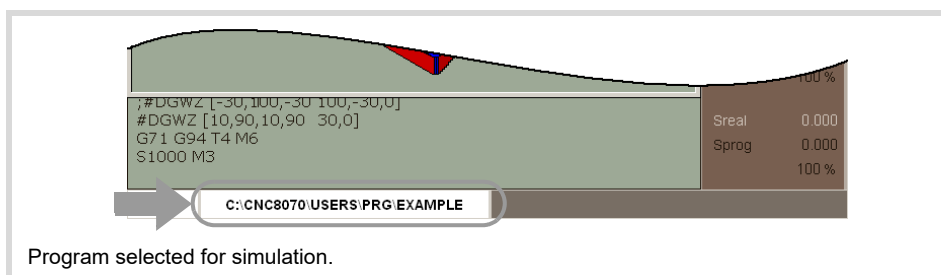


CNCelite
8058 8060
8065 8070

REF: 2508

8.2.4 Program simulation

The graphic window shows the program selected at the editing window and its name appears at the bottom center of the screen.



The program simulation procedure is the following:

- 1 Choose the type of graphic representation, its dimensions and the point of view. This data may also be modified during the simulation of the program.
- 2 Activate the desired simulation options using the icon menu.
- 3 Press the START icon to start the simulation. Every time the START icon is pressed, the editor saves the program being edited in the channel, even when the "save always" option is not active.
- 4 The simulation may be interrupted with the STOP icon or canceled with the RESET icon.

The simulation of the program starts at the first block of the program and ends after executing one of the end-of-program functions "M02" or "M30". As an option, it is possible to define the first and last blocks of the simulation. See ["8.8 Working in the program window."](#) on page 184.

To simulate the program, the CNC assumes the real configuration of the spindles of the channel and the configuration of the machine parameters.

"Single block" mode of simulation.

If the –single block– mode is active, program execution will interrupted at the end of each block; the [START] icon must be pressed again to continue.



The –single block– mode is selected from the icon menu and may be selected before or while simulating the program.

Simulation options

The available simulation options are accessed from the icon menu. Pressing the icon displays a window that shows the following options.



Tool radius compensation

It activates or cancels tool radius compensation to simulate the program.

Tool radius compensation being off, the simulation will ignore the programmed tool radius compensation.



Conditional stop during simulation

It simulates the external "conditional stop" switch.

While active, the program simulation will be interrupted at the blocks having a "conditional stop" function "M01". The simulation will resume when pressing the START icon.

8.

EDISIMU MODE (EDITING AND SIMULATION)

Program editing and simulation

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

8.

EDISIMU MODE (EDITING AND SIMULATION)

Program editing and simulation

**Software and work zone limits.**

Activate or deactivate the software limits and the work zones for program simulation.

- Having this option active, if during simulation the axes reach the software limits or the work zones, the CNC interrupts simulation and issues the corresponding error message.
- Having this option deactivated, the CNC will ignore the software limits and the work zones during simulation. The CNC ignores the option to assume the active zones in execution.

**Block jump**

It simulates the external "block skip" switch.

While active, the blocks having the block-skip character "/" will be ignored (not simulated).

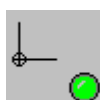
**Cancel channel synchronization**

There is one icon for each channel. It cancels the channel synchronization wait periods during simulation.

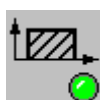
When active, the wait period will end immediately and it will resume the execution of the program.

**Synchronizing spindles.**

There is an icon for each spindle where to indicate the spindle to synchronize with. The '0' value cancels the synchronization.

**Assume the active origins for execution.**

With this option, when starting the simulation or pressing the simulation reset, the CNC applies to the simulation the origins set in the execution environment (for example, the part zero set in jog mode).

**Assume the active zones in execution.**

Assume or not, for simulation, the active zones in execution.

8.2.5 Simulation errors

When an error occurs, the CNC will display a window describing the cause of the error. These errors are displayed in the middle of the screen, regardless of which window is active. Pressing the [ESC] key closes the windows one by one. Use the [↑][↓] keys to see the various windows without closing them.

There are two types of errors. The top of the window shows the category and it will have a different color depending on the type of error it shows.

WARNING

It is just a warning, it does not interrupt the simulation of the program.

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

ERROR

The errors do not interrupt the simulation of the program.

Although the window displaying them may be closed by pressing [ESC], it does not mean that the error status has been taken care of; to do that, press the [RESET] icon. The program can be neither edited nor simulated while the error stays state active.

8.**EDISIMU MODE (EDITING AND SIMULATION)**

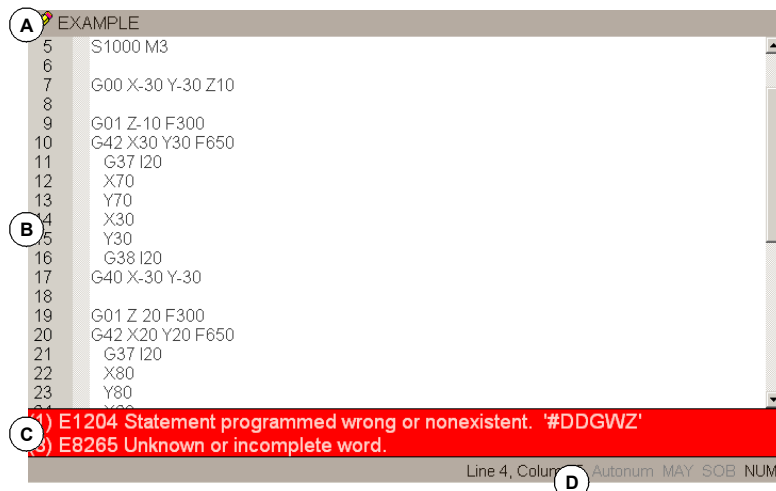
Program editing and simulation

8.3 Editing window

This window may be used to edit, modify or see the contents of a part-program and check the program for syntax errors.

Description of the editing window

The editing window shows the following information:



A Title bar

Name of the program selected for editing. It also indicates whether it is a read-only program or not (if read-only, while simulating or executing). An "*" next to the program means that the program has been modified since last saved (only if automatic program saving is off).

B Edit area.

Line number and area for editing the program.

C Editing errors (if any) and programming assistance. If the text is not display in full, place the focus in this area and move the text using the [←][→][↑][↓] keys.

- This area shows, on a red background, the errors that occur while editing the program or the errors found after running a syntax check of the program.
- This area shows, on a blue background, the contextual assistance offered by the editor when programming commands in high level language.

D Status bar.

Information about cursor position and the status of the editor options such as:

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

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.

Functionalities of the editor.

Group lines at the editor.

The blocks can be grouped in order to expand or shrink them in the same way as for the cycles or profiles. To group the blocks, program the comment (*) at the beginning of the group and the comment () at the end. The blocks between both comments make up a group.

Expand y hide cycles, profiles and grouped blocks.

The editor offers the [ALT]+[+] hotkey to expand y hide cycles, profiles and grouped blocks. If the CNC has a mouse, click on the symbol located to the right of the cycle, profile or group of blocks to expand them and hide them.

8.

EDISIMU MODE (EDITING AND SIMULATION)
Editing window

Having the "Hide cycles/profiles" option is active, the editor only shows the name of the canned cycle or of the profile. Having this option active, when the cursor moves over a hidden element, it expands automatically; when the cursor moves out of the element, it shrinks again.

Zoom at the editor.

The editor has the following hotkeys to increase or decrease the size of the editor font. If the CNC has a mouse with a wheel, the [CTRL] key combined with this wheel can also be used to increase and decrease the size of the text font.

[CTRL]+[+]	Zoom in.
[CTRL]+[-]	Zoom out.

Multi-line blocks.

```
POCK.)
[
.C.M=0
```



The editor adjusts the long blocks to the size of the window dividing the block into several lines. On the right side of each cut line, the editor shows a symbol to indicate that the block continues in the next line.

8.

EDISIMU MODE (EDITING AND SIMULATION)

Editing window

8.3.1 Softkey and hotkey menus.

Horizontal softkey menu.

The options available in the softkey menu are the following.

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.
Geometry and planes.	Access the help for programming inclined planes and geometric help.
Cycle editor.	Access the editor for machining and probing canned cycles.
Profile editor.	Access the profile editor and define a new profile or modify an existing one.
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.
TEACH-IN.	Turn TEACH-IN mode on or off.
Customizing.	Customize the appearance and the properties 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.

8.

EDISIMU MODE (EDITING AND SIMULATION)

Editing window

8.3.2 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 into the block that is being edited.

Entering a help element in the block being edited.

When an element has been selected from the list and [ENTER] is pressed, the editor inserts in the cursor position the element selected in the drop menu.

- If the statement does not have parameters, the editor inserts the whole statement.
- If the statement has parameters, but the text written by the user does not have any parameters, the editor inserts only the fixed part of the statement. If the user has written a parameter, the editor does not insert anything.
- If it is not an axis variable and/or array variable, the editor inserts the whole variable.
- If it is an axis variable and/or array variable, but the user has not written it, the editor inserts only the fixed part of the variable. If it is an axis variable and/or array variable and the user has written it, the editor does not insert anything.

After inserting an element, if it is necessary, the bottom of the screen maintains the contextual assistance for that element to complete the editing of the block. Pressing [ENTER] again, the contextual assistance disappears from the bottom of the screen.

Activating the contextual assistance.

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

8.

EDISIMU MODE (EDITING AND SIMULATION)
Editing window

8.3.3 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.

8.

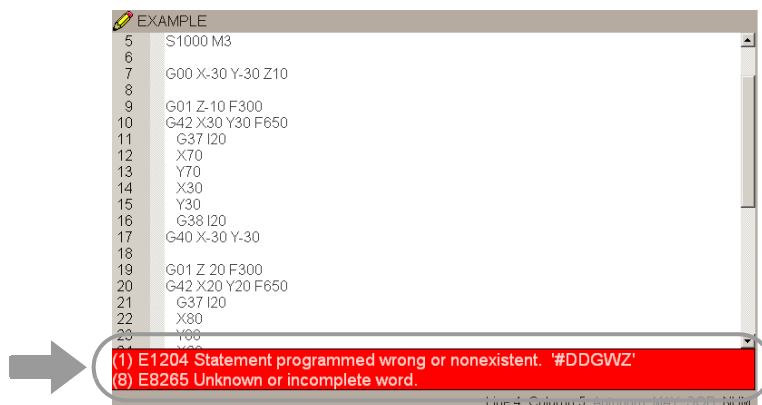
EDISIMU MODE (EDITING AND SIMULATION)

Editing window



8.3.4 Syntax errors when editing

The syntax errors occurred while editing or after running a syntax check will be displayed at the bottom of editing window. To toggle the cursor between the editor and the error listing, press the key combination [CTRL]+[TAB].



Errors while editing

While editing, each block is analyzed when entered. If a syntax error is detected in the block, the error window will display the following information:

- Position of the error in the block.
- Error number and explanatory text.

Errors after a syntax check



The syntax check is executed from the icon menu. The syntax check is not available for programs written in 8055 CNC language.

The syntax check checks all the blocks of the program. If a syntax error is detected, the error window will show the following information.

- Location and name of the program being checked.
- Line number and position of the error within the block.
- Explanation of the error.

Moving the cursor through the errors of the window, the editor will highlight the block containing the selected error. Use the [↑][↓] keys to move the cursor. Press [ENTER] to select the block containing the error or press [ESC] to close the error window.

If the text is not display in full, place the focus in this area and move the text using the [←][→][↑][↓] keys.

8.

EDISIMU MODE (EDITING AND SIMULATION)
Editing window

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

8.4 Working in the editing window.

8.4.1 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 and executed in each channel. When selecting this option, the CNC shows a list of the available programs. See ["3.11 File selection window"](#) on page 64.

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 lower window and the CNC will open an empty program or a predefined template depending on how the editor is configured. See ["8.4.6 Customizing the editor \(general options\)."](#) on page 165.
- 3 Press [ENTER] to accept the selection and open the program or [ESC] to cancel it and close the program listing.

8.4.2 Operations with blocks. 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.

8.4.3 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].

8.4.4 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. |

8.

EDISIMU MODE (EDITING AND SIMULATION)

Working in the editing window.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

8.4.5 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.

This option is only available when the "auto save" option is active. See ["8.4.6 Customizing the editor \(general options\)." on page 165.](#)

File "Save"

This softkey saves the file that is being edited.

This option is only available when the "auto save" option is not active. See ["8.4.6 Customizing the editor \(general options\)." on page 165.](#)

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 "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 "Print"

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

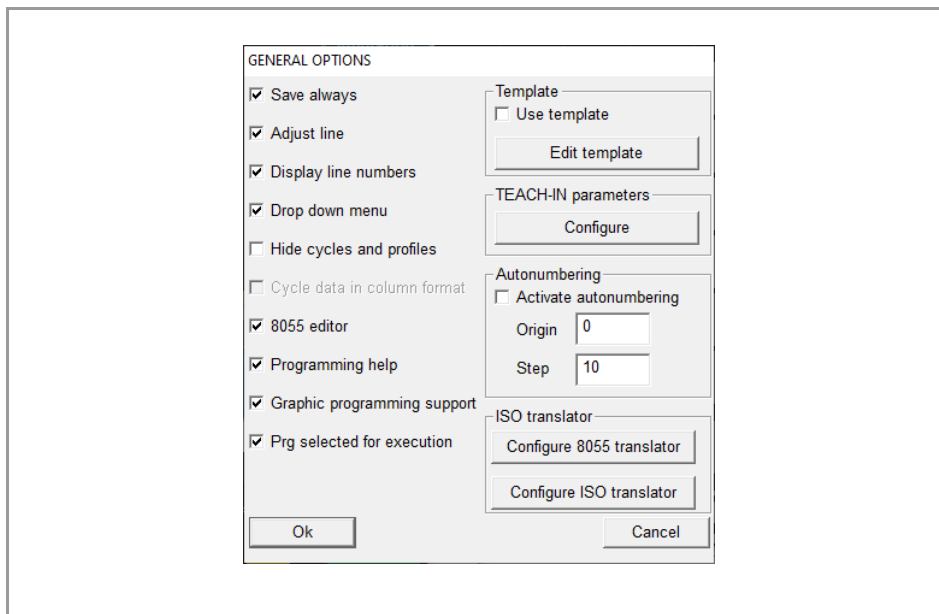
File "Import"

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

- The DXF format is standard for exchanging graphic files. Importing this type of files makes it possible to generate the part-program directly from a drawing. The files must consist of points, lines and arcs. See ["8.4.10 Import DXF files" on page 171.](#)
- 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 program from the list and press [ENTER].

8.4.6 Customizing the editor (general options).



Save always.

Activating the automatic saving of the program; if this option is not active, the program is saved from the softkey menu. When this option is active, the CNC will automatically save the program every time the cursor changes blocks. In large programs (more than 200 kB), the CNC saves the program when the user has not modified the program for about 5 seconds.

Line adjust.

Adjust the long blocks to the size of the window dividing the block into several lines. In large programs (more than 200 kB), the CNC does not adjust the lines.

Show the line number.

Show the line numbers at the editor.

Drop menus.

Organize the horizontal softkey menu from edisimu mode in drop menus.

Hide cycles and profiles.

When opening the program, insert the canned cycle or profile, the editor only shows the name of the element; otherwise, it shows the entire content. In both cases, the content of the canned cycle or the profile can be displayed or hidden by pressing [ALT][–]. In large programs (more than 200 kB), the editor does not hide the canned cycles or the profiles.

Editor 8055.

Activate editing in 8055 CNC language. See ["8.2.2 Editing a program in the 8055 CNC language"](#) on page 149.

8.

EDISIMU MODE (EDITING AND SIMULATION)

Working in the editing window.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Programming assistance.

Activate the contextual assistance for programming commands in high level language. Contextual assistance is not available when using the 8055 CNC language. See ["8.3.2 Contextual programming assistance."](#) on page 159.

Graphic assistance for programming.

While editing the program, this window shows the 2D geometry of the program or of the profile being edited. See ["8.5 Graphic assistance for the program editor."](#) on page 181.

In the editor, open the programme selected to run.

This option displays a softkey in the horizontal menu to open the selected programme in the editor in automatic mode.

Template.

This option activates the use of the template for new programs. The "Edit template" button opens a template at the editor to edit it. Only one template may be in the editor, called "Template.nc" and must be saved in the following folder.

..\Users\Session\Templates

The "Save always" option of the editor sets how the template is saved, automatically or from the softkey menu.

TEACH-IN parameters.

This option sets the behavior of each axis in TEACH-IN mode. Each axis may have one of the following behaviors.

Behavior.	Meaning.
Selected and visible.	The axis is displayed in the TEACH-IN window and it is included in the blocks being edited by pressing the [RECALL] key.
Not selected and visible.	The axis is displayed in the TEACH-IN window, but it is not included in the blocks being edited by pressing the [RECALL] key.
Not selected and hidden.	The axis is neither displayed in the TEACH-IN window nor included in the blocks being edited by pressing the [RECALL] key.

Autonumbering.

This option activates the auto-numbering of blocks and may be used to set the number of the first block and the sequencing step for two consecutive blocks. While auto-numbering is active, the CNC inserts the block number automatically every time a new block is generated (line break).

8.

EDISIMU MODE (EDITING AND SIMULATION)
Working in the editing window.



ISO Translator. Configuring the 8055 translator.

Options for configuring the 8055 CNC program translator.

Translation of tool editors.

Tool correctors programmed in the language of the 8055 can always be translated as D1, if the translator has been so configured. Furthermore, the translator can be configured to add a #WARNINGSTOP or not when translating tool offsets.

Translation of the G72+G16+G15 sequence.

The G72+G16+G15 programmed in the 8055 language may be translated as #CYL when the translator has been configured accordingly. This sequence makes it possible in the 8055 to work with a C axis end mill as if it were a C axis for lathe, to machine cylindrical parts.

Original program.	Converted (translated) program.
N10 G72 C1.2 N20 G16 ZC N30 G15 X	#CYL [Z, C, X47.746483]

Translation of G58, G59 and G158.

- With the selected option, the functions G58 and G59 are translated using the subroutine G558 (configurable). The translation of these functions must be carried out in the CNC where they are going to run, in the channel where they are going to run and with the offset table correctly loaded; otherwise, the zero offset will be incorrect.

Original program (8055).	Converted (translated) program.
G58	G558 O5
G59	G558 O6
Cancellation.	G558

- When the option is not selected, the functions G58 and G59 are translated using the G158 function.

Original program (8055).	Converted (translated) program.
G58	#WARNINGSTOP ["(Approximate translation)"] G158 X=V.A.ORG[5].X Y=V.A.ORG[5].Y Z=V.A.ORG[5].Z (G58)
G59	#WARNINGSTOP ["(Approximate translation)"] G158 X=V.A.ORG[6].X Y=V.A.ORG[6].Y Z=V.A.ORG[6].Z (G59)

8.

EDISIMU MODE (EDITING AND SIMULATION)

Working in the editing window.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

ISO Translator. Configuring the ISO translator.

Configuration table for the translator into the Fagor language from programs written in other languages. To work with programs in the Fagor language, it is not required for the table to be configured. Consult the "Translator of part programs" (man_elite_58_60_65_pptrans.pdf) manual for the available languages and the table meaning.

8.

EDISIMU MODE (EDITING AND SIMULATION)

Working in the editing window.

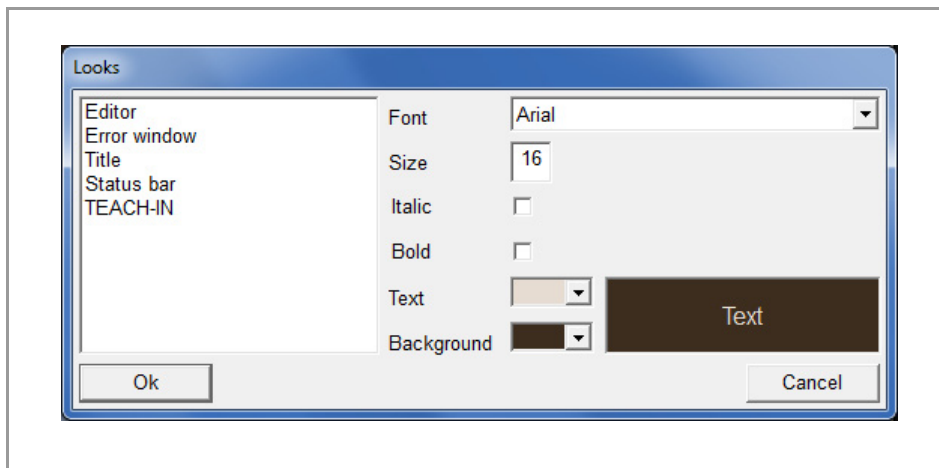


CNCelite
8058 8060
8065 8070

REF: 2508

8.4.7 Customizing the editor (appearance).

This option customizes the appearance (color, font, etc.) of the elements that make up the program editor. After defining the new look, press [ENTER] to accept the changes or [ESC] to ignore them.



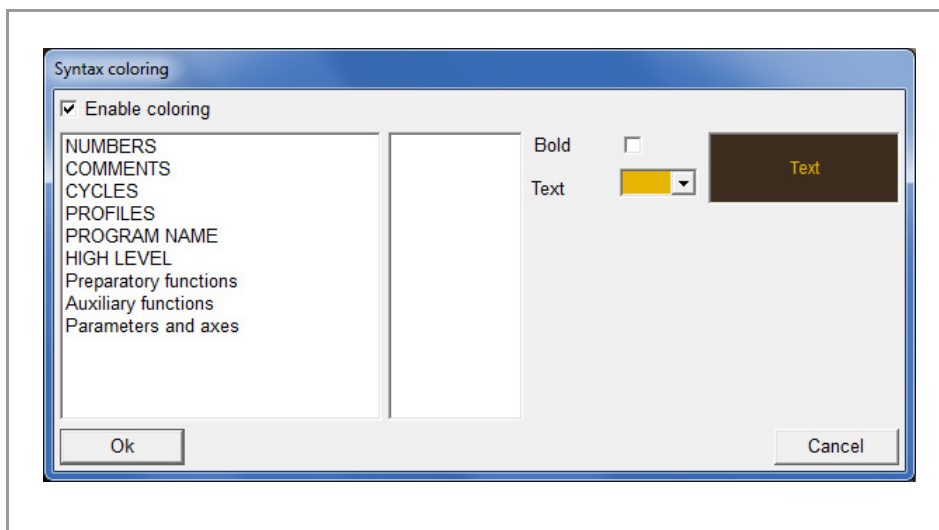
8.

EDISIMU MODE (EDITING AND SIMULATION)

Working in the editing window.

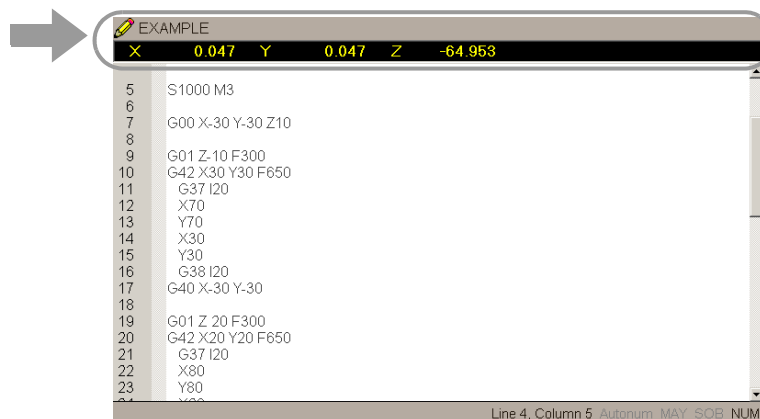
8.4.8 Customizing the editor (syntax highlighting).

This option may be used to customize the color of the elements (functions, comments, etc.) that make up the program. After defining the new look, press [ENTER] to accept the changes or [ESC] to ignore them. In large programs (more than 200 kB), the editor cancels the syntax coloring.



8.4.9 TEACH-IN

The "Teach-in" softkey is used to activate or deactivate the TEACH-IN mode; in this mode, the axes may be moved manually and their position may be assigned to a block. When this mode is active, the top of the editing window shows the position of the axes defined as "visible" for the TEACH-IN mode. See ["8.4.6 Customizing the editor \(general options\)."](#) on page 165.



When TEACH-IN mode is active, it is possible to keep editing the coordinates of the axes directly from the keyboard or they may be assigned the current position of the machine axes. Both editing methods may be used indistinctly, even while defining a block. To define the coordinates of one or several axes using TEACH-IN:

- 1 Move the axes to the desired position using the JOG keys, the handwheels or the MDI mode.
- 2 In the part-program, edit the name of the axes whose position is to be defined or not select any axis if you wish to define the position of all of them.
- 3 Press the [RECALL] key.

If an axis of the channel has been edited, the CNC assigns the current position of that axis as the program coordinate. The axis is displayed in the TEACH-IN window.

If only the block number or an empty line has been edited, it inserts a block with the position of all the axes of the channel defined as "selected" for the TEACH-IN mode.

If a character has been edited other than the axis name or the block number, it does not insert anything and the cursor stays in the same place.

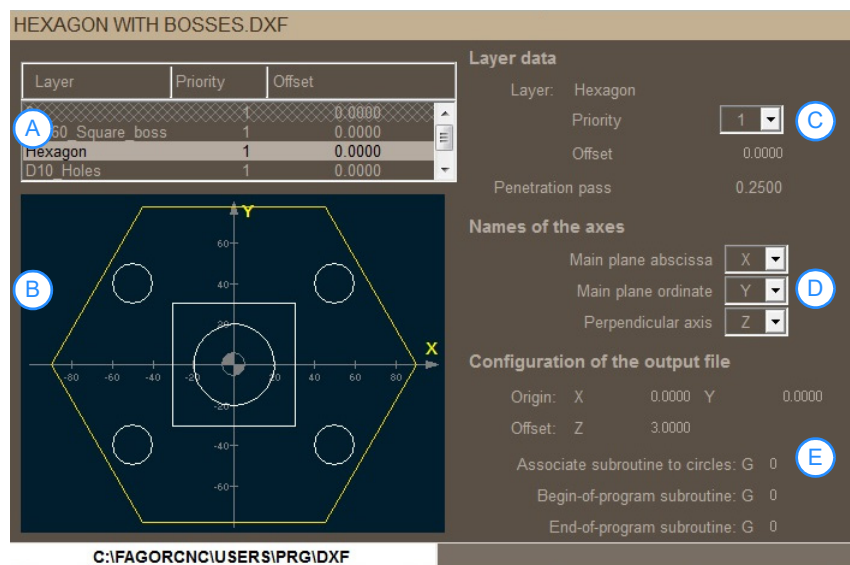
8.4.10 Import DXF files

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 drawing layers.
- B File preview. The preview 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 with the cursor.
- D Name of the axes.
- E Output file configuration.

List of drawing layers.

The dxf file can be divided into layers, where each layer contains different part sections (contours, dimensions, etc). The table indicates the following for each of the layers:

- Name of the layer, as defined in the dxf file.
- Layer priority or order.
- Offset (position) of the layer on the perpendicular axis.
- Dwelling of the layer; enabled or disabled (the layer appears shaded). When a layer is disabled, the CNC does not insert it into the program (for example, coordinate layers or auxiliary lines). To disable a layer, select it with the cursor and press the "Disable layer" softkey. To re-enable the layer, press the same softkey.

Data of the layer selected with the cursor.

The following data must be defined for each layer.

- 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. Priority 1 layers will be the first to be executed, then priority 2 layers and so on. There can be several layers that have the same priority.

- Offset (position) of the layer on the perpendicular axis. The offset (height) allows each layer to be executed for the desired Z coordinate (or that of the relevant perpendicular axis).
- Depth of pass.

Name of the axes.

The work plane and the perpendicular axis must be defined before importing the file into the part-program (abscissa and ordinate) . To import the file into the profile editor, first define the perpendicular axis; the work plane will be the one selected by the editor.

Output file configuration.

- Origen X / Origen Y. 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.
- Subroutine associated with circles. If a subroutine has been defined, 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 (for example, to convert the circles into drill holes). If no subroutine has been defined, the dxf converter converts the circles to segments with G2/G3.
- Subroutine to execute at the beginning of the program.
- Subroutine to execute at the end of the program.

Valid subroutines are G180-189, G380-G399 and G500-G599.

Drawing design

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 if the CNC units are millimeters or 5 decimals if the units are in inches. A greater resolution increases the size of the DXF file unnecessarily because the CNC ignores the excess of resolution.

8.

EDISIMU MODE (EDITING AND SIMULATION)

Working in the editing window.



CNCelite
8058 8060
8065 8070

REF: 2508

8.4.11 Import files from the 8055 (PIM and PIT).

The CNC interprets the PIM and PIT formats as programs in the language of the 8055. Files with PIM extensions contain the lathe and PIT programs. Importing these file types enables programs to be converted from the 8055 language, so that the program can be edited in the CNC, including the editor cycles.

In the horizontal softkeys menu, select "File" > "Import file". The editor displays the list of programs that can be imported. Select the desired file from the list and press [ENTER]. The editor imports and translates the selected program.

The options of translator 8055 can be found in the general editor options. See ["ISO Translator. Configuring the 8055 translator."](#) on page 167.

Programs with editor cycles.

On importing a file, the CNC translates the conversational cycles and displays them in their own language. These cycles can be opened and modified in the editor by pressing the [RECALL] key. After editing, the CNC can insert the cycle inside another program.

8.

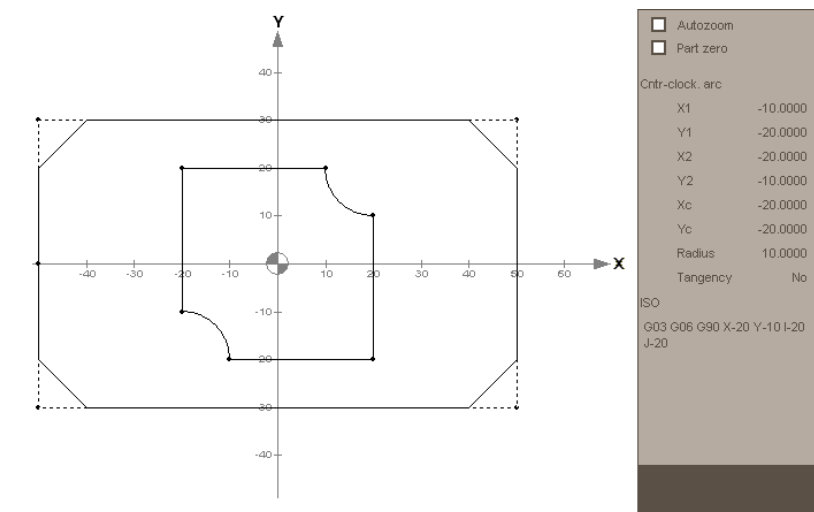
EDISIMU MODE (EDITING AND SIMULATION)

Working in the editing window.

8.4.12 Profile editor.

The profile editor may be accessed from the softkey menu to edit a new one, or selecting a profile of the program and pressing the [RECALL] key. The profile editor shows its options on the softkey menu. See chapter "11 Profile editor".

To return to editing the program, press "End". The softkey menu of the program editor shows the "insert profile" softkey to insert the defined profile into the program. The block for the defined profile is inserted after the block where the cursor is.



8.

EDISIMU MODE (EDITING AND SIMULATION)

Working in the editing window.

8.4.13 Canned cycle editor.

The cycle editor may be accessed from the softkey menu to edit a new one, or selecting a cycle of the program and pressing the [RECALL] key. The softkey menu of the cycle editor shows the available canned cycles.



To return to the program editor, press the [BACK] key. The softkey menu of the program editor shows the "insert cycle" softkey to insert the defined cycle into the program. The block for the defined cycle is inserted after the block where the cursor is.



Canned cycles have a specific manual both for milling and turning. Refer to the documentation included in the CD-Rom that comes with the product for further detail.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

8.4.14 Inclined planes (-M- model)

The inclined-plane editor may be accessed from the softkey menu to edit a new one, or selecting an inclined plane of the program and pressing the [RECALL] key.



The softkey menu of the inclined-plane editor shows the various ways to program inclined planes. To return to the main menu, press the [BACK] key.

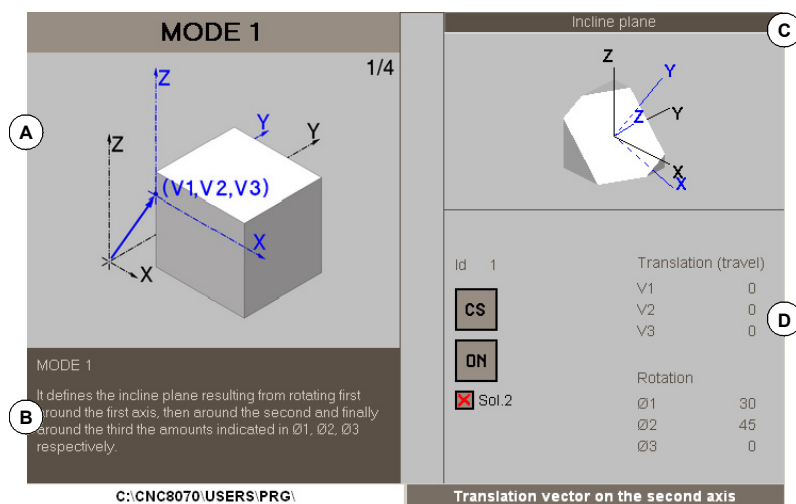
Inclined planes programmed directly in ISO code using the instructions #CS and #ACS can also be restored using the [RECALL] key. This way, it is possible to check the programmed parameters and the resulting inclined plane.

Once the inclined plane has been defined, press [ESC] to quit the editor. The softkey menu will show the "insert inclined plane" softkey to insert the defined inclined plane into the program. The block for the defined inclined plane is inserted after the block where the cursor is.



For further information on how to program inclined planes, instructions #CS and #ACS, refer to the programming manual.

Description of the help screen for inclined planes.



A Help graphics. Sequence of drawings showing each step for defining the inclined plane. When the focus is on a programmable parameter, the sequence will stop and show the relevant explanatory drawing.

B Short explanation on how to program the selected inclined plane.

C Drawing of the resulting inclined plane. The drawing is updated (refreshed) as the user programs the different parameters that affect the geometry of the inclined plane. The drawing shows the following elements.

- The inclined plane in white.
- The reference system of the inclined plane, X' Y' Z', in blue.
- When there is a translation vector (V1, V2, V3), the drawing will show a reference system at the lower left side of the drawing. The distance between this reference system and the cube is not proportional; the graphics is merely approximate.

When the drawing is selected with the focus, the cube may be rotated using the [◀][▶][▲][▼] keys or using the wheel of the mouse.

D Parameters to define the inclined plane.

8.4.15 Geometric-help editor.

The geometric-help editor may be accessed from the softkey menu to edit a new one, or selecting one of the program helps of the program and pressing the [RECALL] key. Geometric helps programmed directly in ISO code can also be recalled by pressing the [RECALL] key to check the programmed parameters.



The softkey menu of the geometric-help editor shows the various programmable helps. To return to the main menu, press the [BACK] key.

Once the help has been defined, press [ESC] to quit the editor. The softkey menu will show the "insert geometric help" softkey to insert the help into the program. The block for the defined geometric help is inserted after the block where the cursor is.



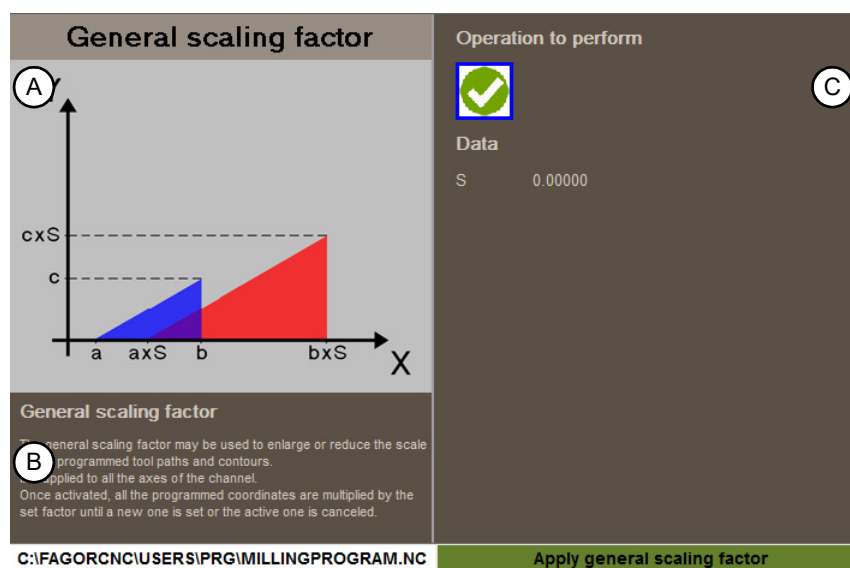
For further information on how to program geometric help, refer to the programming manual.

8.

EDISIMU MODE (EDITING AND SIMULATION)

Working in the editing window.

Description of the screen and softkey menu.



A Help graphics.

B Short explanation on how to program the geometric help.

C Parameters to define the machining operation.

Horizontal softkey menu.

Softkey.	Meaning.
	Scaling factor. Mirror image. Rotate the coordinate system.
	Corner chamfering and rounding. Tangential entry and exit. Square, semi-rounded or rounded corner.
	Plane change.
	Zero offset and preset.

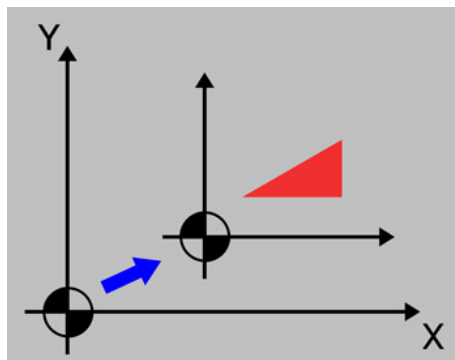
FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Description of the geometric help.

Zero offset and preset.

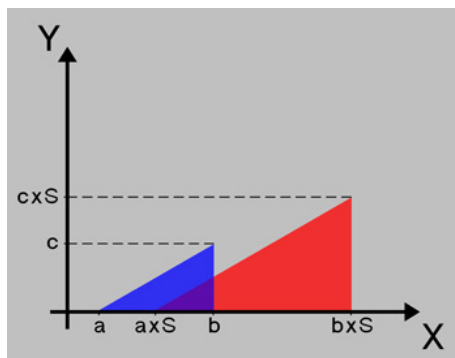
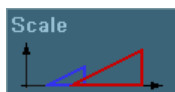


This softkey may be used to select the following options:

- Machine zero (home). Cancel any zero offset and assume machine zero as part zero.
- Absolute zero offset. Set, activate or define + activate absolute zero offsets (G159).
- Incremental zero offset: Set, activate or define + activate incremental zero offsets (G158).
- Coordinate preset: Edit the preset value for the active axes. If you do not wish to preset any of them, leave its field empty (blank).

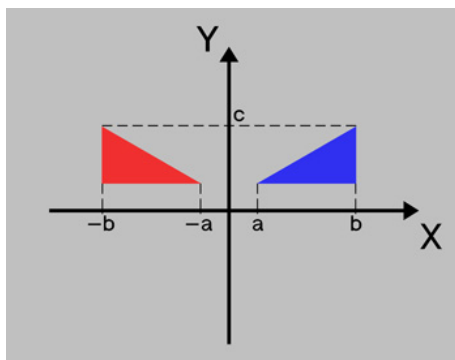
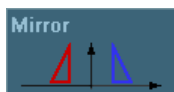
The cycle internally generates an ISO block with function G53, G159, G158 or G92.

Scaling factor.



This softkey may be used to cancel the current scaling factor and activate a new one. In the latter case, the cycle shows the necessary data to define the scaling factor.

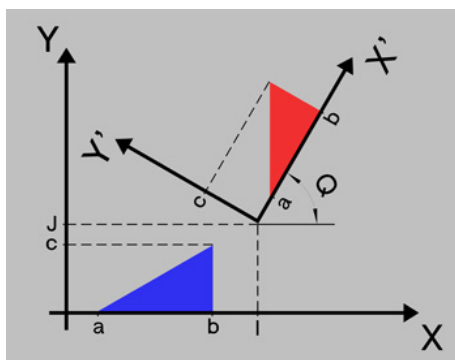
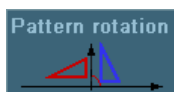
The cycle internally generates an ISO block with function G72.

Mirror image.

This softkey may be used to select the following options:

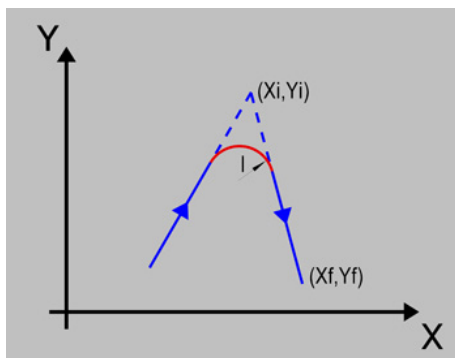
- Select the desired action, cancel the active mirror image, define a new one canceling the previous ones or define a new one and add it to the one currently active.
- Select the axes on which mirror image is applied. For a new mirror image, you may select between the X, Y or X-Y axes. For an additive mirror image, you may select between the X and Y axes. This icon is not shown when selecting mirror image cancellation.

The cycle internally generates an ISO block that contains a combination of functions G10, G11 and G12.

Rotate the coordinate system.

This softkey may be used to cancel the active coordinate (pattern) rotation, define a new one canceling the previous one or define a new one and add it to the currently active one.. When defining or activating a coordinate (pattern) rotation, the cycle shows the necessary data to define the coordinate (pattern) rotation..

The cycle internally generates up to two ISO blocks with function G73.

Corner chamfering and rounding.

This softkey may be used to define corner rounding or chamfering . In either case, the cycle shows the necessary data to define the machining operation.

The cycle internally generates an ISO block with function G36 or G39.

8.

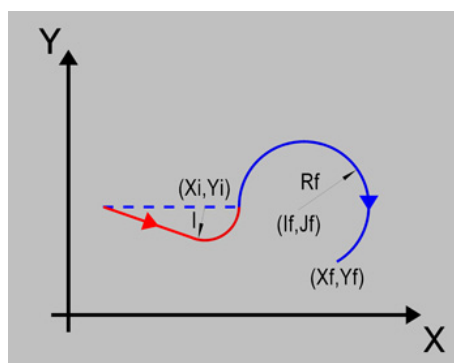
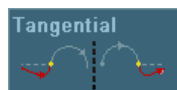
EDISIMU MODE (EDITING AND SIMULATION)

Working in the editing window.

FAGOR
AUTOMATION

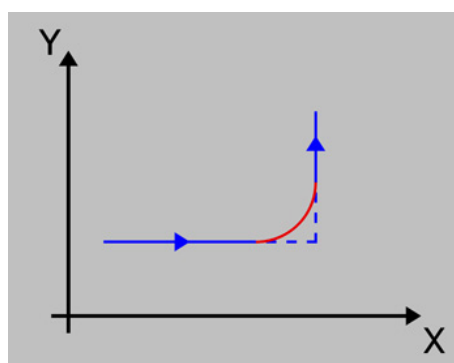
CNCelite
8058 8060
8065 8070

REF: 2508

Tangential entry and exit.

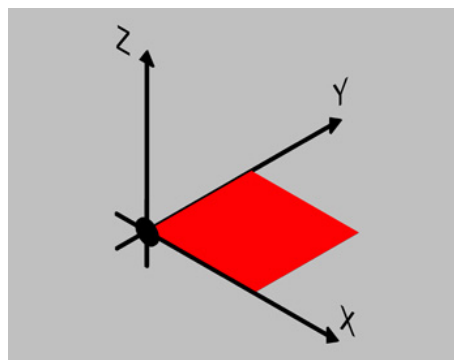
This softkey may be used to define a tangential entry or exit. In either case, the cycle shows the necessary data to define the machining operation.

The cycle internally generates an ISO block with function G37 or G38.

Square, semi-rounded or rounded corner.

This softkey may be used to define the type of corner, round, semi-round or sharp. If necessary, the cycle shows the necessary data to define the type of corner.

The cycle internally generates an ISO block with function G05, G50 or G07.

Plane change.

This softkey may be used to define the work plane, be it a main one (G17, G18 or G19) or one defined by two axes (G20).

The cycle internally generates an ISO block with function G17, G18, G19 or G20.

8.

EDISIMU MODE (EDITING AND SIMULATION)

Working in the editing window.

FAGOR
AUTOMATION

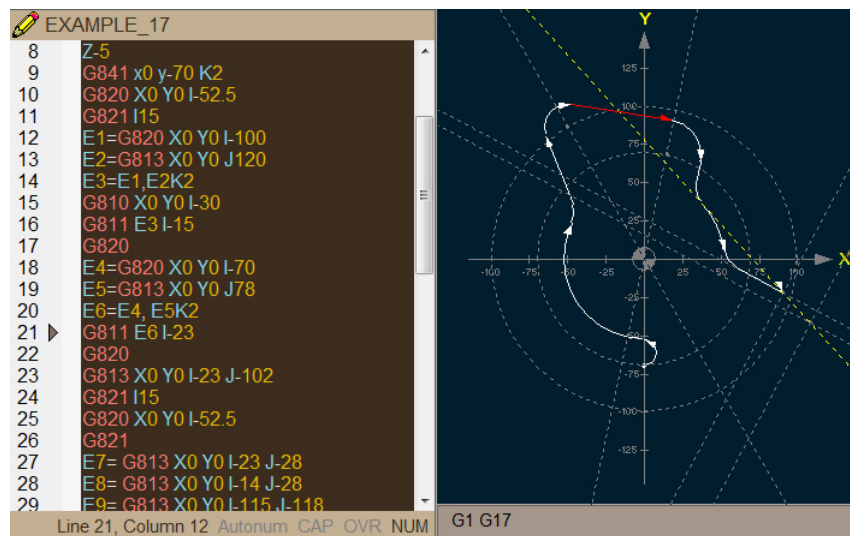
CNCelite
8058 8060
8065 8070

REF: 2508

8.5 Graphic assistance for the program editor.

While editing the program, this window shows the 2D geometry of the program or of the profile being edited. The window shows the geometry from the first block of the program or of the profile up to the block selected with the cursor.

This window only shows 2D geometry, that of the work plane, ignoring coordinate transformations, RTCP, scaling factors, mirror images, etc. To take these elements into consideration, run a simulation of the program. When simulating a program, the CNC replaces this window with the graphics window.



During execution, the window displays the elements as follows:

- The tool path generated by the block selected with the cursor is shown as a red solid line.
- The resolved elements are shown in a gray dashed line.
- The partially resolved blocks (such as circles or straight lines), if any, are shown as a yellow dashed line.
- The canned cycles are shown as a circle at the starting point of the cycle. The radius of the circle will be the same as the radius of the programmed tool.
- If the cursor is inside a profile or ProGTL3 macro, the window shows the profile defined between G841-G842, and G840 or the line selected with the cursor. If the cursor is not in a profile that begins with G841-G842 or with #PROFILE BEGIN, the window shows the entire previous geometry.

8.

EDISIMU MODE (EDITING AND SIMULATION)

Graphic assistance for the program editor.

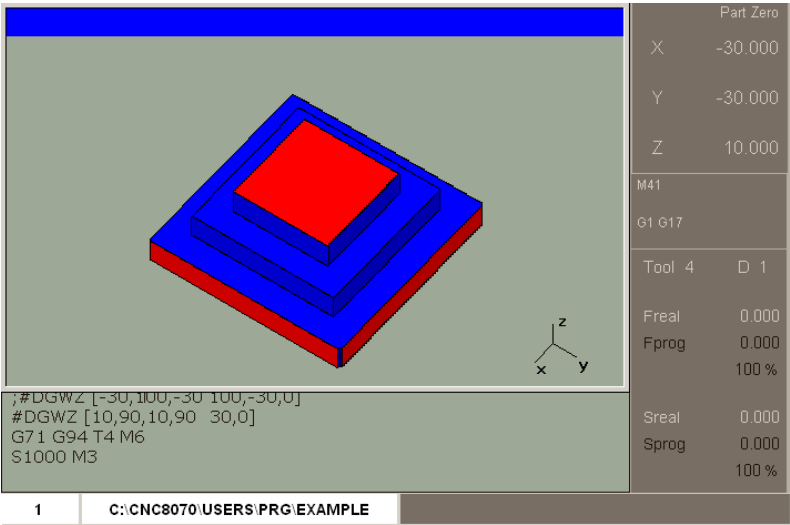
FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

8.6 Graphics window

This window is used to show a graphic representation of the program being simulated and take measurements over the drawing. The CNC has different types of graphics.



8.

EDISIMU MODE (EDITING AND SIMULATION)

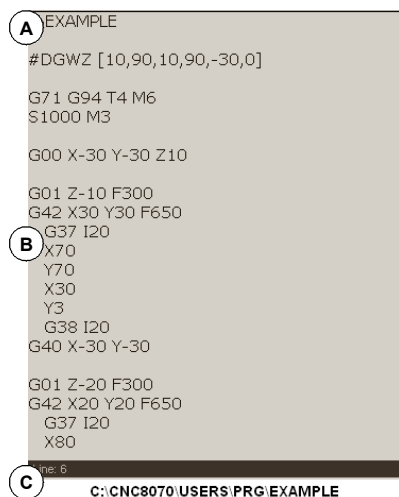
Graphics window

8.7 Program window

This window shows the contents of the program selected for simulation and allows selecting the first and last blocks of the simulation. When not selected, the simulation will begin at the first block of the block and will end after executing one of the end-of-program functions "M02" or "M30". While simulating, the window cursor will show the block being simulated.

Description of the program window

The program window shows the following information:



A Title bar.

Name of the program selected for simulation.

B Program blocks selected for simulation.

While simulating, the cursor will indicate the block being simulated. The "active subroutines" option being active, the window displays information related to the execution of subroutines, canned cycles, repetition blocks and loops.

C Program line.

Line of the program where the cursor is.

Horizontal softkey menu.

When selecting the program window, the horizontal softkey menu will show the options related to this window.

Softkey.	Description.
Block selection.	Select the first and last blocks of the execution.
Display.	Toggle between the various screens of the EDISIMU mode and display information related to the status of the subroutines, canned cycles, repetition blocks and loops.
Exblk.	Execute only the block selected with the cursor.
Graphics.	Show the graphics screen.

8.

EDISIMU MODE (EDITING AND SIMULATION)

Program window

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

8.8 Working in the program window.

8.8.1 Select the first and last blocks of the execution.



After selecting the "block selection" softkey, the horizontal softkey menu shows the following options. To return to the main menu, press the [BACK] key.

Softkey.	Description.
Set beginning	First block for execution or for manual block search.
Stop condition.	Set the condition to end the execution or the manual block search.
Find text.	Find text

Set beginning.

This option sets as starting block for simulation the block selected with the cursor. When not setting the first block, the simulation will begin at the first block of the program.

The first block may be selected using the cursor or the "Find text" option of the softkey menu. The selected block stays active until canceled (selecting another block or selecting the same one again) or simulating the program.

Stop condition.

This option sets the block where the simulation of the program or subroutine will be interrupted. After executing that block, the execution may be resumed with the [START] icon or canceled with the [RESET] icon. If no last block is established, the simulation of the program will end after executing one of the end-of-program functions "M02" or "M30".

Select subroutine.

This option selects the stop condition in a global subroutine which has been called upon from the program. When selecting this option, the CNC shows a list of the available subroutines. After selecting the desired subroutine, it will appear in the program window.

Set stop block.

This option sets as simulation interruption block the block selected with the cursor. If no last block is established, the simulation of the program will end after executing one of the end-of-program functions "M02" or "M30".

The last block may be selected using the cursor or the "Find text" option of the softkey menu. The selected block stays active until canceled (selecting another block or selecting the same one again) or executing the program.

Number of times.

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

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

Find text.

This option shows a dialog box for placing the cursor on a particular line of the program or search for text or a character string in the program.

Go to line.

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

Find text

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

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

8.

EDISIMU MODE (EDITING AND SIMULATION)

Working in the program window.



CNCelite
8058 8060
8065 8070

REF: 2508

8.8.2 Simulate program blocks separately.

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

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

8.

EDISIMU MODE (EDITING AND SIMULATION)

Working in the program window.

8.8.3 Display the status of the program or of the active subroutines.

From the horizontal softkey menu, it is possible to toggle between the display of program blocks and the display of information related to the status of the subroutines, canned cycles, repetition blocks and loops.

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

Description of the information displayed.

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

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

The window shows the following information.

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

8.

EDISIMU MODE (EDITING AND SIMULATION)

Working in the program window.

8.9 Statistics window

This window offers an estimate the total program execution time at 100% of the programmed feedrate and the machining time for each tool. For this execution time estimate, the CNC analyzes the following.

- The machining and positioning time for each tool used in the program.
- The number of "M" functions that are executed.
- The number of tool changes performed.

Description of the statistics window

This statistics window shows the following information:

Total Execution Time (s)		173.321	
Number of M functions:		6	
Number of tool changes:		3	
Positioning and machining in seconds			
TOOL		Positioning	Machining
0		0.000	0.000
1		1.567	18.572
2		0.600	54.015
3		0.840	97.677

A General information.

It shows a time estimate of program execution at 100% of the programmed feedrate, the number of "M" functions executed and the number of tool changes made.

B Machining time for each tool.

It shows a list of the tools used in the program indicating the machining time for each tool and the total positioning time.

C "G" functions active during simulation.

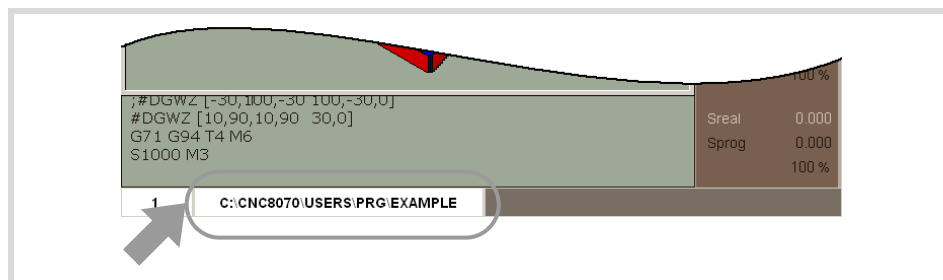
8.

EDISIMU MODE (EDITING AND SIMULATION)

Statistics window

8.9.1 Time estimates

The statistics window shows an estimate of the execution of the program selected in the editing window, and whose name appears at the bottom center of the screen.



The process to estimate time is the following:

- 1 Use the program window to select the first and last block for the execution time estimate. If not selected, the execution time estimate will be done from the first block of the program to the execution of one of the end-of-program functions "M02" or "M30".
- 2 Select the desired simulation options.
- 3 From the vertical softkey menu, start calculating the execution time estimate.



8.

EDISIMU MODE (EDITING AND SIMULATION)
Statistics window

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

8.

EDISIMU MODE (EDITING AND SIMULATION)

Statistics window



CNCelite
8058 8060
8065 8070

REF: 2508

FMC (FAGOR MACHINING CALCULATOR).

9

The FMC application consists of a database of materials to be machined and machining operations (milling and turning) and an interface to choose suitable cutting conditions, particularly the feedrate of the axes (F) and the turning speed of the spindle (S). After selecting a particular material and operation, the FMC offers a range of recommended cutting conditions. The user must choose the desired cutting conditions and a tool among those offered. Using this data, the application calculates the optimal F and S values for the machining.

The user can insert the obtained result in a part-program or canned cycle. The FMC main window allows for the selection of the data to be inserted.

- In the part-program, the FMC enters one block for each selected data. Each block introduced by the FMC is accompanied by a comment indicating the material, operation and associated tool.
- In a canned cycle, the FMC completes the associated data for the feedrate (F), speed (S) and tool number (T).

Example of the blocks introduced by the FMC for the part-program.

```
; This is a comment!!!  
T2 ; ### FMC # Stainless steel # Rectangular Boss # T2  
M6 ; ### FMC # Stainless steel # Rectangular Boss # T2  
G94 F95.49 ; ### FMC # Stainless steel # Rectangular Boss # T2  
G97 S1909.8 ; ### FMC # Stainless steel # Rectangular Boss # T2  
#DMC ON [PWRSP = 0.38, FZMIN = 0.0200, FZMAX = 0.0600] ; ### FMC # Stainless  
steel # Rectangular Boss # T2
```

To take full advantage of the FMC's capabilities, the tools to be used must be fully defined on the tool table - type of tool, diameter and number of teeth.

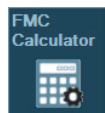
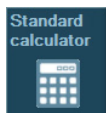
Accessing the FMC calculator.



- In any CNC mode or in the cycle editor, press the [CALC] key (or [CTRL] [K] keys). By calling the calculator in this manner, the FMC allows the calculated values to be inserted into the cycle or in the ISO editor.



- In the canned cycle editor, place the icon on the F or S field and press [RECALL]. By calling the calculator in this manner, the FMC allows the calculated values to be inserted into the cycle.
- The calculator can also be accessed from the task window ([CTRL] [A] keys), the "Accelerators" tab or "calculator" option.



The same key can access both the arithmetic calculator and the FMC calculator. The arithmetic calculator will have a softkey access to the FMC calculator and vice versa.



CNCelite
8058 8060
8065 8070

REF: 2508

Exit the FMC calculator.

- Press [ESC] from any calculator page. The FMC does not modify the program or the cycle.
- Press the “insert” softkey. If the FMC calculator was accessed from the program editor, the calculator inserts the selected blocks into the program. If the FMC calculator was accessed from the cycle editor ([RECALL] key), the calculator completes the feedrate (F), speed (S) and tool number (T) fields for the current operation (roughing, semi-finishing or finishing). If the FMC calculator was accessed from the cycle editor, ([CALC], [CTRL][K] or [CTRL][A] keys), the calculator will not change the cycle data.



9.

FMC (FAGOR MACHINING CALCULATOR).



9.1 FMC calculator.

This page allows the cutting conditions to be calculated for the chosen material and operation and to insert the result into the edited program. The FMC only inserts the selected options (green check mark to the left of the data).

FMC

Material **(A)** Mould and Matrix Operation **(B)** Surface Milling

	Min	Max	Selected value
Cutting speed (m/min)	150.0000	250.0000	165.0000
Feed per tooth (mm/z) (C)	0.1500	0.3000	0.2000
Depth (mm)	3.0000	6.0000	4.0000
Radial wipe (%)	0.00	100.00	60.00

☒ Comment **(D)**

☒ Tool **(E)** T1 Diameter **(G)** 63.0000 mm

☒ M6 **(F)** Number of teeth 1 z

Calculated cutting conditions **(H)**

☒ F 166.72 mm/min

☒ S 833.6 rpm

Cutting power **(I)** 0.00 %

☒ DMC **(J)**

P30 Quality. Coated Hard Metal(TiN+TiC)

- (A) Selected material (the list shows the materials determined from the materials table).
- (B) Selected operation (the list shows the operations determined from the operations table).
- (C) User defined cutting conditions for the selected material and operation; minimum, maximum and selected values for the calculation. For Fagor materials and operations, the minimum and maximum values cannot be modified. When the tool is a "thread" type (external thread or blade), the feedrate per tooth must define the pitch of the thread.
- (D) Comment.
- (E) Tool. Select a tool from the list or write the tool number directly.
- (F) Function M6 (tool change). This field is only visible when the tool field is selected
- (G) Tool information.
- Milling tools; "Diameter" and "Number of teeth" (non-editable values).
 - Turning tools; "Part diameter" (editable value).
 - Other types of tools; "Diameter" and "Number of teeth" (editable values).
- (H) Calculated cutting conditions; feedrate and velocity. The units of the cutting conditions are given in the active units (millimeters or inches). The constant surface speed (m/min or ft/min) is only available on the lathe model or dual-purpose machine.
- (I) Calculated cutting conditions; cutting power and instruction to activate the DMC.
- (J) Coating of the tool selected based on the recommended chosen values.



- The DMC function is subject to the corresponding software option. If this software option is not active, the FMC calculator will not calculate the DMC function.
- The DMC is only available for the Fagor digital servo master spindle; if the master spindle is analog, the FMC calculator cannot calculate the DMC function. The spindle must be enabled for the DMC in the machine parameter DMCPDL.
- DMC is only available for milling operations with "Milling" and "Surface milling" operations. For all other operations, the FMC calculator will not calculate the DMC function.

Vertical softkey menu.

Softkey.	Meaning.
	Insert. This softkey (or the [INS] key) inserts the result obtained from the FMC calculator into the program being edited.
	Copy. This softkey copies the result obtained from the FMC calculator to the clipboard. The contents of the clipboard may be pasted into the ISO program editor. To paste the contents of the clipboard into a program, press [CTRL] + [V].

9.

FMC (FAGOR MACHINING CALCULATOR).

FMC calculator.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

9.2 Materials table.

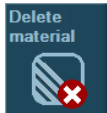
This table lists the materials defined by Fagor, the OEM and the user. The OEM and the user may add or delete their own materials, but they cannot modify or delete those defined by Fagor (with the exception of the KC value).

Materials table						
▲	Description	Germany W	UNE	AISI	KC (N/mm2)	Hidden
🔒	Mould and Matrix	1.2343		H11	1950.0000	☐
🔒	Mould and Matrix 2	1.2080	F-5210		1950.0000	☐
🔒	Carbon Steel	1.1191	F-1140	1045	1700.0000	☐
🔒	Quenching and Tempering	1.7220	F-1250	4135	1700.0000	☐
🔒	Hardening Steel	1.7016	F-1510	5015	1700.0000	☐
🔒	Aluminium		EN AC-AISI12		600.0000	☐
🔒	Aluminium 2			7075	600.0000	☐
(A)	(B) Bronze/Brass (C)		(D) CuZn20Pb2	(E)	(F) 500	(G)
🔒	Stainless Steel	1.4401	F-3543	316	2000.0000	☐
🔒	Stainless Steel 2	1.4021		420	1800.0000	☐
P30 Quality. Coated Hard Metal(TiN+TiC)						

- (A) The materials defined by Fagor are labeled with a lock icon and cannot be modified. OEM and user materials do not have any icon.
- (B) Material description.
- (C) Numbering system for steel, according to EN 10020 (WNR standard numbering).
- (D) Classification of materials according to the UNE standard.
- (E) Classification of materials according to the AISI standard.
- (F) Specific cutting force.
- (G) Hide the material in the FMC window.

Double click on either heading of the first two columns to sort the table by that criterion. The green row indicates the material selected in the FMC window.

Vertical softkey menu.

Softkey.	Meaning.
	Add material. This softkey adds a material to the list.
	Delete material. This softkey deletes a material from the list. Only materials defined by the OEM and the user can be deleted.
	Save table. The table must be saved each time the OEM or the user add a material, modify any value or change the "Hidden" status.

9.

FMC (FAGOR MACHINING CALCULATOR).

Materials table.

FAGOR
AUTOMATION



CNCelite
8058 8060
8065 8070

REF: 2508

9.3 Operations table.

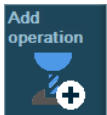
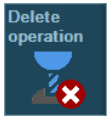
This table lists the operations defined by Fagor, the OEM and the user. The OEM and the user may add or delete their own operations, but they cannot modify or delete those defined by Fagor.

Operations table			
▲	Description	Tool type	Hidden
🔒	Surface Milling	Surface milling	☒
🔒	Rectangular Boss	Milling	☐
🔒	Drilling Canned Cycle	Drilling	☐
🔒	Boring Canned Cycle	Boring	☐
🔒	Rectangular Pocket Canned Cycle	Milling	☐
🔒	Straight Section Turning Canned Cycle	Turning	☐
🔒	Curved Section Turning Canned Cycle	Turning	☐
(A)	(B) Axle Grooving Canned Cycle	Grooving (C)	(D)
🔒	Pattern-repeat Canned Cycle	Turning	☐
🔒	Cylindrical Syde Slot Canned Cycle	Milling	☐
P30 Quality. Coated Hard Metal(TiN+TiC)			

(A) The operations defined by Fagor are labeled with a lock icon and cannot be modified. OEM and user operations do not have any icon.
 (B) Operation description.
 (C) Tool type. These are the default types from the tool table.
 (D) Hide the operation in the FMC window.

Double click on either heading of the first two columns to sort the table by that criterion. The green row indicates the operation selected in the FMC window.

Vertical softkey menu.

Softkey.	Meaning.
	Add material. This softkey adds an operation to the list.
	Delete operation. This softkey deletes an operation from the list. Only operations defined by the OEM and the user can be deleted.
	Save table. The table must be saved each time the OEM or the user add an operation, modify any value or change the "Hidden" status.

9.

FMC (FAGOR MACHINING CALCULATOR).

Operations table.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

9.4 Working with the FMC.

9.

FMC (FAGOR MACHINING CALCULATOR).

Working with the FMC.

To calculate the proper cutting conditions using the FMC, follow these steps.

- 1 Choose a material and an operation and, if necessary, determine the material and the operation from the corresponding screen. See ["9.2 Materials table."](#) on page 194. See ["9.3 Operations table."](#) on page 195.
- 2 Choose a tool from those listed by the FMC, whose type matches those on the tool table for the selected operation. When choosing the tool, be aware that it must be suitable for machining the selected material.



DANGER

If a tool is chosen that is not suitable for the selected material, there may be a risk of tool breakage or "ejection risk".

- 3 The FMC calculates the optimal cutting conditions (F and S) for the chosen material, operation and tool. Select the remaining data to insert into the program (tool, comment, etc.).
- 4 Press the "insert" softkey to insert blocks into the program.

9.5 Backup/Restore. Data backup copy.



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

The utilities mode may be used to make a backup of the CNC configuration (OEM and user data) to be restored later one if necessary. When selecting the data to be included in a backup or restore, the database containing the materials and operations added by the manufacturer or user is in the Tables section under "User Data". See ["19.6 Data safety backup. Backup - Restore"](#) on page 344.

FCAS (FAGOR COLLISION AVOIDANCE SYSTEM).

10

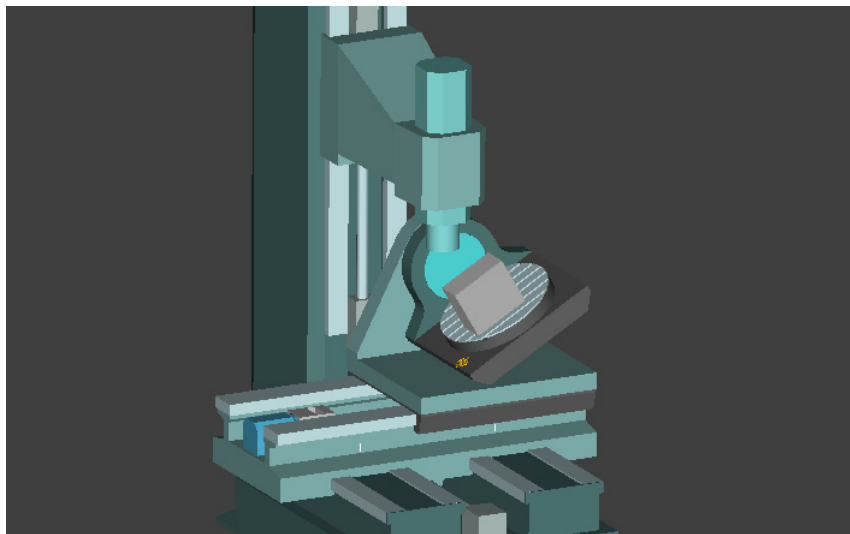


FCAS is only available on single-channel machines that use HD graphics with a model configuration of the machine adjusted to reality (xca file). Default xca files supplied by Fagor are generic, which means they are not suitable for the FCAS option.

The FCAS (Fagor Collision Avoidance System) option monitors automatically, in MDI/MDA, manually and tool inspection movements in real time, so as to avoid collisions between the tool and the machine. Otherwise, the FCAS option does not supervise the movements during the machine home search. When the FCAS option detects the likelihood of a collision, it halts any movement within a predefined safety margin defined by the machine configuration.

The FCAS option requires that the HD graphics to be active and that there is a defined a model configuration of the machine adjusted to reality (.xca file), which includes all its moving parts. The FCAS option can detect collisions with all these drawn parts. After defining the machine configuration, an extra safety region must be drawn for all parts that serves as a safety zone for braking.

To detect collisions of the turning tools, the image of the tool with its tool holder must be defined. These images (stl format) are defined in the tool table ("Toolholders" field).



Machine configuration (xca file).

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

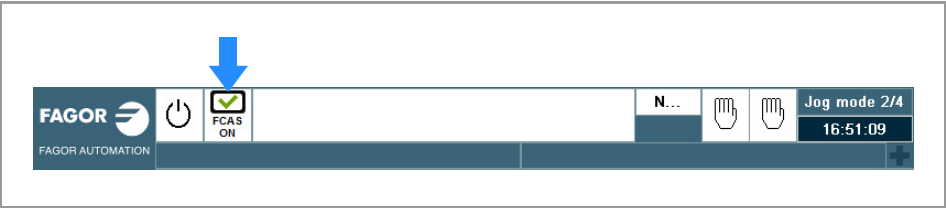
REF: 2508

10.1 FCAS status.



The FCAS status icons are only available when the SOFT FCAS software option is enabled. If it is not enabled, the CNC will not display any icon.

The CNC will display the FCAS status on the top bar.



Icon.	Meaning.
	Active feature.
	Feature is disabled; the machine configuration is not loaded (xca file), the HD graphics are not selected or the PLC has disabled this feature.
 	Blinking icon. • The axes are close to the collision zone. • CPU overload during collision detection calculations. The variable VGCOLLISIONPERF indicates more details behind the cause of the blinking.

10.

FCAS (FAGOR COLLISION AVOIDANCE SYSTEM).

FCAS status.



CNCelite
8058 8060
8065 8070

REF: 2508

10.2 FCAS operation.



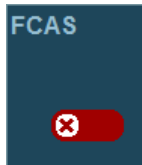
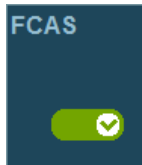
Although the FCAS option reduces the risk of collisions, it does not guarantee that these are completely avoided. Therefore, it is the responsibility of the user to ensure that there are no obstacles that could cause a collision during the machining process, even when the FCAS option is active.

- The FCAS option does not monitor collisions for the part.
- The FCAS option only monitors collisions with the parts of the machine that the OEM has correctly included (position, dimensions, etc.) in the machine configuration. The parts that have not been included in the configuration are not supervised.

The collision control is activated/deactivated from the PLC. The user may do so using the keypad if the OEM has been provided a button or key for this purpose. Additionally, the HD graphics must be selected and the machine configuration must be loaded (xca file). Collision control is not active if the STD graphics (Fagor standard graphics) are selected.

Automatic mode.

For executing a program, the FCAS option can be activated or deactivated from the softkey menu. If the CNC detects that the tool is going to collide, it halts the execution of the program and displays a collision limit reached error.

Softkey.	Meaning.
	FCAS not active.
	FCAS active.

MDI/MDA mode.

If the CNC detects that the tool is going to collide, it halts the execution of the program and displays a collision limit reached error.

Manual mode.

The CNC will start to brake if the direction of the tool approaches a collision zone. The movement will automatically stop when it reaches a collision point and "Collision limit reached" message will be shown. When the collision zone is reached, with the axes idling, the axes can only be moved again in the opposite direction in which they were moving toward the collision zone until leaving the near-collision zone.



During braking, or if a movement in the direction of a collision is insisted on, the axis may go beyond the safety zone.

EDISIMU mode.

For a program simulation, the FCAS option can be activated or deactivated from the softkey menu. If the CNC detects that the tool is going to collide, it halts the program simulation and displays a collision limit reached error, indicating the program line and the block forcing the collision.

10.

FCAS (FAGOR COLLISION AVOIDANCE SYSTEM).

FCAS operation.

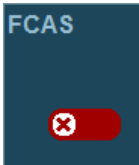
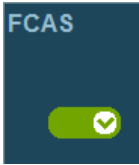
FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

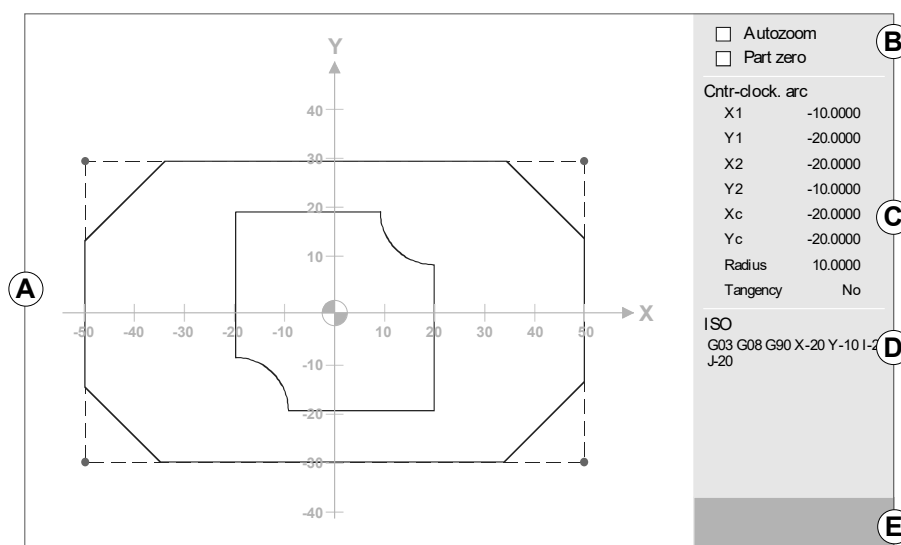
10.**FCAS (FAGOR COLLISION AVOIDANCE SYSTEM).**

FCAS operation.

Softkey.	Meaning.
	Deactivating the FCAS.
	Activating the FCAS.

11.1 Interface description.

The profile editor is used to edit quickly and easily simple rectangular, circular profiles and any type of profile consisting of straight and curved sections. As the profile data is entered, the editor shows a graphic representation of the profile.



- A** Graphic area. Graphic representation of the profile being drawn, axes coordinated with autoscale and name of the axes that make up the plane. The name of the axis indicates the positive direction of the axis.
- B** Status of the autozoom and part zero options, regarding the display of the profile at the editor.
- C** Data entry area.
- D** Translation (conversion) of the selected profile or part of it into ISO code
- E** Area used to enter the values of the corners or the ISO coded text to be added to the element.

Keyboard shortcuts.

These options will not be available when a menu for editing data or selecting items is active at the editor.

Keys.	Meaning.
[←] [→] [↑] [↓]	Move the graphics.
[+] [-]	Enlarge or reduce the display area.
[/]	Keeps the part zero visible at all times.
[*]	Activate autozoom.

Softkey menu.

The options that may be selected from the softkey menu make it possible to edit profiles, modify edited profiles, select the zoom, the work plane, undo the last change and end the editing session. While editing or modifying the profile, the softkey menu offers the option to undo the last operation. Likewise, it offers the option to save the profile at any time.

Softkey.	Meaning.
Edit	Edit a new profile, enlarge an existing profile or import a profile saved in DXF format. See "11.2 Define a new profile, enlarge an existing one or import one from a file." on page 204.
Modify	Modify, insert or delete elements of a profile. See "11.3 Modify a profile and insert corners" on page 213.
Displayed area	Modify the zoom of the graphics area. See "11.4 Configuring the profile editor. Displayed area." on page 215.
Plane	Define the work plane. See "11.5 Configuring the profile editor. Define the work plane." on page 215.
End	End the profile editing session and insert the edited profiles into the program. See "11.6 End the session at the editor." on page 215.
Undo	Undo the last operation.
Save and continue	Save the profile and continue editing. Using this key does not require that the profile be completed.

11.

PROFILE EDITOR

Interface description.

11.1.1 How to use the profile editor.

Several profiles may be edited without having to exit the profile editor. To edit a profile, proceed as follows:

- 1 Define the work plane at the profile editor.
- 2 Select the type of profile to be edited, such as a circular or rectangular profile or any profile.
- 3 For a rectangular or circular profile, define its data and insert it. For any profile, first select the starting point of the profile. Once the first point has been selected, draw the profile, which will be made up of straight and curved sections. If it has corner rounding, chamfering or tangential entries and exits, use one of these methods:
 - Treat them as individual sections when having enough information to define them.
 - Ignore them while defining the profile and, once it has been defined, select the corners that have those characteristics and insert them.
- 4 End the profile editing session by inserting them into the program. The portion of ISO-code program corresponding to the edited profile will be identified with the line "(#PROFILE)" or it will appear framed between the lines "(#PROFILE BEGIN)" and "(#PROFILE END)".

Data editing

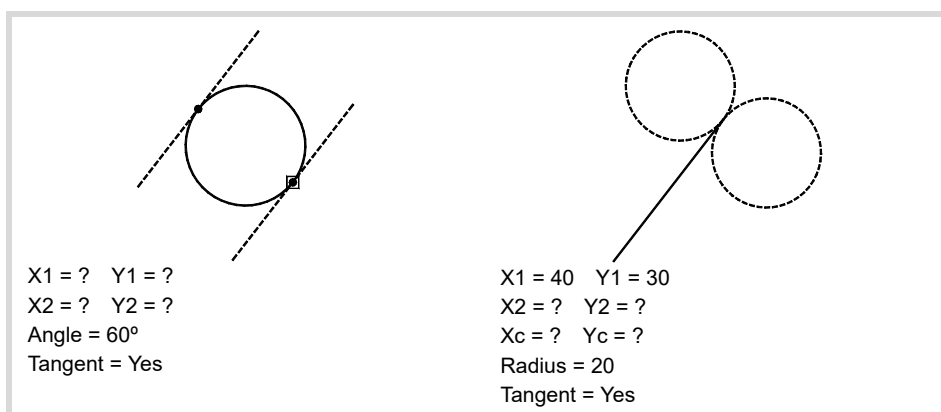
All data need not be defined; but it is recommended to define all the known data. To define the profile data, proceed as follows:

- 1 Press the softkey corresponding to the data to be defined.
- 2 Key in the desired value (which may be a numeric constant or an expression entered via the calculator). Use the [SPACE] key to change the value of a non-numerical data (tangency, direction, etc.). Press [CTRL]+[K] to access the calculator.
- 3 Press [ENTER] to accept the defined value or [ESC] to reject it and return to the previous one. If the entered value is accepted, the CNC will select the next data.
- 4 Once all the data has been defined, press the "Validate" softkey and the CNC will show the profile that has been defined.

Displaying incomplete sections

If there isn't enough data to show the defined section, the CNC will draw as much of it as it knows. The sections that are not fully defined will be shown with a dash line.

If there are more than one possibility, use the arrow keys to view the available options one by one except the ones that could generate tangency errors later on. To select the desired option, press [ENTER]. The sections with several possibilities will be shown in green whereas the rest of the sections will be shown in white.



11.

PROFILE EDITOR
Interface description.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

11.2 Define a new profile, enlarge an existing one or import one from a file.

With the softkey menu of this screen, it is possible to define any profile, a circular or a rectangular one. It can also be used to enlarge a profile already defined or import a profile saved in DXF format.

Softkey.	Meaning.
Profile	Softkey for editing any profile by defining the straight and curved sections that make it up. See " 11.2.1 Define any profile using straight and curved sections. " on page 205.
Circle	Softkey for quickly defining a circular profile. See " 11.2.2 Define a circular profile. " on page 207.
Rectangle	Softkey for quickly defining a rectangular profile. See " 11.2.3 Define a rectangular profile. " on page 208.
Expand a profile	Softkey for adding straight and curved sections to the profile. See " 11.2.4 Enlarge a profile. " on page 208.
Import profile	Softkey for importing a profile in DXF format. See " 11.2.5 Import a profile from a DXF file. " on page 209.

11.

PROFILE EDITOR

Define a new profile, enlarge an existing one or import one from a file.

11.2.1 Define any profile using straight and curved sections.

For any element of the profile, the softkey menu may be used to define the data in Cartesian or Polar coordinates as well as in absolute or incremental coordinates. The softkeys for these options are only available when allowed by the selected data.

Softkey.	Meaning.
Cartesian Polar	Softkey to select either Cartesian or Polar coordinates.
ABS INC	Softkey to select either absolute or incremental coordinates. Being the incremental coordinates active, the editor will show the symbol Δ next to the affected data.
Polar origin	Softkey to define the Polar origin.
Direction	Direction of the profile; clockwise or counterclockwise.
Validate	Validate the profile defined.

Define the starting point of the profile.



When selecting a new profile, the starting point must always be defined first. Once the profile's starting point has been defined, the softkey menu will show the necessary options to define the profile. To return to the main menu, press the [BACK] key.

The starting point may be edited both in Cartesian and Polar coordinates, but always in absolute coordinates.

Define a straight section of the profile.

The end point of the section may be edited both in Cartesian and Polar coordinates as well as in absolute or incremental coordinates. Being the incremental coordinates active, the editor will show the symbol Δ next to the affected data.

Cartesian coordinates.

The data definition area shows the following data.

Data.	Information
X1, Y1	Coordinates of the starting point of the section on each axis of the active plane at the editor.
X2, Y2	Coordinates of the end point of the section on each axis of the active plane at the editor.
Angle	Angle of the profile with the abscissa axis.
Tangency	Tangency with the previous section of the profile.

Polar coordinates.

The data definition area shows the following data.

Data.	Information
r1, θ 1	Radius and Polar angle of the section's starting point.
r2, θ 2	Radius and Polar angle of the section's end point.
Angle	Angle of the profile with the abscissa axis.
Tangency	Tangency with the previous section of the profile.

Define a curved section (clockwise or counterclockwise) of the profile.

The end point and the center of the section may be edited both in Cartesian and Polar coordinates as well as in absolute or incremental coordinates. Both points may have different types of coordinates and, besides, when using Polar coordinates, both points may have a

11.

PROFILE EDITOR

Define a new profile, enlarge an existing one or import one from a file.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

different Polar origin. Being the incremental coordinates active, the editor will show the symbol Δ next to the affected data.

Cartesian coordinates.

The data definition area shows the following data.

Data.	Information
X1, Y1	Coordinates of the starting point of the section on each axis of the active plane at the editor.
X2, Y2	Coordinates of the end point of the section on each axis of the active plane at the editor.
Xc, Yc	Coordinates of the center of the section on each axis of the active plane at the editor.
Radius	Radius of the arc.
Tangency	Tangency with the previous section of the profile.

Polar coordinates.

The data definition area shows the following data.

Data.	Information
r1, θ 1	Radius and Polar angle of the section's starting point.
r2, θ 2	Radius and Polar angle of the section's end point.
rc, θ c	Radius and Polar angle of the section's center point.
Radius	Radius of the arc.
Tangency	Tangency with the previous section of the profile.

Start a new profile.

This softkey may be used to add a new profile to the current one, for example to define a pocket with islands.

Modify the last element of the profile.

This softkey may be used to modify the last element added to the profile.

Undo the last operation.

This softkey may be used to "undo" the last operation made.

Save and continue.

To save the profile and continue editing. Using this key does not require that the profile be completed.

11.

PROFILE EDITOR

Define a new profile, enlarge an existing one or import one from a file.



11.2.2 Define a circular profile.

The softkey menu may be used to define the data in Cartesian or Polar coordinates as well as in absolute or incremental coordinates. The softkeys for these options are only available when allowed by the selected data.

Softkey.	Meaning.
Cartesian Polar	Softkey to select either Cartesian or Polar coordinates.
ABS INC	Softkey to select either absolute or incremental coordinates.
Polar origin	Softkey to define the Polar origin. The softkey is only available when Polar coordinates are active.
Direction	Direction of the profile; clockwise or counterclockwise.
Validate	Validate the profile defined.

The type of coordinates may be changed at any time and the editor will update the displayed values.

- The starting point and the center may be edited both in Cartesian and Polar coordinates, but both points must have the same type of coordinates. Therefore, a change of coordinate type affects both points. If programmed in Polar coordinates, the Polar origin will be the same for both.
- The starting point of the circle may only be edited both in absolute coordinates, whereas the center may be edited in both absolute and incremental coordinates.

Define the data of the profile.

The data definition area shows all the data needed to define a profile. The data shown by the editor depends on the active coordinate type, Cartesian or Polar. Being the incremental coordinates active, the editor will show the symbol Δ next to the affected data.

Cartesian coordinates.

The data definition area shows the following data.

Data.	Information
X1, Y1	Coordinates of the starting point of the profile on each axis of the active plane at the editor.
Xc, Yc	Coordinates of the center of the profile on each axis of the active plane at the editor.
Radius	Radius of the arc.
Direction	Direction of the profile; clockwise or counterclockwise.

Polar coordinates.

The data definition area shows the following data.

Data.	Information
r1, θ 1	Radius and Polar angle of the profile's starting point.
rc, θ c	Radius and Polar angle of the profile's center point.
Radius	Radius of the arc.
Direction	Direction of the profile; clockwise or counterclockwise.

11.

PROFILE EDITOR

Define a new profile, enlarge an existing one or import one from a file.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

11.2.3 Define a rectangular profile.

The softkey menu may be used to define the data in Cartesian or Polar coordinates. The softkey for this options is only available when allowed by the selected data.

Softkey.	Meaning.
Cartesian Polar	Softkey to select either Cartesian or Polar coordinates.
Polar origin	Softkey to define the Polar origin. The softkey is only available when Polar coordinates are active.
Direction	Direction of the profile; clockwise or counterclockwise.
Validate	Validate the profile defined.

The type of coordinates may be changed at any time and the editor will update the displayed values.

- The starting point may be edited both in Cartesian and Polar coordinates.
- The starting point of the rectangle can only be edited in absolute coordinates.

Define the data of the profile.

The data definition area shows all the data needed to define a profile. The data shown by the editor depends on the active coordinate type, Cartesian or Polar.

Cartesian coordinates.

The data definition area shows the following data.

Data.	Information
X1, Y1	Coordinates of the starting point of the profile on each axis of the active plane at the editor.
XL, YL	Length of the profile on each axis of the active plane at the editor.
Angle	Angle of the profile with the abscissa axis.
Direction	Direction of the profile; clockwise or counterclockwise.

Polar coordinates.

The data definition area shows the following data.

Data.	Information
r1, θ1	Radius and Polar angle of the profile's starting point.
XL, YL	Length of the profile on each axis of the active plane at the editor.
Angle	Angle of the profile with the abscissa axis.
Direction	Direction of the profile; clockwise or counterclockwise.

11.2.4 Enlarge a profile.

The softkey menu may be used to select one of the profiles of the editor and continue building it by adding straight and curved sections. After selecting the profile, press the [ENTER] key to enter in editing mode where the softkey menu will show the options to define straight and curved sections. See ["11.2.1 Define any profile using straight and curved sections."](#) on page 205.

11.

PROFILE EDITOR

Define a new profile, enlarge an existing one or import one from a file.

11.2.5 Import a profile from a DXF file.

The DXF format is standard for exchanging graphic files. The CNC can import this type of files and, based on the contours and tool paths it contains, generate the ISO-coded blocks of the part-program.

DXF files may be imported and modified in the program editor and in the profile editor. After selecting the file, define how the various layers of the DXF file are to be interpreted.

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.	Drilling, tapping, etc must be represented with a single point. The CNC interprets the circles that represent the drilling, etc. as paths to be machined. To keep the look of the drawing intact, place these elements on a layer so the CNC can disable this layer when importing the drawing.

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 if 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.

11.

PROFILE EDITOR

Define a new profile, enlarge an existing one or import one from a file.

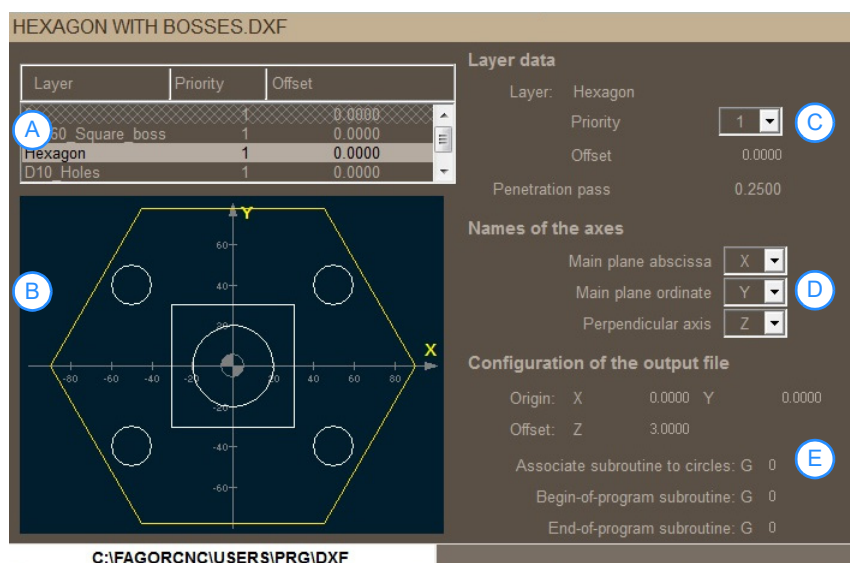


CNCelite
8058 8060
8065 8070

REF: 2508

Importing a profile into the CNC.

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 drawing layers.
- B File preview. The preview 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 with the cursor.
- D Name of the axes.
- E Output file configuration.

List of drawing layers.

The dxf file can be divided into layers, where each layer contains different part sections (contours, dimensions, etc). The table indicates the following for each of the layers:

- Name of the layer, as defined in the dxf file.
- Layer priority or order.
- Offset (position) of the layer on the perpendicular axis.
- Dwelling of the layer; enabled or disabled (the layer appears shaded). When a layer is disabled, the CNC does not insert it into the program (for example, coordinate layers or auxiliary lines). To disable a layer, select it with the cursor and press the "Disable layer" softkey. To re-enable the layer, press the same softkey.

Data of the layer selected with the cursor.

The following data must be defined for each layer.

- 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. Priority 1 layers will be the first to be executed, then priority 2 layers and so on. There can be several layers that have the same priority.
- Offset (position) of the layer on the perpendicular axis. The offset (height) allows each layer to be executed for the desired Z coordinate (or that of the relevant perpendicular axis).
- Depth of pass.

Name of the axes.

The work plane and the perpendicular axis must be defined before importing the file into the part-program (abscissa and ordinate). To import the file into the profile editor, first define the perpendicular axis; the work plane will be the one selected by the editor.

Output file configuration.

- Origin X / Origin Y. 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.

11.

PROFILE EDITOR

Define a new profile, enlarge an existing one or import one from a file.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

- Subroutine associated with circles. If a subroutine has been defined, 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 (for example, to convert the circles into drill holes). If no subroutine has been defined, the dxf converter converts the circles to segments with G2/G3.
- Subroutine to execute at the beginning of the program.
- Subroutine to execute at the end of the program.

Valid subroutines are G180-189, G380-G399 and G500-G599.

11.

PROFILE EDITOR

Define a new profile, enlarge an existing one or import one from a file.



CNCelite
8058 8060
8065 8070

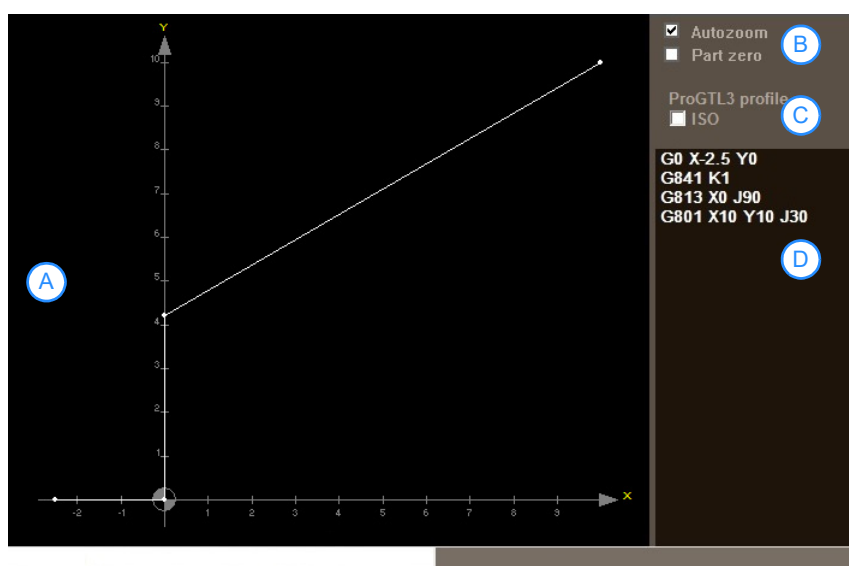
REF: 2508

11.2.6 Editing a profile in ProGTL3 language.

11.

PROFILE EDITOR

Define a new profile, enlarge an existing one or import one from a file.



- A** Graphic area. Graphic representation of the profile being drawn, axes coordinated with autoscale and name of the axes that make up the plane. The name of the axis indicates the positive direction of the axis.
- B** Status of the autozoom and part zero options, regarding the display of the profile at the editor.
- C** Status of the ISO option.
- D** Zone to edit the sections in ProGTL3 programming language.

The softkey menu may be used to perform the following actions.

Softkey.	Meaning.
Displayed area.	Softkey to modify the zoom of the graphics area. See "11.4 Configuring the profile editor. Displayed area." on page 215.
ISO	Softkey to insert the profile into the program using ISO code.
Finish	Softkey to end the profile editor session. Before exiting, the profile editor will prompt for the edited profile to be saved or not. See "11.6 End the session at the editor." on page 215.
Save and continue.	This softkey saves the profile and continues with the editing. Using this key does not require that the profile be completed.

11.3 Modify a profile and insert corners

This softkey menu may be used to change the defined profiles either changing or deleting existing elements, inserting new ones or introducing rounding, chamfering and tangential entries or exits. When selecting this option, the softkey menu will show the necessary options to modify the profile.

Softkey.	Meaning.
Modify element	This softkey may be used to modify any data of a section of the profile.
Insert element	This softkey may be used to insert a new element in any position of the profile.
Delete element	This softkey may be used to delete an element of the selected profile.
Corners	This softkey may be used to include rounding, chamfers, tangential entries or exits in the defined profile.
Additional ISO	This softkey may be used to add an ISO coded line to a previously closed profile. Once the ISO coded line to be added has been entered, confirm the command by pressing the [ENTER] key.
Undo	To "undo" the last operation made.
Save and continue	To save the profile and continue editing. Using this key does not require that the profile be completed.

Modify element

This softkey may be used to modify any data of a section of the profile. Once the desired element has been selected, one may modify the type of section (straight or arc) or its data. Once the element has been modified, press "Validate" to confirm the changes.

The CNC recalculates the new profile according to the data used to define that section and the next one (tangency, angle, etc.)

Insert element

This softkey may be used to insert a new element in any position of the profile. After selecting the section after which the element is to be inserted, select the type of section (straight or arc) to be inserted, define its parameters and press the "Validate" softkey.

The CNC recalculates the new profile according to the data used to define that section and the next one (tangency, angle, etc.)

Delete element

This softkey may be used to delete an element of the selected profile. Once the element to be deleted has been selected, confirm the command by pressing [ENTER]. The CNC recalculates the new profile.

Corner definition

This softkey may be used to include rounding, chamfers, tangential entries or exits in the defined profile. When selecting this option, the softkey menu shows the type of corners that can be inserted.

Softkey.	Meaning.
Rounding	Define a rounding at the profiles corners where it is possible.
Chamfer	Define a chamfer at the corners of the profile where it is possible.
Tangential entry	Add a tangential tool entry at the beginning of the profile.
Tangential exit	Add a tangential tool exit at the end of the profile.

11.

PROFILE EDITOR

Modify a profile and insert corners

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Once the type of corner to be inserted has been selected, the CNC will highlight in red one of the corners of the profile. Use the softkey menu or the following keys to select another element of the profile or select a corner of another profile.

Key.	Meaning.
	Select the corners of the profile.
	Select a corner of another profile.

After selecting the corner of the profile to be modified and the CNC will request the value (radius or size) of the corner.

- For a rounding, enter the rounding radius.
- For a chamfer, enter the size of the chamfer.
- For a tangential entry, enter the entry radius.
- For a tangential exit, enter the exit radius.

After defining the value of the corner, press [ENTER] to insert the corner in the profile. Then, one may select another corner or quit this mode by pressing [ESC].

Additional ISO.

This softkey may be used to add an ISO coded line to a previously closed profile. Once the ISO coded line to be added has been entered, confirm the command by pressing the [ENTER] key.



When the profile is part of a conversational canned cycle, the CNC ignores the feedrate (F) programmed in the additional ISO lines. The CNC uses the feedrates set in the canned cycle.

11.

PROFILE EDITOR

Modify a profile and insert corners

11.4 Configuring the profile editor. Displayed area.

The softkey menu of this screen may be used to modify the zoom of the graphics area. When accessing the "display area" menu, the softkey menu shows the following options:

Softkey.	Meaning.
Zoom + Zoom -	Enlarge or reduce the display area.
Optimum area	Select the best zoom, i.e. it places the profile in the center of the graphics window and selects the best zoom possible to show the whole profile.
Part zero	Keeps the part zero visible at all times.
Autozoom	Activate autozoom. When applying Autozoom, every time a new section is inserted which goes beyond the screen, the profile will automatically be centered and zoomed in or out to show the whole profile. This way, it will show the whole profile again.
Validate	Confirm the changes made.

When at the editor no menu is active to edit data or select elements, these options may be applied with their corresponding hotkeys. See "[Keyboard shortcuts.](#)" on page 201.

11.5 Configuring the profile editor. Define the work plane.

The softkey menu of this screen may be used to modify the axes of the plane and their directions. At the lathe model, the direction and position of the axes are defined by machine parameter GRAPHTYPE. When accessing the plane menu, the softkey menu shows the following options:

Softkey.	Meaning.
Abscissa axis	Select the abscissa axis.
Abscissa direction	Select the direction of the abscissa axis.
Ordinate axis	Select the ordinate axis.
Ordinate direction	Select the direction of the ordinate axis.
Validate	Confirm the changes made.

When loading a profile, the profile editor configures the axes according to the extension of the file containing the profile. The extension of the profile must begin with "p" followed by the name of the abscissa and ordinate axes. For example, when loading the profile name.pxz, the profile editor sets the X axis as abscissa and Z as ordinate.

11.6 End the session at the editor.

The "End" softkey ends the profile editing session. Before exiting the profile editor, it will offer the option to save or not the edited profile.

Softkey.	Meaning.
Save profile	Insert the profile in the program and exit the profile editor. If the profile has been resolved, the CNC will insert it in the part program that is being edited. If the editor cannot resolve the profile due to lack of data, the CNC will issue the relevant message.
Do not save profile	Do not insert the profile in the program and it exit the profile editor.
Continue	Do not insert the profile in the program and continue editing the profile.

11.

PROFILE EDITOR

Configuring the profile editor. Displayed area.

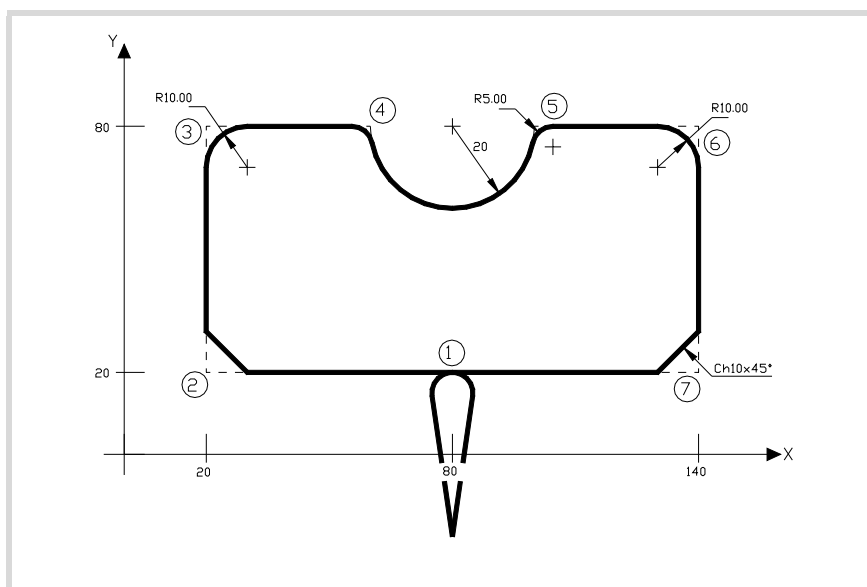
FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

11.7 Examples of how to define profiles.

11.7.1 Profile editor. Example 1 (milling).



Profile definition without rounding, chamfers or tangential entries and exits.

Section.	Geometry.		
Starting point	X = 80	Y = -20	
Straight	X = 80	Y = 20	
Straight	X = 20	Y = 20	
Straight	X = 20	Y = 80	
Straight	X = 60	Y = 80	
Counterclockwise arc	X = 100	Y = 80	Center X = 80 Center Y = 80 Radius = 20
Straight	X = 140	Y = 80	
Straight	X = 140	Y = 20	
Straight	X = 80	Y = 20	
Straight	X = 80	Y = -20	

Definition of rounding, chamfers and tangential entries and exits

Select the "Corners" option. Press [ESC] to quit the "Corners" option.

Corner		
Tangential entry	Select point "1"	Assign radius = 5
Chamfer	Select point "2"	Assign size = 10
Rounding	Select point "3"	Assign radius = 10
Rounding	Select point "4"	Assign radius = 5
Rounding	Select point "5"	Assign radius = 5
Rounding	Select point "6"	Assign radius = 10
Chamfer	Select point "7"	Assign size = 10
Tangential exit	Select point "1"	Assign radius = 5

End of editing

Select the "END" option and save the profile. The CNC quits the profile editor and inserts the profile in the part-program.

11.

PROFILE EDITOR

Examples of how to define profiles.

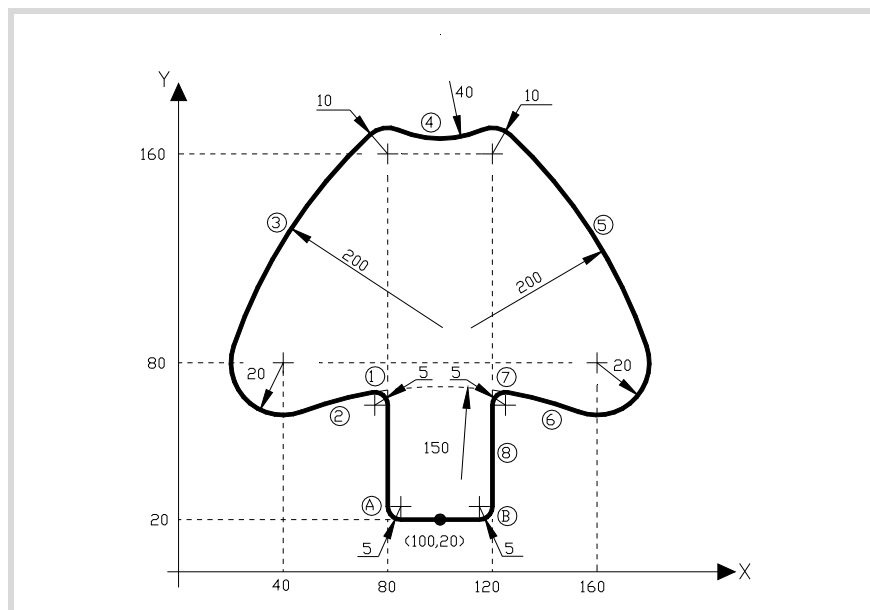
FAGOR
AUTOMATION



CNCelite
8058 8060
8065 8070

REF: 2508

11.7.2 Profile editor. Example 2 (milling).



Definition of a profile without rounding.

Section.	Geometry.		
Starting point	X = 100 Y = 20		
Straight	X = 80 Y = 20		
Straight	X = 80	Angle = 90	
Counterclockwise arc (1)	Center X = 75	Radius = 5	Tangency = Yes
Counterclockwise arc (2)	Center X = 100	Radius = 150	Tangency = Yes
Clockwise arc	Center X = 40 Center Y = 80	Radius = 20	Tangency = Yes
- The CNC shows the options for section 2. Select the correct one. - The CNC shows the options for section 1. Select the correct one.			
Clockwise arc (3)		Radius = 200	Tangency = Yes
Clockwise arc	Center X = 80 Center Y = 160	Radius = 10	Tangency = Yes
The CNC shows the options for section 3. Select the correct one.			
Counterclockwise arc (4)		Radius = 40	Tangency = Yes
Clockwise arc	Center X = 120 Center Y = 160	Radius = 10	Tangency = Yes
The CNC shows the options for section 4. Select the correct one.			
Clockwise arc (5)		Radius = 200	Tangency = Yes
Clockwise arc	Center X = 160 Center Y = 80	Radius = 20	Tangency = Yes
The CNC shows the options for section 5. Select the correct one.			
Counterclockwise arc (6)	Center X = 100	Radius = 150	Tangency = Yes
The CNC shows the options for section 6. Select the correct one.			
Counterclockwise arc (7)	Center X = 125	Radius = 5	Tangency = Yes
The CNC shows the options for section 7. Select the correct one.			
Straight (8)	X = 120 Y = 20		Tangency = Yes
The CNC shows the options for section 8. Select the correct one.			
Straight	X = 100 Y = 20		

11.

PROFILE EDITOR

Examples of how to define profiles.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Rounding definition "A" and "B".

Select the "Corners" option. Press [ESC] to quit the "Corners" option.

Corners.

Rounding	Select point "A"	Assign radius = 5
Rounding	Select point "B"	Assign radius = 5

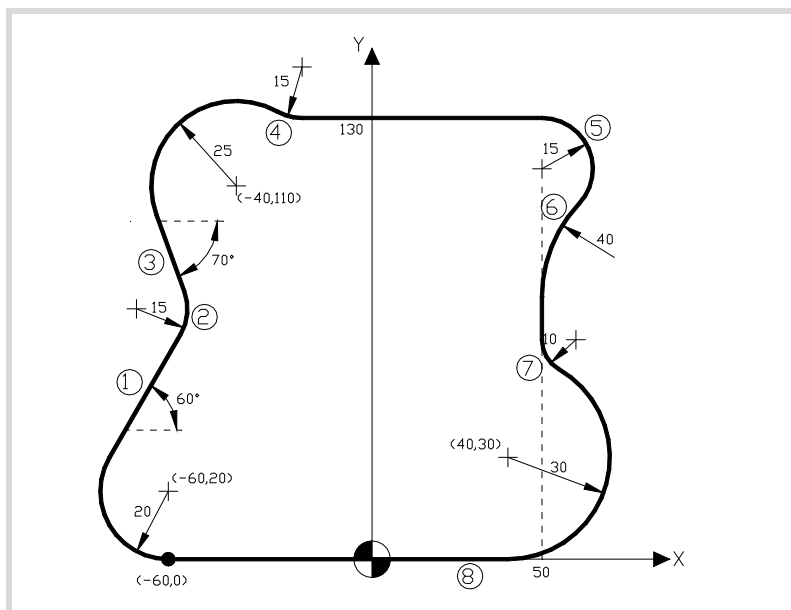
End of editing

Select the "END" option and save the profile. The CNC quits the profile editor and inserts the profile in the part-program.

11.**PROFILE EDITOR**

Examples of how to define profiles.

11.7.3 Profile editor. Example 3 (milling).



Profile definition.

Section.	Geometry.		
Starting point	X = -60 Y = 0		
Clockwise arc	Center X = -60 Center Y = 20	Radius = 20	
Straight (1)		Angle = 60	Tangency = Yes
• The CNC shows the options for section 1. Select the correct one.			
Counterclockwise arc (2)		Radius = 15	Tangency = Yes
Straight (3)		Angle = -70	Tangency = Yes
Clockwise arc	Center X = -40 Center Y = 110	Radius = 25	Tangency = Yes
• The CNC shows the options for section 3. Select the correct one.			
• The CNC shows the options for section 2. Select the correct one.			
Counterclockwise arc (4)		Radius = 15	Tangency = Yes
Straight	Y = 130	Angle = 0	Tangency = Yes
• The CNC shows the options for section 4. Select the correct one.			
Clockwise arc (5)	Center X = 50	Radius = 15	Tangency = Yes
• The CNC shows the options for section 5. Select the correct one.			
Counterclockwise arc (6)		Radius = 40	Tangency = Yes
Straight	X = 50	Angle = 270	Tangency = Yes
• The CNC shows the options for section 6. Select the correct one.			
Counterclockwise arc (7)		Radius = 10	Tangency = Yes
Clockwise arc	Center X = 40 Center Y = 30	Radius = 30	Tangency = Yes
• The CNC shows the options for section 7. Select the correct one.			
Straight (8)	X = -60 Y = 0		Tangency = Yes
• The CNC shows the options for section 8. Select the correct one.			

End of editing

Select the "END" option and save the profile. The CNC quits the profile editor and inserts the profile in the part-program.

11.

PROFILE EDITOR

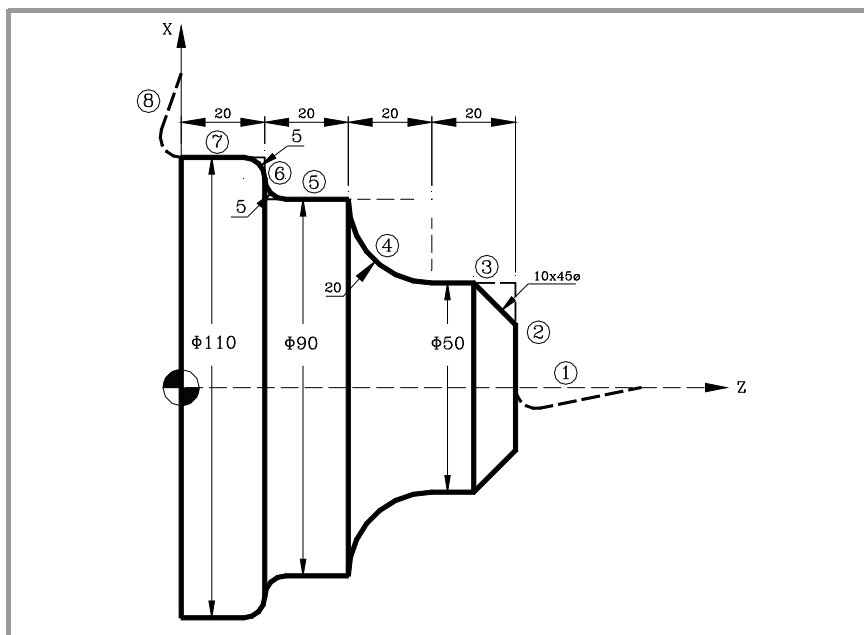
Examples of how to define profiles.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

11.7.4 Profile editor. Example 4 (lathe)



Profile definition without rounding, chamfers or tangential entries and exits.

Section.	Geometry.		
Starting point	Z = 100 X = 0		
Straight (1)	Z = 80 X = 0		
Straight (2)	Z = 80 X = 50		
Straight (3)	Z = 60 X = 50		
Clockwise arc (4)	Z = 40 X = 90	Center Z = 60 Center X = 90	Radius = 20
Straight (5)	Z = 20 X = 90		
Straight (6)	Z = 20 X = 110		
Straight (7)	Z = 0 X = 110		
Straight (8)	Z = 0 X = 150		

Definition of rounding, chamfers and tangential entries and exits

Select the "Corners" option. Press [ESC] to quit the "Corners" option.

Corners.		
Tangential entry	Select the corner "1-2"	Assign radius = 5
Chamfer	Select the corner "2-3"	Assign size = 10
Rounding	Select the corner "5-6"	Assign radius = 5
Rounding	Select the corner "6-7"	Assign radius = 5
Tangential exit	Select the corner "7-8"	Assign radius = 5

End of editing

Select the "END" option and save the profile. The CNC quits the profile editor and inserts the profile in the part-program.

11.

PROFILE EDITOR

Examples of how to define profiles.

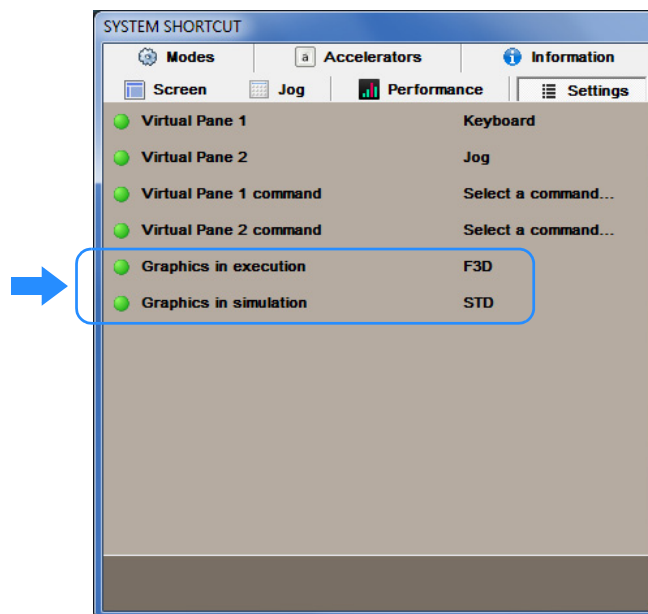
GRAPHIC ENVIRONMENT (MILL MODEL)

12

Solid and linear graphics for the execution and simulation of part-programs and canned cycles of the editor. During machining, linear graphs display the tool path in real time, as well as solid graphics for the tool removing material from the part. These graphics can display up to 4 views of the machining process. Measurements can also be made for the part and sections can even be made for the piece.

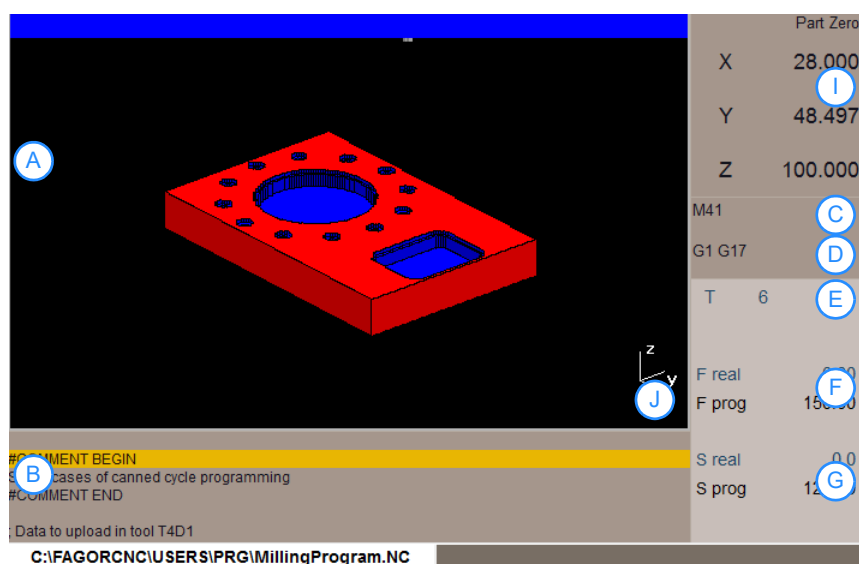
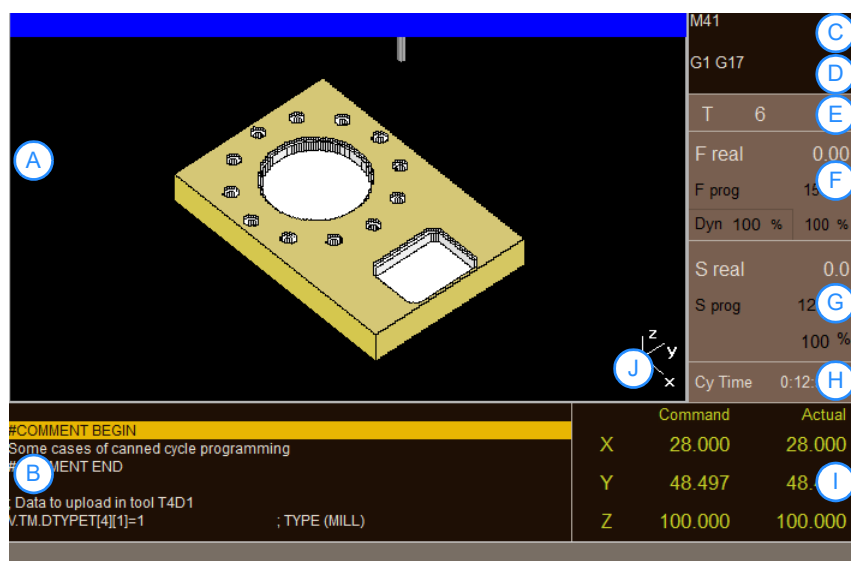
HD graphics and standard Fagor graphics.

The CNC can toggle between Fagor standard graphics and HD graphics from the task window ([CTRL] + [A]), "Settings" tab. The CNC application must be restarted.



12.1 Description of the graphic environment.

The graphic environment displays a graphic representation of the program that is being executed or simulated and take measurements on the graphics.



- A Graphic program representation.
- B Program blocks.
- C Active "M" functions.
- D Active "G" functions and high-level commands.
- E Tool information.
- F "F" axis feedrate.
- G "S" spindle speed.
- H Execution time "Cy Time" of the program.
- I Position of the axes.
- J Graph orientation.

Graphic program representation.

Shows a graphic representation of the tool paths or of the part as the program is being executed or simulated.

Program blocks.

It shows data on the selected program for execution and selects the first and final execution blocks. During execution, the cursor shows the block being executed.

Tool information.

- Number of the active tool "T".
- Active "D" tool offset.

"F" axis feedrate.

- Simulation graphics.
 - "F real" real feedrate.
 - "F prog" programmed feedrate.
- Execution graphics.
 - "F real" real feedrate.
 - "F prog" programmed feedrate.
 - Feedrate percentage active on the CNC (jog key, program or PLC).
 - Dynamic percentage "Dyn" machining.
 - The text "F real" appears in red indicating that the PLC is inhibiting the movement of the axes (FEEDHOL mark active).

"S" spindle speed.

- Simulation graphics.
 - Real speed "S real" ("S1 real", "S2 real", etc).
 - Programmed speed "S prog" ("S1 prog", "S2 prog", etc).
 - If the text "Sreal" appears in red, it means that the PLC is inhibiting the movement of the spindle (INHIBIT mark active).
- Execution graphics.
 - Real speed "S real" ("S1 real", "S2 real", etc).
 - Programmed speed "S prog" ("S1 prog", "S2 prog", etc).
 - Active speed percentage for the CNC (JOG keypad, program or PLC).
 - If the text "Sreal" appears in red, it means that the PLC is inhibiting the movement of the spindle (INHIBIT mark active).
 - The screen only shows the data on one spindle. If there are several spindles in the channel, the data on the next spindle may be displayed by pressing the "S" key.

Position of the axes.

- Simulation graphics.
 - Current position of the axes as compared to part zero.
- Execution graphics.
 - 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).

12.

GRAPHIC ENVIRONMENT (MILL MODEL)

Description of the graphic environment.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

12.

GRAPHIC ENVIRONMENT (MILL MODEL)

Description of the graphic environment.

Graph orientation.

This zone shows the work plane appearing in the display area and an illustration representing the size of the graph and the portion of the graphic area selected with the zoom. On 3D graphics, the illustration shows the point of view of the graph displayed and it may be changed by the operator.

Key.	Meaning.
	Select a new point of view.
	Assumes the new point of view selected.
	Cancels the selected point point of view.

12.1.1 Softkey menus.

Vertical softkey menu.

The vertical softkey menu shows the options related to the operating mode that called upon the graphic environment (EDISIMU mode for simulation, automatic mode for execution, etc.).

Horizontal softkey menu (graphic area).

When selecting this area, the horizontal softkey menu shows the necessary options for interacting with graphics.

Softkey.	Description.
"Type of graphics"	Select the type of graphics.
"ZOOM"	Enlarge or reduce the whole graph shown or part of it.
"Dimensions"	Define the size of the graphics representation.
"Point of view"	Change the point of view of the graph and show it from another perspective.
"Measurement"	Measure the distance between two points.
"Clear screen"	Clear the screen or delete the graphics displayed.
"Colors"	Set the appearance of the graphics on display.
"Options"	Set the appearance and some options of the graphic window.
" +Real"	Display either the real or the theoretical tool path.
" +Error"	Display the real path, but enlarging the error with respect to the theoretical path. Pressing this softkey also activates the one corresponding to the real path.
"Speed"	Change the simulation speed.

12.

GRAPHIC ENVIRONMENT (MILL MODEL)

Description of the graphic environment.

12.2 Type of graphics

When selecting this option, the softkey menu shows the types of graphics available. The various types of graphics may be line (3D lines, XY, XZ, YZ and Combined) or solid (Sections and 3D solid). Line graphics show the tool path with lines of different colors and solid graphics show an image of the part.

The type of graphics selected will remain active until another type is selected or the graphic display is deactivated or the CNC is turned off. Likewise, when changing the type of graphics, the CNC will maintain the graphic conditions (zoom, graphic parameters, display area, etc.), defined for the last graphic.

"3D lines" Graphics

This type of graphics displays a three-dimensional graph of the tool paths.

"Sections" Graphics

This type of graphic displays a top view (XY plane) of the part; it shows the machining depths in different tones. It also displays the XZ and YZ sections for the areas shown by the indicators of the top view.

These indicators may be moved around using the [↑][↓][←][→] keys, to display the different sections of the part. The CNC shows dynamically the new section being selected.

The indicators may be moved at any time even while executing the program.

"XY" - "XZ" - "YZ" graphics

This type of graphic displays the tool paths in the XY, XZ or YZ plane.

"Combined" graphics

This type of graphic divides the display area in four quadrants and displays the tool path corresponding to each plane XY, XZ, YZ and to the 3D view.

"Solid 3D" Graphics

This type of graphics displays a three-dimensional graph of the machining of the part. Starting out with a 3D block which is "machined" as the program is executed or simulated.

12.

GRAPHIC ENVIRONMENT (MILL MODEL)

Type of graphics



12.3 Zoom

The zoom option may be used to enlarge or reduce the whole graph shown or part of it. This option is not available in the "Combined" type of graphics.

After selecting the "Zoom" option, a zoom frame will appear over the graphics. This frame may be enlarged, reduced and moved around over the graphics already displayed in order to select a particular portion of it to zoom into or out of.

Key.	Meaning.
   	To move the zoom frame around.
 	To change the size of the zoom frame.
	To validate and display the selected zoom area.

The graphics at the lower right-hand side of the screen shows two figures. The one shown with lines only, indicates the dimensions of the display area and the one with colored sides indicates the portion selected with the zoom.



When selecting this option, the softkey menu shows the available zoom options. To return to the main menu, press the [BACK] key.

Zoom "Initial"

This option restores the size of the display area set via program or using the "Dimensions" option.

Zoom "Automatic"

The CNC uses the zoom that it considers best according to the movements programmed.

Zoom "Previous"

This option displays up to two zooms defined earlier. After the second one, it shows again the one defined last.

Zoom "Limits"

Only for the "Sections" type. In this graphics, zooming is done by moving the indicators that appear framing the graphic sections.

With this option, it is possible to select the axis whose indicator is to be moved. The indicator may also be selected with the [+] and [-] keys of the numeric keyboard in a rotary fashion (Xmin, Xmax, Ymin, Ymax, Zmin, Zmax).

Zoom "Edit"

It is used to manually edit the zoom values. It is edited in the dialog area of the graphic window that shows the dimensions of the zoom frame

12.

Zoom

GRAPHIC ENVIRONMENT (MILL MODEL)

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

12.4 Dimensions

This softkey is used to define the size of the graphic representation by setting the maximum and minimum coordinates of the graphics on each axis.



When selecting this option, the softkey menu shows the options available for setting the dimensions. To return to the main menu, press the [BACK] key.

Dimensions "Automatic"

The CNC sets the dimensions that it considers best according to the movements programmed.

Dimensions "Edit"

The CNC lets manually edit the dimension values. It is edited in the graphic window that shows the dimensions of the graphics on each axis.

12.5 Point of view

This softkey is used to select the point of view on 3D graphics. This option is only available for the types of graphics "Combined", "3D lines", "Sections" and "Solid 3D".

The orientation of the graphics may be directly selected at the graphic window by orienting the XY plane and the Z axis. The XY plane may be rotated 360° and the Z axis 90°. The figure at the lower right-hand side of the screen shows the point of view currently selected.

Key.	Meaning.
	To orient the XY plane.
	To orient the Z axis.
	To validate and display the selected point of view.
	To cancel the selected point point of view.



When selecting this option, the softkey menu shows the options available for selecting the point of view. To return to the main menu, press the [BACK] key.

Point of view "Edit"

It is used to manually edit the orientation of the axes of the graphics. It is edited in the dialog area of the graphic window that shows the current orientation of the axes.



12.6 Measurement

This softkey may be used to measure the distance between two points. This option is only available for the types of graphics "XY", "XZ", "YZ" and "Solid 3D".

When selecting this option, the section being measured will appear on the graphics with two cursors and a dashed line. The cursor currently selected will appear in red.

Key.	Meaning.
   	To move the zoom frame around.
	To change the size of the zoom frame.

The dialog area will show the coordinates of both cursors, the distance between them on the straight line and the components of that distance on the axes of the active plane. The coordinates of the selected cursor will appear in red.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the [BACK] key.

Measurement "Point1" & "Point2"

This option is used to select the cursor to be moved (same as using the [+] key).

Measurement "Edit"

This option is used to manually edit the position of the cursors. It is edited in the dialog area of the graphic window that shows the position of both cursors.

12.

GRAPHIC ENVIRONMENT (MILL MODEL)

Measurement

12.7 Clear screen

This softkey is used to clear the screen or delete the graphics displayed. If a solid graphic type is selected, the graphic representation will be reset and it will return to its initial state without machining.

12.

GRAPHIC ENVIRONMENT (MILL MODEL)

Clear screen

12.8 Colors

This softkey is used to change the colors used in the graphic representation.

- In the line graphics, it is possible to choose the color for each type of tool path; the color for rapid movements, paths with tool compensation, etc.

The real coordinates are only available for the execution of the program. The real coordinate is the actual position of the tool which differs from the command coordinate in the amount of following error (axis lag).

- In solid graphics, only the color of each side of the solid may be selected.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the [BACK] key.

Colors "Apply"

It assumes the new colors and applies them to the blocks drawn next. If the new colors are not applied, the graphics are drawn with the old colors.

Colors "Edit"

It is used to select the new colors for the graphics. They are selected in the dialog area of the graphic window that show the current colors.

Key.	Meaning.
	To move the cursor through the data.
	To open and close the color palettes.
	To move the cursor through the colors of the palette.
	To cancel the selected colors.



12.9 Options

This softkey is used to set the appearance and some functions of the graphic window. These options may be used at any time, even while executing a program.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the [BACK] key.

Option "Activate"

This softkey may be used to activate the graphics. The softkey appears pressed when this option is enabled. The status of this softkey cannot be modified while executing or simulating a program.

When the graphic representation is deactivated and activated, the current graphic is erased; but the display conditions are kept active (type of graphics, zoom, graphic parameters and display area) that were active before that mode was deactivated.

Option "Simple"

This softkey shows the single window for graphics. The softkey appears pressed when this option is enabled.

The single window hides the dialog and data areas on the right-hand side of the graphic window so the drawing occupies the whole graphic window.

Option "Hide Tool"

This softkey is used to hide or show the tool while simulating in "3D solid" graphics. The softkey appears pressed when this option is enabled.

Lathe tools are never displayed regardless of the status of this softkey. When it is a lathe tool, this softkey may be activated to hide the tool, but it cannot be deactivated to show the tool.

Option "Print"

This softkey may be used to print the graphics in the pre-determined printer or save it as a file (bmp format) at the CNC. When selecting the "File" option, it will be saved in the folder "C:\Cnc8070\Users\Reports\"; the file name may be selected using the "Print configuration" softkey.

When selecting this option, the CNC will show a dialog box requesting the print destination (printer or file). After selecting the destination, press [ENTER] to print it or [ESC] to cancel it.

Option "Print setup"

It is used to set the printing configuration. When selecting this option, the CNC shows a dialog box where the following may be defined:

- The title of the graphics that will appear next to it in the print.
- The name of the file where the graphics will be stored when printing out to a file.

After filling out the data, press [ENTER] to accept them or [ESC] to cancel them.

12.

GRAPHIC ENVIRONMENT (MILL MODEL)
Options

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

12.10 Real coordinates

This softkey is used to draw the real tool path or the theoretical tool path. This option is only available in line type graphics.

This option is only available when executing the program; not when simulating it. When selecting this option (the softkey will appear pressed), the CNC draws the actual (real) tool path.

12.11 Real coordinates with enlarged error

This softkey shows the real path, but enlarging the error with respect to the theoretical path. The error enlargement factor may be set after pressing the softkey.

This option is only available when executing the program; not when simulating it. When selecting this option (the softkey will appear pressed), the CNC draws the actual (real) tool path with enlarged error. Pressing this softkey also activates the one corresponding to the real path.

12.12 Simulation speed

This softkey is used to change the simulation speed.



When selecting this option, the softkey menu shows the options to set the simulation speed. To return to the main menu, press the [BACK] key.

This option is only available when simulating the program; not when executing it.

Speed "Edit"

It is used to select the new simulation speed. It is selected using the graduated ruler that indicates the active simulation speed.

Key.	Meaning.
 	To move the ruler cursor.

12.

GRAPHIC ENVIRONMENT (MILL MODEL)

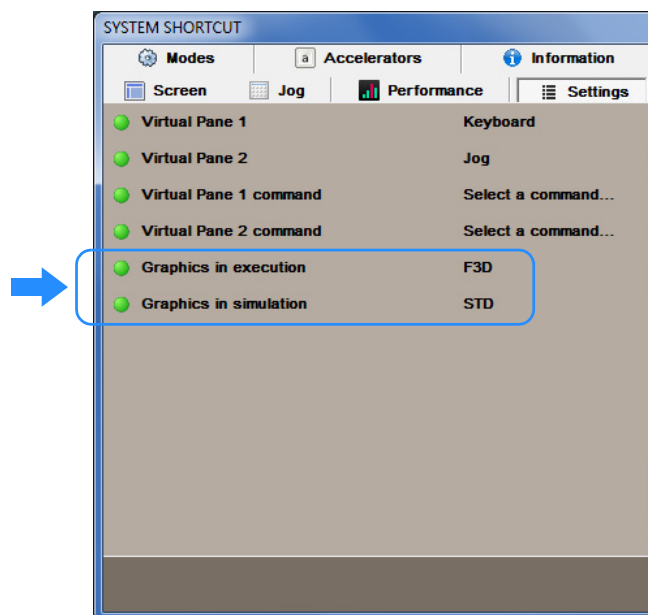
Real coordinates

13.1 Graphic environment.

High definition solid 3D graphics for the execution and simulation of part-programs and canned cycles of the editor. During machining, the HD graphics display the tool removing the material from the part in real time, allowing for the condition of the part to be seen at all times. HD graphics can display up to 4 views of the part, where each can be rotated, zoomed in or zoomed out. Measurements can also be made on the part and even sections on the piece from any angle.

HD graphics and standard Fagor graphics.

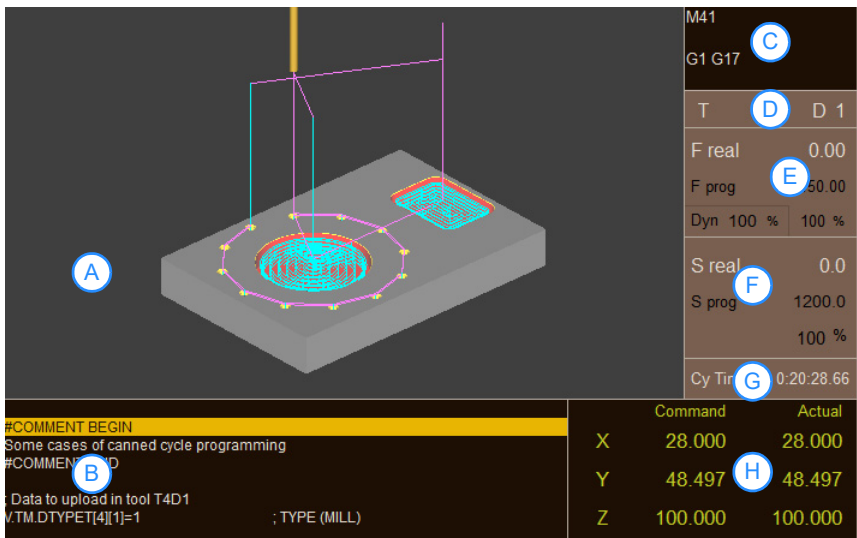
Using active HD graphics, the standard Fagor graphics can be accessed through the task window [CTRL] + [A], "Settings" tab.



13.1.1 HD graphics in automatic mode.

The automatic mode displays a graphical representation of the program being executed.

13. GRAPHICS HD (.M- MODEL).
Graphic environment.



	Command	Actual
X	28.000	28.000
Y	48.497	48.497
Z	100.000	100.000

A. The graphical area displays a graphical representation of the tool paths, the solid or both.
B. Program or executed subroutine.
C. Active functions "M" and "G".
D. Tool number "T" and tool offset "D".
E. Real feedrate (F real), programmed feedrate (F prog) and axes override. Dynamic (Dyn) override.
F. Real speed (S real), programmed speed (S prog) and spindle override.
G. Execution time (Cy Time) of the program.
H. Programmed coordinate (Command) and axis position regarding the part zero referring to the tool tip.

Vertical softkey menu.

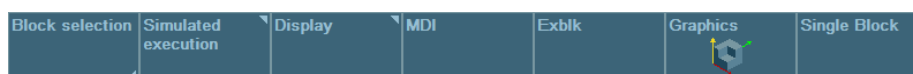
The vertical softkey menu displays the options depending on the operating mode calling the graphic environment; in this case, automatic mode. Refer to the chapter on "Automatic mode" for more details on the functionality of these softkeys.

Horizontal softkey menu.



The horizontal softkey menu changes depending on the area in which the focus is located; graphics area or program area. Use the [FOCUS] key to change the focus area.

- When the focus is on the program area, the softkey menu displays the options to manage the program execution. The softkey menu is the same as the automatic mode. Refer to the chapter on "Automatic mode" for more details on the functionality of these softkeys.

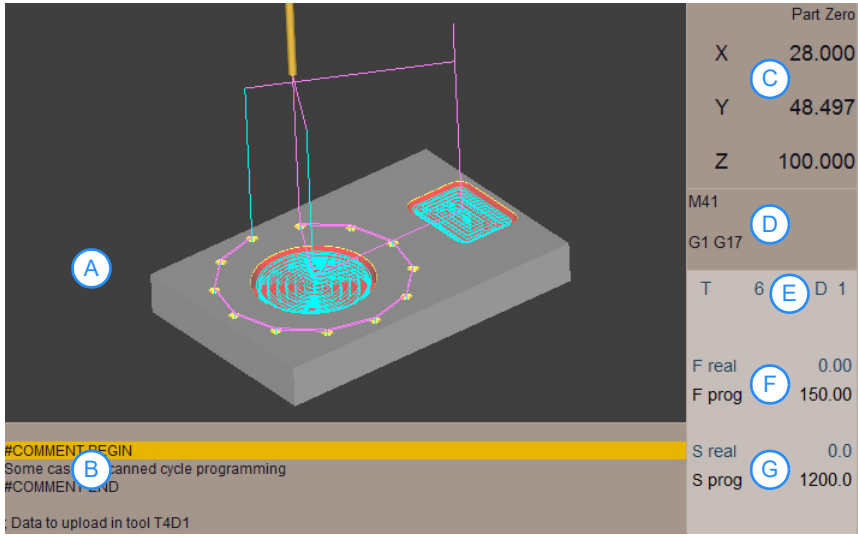


- When the focus is on the graphics area, the softkey menu displays the options to work with the graphics. The softkey menu is similar to the graphics of the edisimu mode.



13.1.2 HD graphics in edisimu mode.

The edisimu mode displays a graphical representation of the program being simulated.



The screenshot shows the edisimu mode interface. On the left, a 3D model of a part being machined is displayed. On the right, a data panel shows the following information:

Part Zero	
X	28.000
Y	48.497
Z	100.000
M41	
G1 G17	
T 6	D 1
F real	0.00
F prog	150.00
S real	0.0
S prog	1200.0

At the bottom, a program editor shows the following code:

```
#COMMENT BEGIN
Some cas...anned cycle programming
#COMMENT END
Data to upload in tool T4D1
```

Labels A through G point to specific elements in the interface:

- A: The graphical area displays a graphical representation of the tool paths, the solid or both.
- B: Program or simulated subroutine.
- C: Axis position regarding the part zero referring to the tool tip.
- D: Active functions "M" and "G".
- E: Tool number "T" and tool offset "D".
- F: Real feedrate (F real), programmed feedrate (F prog) of the axes.
- G: Real speed (S real), programmed speed (S prog) of spindle.

Vertical softkey menu.

The vertical softkey menu displays the options depending on the operating mode calling the graphic environment; in this case, edisimu mode. Refer to the chapter on "edisimu mode" for more details on the functionality of these softkeys.

Horizontal softkey menu.



The horizontal softkey menu changes depending on the area in which the focus is located; graphics area or program area. Use the [FOCUS] key to change the focus area.

- When the focus is on the program area, the softkey menu displays the options used to manage the program simulation. The softkey menu is the same as the edisimu mode. Refer to the chapter on "edisimu mode" for more details on the functionality of these softkeys.



- When the focus is on the graphics area, the softkey menu displays the options to work with the graphics. The softkey menu is similar to the graphics of the automatic mode.



13.1.3 Horizontal softkey menu.

Softkey.	Function.
	Type of view. - Select the type of view. - Select the parts to view.
	Configuration. - Configuring the graphics window. - Configuring and activating the sections. - Configure the colors of the tool path and solid. - General configuration of the graphics. - Cancel the graphics. - Load a machine configuration (xca file).
	Actions. - Move sections. - Print the diagram.
	Delete. Removes the tool paths and resets the solid to its initial dimensions.
	Dimensions. - Defines the type and size of the parts. - Deletes parts. - Saves and deletes parts.
	Measurement. Measure the distance between two points.
	Ver. - Views the tool paths and the solid. - Simulation speed.

13.

GRAPHICS HD (.M- MODEL).

Graphic environment.

13.2 Move, rotate and zoom in or out on the graphic.

When no option has been selected on the softkey menu, it is possible to act upon the graphic as follows.

Move the graphic.

To move the part of the active view, move the mouse while holding its right button or use the following keys.

Key.	Meaning.
 	Move the part to the left or to the right.
 	Move the part up or down.

Rotate the graphic.

To rotate the part of the active view, move the mouse while holding its left button or use the following keys [[↑][↓]][←][→].

Zoom the graphic.

To zoom in or out on the active view, turn the mouse wheel or use the [+][-] keys.

13.

GRAPHICS HD (.M- MODEL).

Move, rotate and zoom in or out on the graphic.

13.3 Select the type of view.



This option is used to select the point of view of the part. Since the screen may be divided into 4 windows, the softkey only affects the active window (the one having the focus). The screen may be selected with the tab key or with the keys [1] through [4] (depending on the window to be selected).

- Top view.
- Front view.
- Left view.
- Right view.
- First user view.
- Second user view.
- Save the current view as user view.
- Select the part to be displayed.

User views.

The top view, front view and side views are predetermined views that may be moved or rotated with the mouse or keyboard. The new position and orientation of the part may be saved as a user view. The CNC keeps the saved user views even after being turned off.

Select the part to be displayed.

The graphics may have defined up to four different parts. Each of the four windows that the screen may be divided into may display the same part or different parts. The following softkeys only affect the active window (the one having the focus).

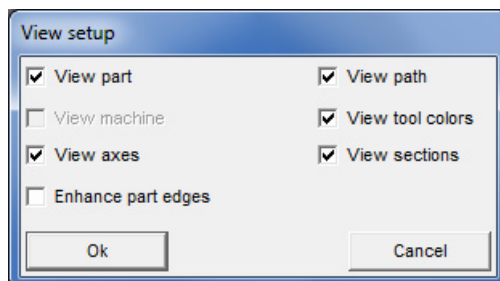
Softkey.	Function.
	General view. In machines with rotary axes, this softkey allows the for the rotating part or tool to be viewed.
	Displays the first part associated with the channel. Each part may be associated with one or more channels. See "13.13 Editing, displaying and hiding parts." on page 247. When changing channels, the displayed part changes. If several channels are working on the same part, all of them can show the same part.
	View part 1. The menu shows a softkey per available part.



13.4 Configuring the graphics window (properties of each window).



This softkey is used to configure the graphic properties of the active window, so each of the four windows dividing the screen may have different properties. For example, a window may show only the part to be machined and another one only the machining tool paths. The active screen may be changed using the tab key or keys [1] through [4] (depending on the window to be selected).



Properties.	Meaning.
View part.	To configure the window to view the part. To view the part, the "View solid" option must be activated using the "View" softkey. See chapter "13 Viewing the tool paths and the solid."
View tool path.	Configure the window to show the tool paths. To view the tool paths, the option "View path" must be activated using the "View" softkey. See chapter "13 Viewing the tool paths and the solid."
View machine.	Not used.
View tool colors.	The diagram shows the tool path of each tool in the color assigned to it. See "13.6 Configure the colors for the tool path and solid." on page 242.
View axes.	The diagram shows the main axes of the origin channel for the part. The origin of the axes coincides with the part zero. See "13.13 Editing, displaying and hiding parts." on page 247.
View sections.	Configure the window to display the active sections. See "13.5 Configuring and activating the sections." on page 240.
Enhance part edges.	The diagram enhances the edges of the part.

13.

GRAPHICS HD (.M. MODEL).

Configuring the graphics window (properties of each window).

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

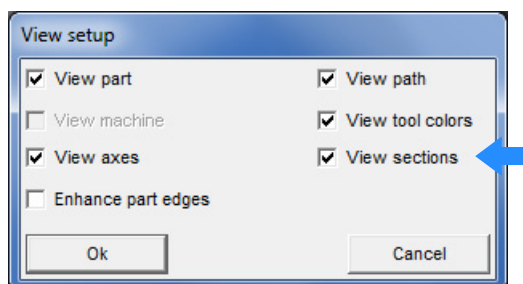
REF: 2508

13.5 Configuring and activating the sections.

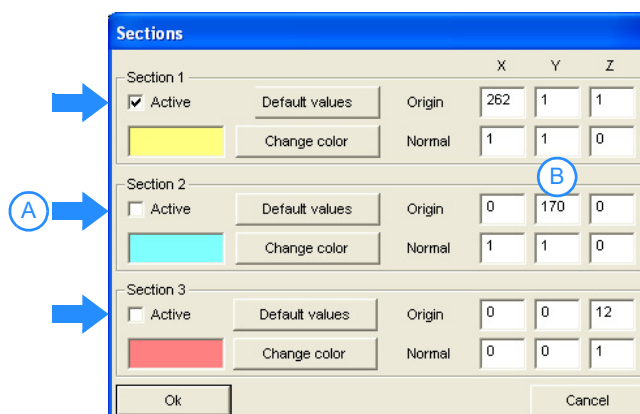


This softkey defines up to three planes used to divide the diagram. The sections are global to all the defined parts. If there is a section defined in X100, it will cut all the parts that reach this coordinate.

Although the definition of the sections is global for all the defined windows, each window may have different sections active. To view the sections in a window, one must also enable the "View sections" option in the window properties. See *"Configuring the graphics window (properties of each window)." on page 239.*



Configuring the sections.



A. Activate the sections in the active window.

B. Section configuration for all windows.

Define the sections (default values).

The "default values" button defines three sections, each of these are perpendicular to one of the axes and cut the part in half.

Define the sections.

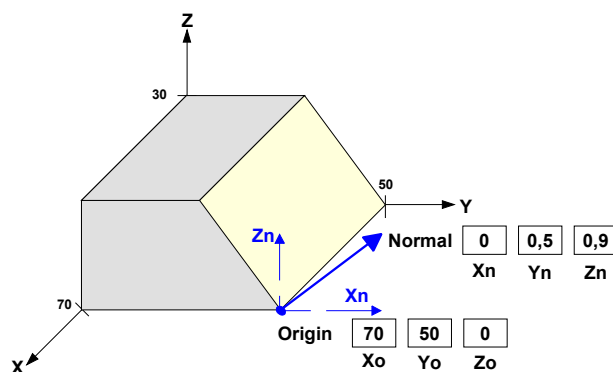
The resulting section will be a plane perpendicular to the vector defined in "Normal" and it goes through the point defined in "Origin". The color of each section may also be defined.

- The "Origin" data defines the origin of the section.
- The "Normal" data defines the components of the vector normal to the plane, that defines the orientation of the plane.

13.

GRAPHICS HD (.M- MODEL).

Configuring and activating the sections.



Activate the sections.

Each section may be activated individually.

To view the sections on the graphic, one must also enable the "View sections" option in the window properties. See ["13.4 Configuring the graphics window \(properties of each window\)."](#) on page 239. If the "View sections" option is not enabled, the graphic does not show any section even if they are active. If the "view sections" option is enabled, the diagram shows the active sections.

13.

GRAPHICS HD (.M- MODEL).
Configuring and activating the sections.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

13.6 Configure the colors for the tool path and solid.



This softkey is used to configure the colors for the part and for the tool paths. This configuration is global for all the windows.

Path colors.

This option is used to assign colors to the following tool paths.

- Rapid traverse (G0).
- Compensated movements.
- Uncompensated movements.
- Canned cycles.
- Tapping.

Part colors.

This option is used to change the background colors, the colors of the machined areas and the look of the part. The selected values remain until the CNC is turned off.

- Background color.
- Color of the machining operations carried out with the first five tools used on the graphic. Here only the colors are defined, to display them, the "view tool colors" option must be active in the properties of the view.
- Material used to represent the solid part. The "Advanced" button may be used to change the properties of the selected material. Besides the various color components (diffuse, ambient, etc.), the brightness and transparency of the part can also be defined. Transparency may take values between 0 and 1. If the value of the transparency is other than zero, the solid part becomes transparent.

13.

GRAPHICS HD (.M- MODEL).

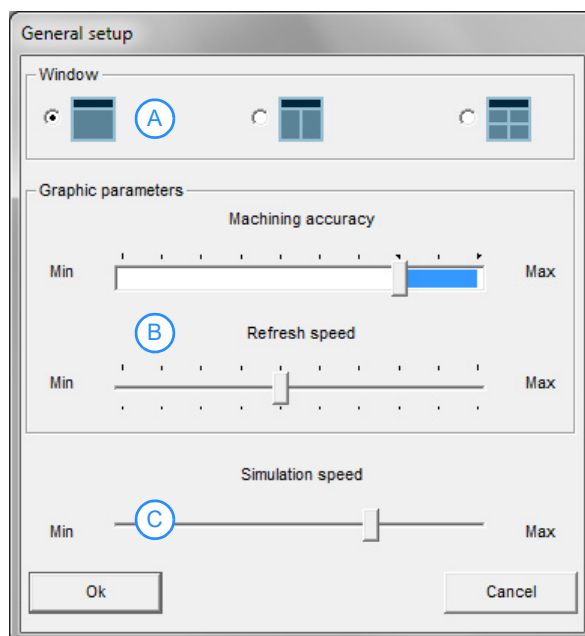
Configure the colors for the tool path and solid.



13.7 General configuration of the graphics.



This softkey may be used to configure the properties of the graphic environment, those affecting all the windows.



- A. Number of windows.
- B. Graphic parameters.
- C. Simulation speed.

Windows.

The graphic environment may be a full-screen window or may be divided into 2 or 4 windows, each with different properties (for example a different view of the part). The screen may be selected with the tab key or with the keys [1] through [4] (depending on the window to be selected).

Graphic parameters.

These parameters influence the speed and quality of the graphics; the higher the value of these parameters, the greater the graphics quality, but the slower the graphics speed.

- Machining accuracy. This selection bar allows to choose whether the graph considers the machine dynamics (blue zone) or not (white zone). Selecting to include machine dynamics increases the simulation time which depends on the dynamics.
- Refresh speed.

Simulation speed (edisimu mode only).

This bar allows the simulation speed to be determined under edisimu mode. The speed may be modified during the simulation using the "View" softkey. See ["13.18 Simulation speed \(edisimu mode only\)."](#) on page 251.

13.

GRAPHICS HD (.M. MODEL).
General configuration of the graphics.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

13.8 Configuration. Cancel the graphics.



This softkey allows the graphics to be deactivated. When the graphics are deactivated, it is not possible to graphically represent the program being executed or simulated. Graphics under automatic and edisimu modes are deactivated independently.

13.9 Configuration. Load machine.



This softkey may be used to load a new machine configuration in HD graphics (xca files). The CNC offers several xca files, one per model, containing the definition and configuration of the machine for HD graphics. On power-up, the CNC assumes the last file used.

If the physical configuration of the machine changes during execution (for example, spindle change with different number of axes), the corresponding xca file must be loaded so the graphics show the changes. The xca files may be loaded either from the softkey menu or from the program using the instruction #DEFGRAPH.

When changing the machine configuration, the CNC saves the part displayed on the screen automatically as LastPiece.stl into the folder ../Users/Grafdata, and it recovers it after the new configuration.



The machine configuration files supplied by Fagor consist of a single file, the xca. These files cover most configurations; therefore, new xca will have to be generated when the machine has some special requirement that affects graphics.

When an OEM creates his own configuration files, for each xca file, he must create a ".def" file with the same name that completes the configuration of the axes involved in the kinematics. Both files must be copied when saving the configuration file in another folder.

13.

GRAPHICS HD (.M- MODEL).

Configuration. Cancel the graphics.



13.10 Actions. Move the active sections.



This option may be used to move the active sections. To move the sections, the active view must have the "view sections" option enabled and it must also have a section active. To end the possibility to move sections, press the [ESC] key or press the same softkey again. With this option active, the bottom of the screen shows a message indicating the active section and its data.

Select the section to move.

One of the sections is always selected (the one highlighted in orange). To change the section to be moved, use the tab key or with the [1] [2] [3] keys (depending on the section number). The section to be moved appears in orange and the rest in the color they have been defined with.

Move sections.

- If the section to be moved is perpendicular to one of the axes of the trihedron, like the default sections, the [↑][→] keys move the section in the positive direction of the axis and the [↓][←] keys move it in the negative direction of the axis.
- If the section is not perpendicular to any of the axes of the trihedron, it is necessary to select one of the axes of the trihedron (hold one of the keys [X] [Y] [Z] down) and use the [↑][→] keys move the section in the positive direction of the axis and the [↓][←] keys move it in the negative direction of the axis. In the jog mode it is not possible to move these sections, as the selection of axes applies to movements in jog.

13.

GRAPHICS HD (.M- MODEL).

Actions. Move the active sections.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

13.11 Actions. Print the graphic.



This option may be used to print the graphic out to the predetermined printer or out to a file.

Print the graphic.



This softkey may be used to print the graphics in the pre-determined printer or save it as a file (bmp format) at the CNC. When printing the graphic to a file, it will be saved in the folder "..\Users\Reports\"; the file name may be selected using the "Print configuration" softkey.

When selecting this option, the CNC will show a dialog box requesting the print destination (printer or file). After selecting the destination, press [ENTER] to print it or [ESC] to cancel it.

Print configuration.



This softkey may be used to set the printing properties. When selecting this option, the CNC shows a dialog box where the following may be defined:

- The title of the graphics that will appear next to it in the print.
- The name of the file where the graphics will be stored when printing out to a file.

After filling out the data, press [ENTER] to accept them or [ESC] to cancel them.

13.12 Delete the graphic.



This softkey deletes the tool paths and resets the solid to its initial dimensions.

13.

GRAPHICS HD (.M- MODEL).

Actions. Print the graphic.



13.13 Editing, displaying and hiding parts.



This softkey allows up to four pieces to be defined, either rectangular or cylindrical, and to be assigned to various channels.

Rectangular part.

Edit part

Part number: (A) 1 Reference origin: (B) C1

Part type: (C) RECT

X Min: -73.9 X Max: 140.9

Y Min: (E) -67.2 Y Max: 67.2

Z Min: -32 Z Max: 112

Associated with: ☒ C1 (F) ☐ C2 ☐ C3

Work with the part

☒ (G) ☐ (H)

Ok Cancel

Cylindrical part.

Edit part

Part number: (A) 1 Reference origin: (B) C1

Part type: (C) CYL Part axis: (D) X

X Min: (E) -65 X Max: 130

Inside diameter: -65 Outside diameter: 65

Associated with: ☒ C1 (F) ☐ C2 ☐ C3

Work with the part

☒ (G) ☐ (H)

Ok Cancel

- A. Part number.
- B. Reference channel; used to define the origins and the dimensions.
- C. Part type; rectangular or circular.
- D. Longitudinal axis of the part (cylindrical parts only).
- E. Part dimensions. To include a zero offset in the part, edit the zero offset at the minimum values of the axes.
- F. Channels associated with the part, for displaying. Since several channels may work on a part, in order to be able to keep displaying the machining of that part when changing channels, it is necessary to define which channels are working on that part.
- G. Show the part
- H. Hide the part.



The dimensions of the part correspond to the part zero (G92, G54 to G59, G159) active at the time of the activating the part, for this reason it is advisable to first activate the part zero and then the part.

13.

GRAPHICS HD (.M. MODEL).
Editing, displaying and hiding parts.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

13.14 Automatic dimensions.



The CNC erases the graphics and sets the dimensions that it considers as optimal, depending on the programmed movements, in the four windows dividing the screen.

13.15 Saving part / Loading part.



These softkeys may be used to save and load the parts as stl files (STereoLithography) to be used in a quick prototyping (for example, 3D printers). The stl files are saved in the folder ../Users/Grafdata.

In a machine configuration file exchange (xca file), the CNC automatically saves the current part as LastPiece.stl and retrieves it after activating the new machine configuration.

13.

GRAPHICS HD (.M- MODEL).

Automatic dimensions.

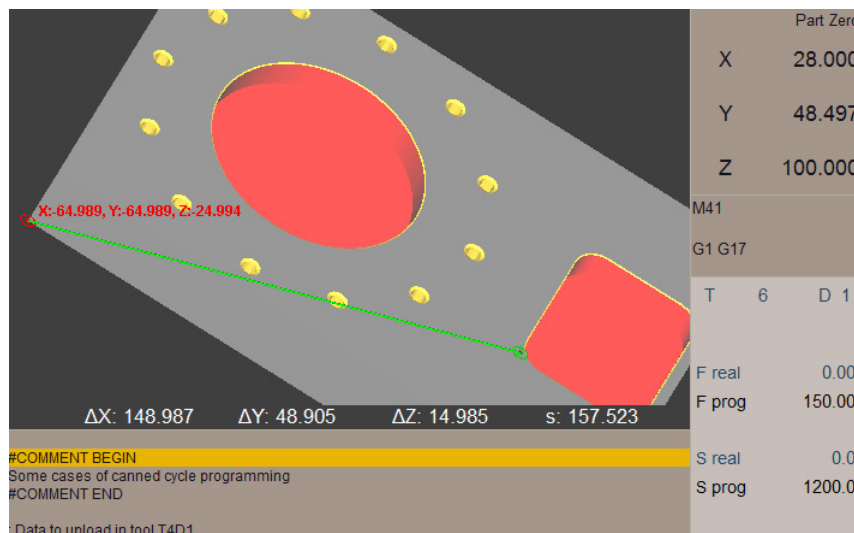


13.16 Measure the part.



This option may be used to measure the distance between two points. After selecting this option, the line drawn between two measuring points above the diagram and the distance between the both points on each of the axes (ΔX , ΔY , ΔZ) and the distance in a straight line is shown below. Once measured, press the [ESC] key or press the same softkey again.

If the graphics environment is divided into several windows, the measuring points are only visible in the active window.



For 3D views, the movement of the measuring points is limited to the part. To measure distances beyond the part (tool paths), a 2D view must be selected (top, front, left or right).

Moving the measuring points.

The active measuring point, indicated in red, can be moved with the mouse or keyboard. It is recommended to use the mouse.



To change the active measuring point, use the tab key or place the cursor over it.

- Use the keys [↑][↓][←][→] to move the active measuring point.
- Use the left mouse button to drag the measuring points.
- Click the mouse on any point on the screen, the active measuring point moves to that point automatically.

13.

GRAPHICS HD (.M- MODEL).

Measure the part.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

13.17 Viewing the tool paths and the solid.

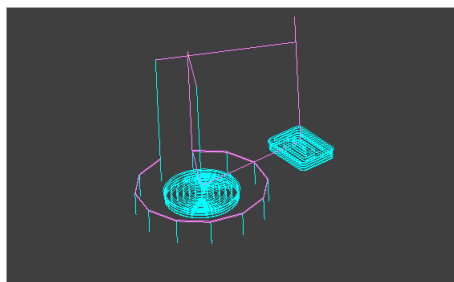


This softkey toggles between the various types of diagrams. Each of the four windows dividing the screen may display the diagram in a different way; to do this, the options "View part" and "View path" must be selected in the window properties. See ["Configuring the graphics window \(properties of each window\)."](#) on page 239.

13.

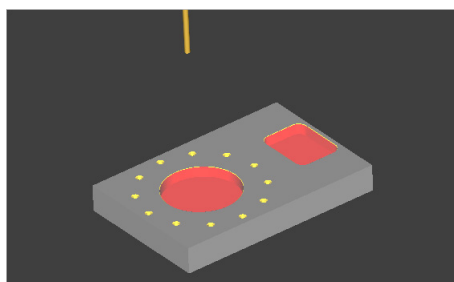
GRAPHICS HD (.M- MODEL).

Viewing the tool paths and the solid.



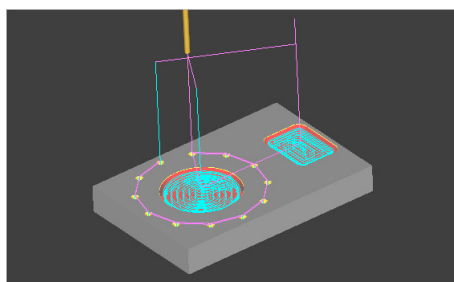
Viewing tool paths.

This option only displays the tool paths. The colors of the tool paths can be changed for the softkey "Configuration" > "Colors" > "Path".



Viewing the solid.

This option only shows the solid to be machined. Each surface is represented by a color associated with the corresponding machine tool. The colors of the tools can be changed for the softkey "Configuration" > "Colors" > "Solid".



Viewing tool paths and solids.

This option only displays the solid to be machined and the tool paths.



13.18 Simulation speed (edisiimu mode only).



This softkey allows the simulation speed to be changed under edisiimu mode. After selecting this option, the CNC displays a cursor for setting the simulation speed. To move the cursor, use the mouses or the [SHIFT][←] or [SHIFT][→] keys.

Simulation speed

70

Z100 ; Positioning along longitudinal axis (Z) to a safety zone
G88 G99 X0 Y0 Z2 I-10 D2 J35 B3 L0.5 H500 V50 ; Circular pocket canned cycle (G
G87 G98 X105 Y0 Z2 I-10 D2 J21 K28 M1 Q8 B3 L1 H480 V30 ; Rectangular pocket canned cyc
G0 G80 Z100 ; Canned cycle cancellation (G80) and positioning to safety Z (Z
T11 D1 ; Call tool (T) and offset (D) for center punching and chamfering
M6 ; Execute tool change

Part Zero

X 90.330
Y 22.979
Z -5.000

M3 M6 M41
G1 G17 G61 G87 G91

T 4 D 1

F real 480.00
F prog 480.00

S real 1000.0
S prog 1000.0

13.

GRAPHICS HD (.M- MODEL).
Simulation speed (edisiimu mode only).



CNCelite
8058 8060
8065 8070

REF: 2508

13.

GRAPHICS HD (.M. MODEL).

Simulation speed (edisi mu mode only).



CNCelite
8058 8060
8065 8070

REF: 2508

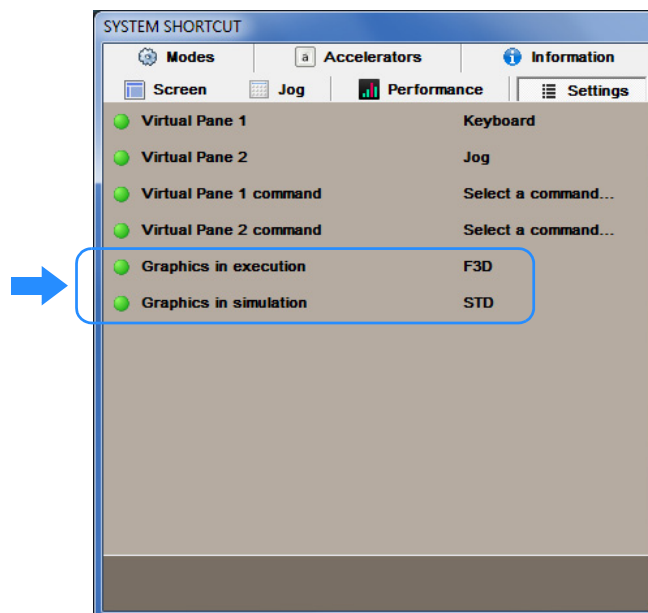
GRAPHIC ENVIRONMENT (LATHE MODEL)

14

Solid and linear graphics for the execution and simulation of part-programs and canned cycles of the editor. During machining, linear graphs display the tool path in real time, as well as solid graphics for the tool removing material from the part. These graphs can show C-axis machining processes and also allow measurements to be made on the part.

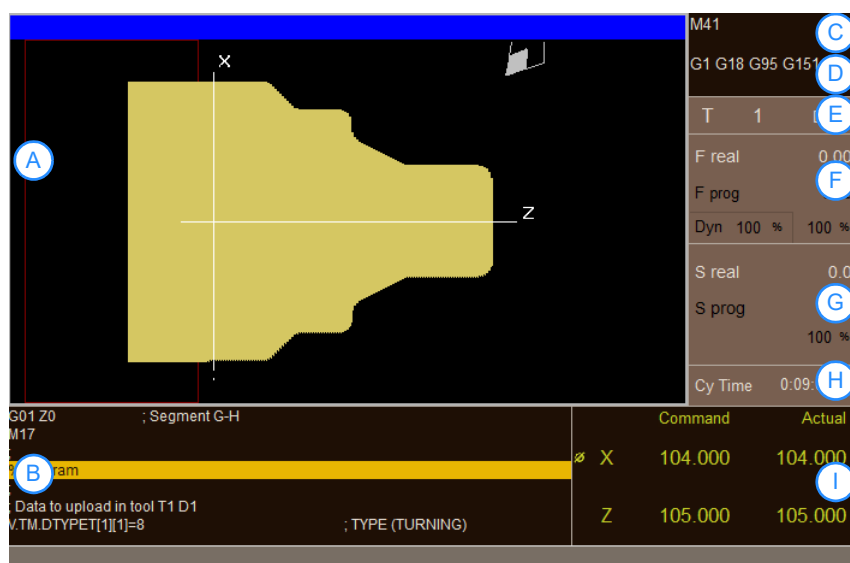
HD graphics and standard Fagor graphics.

The CNC can toggle between Fagor standard graphics and HD graphics from the task window ([CTRL] + [A]), "Settings" tab. The CNC application must be restarted.

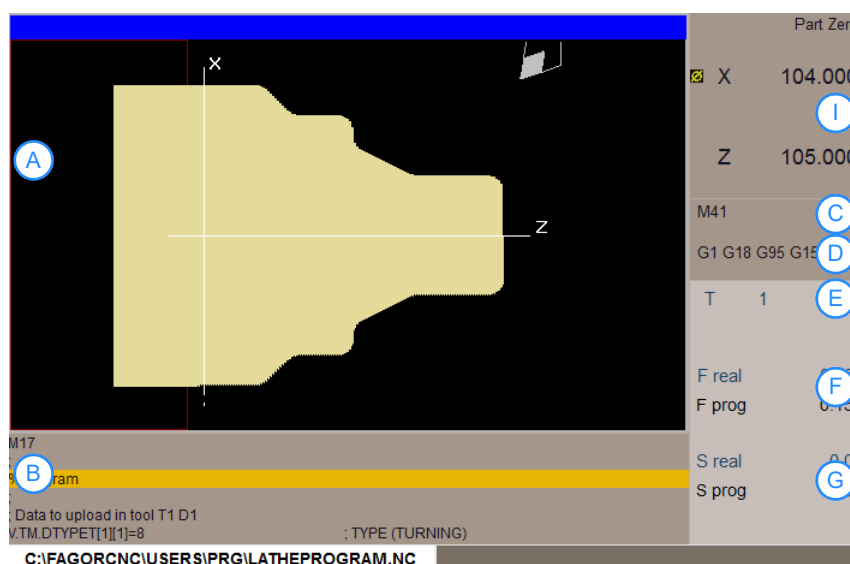


14.1 Description of the graphic environment.

The graphic environment displays a graphic representation of the program that is being executed or simulated and take measurements on the graphics.



Execution graphics.



Simulation graphics.

- A Graphic program representation.
- B Program blocks.
- C Active "M" functions.
- D Active "G" functions and high-level commands.
- E Tool information.
- F "F" axis feedrate.
- G "S" spindle speed.
- H Execution time "Cy Time" of the program.
- I Position of the axes.

Graphic program representation.

Shows a graphic representation of the tool paths or of the part as the program is being executed or simulated.

Program blocks.

It shows data on the selected program for execution and selects the first and final execution blocks. During execution, the cursor shows the block being executed.

Tool information.

- Number of the active tool "T".
- Active "D" tool offset.

"F" axis feedrate.

- Simulation graphics.
 - "F real" real feedrate.
 - "F prog" programmed feedrate.
- Execution graphics.
 - "F real" real feedrate.
 - "F prog" programmed feedrate.
 - Feedrate percentage active on the CNC (jog key, program or PLC).
 - Dynamic percentage "Dyn" machining.
 - The text "F real" appears in red indicating that the PLC is inhibiting the movement of the axes (FEEDHOL mark active).

"S" spindle speed.

- Simulation graphics.
 - Real speed "S real" ("S1 real", "S2 real", etc).
 - Programmed speed "S prog" ("S1 prog", "S2 prog", etc).
 - If the text "Sreal" appears in red, it means that the PLC is inhibiting the movement of the spindle (INHIBIT mark active).
- Execution graphics.
 - Real speed "S real" ("S1 real", "S2 real", etc).
 - Programmed speed "S prog" ("S1 prog", "S2 prog", etc).
 - Active speed percentage for the CNC (JOG keypad, program or PLC).
 - If the text "Sreal" appears in red, it means that the PLC is inhibiting the movement of the spindle (INHIBIT mark active).
 - The screen only shows the data on one spindle. If there are several spindles in the channel, the data on the next spindle may be displayed by pressing the "S" key.

Position of the axes.

- Simulation graphics.
 - Current position of the axes as compared to part zero.
- Execution graphics.
 - 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).

14.

GRAPHIC ENVIRONMENT (LATHE MODEL)

Description of the graphic environment.

FAGOR
AUTOMATION 
CNCelite
8058 8060
8065 8070

REF: 2508

Graph orientation.

For 3-D graphics, the user can modify the orientation (point of view) of the displayed graph.

Key.	Meaning.
	Select a new point of view.
	Assumes the new point of view selected.
	Cancels the selected point point of view.

14.

GRAPHIC ENVIRONMENT (LATHES MODEL)

Description of the graphic environment.

14.1.1 Softkey menus.

Vertical softkey menu.

The vertical softkey menu shows the options related to the operating mode that called upon the graphic environment (EDISIMU mode for simulation, automatic mode for execution, etc.).

Horizontal softkey menu (graphic area).

When selecting this area, the horizontal softkey menu shows the necessary options for interacting with graphics.

Softkey.	Description.
"Type of graphics"	Select the type of graphics.
"ZOOM"	Enlarge or reduce the whole graph shown or part of it.
"Dimensions"	Define the size of the graphics representation.
"Measurement"	Measure the distance between two points.
"Clear screen"	Clear the screen or delete the graphics displayed.
"Colors"	Set the appearance of the graphics on display.
"Options"	Set the appearance and some options of the graphic window.
" +Real"	Display either the real or the theoretical tool path.
" +Error"	Display the real path, but enlarging the error with respect to the theoretical path. Pressing this softkey also activates the one corresponding to the real path.
"Speed"	Change the simulation speed.

14.

GRAPHIC ENVIRONMENT (LATHE MODEL)

Description of the graphic environment.

14.2 Type of graphics

When selecting this option, the softkey menu shows the types of graphics available. The various types of graphics may be grouped into line graphics (XZ, XC, ZC and combined) and solid graphics (Solid XZ, Solid XC, Solid ZC). Line graphics show the tool path with lines of different colors and solid graphics show an image of the part.

The type of graphics selected will remain active until another type is selected or the graphic display is deactivated or the CNC is turned off. Likewise, when changing the type of graphics, the CNC will maintain the graphic conditions (zoom, graphic parameters, display area, etc.), defined for the last graphic.

"XZ" - "XC" - "ZC" graphics

This type of graphic displays the tool paths in the XZ, XC or ZC plane.

"Combined" graphics

This type of graphic divides the display area in four quadrants and displays the tool path corresponding to each plane planos XZ, XC or ZC.

"Solid 3D" Graphics

This type of graphics displays a three-dimensional graph of the machining of the part. Starting out with a solid block which is "machined" as the program is executed or simulated.

14.

GRAPHIC ENVIRONMENT (LATHE MODEL)

Type of graphics



14.3 Zoom

The zoom option may be used to enlarge or reduce the whole graph shown or part of it. This option is not available in the "Combined" type of graphics.

After selecting the "Zoom" option, a zoom frame will appear over the graphics. This frame may be enlarged, reduced and moved around over the graphics already displayed in order to select a particular portion of it to zoom into or out of.

Key.	Meaning.
   	To move the zoom frame around.
 	To change the size of the zoom frame.
	To validate and display the selected zoom area.

A window is displayed on the graphics at the lower right-hand side of the screen. This window indicates the graphic area selected with the zoom.



When selecting this option, the softkey menu shows the available zoom options. To return to the main menu, press the [BACK] key.

Zoom "Initial"

This option restores the size of the display area set via program or using the "Dimensions" option.

Zoom "Automatic"

The CNC uses the zoom that it considers best according to the movements programmed.

Zoom "Previous"

This option displays up to two zooms defined earlier. After the second one, it shows again the one defined last.

Zoom "Edit"

It is used to manually edit the zoom values. It is edited in the dialog area of the graphic window that shows the dimensions of the zoom frame

14.

Zoom

GRAPHIC ENVIRONMENT (LATHE MODEL)

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

14.4 Dimensions

It is used to define the size of the graphic representation by setting the inside and outside diameters of the part and the maximum and minimum coordinates of the graphics on the longitudinal axis.



When selecting this option, the softkey menu shows the options available for setting the dimensions. To return to the main menu, press the [BACK] key.

Dimensions "Automatic"

The CNC sets the dimensions that it considers best according to the movements programmed.

Dimensions "Edit"

The CNC lets manually edit the dimension values. It is edited in the dialog area of the graphic window that shows the dimensions of the graphics.

14.5 Measurement

This softkey may be used to measure the distance between two points. This option is not available in the "Combined" type of graphics.

When selecting this option, the section being measured will appear on the graphics with two cursors and a dashed line. The cursor currently selected will appear in red.

Key.	Meaning.
	To move the zoom frame around.
	To change the size of the zoom frame.

The dialog area will show the coordinates of both cursors, the distance between them on the straight line and the components of that distance on the axes of the active plane. The coordinates of the selected cursor will appear in red.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the [BACK] key.

Measurement "Point1" & "Point2"

This option is used to select the cursor to be moved (same as using the [+] key).

Measurement "Edit"

This option is used to manually edit the position of the cursors. It is edited in the dialog area of the graphic window that shows the position of both cursors.

14.6 Clear screen

This softkey is used to clear the screen or delete the graphics displayed. If a solid graphic type is selected, the graphic representation will be reset and it will return to its initial state without machining.

14.7 Colors

This softkey is used to change the colors used in the graphic representation.

- In the line graphics, it is possible to choose the color for each type of tool path; the color for rapid movements, paths with tool compensation, etc.

The real coordinates are only available for the execution of the program. The real coordinate is the actual position of the tool which differs from the command coordinate in the amount of following error (axis lag).

- In solid graphics, it is possible to choose the color of the solid, of the tool, etc.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the [BACK] key.

Colors "Apply"

It assumes the new colors and applies them to the blocks drawn next. If the new colors are not applied, the graphics are drawn with the old colors.

Colors "Edit"

It is used to select the new colors for the graphics. They are selected in the dialog area of the graphic window that show the current colors.

Key.	Meaning.
	To move the cursor through the data.
	To open and close the color palettes.
	To move the cursor through the colors of the palette.
	To cancel the selected colors.

14.

GRAPHIC ENVIRONMENT (LATHE MODEL)

Clear screen

14.8 Options

This softkey is used to set the appearance and some functions of the graphic window. These options may be used at any time, even while executing a program.



When selecting this option, the softkey menu shows the available options. To return to the main menu, press the [BACK] key.

Option "Activate"

This softkey may be used to activate the graphics. The softkey appears pressed when this option is enabled. The status of this softkey cannot be modified while executing or simulating a program.

When the graphic representation is deactivated and activated, the current graphic is erased; but the display conditions are kept active (type of graphics, zoom, graphic parameters and display area) that were active before that mode was deactivated.

Option "Simple"

This softkey shows the single window for graphics. The softkey appears pressed when this option is enabled.

The single window hides the dialog and data areas on the right-hand side of the graphic window so the drawing occupies the whole graphic window.

Option "Lines"

This softkey hides the solid part of the graphics and only shows the tool paths. The softkey appears pressed when this option is enabled.

Option "Print"

This softkey may be used to print the graphics in the pre-determined printer or save it as a file (bmp format) at the CNC. When selecting the "File" option, it will be saved in the folder "C:\Cnc8070\Users\Reports\"; the file name may be selected using the "Print configuration" softkey.

When selecting this option, the CNC will show a dialog box requesting the print destination (printer or file). After selecting the destination, press [ENTER] to print it or [ESC] to cancel it.

Option "Print setup"

It is used to set the printing configuration. When selecting this option, the CNC shows a dialog box where the following may be defined:

- The title of the graphics that will appear next to it in the print.
- The name of the file where the graphics will be stored when printing out to a file.

After filling out the data, press [ENTER] to accept them or [ESC] to cancel them.

14.

GRAPHIC ENVIRONMENT (LATHE MODEL)

Options



14.9 Real coordinates

This softkey is used to draw the real tool path or the theoretical tool path. This option is only available in line type graphics.

This option is only available when executing the program; not when simulating it. When selecting this option (the softkey will appear pressed), the CNC draws the actual (real) tool path.

14.10 Real coordinates with enlarged error

This softkey shows the real path, but enlarging the error with respect to the theoretical path. The error enlargement factor may be set after pressing the softkey.

This option is only available when executing the program; not when simulating it. When selecting this option (the softkey will appear pressed), the CNC draws the actual (real) tool path with enlarged error. Pressing this softkey also activates the one corresponding to the real path.

14.11 Simulation speed

This softkey is used to change the simulation speed.



When selecting this option, the softkey menu shows the options to set the simulation speed. To return to the main menu, press the [BACK] key.

This option is only available when simulating the program; not when executing it.

Speed "Edit"

It is used to select the new simulation speed. It is selected using the graduated ruler that indicates the active simulation speed.

Key.	Meaning.
 	To move the ruler cursor.

14.

GRAPHIC ENVIRONMENT (LATHE MODEL)

Real coordinates

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

14.

GRAPHIC ENVIRONMENT (LATHE MODEL)

Simulation speed



CNCelite
8058 8060
8065 8070

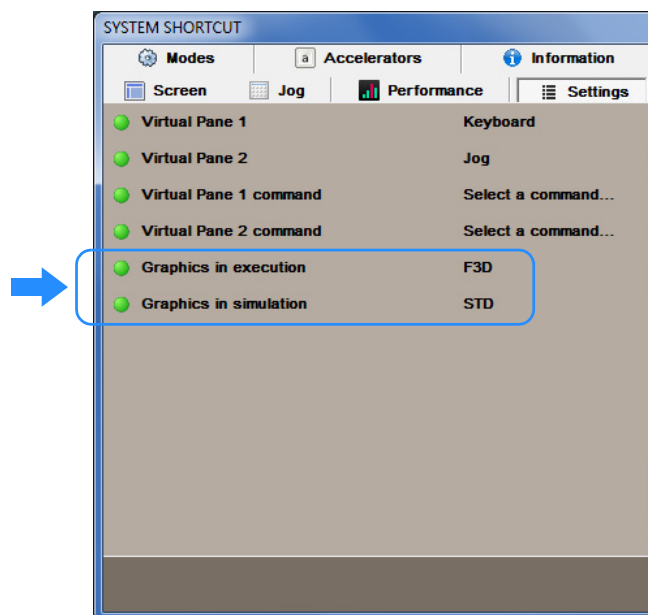
REF: 2508

15.1 Graphic environment.

High definition solid 3D graphics for the execution and simulation of part-programs and canned cycles of the editor. During machining, the HD graphics display the tool removing the material from the part in real time, allowing for the condition of the part to be seen at all times. HD graphics can display up to 4 views of the part, where each can be rotated, zoomed in or zoomed out. Measurements can also be made on the part and even sections on the piece from any angle.

HD graphics and standard Fagor graphics.

Using active HD graphics, the standard Fagor graphics can be accessed through the task window [CTRL] + [A], "Settings" tab.



15.1.1 HD graphics in automatic mode.

The automatic mode displays a graphical representation of the program being executed.

15.

GRAPHICS HD (.T. MODEL).

Graphic environment.

The screenshot shows the automatic mode interface. The main area displays a 3D model of a part being machined. The vertical softkey menu on the right contains the following items:

- A**: The graphical area displays a graphical representation of the tool paths, the solid or both.
- B**: Program or executed subroutine.
- C**: Active functions "M" and "G".
- D**: Tool number "T" and tool offset "D".
- E**: Real feedrate (F real), programmed feedrate (F prog) and axes override. Dynamic (Dyn) override.
- F**: Real speed (S real), programmed speed (S prog) and spindle override.
- G**: Execution time (Cy Time) of the program.
- H**: Programmed coordinate (Command) and axis position regarding the part zero referring to the tool tip.

Vertical softkey menu.

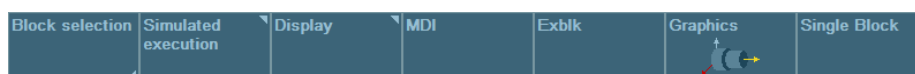
The vertical softkey menu displays the options depending on the operating mode calling the graphic environment; in this case, automatic mode. Refer to the chapter on "Automatic mode" for more details on the functionality of these softkeys.

Horizontal softkey menu.



The horizontal softkey menu changes depending on the area in which the focus is located; graphics area or program area. Use the [FOCUS] key to change the focus area.

- When the focus is on the program area, the softkey menu displays the options to manage the program execution. The softkey menu is the same as the automatic mode. Refer to the chapter on "Automatic mode" for more details on the functionality of these softkeys.

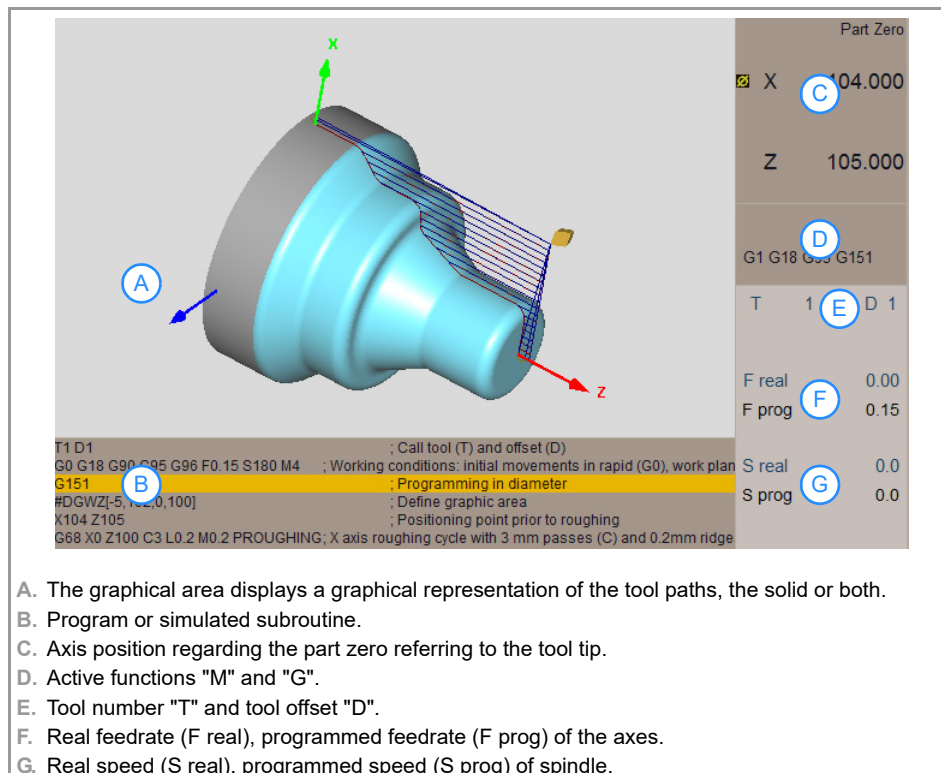


- When the focus is on the graphics area, the softkey menu displays the options to work with the graphics. The softkey menu is similar to the graphics of the edisimu mode.



15.1.2 HD graphics in edisimu mode.

The edisimu mode displays a graphical representation of the program being simulated.



Vertical softkey menu.

The vertical softkey menu displays the options depending on the operating mode calling the graphic environment; in this case, edisimu mode. Refer to the chapter on "edisimu mode" for more details on the functionality of these softkeys.

Horizontal softkey menu.



The horizontal softkey menu changes depending on the area in which the focus is located; graphics area or program area. Use the [FOCUS] key to change the focus area.

- When the focus is on the program area, the softkey menu displays the options used to manage the program simulation. The softkey menu is the same as the edisimu mode. Refer to the chapter on "edisimu mode" for more details on the functionality of these softkeys.



- When the focus is on the graphics area, the softkey menu displays the options to work with the graphics. The softkey menu is similar to the graphics of the automatic mode.



15.1.3 Horizontal softkey menu.

Softkey.	Function.
	Type of view. - Select the type of view. - Select the parts to view.
	Configuration. - Configuring the graphics window. - Configuring and activating the sections. - Configure the colors of the tool path and solid. - General configuration of the graphics. - Cancel the graphics. - Load a machine configuration (xca file).
	Actions. - Move sections. - Print the diagram.
	Delete. Removes the tool paths and resets the solid to its initial dimensions.
	Dimensions. - Defines the type and size of the parts. - Deletes parts. - Saves and deletes parts.
	Measurement. Measure the distance between two points.
	Ver. - Views the tool paths and the solid. - Simulation speed.

15.

GRAPHICS HD (.T. MODEL).

Graphic environment.

15.2 Move, rotate and zoom in or out on the graphic.

When no option has been selected on the softkey menu, it is possible to act upon the graphic as follows.

Move the graphic.

To move the part of the active view, move the mouse while holding its right button or use the following keys.

Key.	Meaning.
 	Move the part to the left or to the right.
 	Move the part up or down.

Rotate the graphic.

To rotate the part of the active view, move the mouse while holding its left button or use the following keys [[↑][↓]][←][→].

Zoom the graphic.

To zoom in or out on the active view, turn the mouse wheel or use the [+][-] keys.

15.

GRAPHICS HD (·T· MODEL).

Move, rotate and zoom in or out on the graphic.

15.3 Select the type of view.



This option is used to select the point of view of the part. Since the screen may be divided into 4 windows, the softkey only affects the active window (the one having the focus). The screen may be selected with the tab key or with the keys [1] through [4] (depending on the window to be selected).

- Top view.
- Front view.
- Left view.
- Right view.
- First user view.
- Second user view.
- Save the current view as user view.
- Select the part to be displayed.

User views.

The top view, front view and side views are predetermined views that may be moved or rotated with the mouse or keyboard. The new position and orientation of the part may be saved as a user view. The CNC keeps the saved user views even after being turned off.

Select the part to be displayed.

The graphics may have defined up to four different parts. Each of the four windows that the screen may be divided into may display the same part or different parts. The following softkeys only affect the active window (the one having the focus).

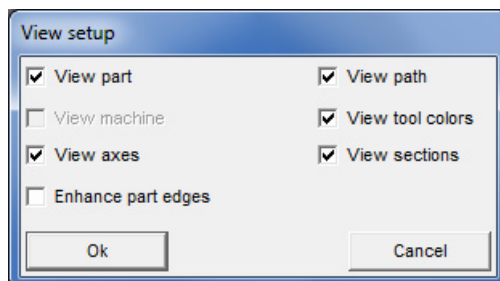
Softkey.	Function.
	General view. In machines with rotary axes, this softkey allows the for the rotating part or tool to be viewed.
	Displays the first part associated with the channel. Each part may be associated with one or more channels. See "15.13 Editing, displaying and hiding parts." on page 278. When changing channels, the displayed part changes. If several channels are working on the same part, all of them can show the same part.
	View part 1. The menu shows a softkey per available part.



15.4 Configuring the graphics window (properties of each window).



This softkey is used to configure the graphic properties of the active window, so each of the four windows dividing the screen may have different properties. For example, a window may show only the part to be machined and another one only the machining tool paths. The active screen may be changed using the tab key or keys [1] through [4] (depending on the window to be selected).



Properties.	Meaning.
View part.	To configure the window to view the part. To view the part, the "View solid" option must be activated using the "View" softkey. See chapter "15 Viewing the tool paths and the solid."
View tool path.	Configure the window to show the tool paths. To view the tool paths, the option "View path" must be activated using the "View" softkey. See chapter "15 Viewing the tool paths and the solid."
View machine.	Not used.
View tool colors.	The diagram shows the tool path of each tool in the color assigned to it. See "15.6 Configure the colors for the tool path and solid." on page 273.
View axes.	The diagram shows the main axes of the origin channel for the part. The origin of the axes coincides with the part zero. See "15.13 Editing, displaying and hiding parts." on page 278.
View sections.	Configure the window to display the active sections. See "15.5 Configuring and activating the sections." on page 272.
Enhance part edges.	The diagram enhances the edges of the part.

15.

GRAPHICS HD (.T. MODEL).

Configuring the graphics window (properties of each window).

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

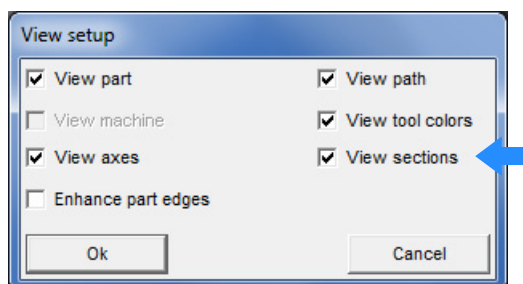
REF: 2508

15.5 Configuring and activating the sections.

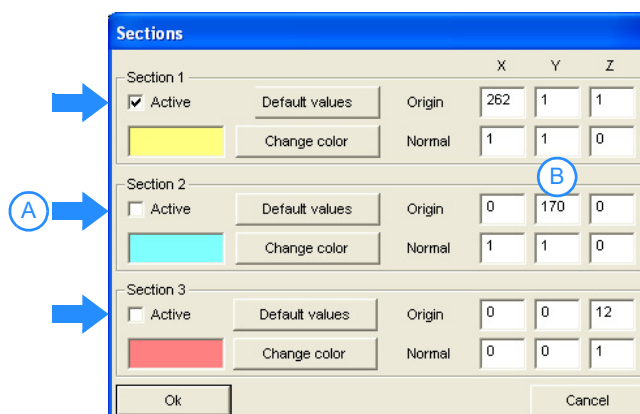


This softkey defines up to three planes used to divide the diagram. The sections are global to all the defined parts. If there is a section defined in X100, it will cut all the parts that reach this coordinate.

Although the definition of the sections is global for all the defined windows, each window may have different sections active. To view the sections in a window, one must also enable the "View sections" option in the window properties. See *"Configuring the graphics window (properties of each window)." on page 271.*



Configuring the sections.



- A. Activate the sections in the active window.
B. Section configuration for all windows.

Define the sections (default values).

The "default values" button defines three sections, each of these are perpendicular to one of the axes and cut the part in half.

Define the sections.

The resulting section will be a plane perpendicular to the vector defined in "Normal" and it goes through the point defined in "Origin". The color of each section may also be defined.

- The "Origin" data defines the origin of the section.
- The "Normal" data defines the components of the vector normal to the plane, that defines the orientation of the plane.

Activate the sections.

Each section may be activated individually.

To view the sections on the graphic, one must also enable the "View sections" option in the window properties. See *"15.4 Configuring the graphics window (properties of each window)." on page 271.* If the "View sections" option is not enabled, the graphic does not show any section even if they are active. If the "view sections" option is enabled, the diagram shows the active sections.

15.

GRAPHICS HD (.T. MODEL).
Configuring and activating the sections.

15.6 Configure the colors for the tool path and solid.



This softkey is used to configure the colors for the part and for the tool paths. This configuration is global for all the windows.

Path colors.

This option is used to assign colors to the following tool paths.

- Rapid traverse (G0).
- Compensated movements.
- Uncompensated movements.
- Canned cycles.
- Tapping.

Part colors.

This option is used to change the background colors, the colors of the machined areas and the look of the part. The selected values remain until the CNC is turned off.

- Background color.
- Color of the machining operations carried out with the first five tools used on the graphic. Here only the colors are defined, to display them, the "view tool colors" option must be active in the properties of the view.
- Material used to represent the solid part. The "Advanced" button may be used to change the properties of the selected material. Besides the various color components (diffuse, ambient, etc.), the brightness and transparency of the part can also be defined. Transparency may take values between 0 and 1. If the value of the transparency is other than zero, the solid part becomes transparent.

15.

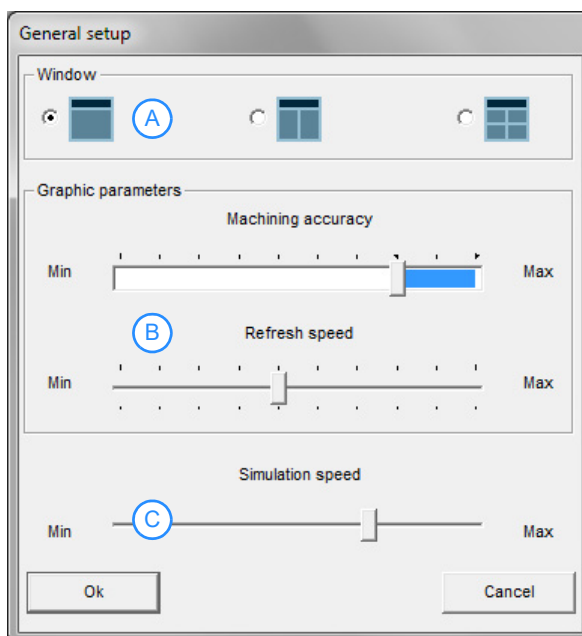
GRAPHICS HD (.T. MODEL).

Configure the colors for the tool path and solid.

15.7 General configuration of the graphics.



This softkey may be used to configure the properties of the graphic environment, those affecting all the windows.



- A. Number of windows.
- B. Graphic parameters.
- C. Simulation speed.

Windows.

The graphic environment may be a full-screen window or may be divided into 2 or 4 windows, each with different properties (for example a different view of the part). The screen may be selected with the tab key or with the keys [1] through [4] (depending on the window to be selected).

Graphic parameters.

These parameters influence the speed and quality of the graphics; the higher the value of these parameters, the greater the graphics quality, but the slower the graphics speed.

- Machining accuracy. This selection bar allows to choose whether the graph considers the machine dynamics (blue zone) or not (white zone). Selecting to include machine dynamics increases the simulation time which depends on the dynamics.
- Refresh speed.

Simulation speed (edisimu mode only).

This bar allows the simulation speed to be determined under edisimu mode. The speed may be modified during the simulation using the "View" softkey. See ["15.18 Simulation speed \(edisimu mode only\)."](#) on page 282.

15.

GRAPHICS HD (.T. MODEL).

General configuration of the graphics.

15.8 Configuration. Cancel the graphics.



This softkey allows the graphics to be deactivated. When the graphics are deactivated, it is not possible to graphically represent the program being executed or simulated. Graphics under automatic and edisimu modes are deactivated independently.

15.9 Configuration. Load machine.



This softkey may be used to load a new machine configuration in HD graphics (xca files). The CNC offers several xca files, one per model, containing the definition and configuration of the machine for HD graphics. On power-up, the CNC assumes the last file used.

If the physical configuration of the machine changes during execution (for example, spindle change with different number of axes), the corresponding xca file must be loaded so the graphics show the changes. The xca files may be loaded either from the softkey menu or from the program using the instruction #DEFGRAPH.

When changing the machine configuration, the CNC saves the part displayed on the screen automatically as LastPiece.stl into the folder ../Users/Grafdata, and it recovers it after the new configuration.



The machine configuration files supplied by Fagor consist of a single file, the xca. These files cover most configurations; therefore, new xca will have to be generated when the machine has some special requirement that affects graphics.

When an OEM creates his own configuration files, for each xca file, he must create a ".def" file with the same name that completes the configuration of the axes involved in the kinematics. Both files must be copied when saving the configuration file in another folder.

15.

GRAPHICS HD (-T- MODEL).
Configuration. Cancel the graphics.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

15.10 Actions. Move the active sections.



This option may be used to move the active sections. To move the sections, the active view must have the "view sections" option enabled and it must also have a section active. To end the possibility to move sections, press the [ESC] key or press the same softkey again. With this option active, the bottom of the screen shows a message indicating the active section and its data.

Select the section to move.

One of the sections is always selected (the one highlighted in orange). To change the section to be moved, use the tab key or with the [1] [2] [3] keys (depending on the section number). The section to be moved appears in orange and the rest in the color they have been defined with.

Move sections.

- If the section to be moved is perpendicular to one of the axes of the trihedron, like the default sections, the [↑][→] keys move the section in the positive direction of the axis and the [↓][←] keys move it in the negative direction of the axis.
- If the section is not perpendicular to any of the axes of the trihedron, it is necessary to select one of the axes of the trihedron (hold one of the keys [X] [Y] [Z] down) and use the [↑][→] keys move the section in the positive direction of the axis and the [↓][←] keys move it in the negative direction of the axis. In the jog mode it is not possible to move these sections, as the selection of axes applies to movements in jog.

15.

GRAPHICS HD (.T. MODEL).

Actions. Move the active sections.



15.11 Actions. Print the graphic.



This option may be used to print the graphic out to the predetermined printer or out to a file.

Print the graphic.



This softkey may be used to print the graphics in the pre-determined printer or save it as a file (bmp format) at the CNC. When printing the graphic to a file, it will be saved in the folder "..\Users\Reports\"; the file name may be selected using the "Print configuration" softkey.

When selecting this option, the CNC will show a dialog box requesting the print destination (printer or file). After selecting the destination, press [ENTER] to print it or [ESC] to cancel it.

Print configuration.



This softkey may be used to set the printing properties. When selecting this option, the CNC shows a dialog box where the following may be defined:

- The title of the graphics that will appear next to it in the print.
- The name of the file where the graphics will be stored when printing out to a file.

After filling out the data, press [ENTER] to accept them or [ESC] to cancel them.

15.12 Delete the graphic.



This softkey deletes the tool paths and resets the solid to its initial dimensions.

15.

GRAPHICS HD (-T- MODEL).

Actions. Print the graphic.

15.13 Editing, displaying and hiding parts.



This softkey allows up to four pieces to be defined, either rectangular or cylindrical, and to be assigned to various channels.

Rectangular part.

Part number: (A) 1 Reference origin: (B) C1

Part type: (C) RECT

X Min: -73.9 X Max: 140.9

Y Min: (E) -67.2 Y Max: 67.2

Z Min: -32 Z Max: 112

Associated with: ☒ C1 (F) ☐ C2 ☐ C3

Work with the part

☒ (G) ☐ (H)

Ok Cancel

Cylindrical part.

Part number: (A) 1 Reference origin: (B) C1

Part type: (C) CYL Part axis: (D) X

X Min: (E) -65 X Max: 130

Inside diameter: -65 Outside diameter: 65

Associated with: ☒ C1 (F) ☐ C2 ☐ C3

Work with the part

☒ (G) ☐ (H)

Ok Cancel

- A. Part number.
- B. Reference channel; used to define the origins and the dimensions.
- C. Part type; rectangular or circular.
- D. Longitudinal axis of the part (cylindrical parts only).
- E. Part dimensions. To include a zero offset in the part, edit the zero offset at the minimum values of the axes.
- F. Channels associated with the part, for displaying. Since several channels may work on a part, in order to be able to keep displaying the machining of that part when changing channels, it is necessary to define which channels are working on that part.
- G. Show the part
- H. Hide the part.



The dimensions of the part correspond to the part zero (G92, G54 to G59, G159) active at the time of the activating the part, for this reason it is advisable to first activate the part zero and then the part.

15.

GRAPHICS HD (-T- MODEL).

Editing, displaying and hiding parts.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

15.14 Automatic dimensions.



The CNC erases the graphics and sets the dimensions that it considers as optimal, depending on the programmed movements, in the four windows dividing the screen.

15.15 Saving part / Loading part.



These softkeys may be used to save and load the parts as stl files (STereoLithography) to be used in a quick prototyping (for example, 3D printers). The stl files are saved in the folder ../Users/Grafdata.

In a machine configuration file exchange (xca file), the CNC automatically saves the current part as LastPiece.stl and retrieves it after activating the new machine configuration.

15.

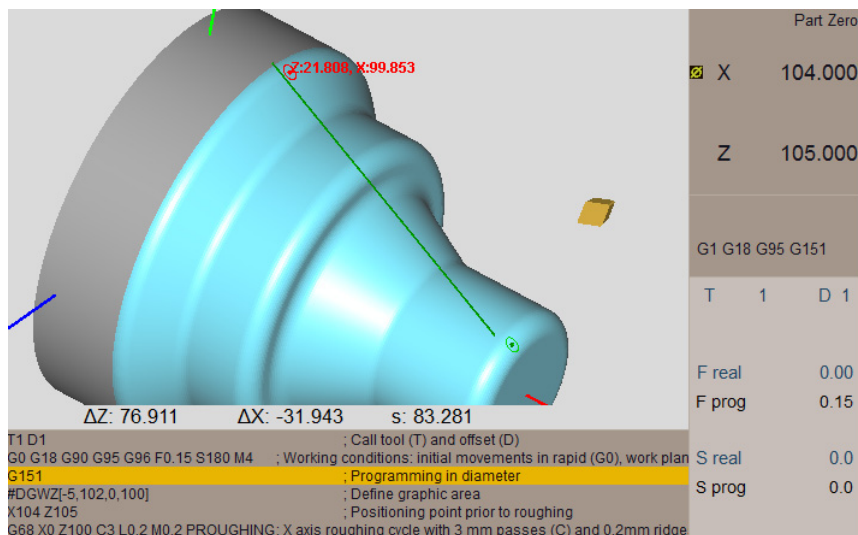
GRAPHICS HD (-T- MODEL).
Automatic dimensions.

15.16 Measure the part.



This option may be used to measure the distance between two points. After selecting this option, the line drawn between two measuring points above the diagram and the distance between the both points on each of the axes (ΔX , ΔY , ΔZ) and the distance in a straight line is shown below. Once measured, press the [ESC] key or press the same softkey again.

If the graphics environment is divided into several windows, the measuring points are only visible in the active window.



For 3D views, the movement of the measuring points is limited to the part. To measure distances beyond the part (tool paths), a 2D view must be selected (top, front, left or right).

Moving the measuring points.

The active measuring point, indicated in red, can be moved with the mouse or keyboard. It is recommended to use the mouse.



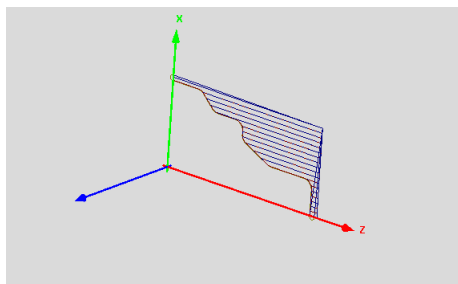
To change the active measuring point, use the tab key or place the cursor over it.

- Use the keys [$\uparrow$][$\downarrow$][$\leftarrow$][$\rightarrow$] to move the active measuring point.
- Use the left mouse button to drag the measuring points.
- Click the mouse on any point on the screen, the active measuring point moves to that point automatically.

15.17 Viewing the tool paths and the solid.

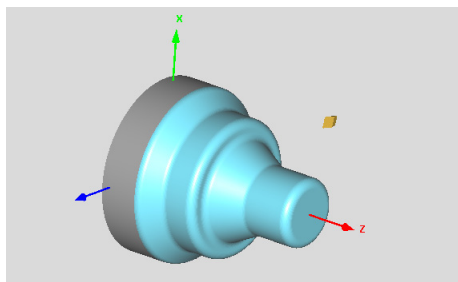


This softkey toggles between the various types of diagrams. Each of the four windows dividing the screen may display the diagram in a different way; to do this, the options "View part" and "View path" must be selected in the window properties. See ["Configuring the graphics window \(properties of each window\)."](#) on page 271.



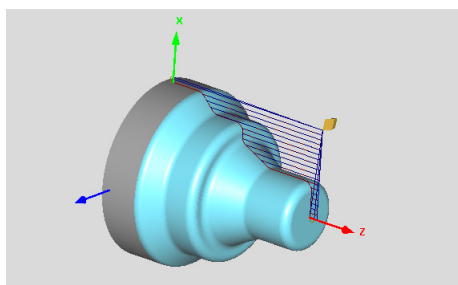
Viewing tool paths.

This option only displays the tool paths. The colors of the tool paths can be changed for the softkey "Configuration" > "Colors" > "Path".



Viewing the solid.

This option only shows the solid to be machined. Each surface is represented by a color associated with the corresponding machine tool. The colors of the tools can be changed for the softkey "Configuration" > "Colors" > "Solid".



Viewing tool paths and solids.

This option only displays the solid to be machined and the tool paths.

15.

GRAPHICS HD (.T- MODEL).

Viewing the tool paths and the solid.

FAGOR
AUTOMATION

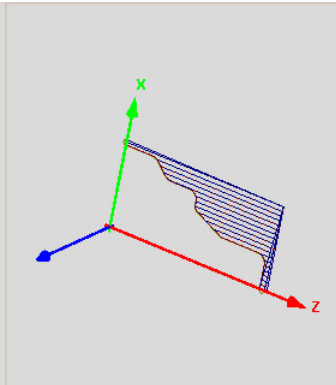
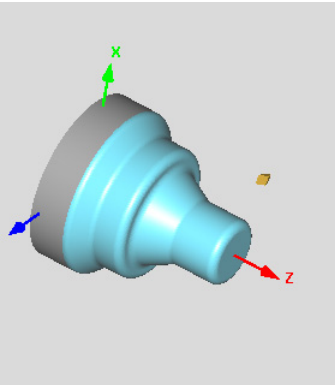
CNCelite
8058 8060
8065 8070

REF: 2508

15.18 Simulation speed (edisiimu mode only).



This softkey allows the simulation speed to be changed under edisiimu mode. After selecting this option, the CNC displays a cursor for setting the simulation speed. To move the cursor, use the mouses or the [SHIFT][←] or [SHIFT][→] keys.



Part Zero

X	104.000
Z	105.000

G1 G18 G95 G151

T	1	D	1
F real	0.00		
F prog	0.15		
S real	0.0		
S prog	0.0		

Simulation speed ————— 70 ————— +

T1 D1 ; Call tool (T) and offset (D)
G0 G18 G90 G95 G96 F0.15 S180 M4 ; Working conditions: initial movements in rapid (G0), work plan
G151 ; Programming in diameter
#DGWZ[-5,102,0,100] ; Define graphic area
X104 Z105 ; Positioning point prior to roughing
G68 X0 Z100 C3 L0.2 M0.2 PROUGHING; X axis roughing cycle with 3 mm passes (C) and 0.2mm ridge

15.

GRAPHICS HD (·T· MODEL).

Simulation speed (edisiimu mode only).



CNCelite
8058 8060
8065 8070

REF: 2508

The MDI mode lays over all the other work modes in such a way that when quitting the MDI mode by pressing [ESC], the CNC goes into the work mode from where the MDI mode was accessed.

The channel does not allow accessing its MDI/MDA mode if a program is being executed in 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.

16.1 Interface description.

Standard window (MDI).

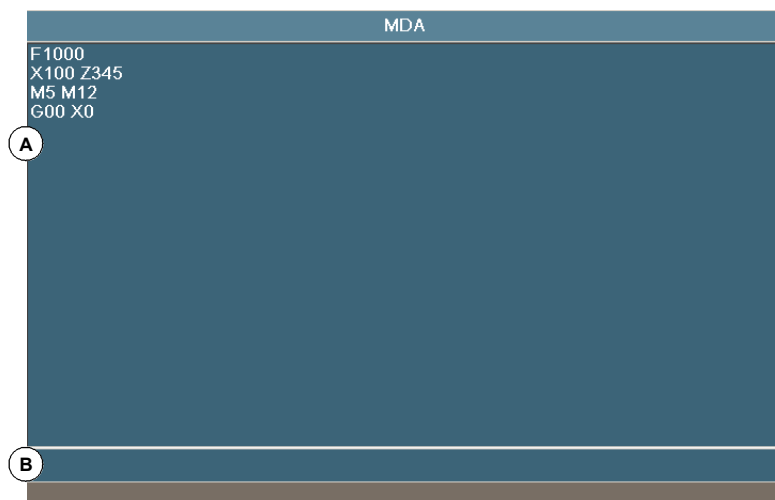
This window only shows the edit line of the MDI mode. In the MDI window, it is possible to edit and execute new blocks or recover blocks saved in the block history, which may be modified before executing them.



- 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.

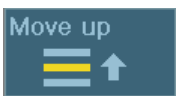


- A History of blocks in MDI mode. Every time a new block is edited, it is added to this history.
B Edit line where the blocks to be executed are edited. Blocks are edited one by one.

16.1.1 Softkey menus.

Horizontal softkey menu.

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. Standard window (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.

16.2 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 pop-up the history and scroll 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.

16.

MDI/MDA MODE

Edit and execute individual blocks.

16.3 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

To save the block history as an independent program, press the "Save" softkey and the CNC will show a list of the programs saved at the CNC.

- 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.

16.**MDI/MDA MODE**
Block history.

17.1 User table presentation.

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.

Some tables are common to all the channels and others belong to each channel. In this case, by default, they show the ones of the active channel; but it is possible to access those of any other channel from the vertical softkey menu.

The zero offset tables and fixture offset tables are common to all the channels; however, in each channel they show the axes of that channel. When applying an offset in a channel, it is only applied to the axes that are part of the channel at the time. In order to activate a fixture zero or part zero offset, those values must be previously stored in the relevant CNC table.

17.1.1 Softkey menus.

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.
	Accessing the tables of other channels. With some tables, only the data of the active channel are displayed, this softkey is used to show the tables of the other channels. This softkey will only be available when using channels.
	Select the axes (zero offset tables) and channels (parameter tables) to display in the tables. When using several channels, only those axes assigned to the active channel may be accessed.
	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.

17.

USER TABLES

User table presentation.

17.2 Zero offset tables

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

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 table shows the axes and spindles that are in the channel at the time; in other words, after swapping axes or spindles between channels, the CNC updates the table. The zero offsets associated with the possible C axes are always visible, even when the C axis is not active.



When accessed from the a channel, the table only shows the axes and spindles of that channel. The vertical softkey menu may be used to access the zero offsets of the rest of the channels.

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.

17.

USER TABLES
Zero offset tables

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 Y (mm)	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
B G56 (G159=3)	00000.0000	00000.0000	00000.0000
G57 (G159=4)	00000.0000	00000.0000	00000.0000
G58 (G159=5)	00000.0000	C 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
G159=10	00000.0000	00000.0000	00000.0000
G159=11	00000.0000	00000.0000	00000.0000
G159=12	00000.0000	00000.0000	00000.0000

- 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 Y (mm)	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
B Δ	00003.0000	00000.0000	00000.0000
G55 (G159=2)	00000.0000	00000.0000	00000.0000
C Δ	00000.0000	00000.0000	00000.0000
G56 (G159=3)	00010.0000	00000.0000	00000.0000
Δ	00000.0000	00000.0000	00000.0000
G57 (G159=4)	00000.0000	D 00000.0000	00000.0000
Δ	00000.0000	00000.0000	00000.0000
G58 (G159=5)	00000.0000	00000.0000	00000.0000
Δ	00000.0000	00000.0000	00000.0000
G59 (G159=6)	00000.0000	00000.0000	00000.0000
Δ	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.

17.

USER TABLES
Zero offset tables

17.3 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.



When accessed from the a channel, the table only shows the axes of that channel. The vertical softkey menu may be used to access the zero offsets of the rest of the channels.

Channel 1 : Fixture offsets			
Fixture	X (mm)	Y (A)	Z (mm)
1	00000.0000	00000.0000	00000.0000
2	00000.0000	00000.0000	00000.0000
3	00000.0000	00000.0000	00000.0000
4	00000.0000	00000.0000	00000.0000
(B)	00000.0000	00000.0000	00000.0000
6	00000.0000	00000.0000	00000.0000
7	00000.0000	00000.0000	00000.0000
8	00000.0000	0 (C)	00000.0000
9	00000.0000	00000.0000	00000.0000
10	00000.0000	00000.0000	00000.0000

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.

17.

USER TABLES

Fixture table

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

17.4 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.

Local P parameters 1		
Parameter	CH1	CH2
P0 - A	0	0
P1 - B	0	0
P2 - C	0	0
P3 - D	0	0
P4 - E	0	0
P5 - F	0	0
P6 - G	0	0
P7 - H	0	0
P8 - I	0	0
P9 - J	0	0
P10 - K	0	0
P11 - L	0	0
P12 - M	0	0
P13 - N	0	0

A Parameters of channel 1.

B Parameters of channel 2.

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	Value	Comment
P10000	101.965800	Comments
P10001	100	Comments
P10002	23	Comments
P10003	1	Comments
P10004	0.168541	Comments
P10005	1.002580	Comments
P10006	-5813	Comments
P10007	-12.025800	Comments
P10008	0	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 UCPCComments.txt and it is possible to have one file per language. These files are saved in the folder "../MTB /data /Lang".

17.

USER TABLES

Arithmetic parameter tables

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 "N") in such a way that "A" is the same as P0 and "Z" is the same as P25. That is why the local parameter tables show the parameter number next to their associated letter.

The parameter values may be set directly in the table or from the PLC or via 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).

17.

USER TABLES

Arithmetic parameter tables



CNCelite
8058 8060
8065 8070

REF: 2508

17.5 Active offsets table.

Within the user tables, the "Active Origins" table displays the status of all functions that affect the part zero (G92, G159, G201, etc). When 5-axis (or 3+2) transformations are active, it also displays information relevant to the construction of the plane.

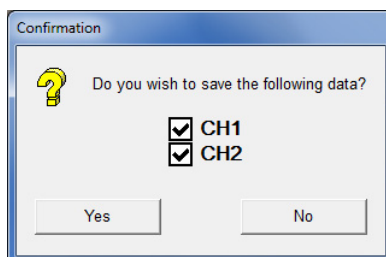
Channel 1 : Active Offsets							
	X (mm)	Y (mm)	Z (mm)	C (deg.)	A (deg.)	U (deg.)	V (deg.)
PLCOF	00000.0000	00000.0000	00000.0000	00000.000	00000.000	00000.000	00000.000
KINTIP=ON							
RTCP HEAD=DYN							
RTCP TABLE=ON							
HEAD=3	00000.0000	00000.0000	00000.0000	00000.000	00000.000	00000.000	00000.000
FIX=0	00000.0000	00000.0000	00000.0000	00000.000	00000.000	00000.000	00000.000
ACS	00000.0000	00000.0000	00000.0000				
ROT	00000.0000	00000.0000	00000.0000				
G159=1	-0200.0000	-0100.0000	00100.0000	00000.000	00000.000	00000.000	00000.000
Δ	00000.0000	00000.0000	00000.0000	00000.000	00000.000	00000.000	00000.000
G158	00000.0000	00000.0000	00000.0000	00000.000	00000.000	00000.000	00000.000
CS	00000.0000	00000.0000	00000.0000				
ROT	00000.0000	00000.0000	00000.0000				
G92	00000.0000	00000.0000	00000.0000	00000.000	00000.000	00000.000	00000.000
G101	00000.0000	00000.0000	00000.0000	00000.000	00000.000	00000.000	00000.000
G201	00000.0000	00000.0000	00000.0000	00000.000	00000.000	00000.000	00000.000
PLANE=G17							
MIRROR	00000.0000	00000.0000	00000.0000				
SCALE ORG	00000.0000	00000.0000	00000.0000	00000.000	00000.000	00000.000	00000.000

Vertical softkey menu.

Softkey.	Description.
	Select the zero offsets to be displayed in the tables. The selection affects all the channels.

17.6 Operations with tables

If the arithmetic parameter tables show the parameters of several channels, the softkeys for initializing, loading, saving and printing the arithmetic parameter tables ask the user to select the channels.



17.

USER TABLES

Operations with tables

17.6.1 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.

Key.	Meaning.
   	Move the cursor.
 	Move the cursor page by page.
 	Move the cursor to the beginning / end of the table.
	Accept the value.
	Reject the value.

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.

17.6.2 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.

17.

USER TABLES
Operations with tables



17.6.3 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.

Find

Find text

Origin

From begining

From cursor

Key.	Meaning.
	Move the cursor.
	It selects an option from a group.
ENTER	Start the search.
ESC	Cancel the search and close the dialog box.



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.

17.

USER TABLES

Operations with tables



CNCelite
8058 8060
8065 8070

REF: 2508

18.1 Presentation of the tool tables and magazine tables.



This operating mode consists of several tables. The various tables may be selected using the horizontal softkeys. If while one of the tables is selected, the [BACK] key is pressed, that table will be deselected.

- Tool table.
- Active-tools table.
- Table for the status of the tool change process.
- Tool magazine tables.

In order to load a tool in the magazine or in the spindle, that tool must have been previously defined in the corresponding table of the CNC.

18.1.1 Softkey menus.

Horizontal softkey menu.

Softkey	Table
	Tool table. This table defines the tools available and the data associated with each one of them. This table is common to all the magazines.
	Active-tools table. This table shows the tool that is active in each channel and the data associated with it.
	Change process. This table monitors the tool changes being executed in each channel.
 	Tool magazine table (there is one table per magazine) For each magazine, it shows the tool distribution and the remaining life time of each tool (if tool life monitoring is active). If the magazine has a changer arm, it shows the tool located in it. The description of the magazine type can also be shown. The icon associated with this table depends on the software configuration (lathe or mill).

When selecting one of these tables, it is displayed on the screen and the vertical icon menu shows the icons associated with that table. Later sections of this chapter show a more detailed description of the icons and operations that may be carried out in each table.

Vertical softkey menu.

If no table is currently selected, the vertical softkey menu will show the following softkeys to save or load all the tables. See ["18.1.3 Save and load the tables"](#) on page 302.



Save the tables

Saves the data of all the tables.



Recall the tables

It restores the data previously saved for all the tables.

18.1.2 Search for a text in the tables

It is possible to search for a text or a value in the list of tools and magazine positions. The search is carried out from the vertical softkey menu.



This icon starts the search. Once the icon has been pressed, the CNC will display a dialog box to define the search criteria. The defined criteria is maintained until a new one is defined.



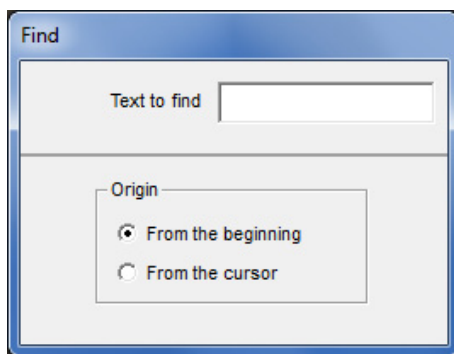
This icon is shown when a search criteria has been defined; it makes it possible to search for the next match using the current search criteria.

To start or resume the search, press [ENTER] and the focus will position on the first match found. Every time one of the icons is pressed, it offers the option to search for the next match or to define a new search. The search may be canceled by pressing the [ESC] key.

Define the search criteria.

Every time one of these icons is pressed, it shows a dialog box to define the search criteria. The following may be defined in this dialog box:

- The text or value to search for.
- The beginning point of the search, namely either from the beginning of the table or from the cursor position.



Key.	Meaning.
	Move the cursor.
	It selects an option from a group.
	Start the search.
	Cancel the search and close the dialog box.

18.

TOOL AND MAGAZINE TABLE

Presentation of the tool tables and magazine tables.

18.1.3 Save and load the tables

From the vertical softkey menu, it is possible to make a backup copy of the table data (recommended). With these files, it is possible to recover the table data when needed.

These tables are saved in the an ASCII file. These files may be saved at the CNC, in a floppy disk or at another device (CNC, PC, etc.) connected through Ethernet. By default, they are saved in the folder "C:\CNC8070\ MTB\ DATA" or in the last folder selected by the user.

These tables are saved in the following files. The table monitoring the change processes is informative only; it is not copied anywhere.

Table	File
Tool table.	Tools.TPT
Active-tools table.	Toolsact.TPA
Tool magazine table (if there is only one magazine).	Magazine.TPM
Tool magazine table (if there are several magazines).	Magazine1.TPM ... Magazine4.TPM

Considerations for saving and loading tables.

Although each table may be saved separately, it is recommended to always have a copy of all the tables. Also, the following must be borne in mind when loading the tables:

- Loading the tool table initializes the magazine tables and the active-tools table. When changing the list of available tools, it may not be coherent with the tool distribution in the magazine or in the spindles. This is why after loading this table it is necessary to define (or load) the magazine tables and the active-tools table, if any, in that order.
- Loading the magazine table initializes the active-tools table. This is because when loading the magazine tables, the new tool arrangement may not be coherent with the active tools. This is why after loading this table, it is necessary to load the active-tools table.

Save the tables

The tables may be saved one by one or all of them simultaneously. In either case, the action is carried out from the vertical softkey menu using one of the following icons.



Saving all the tables at the same time. To save all the tables at the same time, no table must be selected.



Saving each table separately. To save the tables one by one, select each table from the horizontal softkey menu.

After pressing the icon, the CNC will ask where to save the data files. Select the desired folder and press [ENTER]. The selection process may be canceled by pressing the [ESC] key.

Recall the tables

It may be used to restore the data, in ASCII format, of the tables from the files. The tables may be loaded one by one or all of them simultaneously. In either case, the action is carried out from the vertical softkey menu using one of the following icons.



Loading all the tables at the same time. To load all the tables at the same time, no table must be selected.



Loading each table separately. To load the tables one by one, select each table from the horizontal softkey menu. To load the data of all the tables (recommended), follow a particular loading order to guarantee data consistency.

After pressing the icon, the CNC will ask the location of the data files. Select the desired folder and press [ENTER]. The selection process may be canceled by pressing the [ESC] key.

18.

TOOL AND MAGAZINE TABLE
Presentation of the tool tables and magazine tables.

Order (sequence) to follow when loading the different tables.

As mentioned earlier, there are two ways to load the table data from the ASCII files; loading all of them at the same time or selecting each table and loading its data.

Loading all the tables at the same time.

In this case, it is up to the CNC to set the order (sequence) used to load the data.

Order (sequence) to follow when loading the tables one by one.

This way, each table is selected and its data loaded. In this case, the following sequence must be followed when loading the tables.

1 First load the tool table.

This table defines the system tools. Loading the tool table initializes the tables of the magazines and the active-tools table.

2 Second, load the tool magazine tables.

Since the tool list has changed, if the magazine tables are not loaded, it is not possible to assure that the new tools are the ones located in the magazine tool or in the active-tools table.

3 Finally, load the active-tools table.

When loading the tables of magazines, it is unsure that the current tool positions will be compatible with the active tools.

18.

TOOL AND MAGAZINE TABLE

Presentation of the tool tables and magazine tables.

18.1.4 Printing the tables

Some of the tables may be printed in a printer accessible from the CNC or as a file (PRN format). When the tables are saved as a file, it may be saved at the CNC, in a floppy disk or at any other device (CNC, PC, etc.) connected through Ethernet. By default, the files are saved in the folder "C:\CNC8070\ USERS\ Reports".

In either case, the action is carried out from the vertical softkey menu.



This softkey starts printing. When pressing this icon, the CNC will show a dialog box requesting the print destination for the table (printer or file).

After selecting the target, press [ENTER] to start printing. Press [ESC] to cancel the selection.

18.

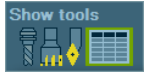
TOOL AND MAGAZINE TABLE

Presentation of the tool tables and magazine tables.



18.2 Tool table

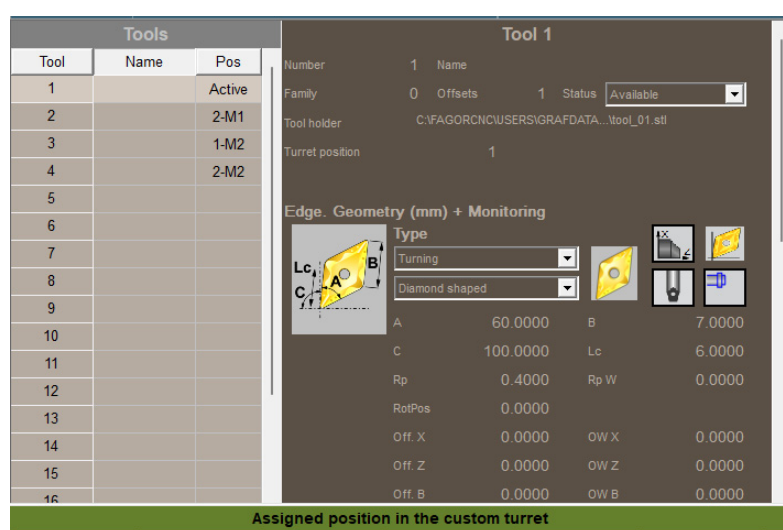
The CNC offers two tool tables. Use the following softkeys to select each one of them.

Softkey	Table
	Softkey available when the tool table is selected in simple mode, it toggles to full mode.
	Softkey available when the tool table is selected in full mode, it toggles to simple mode.

Tool table (full mode)

The table is divided in two panels. The left panel shows the list of the available tools and the right panel shows the data of the tool selected on the list.

This table may be used to set all the tool data. It is also possible to add or remove tools and offsets from the table.



Tools		
Tool	Name	Pos
1		Active
2		2-M1
3		1-M2
4		2-M2
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		

Tool 1			
Number	1	Name	
Family	0	Offsets	1
Status	Available		
Tool holder	C:\FAGORCNC\USERS\IGRAFDATA_tool_01.stl		
Turret position	1		
Edge Geometry (mm) + Monitoring			
Type	Turning		
Diamond shaped	Diamond shaped		
A	60.0000	B	7.0000
C	100.0000	Lc	6.0000
Rp	0.4000	Rp W	0.0000
RotPos	0.0000		
Off. X	0.0000	OW X	0.0000
Off. Z	0.0000	OW Z	0.0000
Off. B	0.0000	OW B	0.0000

Assigned position in the custom turret

Tool table (simple mode)

The table shows the list of tools with most common data. This table can only be used to set some tool data. Unlike the full mode, the simple mode does not allow adding or removing tools or offsets from the table.

M TOOLS: STANDARD										
T	D	Mz	Name	L	L W	R	R W	Rp	Rp W	
1	1	1		75.0000	0.0000	25.0000	0.0000	0.0000	0.0000	
2	1	1		100.0000	0.0000	7.5000	0.0000	0.0000	0.0000	
3	1	1		75.0000	0.0000	5.0000	0.0000	1.0000	0.0000	
4	1	1		70.0000	0.0000	4.0000	0.0000	0.0000	0.0000	
5	1	1		85.0000	0.0000	2.5000	0.0000	0.0000	0.0000	
6	1	1		85.0000	0.0000	2.5000	0.0000	0.0000	0.0000	
7	1	1		80.0000	0.0000	5.0000	0.0000	5.0000	0.0000	
8	1	1		65.0000	0.0000	3.0000	0.0000	0.0000	0.0000	
9	1	1		100.0000	0.0000	10.0000	0.0000	0.4000	0.0000	
10	1	1		85.0000	0.0000	7.5000	0.0000	0.0000	0.0000	
11	1	1		65.0000	0.0000	5.0000	0.0000	0.0000	0.0000	
12	1	1		125.0000	0.0000	12.0000	0.0000	0.0000	0.0000	
13	1	1		135.0000	0.0000	25.0000	0.0000	0.0000	0.0000	

Radius

18.

TOOL AND MAGAZINE TABLE
Tool table

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

About the tools

Each tool is identified by its number, that is unique for the whole system; it cannot be repeated in different magazines nor in ground tools. This table indicates the position and the magazine where each tool is located, whether it is a ground tool or the tool is active in a channel.

Ground tools

A ground tool is a tool that is not stored in any magazine and is loaded manually when requested. Ground tools are also defined in the tool table, but they are not associated with any magazine position.

Ground tool loading and unloading is global to the system; it is not associated with any particular magazine or channel.

18.

TOOL AND MAGAZINE TABLE
Tool table



CNCelite
8058 8060
8065 8070

REF: 2508

18.3 Tool table (full mode).

This table defines the tools available and the data associated with each one of them. The tool list is common to the whole system, i.e. is common to all the available magazines. Once the tools have been defined, they may be distributed in the various magazines.

18.3.1 Vertical softkey menu.

The options shown for the tool table are. Bear in mind that the table is divided into two panels. There are options that are valid for both panels and options that are only available on one of them.

Softkey.	Description.
	Show more options on the softkey menu.
	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.
	Search for a text on a tool list. See "18.1.2 Search for a text in the tables" on page 301.
	Add a new tool to the list. This icon is only available for the tool list.
	Remove a tool from the list. A tool cannot be deleted if it is in the tool magazine. This icon is only available for the tool list.
	Delete the data where the cursor is. When deleting a data, it assumes its default value. This icon is only available for tool data.
	Configure the data shown in the tool table. This icon is only available for tool data.
	It initializes the tool table. Initializing the tables eliminates all the tools from the list. It also initializes the active-tools table and the magazine tables because the available tools have been erased. The CNC will request confirmation of the command.
	Saves the table data in a file. See "18.1.3 Save and load the tables" on page 302.
	Recall the table data previously saved in a file. Bear in mind that loading the tool table initializes the magazine tables and the active-tools table. See "18.1.3 Save and load the tables" on page 302.
	Print the table in the pre-determined printer or save it as a file (prn format) at the CNC. See "18.1.4 Printing the tables" on page 304.

18.

TOOL AND MAGAZINE TABLE

Tool table (full mode).

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

18.

TOOL AND MAGAZINE TABLE
Tool table (full mode).

Softkey.	Description.
	Copy the data of the offset being displayed onto the clipboard. The data saved may be pasted to a new offset.
	Paste the offset data that is saved on the clipboard.

18.3.2 The tool list

The tool list appears on the left panel of the tool table. The list shows the available tools and their position. The CNC updates data of the list every time a tool change is carried out.

Tool number

The tool number is assigned automatically when adding the tool to the list and may be modified by the user in the data window.

Tool name

Name identifying the tool defined by the user in the data window. It may be edited directly on the list.

Tool position

It indicates the position of the tool, in a magazine, in the spindle or in the claws of the tool changer arm.

C1-C4 It is in one of the spindles.

M1-M4 It is in one of the magazines. In this case, it also indicates its magazine position.

CH1-CH2 It is in the holders of the tool changer arm.

If none of these positions is indicated, it means that it is a ground tool. Ground tools are not stored in the magazine and are loaded manually when requested.

How to move around on the tool list

Key.	Meaning.
	Scroll list up and down line by line.
	Move the cursor page by page through the list.
	Move the cursor to the beginning or end of the list.
	Access the data of the selected tool.

18.3.3 Description of the tool data

The right panel of the tool table shows the data of the tool selected on the list. This data must be defined by the user. If the CNC has tool life monitoring, the CNC will be in charge of updating the value of the actual (real) life.

Configuring the tool data display

The data being shown in the tool table may be set up from the vertical softkey menu.



Softkey to access the setup panel of the tool table. When pressing this softkey, the CNC shows a dialog box to carry out the following actions.

- Select the data to be shown in the tool table. To show or hide a data, activate or cancel its check box.
- Assign a name or text to any of the 4 "custom" parameters available in each tool.
- Define whether the wear is entered either with an incremental or with an absolute value.

Key.	Meaning.
	Move the cursor.
	Activate/deactivate the selection box.
	Accept the configuration defined.
	Cancel the defined configuration.

Description of the tool data

The tool data is grouped as follows:

- Tool identification.
- Tool geometry.
- Tool monitoring.
- Information for the magazine.
- Custom.
- Comment.

Tool identification

Information identifying the tool: number, name, family, number of tool offsets and status.

Tool number

The tool number is assigned automatically when the tool is added to the list. This number may be changed if it is not in the magazine, in the spindle or on the tool changer arm.

The tool number may be any integer between 1 and 999999999; by default, it is assigned the first available value on the list. When entering an existing tool number, the CNC displays the data for that tool.

Tool name

Name identifying the tool. This data can also be defined on the tool list. The tool name may be up to 32-characters long.

18.

TOOL AND MAGAZINE TABLE

Tool table (full mode).

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

18.

TOOL AND MAGAZINE TABLE
Tool table (full mode).**Tool family**

A tool family is a group of tools that share similar characteristics. This information is used when using an automatic tool changer so the CNC can replace the worn-out or rejected tool with a similar one.

When requesting a new tool, the CNC checks whether it is worn out (real life greater than nominal life) or it has been rejected. If so, it selects the next tool in the table that belongs to the same family.

The family of a tool may be any integer between 0 and 99999999. The ·0· family is the same as not having a family; i.e. the tools belonging to the ·0· family cannot be replaced with another one.

Tool offsets

Number of tool offsets. Each tool offset has different geometry and monetary data associated with it. A tool can have up to 8 offsets. When a tool has several offsets, their numbers must be correlative (non-skipping).

Tool status

It indicates the tool status. The possible status are:

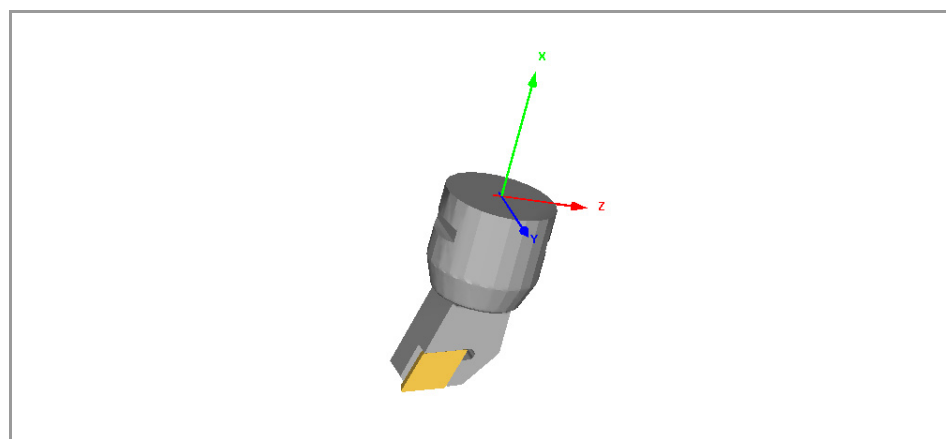
Status	Meaning
Available	The tool is available.
Worn out	The "real life" is greater than the "nominal life".
Rejected	The tool has been rejected by the PLC.

When using tool life monitoring, the "worn-out" and "rejected" indicators are also set by the CNC when any of the previous cases occur.

"Tool holder" field.

Tool holder shape file (stl format) for HD graphics. By default, the files are in the ..Users/Grafdata/Tools folder. To select a file, press [RECALL].

It is the user's responsibility to ensure that the dimensions of the tool holder in the stl file are correct. Otherwise, gaps or overlaps may appear between the insert and the tool holder. For the tool tip to be in the actual position, and the tool holder orientation to be correct, the drawing must have the following axis configuration.

**"Position in turret" field.**

This field is only displayed if there is a "Custom Turret" type magazine. Position assigned to the tool in the custom turret. If the tool position is not fixed (interchangeable tool), set the position to -1. In this case, the variable (V.)G.NXPTURRET defines the position for the next tool.

Tool geometry

This area shows the data about the tool type and its dimensions. The geometry data depends on the type of tool. The table only shows the data that makes sense for the selected tool.

While defining the data, it shows various information graphics depending on the data being defined. On the other hand, the bottom of the screen shows the description of the data currently selected.

The data related to the geometry may be accessed with the following hotkeys:

Hotkey	Access
L	Length, length wear and edge length.
R	Radius, radius wear, nose (tip) radius and nose (tip) radius wear.
A	Penetration angle
O	Offsets on each axis.

Offset selection

The geometry data is associated with the tool offset. If the tool has been defined with several offsets, it shows the number of the offset whose data is displayed, and it also allows selecting the previous or next offset. To change offsets, place the focus on the buttons and press [SPACE].



Offset number and selection of the previous or next offset. In this case, it will display the data of the second tool offset.

Operation type and tool

Regardless of the software installed, it is possible to define both milling and lathe tools. The tool is defined depending on the operation it can carry out. If it is not the right tool for any of the proposed operations, it must be assigned the operation "Others".

Once the operation has been selected, the screen will show the available tools. It will show a help graphic with the selected tool type.

The data to define the geometry of the tool will be relevant to the selected tool. It will only show the data that makes sense for the selected tool.

Operation	Tool type
Milling (A) Flat endmill. (B) Toric endmill. (C) Ball endmill. (D) Disk endmill.	 (A)  (B)  (C)  (D)
Reaming (A) Reamer.	 (A)
Grooving/cutoff (A) Square.	 (A)
Drilling (A) Drill bit.	 (A)
Boring (A) Quill.	 (A)
Turning (A) Diamond. (B) Square. (C) Round.	 (A)  (B)  (C)

18.

TOOL AND MAGAZINE TABLE

Tool table (full mode).

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

18.

TOOL AND MAGAZINE TABLE
Tool table (full mode).

Operation	Tool type
Surface milling (A) Surface milling endmill.	 (A)
Tapping (A) Cutter. (B) Tap.	 (A)  (B)
Measuring probe	
Others	To define the tools that do are not suitable for the proposed operations.

Axis orientation in turning.

It is defined with an icon that is only displayed when defining a turning (lathe) tool.

The orientation of the axes is determined by the type of lathe (horizontal or vertical), the position of the turret and the spindle position (on the right or on the left).

Location code or tool calibration point.

The location code indicates which is the calibrated tool tip and, therefore, the point controlled by the CNC to apply radius compensation. The location code depends on the orientation of the machine axes. It is defined with an icon that is only displayed when defining a turning (lathe) tool.

Tool-holder orientation.

It is defined by an icon. The orientation of the tool holder indicates whether it is a tool for horizontal or radial machining. For turning tools, the meaning of this icon depends on the orientation of the tool axes.

		Horizontal lathe. Radial machining.
		Vertical lathe. Longitudinal machining.

Spindle turning direction

Use the following icons to define the spindle turning direction.

Icon	Spindle turning direction
	Undefined spindle turning direction.
	Clockwise spindle turning direction.
	Counterclockwise spindle turning direction.

The new tools do not have a pre-determined spindle turning direction; during execution, the spindle turns in the programmed direction (M03/M04).

When assigning a turning direction to a tool in the table, the CNC will verify, during execution, that the turning direction in the table is the same as the one programmed (M03/M04). If the two directions are not the same, the CNC will display the corresponding error message. The CNC verifies this every time an M03, M04 or M06 is programmed.



"L" - Tool length

This data is only shown on tools that are not for turning. The dimensions of the turning tools are defined with the offsets.

"R" - Tool radius

This data is only shown on tools that are not for turning. The dimensions of the turning tools are defined with the offsets.



- L Tool length.
- R Tool radius.
- Lc Cutting length.

"LW" "RW" - Tool length and radius wear

Tool radius wear and length wear offset. The CNC adds the wear value to the nominal length and radius to calculate the real tool length ($L + LW$) and real tool radius ($R + RW$).

Initial length	Wear	Final length
50	0.2	50.2
50	-0.2	49.8
-50	0.2	-49.8
-50	-0.2	-50.2

In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute. In either case, deleting the wear value or setting it to 0 implies resetting the amount of wear to 0. See ["Select the type of wear values to enter, incremental or absolute."](#) on page 315.

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

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

"A" - Cutter angle

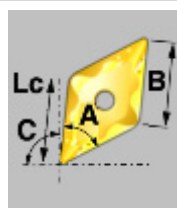
This data is only shown for turning tools. For the grooving and cut-off tool, this data assumes a value of 90°.

"C" - Cutting angle

This data is only shown for turning tools. For the grooving and cut-off tool, this data assumes a value of 90°.

"B" - Cutter width

This data is only shown for turning tools.



- A Cutter angle.
- C Cutting angle.
- B Cutter width.
- Lc Cutting length.

18.

TOOL AND MAGAZINE TABLE

Tool table (full mode).

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

"Rp" - Tool tip radius

Tool tip radius. For the grooving and cut-off tool, this data assumes a value of 0.

"RpW" - Tool tip radius wear

Tool tip radius wear. The CNC adds the wear value to the nominal tool tip radius to calculate the actual (real) tool tip radius ($R_p + R_{pW}$).

In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute. In either case, deleting the wear value or setting it to 0 implies resetting the amount of wear to 0. See ["Select the type of wear values to enter, incremental or absolute."](#) on page 315.

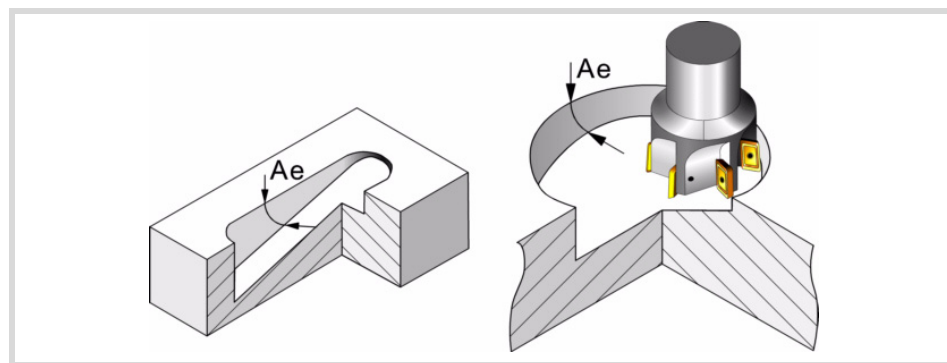
Using incremental wear, the value entered by the user will be added (or subtracted if it is negative) to the absolute value of the wear. After pressing [ENTER] to accept the new value, the wear field will show the resulting absolute value.

"Lc" - Cutting length

Length of the cutting edge.

"Ae" - Entry angle

This data is only shown on tools that are not for turning. Penetration angle for pocket milling.

**"Zn" - Number of teeth.**

This information is only available for "Milling", "Drilling", "Surface milling", "Reaming", "boring" and "Other" tools. This data is necessary to work using the DMC function, so that the CNC can limit the feedrate per tooth between the minimum and maximum.

"RotPos" field.

This field is only displayed if the tool can be oriented with a rotary axis. Angle of the rotary axis of the tool when the offsets were measured. Angle 0° corresponds to the tool perpendicular to the abscissa axis (usually Z).

Tool (length) offsets and offset wear.

The offsets are used to define the tool dimensions in each axis. The dimensions of the turning tools are defined using these offsets; either these offsets or tool length and radius may be used for the dimensions of the rest of the tools.

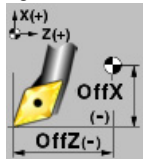
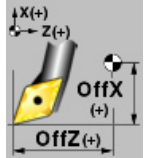
On tools that are not just for turning, e.g. endmills and drill bits, the offsets may also be used to define the tool position when using a tool holder or an intermediate tool. In this case, the tool dimensions are defined with the radius and the length.

18.

TOOL AND MAGAZINE TABLE
Tool table (full mode).



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

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

In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute. In either case, deleting the wear value or setting it to 0 implies resetting the amount of wear to 0. See ["Select the type of wear values to enter, incremental or absolute."](#) on page 315.

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

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

Select the type of wear values to enter, incremental or absolute.

The tool offset wear (for length, radius, tip radius and offsets) may be entered with either absolute or incremental values. The value shown by their respective fields in the tool table are always absolute.



After pressing this icon, the CNC shows the dialog box for configuring the tool table. Here, the user can define whether the wear values being entered are incremental (box checked) or absolute (empty unchecked). By default, the wear value being entered will be absolute.

When selecting an incremental wear, it is possible to define the maximum increment possible; by default 0.5 mm (0.019685 inch). This data limits the wear increment to be edited, not the maximum absolute wear possible. If no value is defined, empty data, there will be no limit for the wear increment to be entered.

The option selected in the tool table will also be applied in the tool calibration mode.

18.

TOOL AND MAGAZINE TABLE

Tool table (full mode).

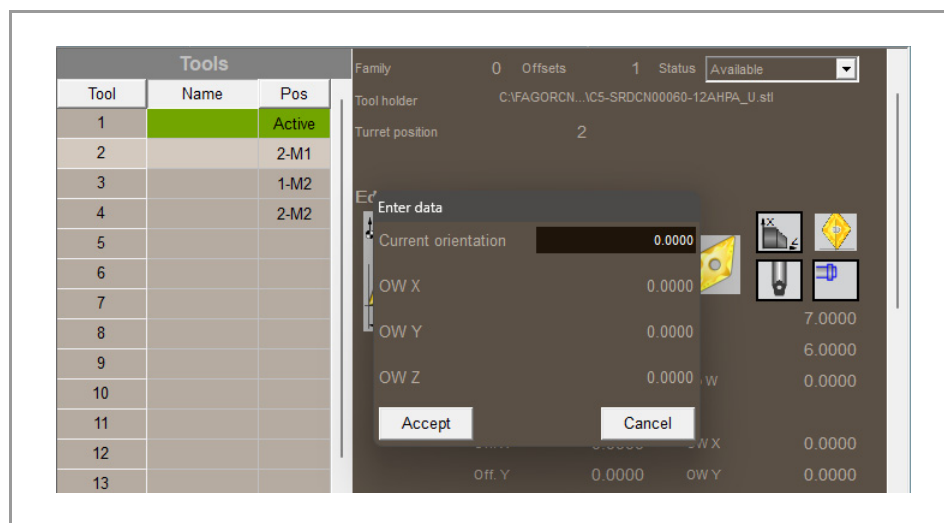
FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Definition of offset wear for tool orientations other than the one indicated in the RotPos field.

This field is only displayed if the tool can be oriented with a rotary axis. By selecting one of the OW X, OW Y, etc fields and pressing [RECALL], the CNC displays a window to define the offset wear and the B-axis position at which they have been measured. When the values are accepted, the CNC recalculates them for the orientation defined in the RotPos field of the main screen table.



18.

TOOL AND MAGAZINE TABLE
Tool table (full mode).

Tool life monitoring

This group shows the data related to tool life monitoring. If a tool has been defined with several offsets, it is possible to manage the status of each one.

When managing the live of two or more offsets of the same tool, the list of the magazine positions (magazine table) only shows whether the tool has been rejected or not.

Type

It is used to activate and select the type of tool life monitoring.

- Life in time (in minutes).
- Life in operations.
- Wear by distance traveled. Option for grinding wheels.

Life in time.

The table shows the fields:

- Real (actual) life. Time the tool has been machining. The CNC updates this value when the tool is being used.
- Nominal life. Permitted machining time.

Life in operations.

The table shows the fields:

- Real (actual) life. Number of operations performed. The CNC updates this value when the tool is being used.
- Nominal life. Number of permitted operations.

Wear by distance traveled.

Recommended option for grinding tools where radius wear is proportional to the distance traveled. The table shows the fields:

- Wear per meter. Wear coefficient (millimeters of wear per machined meter). The wear per meter cannot be greater than 1/3 of the tool radius. The CNC takes this factor into account when the tool is working with radius compensation activated.
- Minimum radius. The minimum radius must be less than the sum of the radius and the wear of the radius. Should the tool radius reach the minimum radius during machining due to wear compensation, the CNC does not deactivate the wear compensation and

it is up to the criteria of the OEM (PLC operation) to decide what action to take. By default, once the program execution is finished, after a reset or tool change, the status of the tool changes from available to rejected.

Tool magazine

This information cannot be modified if the tool is in the spindle, on the changer arm or in the magazine.

Size

Tool size. The size determines the number of positions (pockets) the tool occupies in the magazine.

Size	Magazine positions it occupies.
Small	It only occupies its position. It takes no extra position.
Medium	It takes half of an additional position to the right and to the left.
Large	It takes a full additional position to the right and to the left.
Custom	The user defines the number of additional positions the tool occupies to its right and to its left.

Positions to the right / Positions to the left

Space reserved in the magazine for the tool to the right and to the left of its position.

This data can be defined when the tool size is "custom".

Special

The tool always occupies the same position in the magazine.

Custom

Data defined by the manufacturer. On the setup panel of the tool table, it is possible to assign a text to any of the 4 parameters.

Data 1 / Data 2

These data show, in numerical format, the information selected by the manufacturer.

Data 3 / Data 4

These data show, in binary format, the information selected by the manufacturer.

Comment

Comment associated with the tool.

18.

TOOL AND MAGAZINE TABLE

Tool table (full mode).

18.4 Tool table (simple mode).

The tools in the table appear sorted by the magazine they belong to starting from the first magazine and leaving the ground tools for last. The tools of a magazine are sorted by tool number and by offset number.

18.4.1 Vertical softkey menu.

The options shown for the tool table are.

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.
	The CNC is showing milling tool information (standard page or offsets); this softkey shows the information of the turning tools.
	The CNC is showing turning tool information (standard page or offsets); this softkey shows the information of the milling tools.
	Show the standard page (screen) of the tool table. The standard page (screen) shows the main data of the tools.
	Show the offset page (screen) of the tool table. The offset page (screen) shows the offsets of the tools.

18.4.2 Configuring the tool data display

The data being shown in the tool table may be set up in the full tool table from the vertical softkey menu.



Softkey to access the setup panel of the tool table. When pressing this softkey, the CNC shows a dialog box to carry out the following actions.

- Select the data to be shown in the tool table. To show or hide a data, activate or cancel its check box.
- Define whether the wear is entered either with an incremental or with an absolute value.

18.4.3 Data of the M tools (standard page - screen-).

The standard page (screen) shows the main data of the tools, in this case, the tools that not specific for turning. The table highlights in color the active tool.

M TOOLS: STANDARD										
T	D	Mz	icon	Name	L	L W	R	R W	Rp	Rp W
1	1	1			75.0000	0.0000	25.0000	0.0000	0.0000	0.0000
2	1	1			100.0000	0.0000	7.5000	0.0000	0.0000	0.0000
3	1	1			75.0000	0.0000	5.0000	0.0000	1.0000	0.0000
4	1	1			70.0000	0.0000	4.0000	0.0000	0.0000	0.0000
5	1	1			85.0000	0.0000	2.5000	0.0000	0.0000	0.0000
6	1	1			85.0000	0.0000	2.5000	0.0000	0.0000	0.0000
7	1	1			80.0000	0.0000	5.0000	0.0000	5.0000	0.0000
8	1	1			65.0000	0.0000	3.0000	0.0000	0.0000	0.0000
9	1	1			100.0000	0.0000	10.0000	0.0000	0.4000	0.0000
10	1	1			85.0000	0.0000	7.5000	0.0000	0.0000	0.0000
11	1	1			65.0000	0.0000	5.0000	0.0000	0.0000	0.0000
12	1	1			125.0000	0.0000	12.0000	0.0000	0.0000	0.0000
13	1	1			135.0000	0.0000	25.0000	0.0000	0.0000	0.0000
Radius										

Column.	Description.
T	Tool number.
D	Tool offset number.
Mz	Magazine it belongs to.
icon	Type of tool offset.
Name	Tool name.
L	Length.
L W	Length wear.
R	Radius.
R W	Radius wear.
Rp	Tip radius.
Rp W	Tool tip radius wear.
Lc	Cutting length.
Ae	Entry angle.
icon	Turning direction.
Zn	Number of teeth.

18.

TOOL AND MAGAZINE TABLE

Tool table (simple mode).

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

18.4.4 Data of the T tools (standard page - screen-).

The standard page (screen) shows the main data of the tools, in this case, the ones for turning. The table highlights in color the active tool.

T TOOLS: STANDARD											
T	D	Mz			Name	A	B	C	Lc	Rp	Rp V
15	1	1				89.0000	5.0000	91.0000	4.0000	1.5000	0.000
16	1	1				90.0000	6.0000	0.0000	25.0000	3.0000	0.000
17	1	1				90.0000	4.0000	90.0000	2.5000	0.0000	0.000

Cutter's cutting angle

Column.	Description.
T	Tool number.
D	Tool offset number.
Mz	Magazine it belongs to.
icon	Type of tool offset.
icon	Location code (shape).
icon	Tool-holder orientation.
Name	Tool name.
A	Cutter angle.
B	Cutter width.
C	Cutting angle of the cutter.
Lc	Cutting length.
Rp	Tip radius.
Rp W	Tool tip radius wear.



18.4.5 Data of the M/T tools (offset page - screen-).

The offset page (screen) shows the offsets of the tools. The table highlights in color the active tool.

T TOOLS: OFFSETS										
T	D	Mz		Name	Off. X	OW X	Off. Y	OW Y	Off. Z	OW Z
15	1	1			0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
16	1	1			0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
17	1	1			0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
Edge										

Column.	Description.
T	Tool number.
D	Tool offset number.
Mz	Magazine it belongs to.
Name	Tool name.
Off. X	Tool offset on each axis.
OW. X	Tool offset wear on each axis.

18.4.6 Editing the tool table.

With the simple tool table, it is only possible to edit the data of the tools. Neither tools nor offsets can be added or removed, uploaded or downloaded or change tools from one magazine to another.

The table allows editing all the fields except the following. These fields may be modified at the full tool table.

- Tool number.
- Tool offset number.
- Magazine it belongs to.
- Type of tool offset.
- Location code (shape).
- Tool-holder orientation.
- Turning direction.

18.

TOOL AND MAGAZINE TABLE
Tool table (simple mode).

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

18.5 Operations with the tool table (full mode).

18.5.1 Editing the tool table

Initialize table

The table can only be initialized when the CNC is "READY". The table is initialized from the vertical softkey menu.



When resetting the table, all the tools are deleted from the list, including those in the spindle and on the tool changer arm. It also initializes the active-tools table and the magazine tables because the available tools have been erased.

Adding/removing a tool to/from the list

A tool may be added or deleted only when it is selected on the tool list. Both operations are carried out from the vertical softkey menu.



Add a tool to the list.

It adds a new tool to the list. The tool is added to the list at the first free position.



Remove a tool from the list.

It removes a tool from the list. A tool cannot be removed if it is in the magazine, in the spindle or on the tool changer arm.

Data editing

Proceed as follows to fill out the tool table data:

- 1 Select the tool to be set from the list and press [ENTER] to access its data.
- 2 Configure the table to show only the data that may be defined hiding the rest.
- 3 Define the tool data. Every time a new value is defined, press [ENTER] to validate it.

When a tool has several offsets, all the data of an offset may be copied to another offset. This operation is carried out from the softkey menu.



These softkeys may be used to copy on to the clipboard the data of the offset being displayed and then paste it to another offset.

- 4 Make a safety backup copy of the tool data out to a file.

Key.	Meaning.
	Move the cursor through the table data.
	Accept the new value.
	Activate/deactivate the selection box. Open/close pop-up menus.
	Move the cursor within a data.
	Select the line before or after the comment.
	Move the cursor to the beginning or end of the comment line.

18.6 Active-tools table

This table shows the list of available tools and which one is active in each channel.

The table is divided in two panels. To switch panels, press the [FOCUS] key.



Tools		
Tool	Name	Position
1		1
2		2
3		3
4		C1
5		5
6		6
7		7
8		8
9		9
10		10
11		11
12		12
13		13
14		14
15		15
16		16

(A) Tool listing.

Active tools

Channel 1

Channel 2

T

4

2

Name

Status

Family

0

0

Active d

1

1

Remaining life

Not mont.

Not mont.

L

65

10

R

8

4

Tool of the position selected on the list

T

10

Name

0

Status

Family

0

D

1

Remaining life

Not mont.

L

65

R

4

(B) Active tools.

(C) Tool selected on the list.

The left panel shows the list of the available tools and the right panel shows the data of the active tool in each channel.

Tool listing

For each tool, it indicates the position and the magazine where it is located, whether it is a ground tool or the tool is active in a channel. The CNC updates data of the list every time a tool change is carried out.

It is the same list that appears in the tool table. See "18.3.2 The tool list" on page 308.

Active tools

It shows the data of the active tool in each channel and also the data of the tool selected on the list. It is also possible to change the tool of the spindle. See "18.6.2 Changing the tool of the spindle" on page 324.

The tool data cannot be edited on this screen. The data shown here is defined in the tool table.

18.

TOOL AND MAGAZINE TABLE

Active-tools table



CNCelite
8058 8060
8065 8070

REF: 2508

18.6.1 Softkey menus.

Vertical softkey menu.

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.
	It initializes the tool table. Initializing the tables eliminates all the tools from the list. It also initializes the active-tools table and the magazine tables because the available tools have been erased. The CNC will request confirmation of the command.
	Saves the table data in a file. See "18.1.3 Save and load the tables" on page 302.
	Recall the table data previously saved in a file. Bear in mind that loading the tool table initializes the magazine tables and the active-tools table. See "18.1.3 Save and load the tables" on page 302.
	Print the table in the pre-determined printer or save it as a file (prn format) at the CNC. See "18.1.4 Printing the tables" on page 304.

18.6.2 Changing the tool of the spindle

It is possible to change the tool of the spindles from the active tools panel. The tool to be placed must be defined in the tool table. To change the active tool, follow these steps.

- 1 Use the cursor to select the active tool to be modified and enter the number of the new tool.
- 2 Press the [CYCLE START] key to load the tool automatically or the [ENTER] key to update the positions list after a manual tool change.
- 3 The tool list will show that the tool is in the spindle.



18.7 Table for the status of the tool change process

This screen monitors the tool changes being executed in each channel.



It indicates the following for each tool change process:

- The type of change selected and the magazine involved.
The type of change refers to the type of operation being carried out or to be carried out when executing function M06. The possible types of change are load a tool into the spindle or load and unload a ground tool from the magazine through the spindle.
- The "execution " LED informs that the tool change process is running and that the magazine is carrying out an operation.
- The "status" LED informs on whether the magazine is in an error state or not. To eliminate the error condition, press the reset softkey in the magazine table.



18.

TOOL AND MAGAZINE TABLE
Table for the status of the tool change process



CNCelite
8058 8060
8065 8070

REF: 2508

18.8 Magazine table

Up to four different magazines may be configured. Each magazine has a table showing the tool distribution in the magazine and which table is in the spindle and on each holder of the changer arm (if any). The CNC updates the table data every time a tool is changed.

18.

TOOL AND MAGAZINE TABLE

Magazine table

Magazine positions				Information	
Position	Tool	Remaining life	Famil		
1	1	Not monit.	0	Magazine status Resting Status Tool changer arm CH1 CH2 Tool of the position selected on the list T 1 Left pos. 0 Name Status Right Pos. Small 0 Special	

(A)Magazine positions.

(B)Magazine information.

The table is divided in two panels.

The left panel shows the list of magazine positions. For each position, it indicates whether it is free, disabled or has a tool. For each tool, it shows the remaining life time (when using tool life monitoring) and the family it belongs to.

The right panel is informative only; it cannot be selected. This panel shows information about the magazine status, about the tools in the tool changer arm and about the tool selected on the list. Optionally, it may show information on the type and configuration of the magazine.

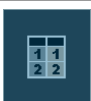


CNCelite
8058 8060
8065 8070

REF: 2508

18.8.1 Softkey menus.

Vertical softkey menu.

Softkey.	Description.
	Show more options on the softkey menu.
	Load / unload a tool to / from the tool changer arm. This icon is only available in magazines with tool changer arm. See "18.9.2 Load / unload a tool to / from the tool changer arm" on page 333.
	Search for a text in the table. When selecting this option, the CNC shows a dialog box requesting the text to be found. See "18.1.2 Search for a text in the tables" on page 301.
	Load a tool into the magazine. See "18.9.1 Loading / unloading tools to / from the magazine" on page 331.
	Unload the tool from the magazine. See "18.9.1 Loading / unloading tools to / from the magazine" on page 331.
	Initialize (reset) the magazine table. Initializing the tables eliminates all the tools from the list. It also initializes the active-tools table because the tool arrangement in the magazine may have changed. The CNC will request confirmation of the command.
	Saves the table data in a file. See "18.1.3 Save and load the tables" on page 302.
	Recall the table data previously saved in a file. Bear in mind that loading the magazine tool initializes the active-tools table. See "18.1.3 Save and load the tables" on page 302.
	Print the table in the pre-determined printer or save it as a file (prn format) at the CNC. See "18.1.4 Printing the tables" on page 304.
	Enable or disable the magazine position selected with the cursor.
	Reset the magazine. Eliminates the error status of the tool manager. This icon is only available when an error occurs at the tool manager.
	Magazine information. Show or hide the magazine data.
	Initialize the magazine data. The CNC will request confirmation of the command. It initializes all the magazine positions assigning T1 to position 1, T2 to position 2 and so on. The tools must exist and must not be in another magazine. It also initializes the list of tools and the active-tools table because the tool arrangement in the magazine may have changed.

18.

TOOL AND MAGAZINE TABLE
Magazine tableFAGOR
AUTOMATIONCNCelite
8058 8060
8065 8070

REF: 2508

18.8.2 List of magazine positions

The list of magazine positions appears on the left panel of the tool magazine table. For each position, it indicates whether it is free, disabled or has a tool. For each tool, it shows the remaining life time (when using tool life monitoring) and the family it belongs to.

Magazine position

The status of a position is indicated with a symbol next to the position number.

- The position is free (white filled circle).
- The position is semi-free (black-and-white filled circle).
- The position is occupied (black filled circle).
- The position is disabled (red filled circle).

Tool number

Number of the tool occupying the magazine position.

Remaining life

If the tool life monitoring is active, shows the remaining life, either in machining time or in the number of operations to be carried out or the tool status (rejected or worn out).

Tool family

Family the tool belongs to, defined by the user in the tool table.

18.

TOOL AND MAGAZINE TABLE
Magazine table

18.8.3 Magazine information

The right panel shows different information about the status of the tool magazine and the tool changer arm. This information is grouped as follows:

Information

A

Magazine status

Resting

Status

B

Tool changer arm

CH1

CH2

C

Tool of the position selected on the list

T

1

Left pos.

0

Name

Small

D

Status

Right Pos.

0

Special

(A) Magazine status.

(B) Tool in the claws of the changer arm.

(C) Tool selected on the list.

(D) Magazine data.

Magazine status

This area shows the operation being carried out in the magazine.

In standby: The magazine is in standby.

Loading: A tool is being loaded into the magazine.

Unloading: A tool is being unloaded from the magazine.

The "Status" led informs the user whether the magazine is in an error state or not. To eliminate the error condition, press the reset softkey in the magazine table.

Change status

If the magazine is involved in a tool change at a given time, it shows the information about the status of that change; the operation is carried out when executing an M06, manager status (in execution or at rest) and the status of the change process (whether it is in error or not).

Tool information

It shows the data related to the magazine of the tool selected from the list. The "status" led informs about the tool status.

Green The tool is available.

Yellow The tool life is about to expire.

Red The tool is expired (worn out) or has been rejected by the PLC.

18.

TOOL AND MAGAZINE TABLE

Magazine table

CNCelite
8058 8060
8065 8070

REF: 2508

•329•

Tools in the changer arm

This area shows which are the tools that occupy each claw (holder) of the changer. It shows the number and size of each tool.

Magazine data



Description of the type of magazine. To show and hide this information, press the information softkey.

18.

TOOL AND MAGAZINE TABLE
Magazine table



CNCelite
8058 8060
8065 8070

REF: 2508

18.9 Operations with the magazine table

18.9.1 Loading / unloading tools to / from the magazine

The tools may be loaded and unloaded to/from the magazine manually or automatically. When done manually, the positions list must be updated.

Manual load/unload

The operator places the tools directly in the magazine without using the CNC. Then, the positions list must be updated.

Automatic load/unload

The operator places the tools in the spindle and the CNC places the tool in the magazine. The positions list is updated automatically.

Initialize table

The table can only be initialized when the CNC is "READY". The table is initialized from the vertical softkey menu. The CNC offers two ways to initialize the table.



Initializing the table eliminates all the data about the position of the tools in the magazine. It also initializes the active-tools table because the tool arrangement in the magazine has been changed.



When initializing the table, it initializes the magazine positions assigning T1 to position 1, T2 to position 2 and so on. The tools must exist and must not be in another magazine.

It also initializes the active-tools table because the tool arrangement in the magazine may have changed.

Load a tool



The tool is loaded into the magazine using the vertical softkey menu. Only the tools defined in the tool table may be loaded into the magazine and they must be defined as ground tools. In other words, they must not be in any position of the magazine, of the spindle or in the claws of the tool changer.



Even if the magazines are configured for not admitting ground tools while machining, they may be loaded into the magazine through this maneuver.

Depending on tool size, when it is loaded into the magazine, it may affect several positions.

Magazine management using the "Load tool" softkey.

This type of management is valid for loading tools both automatically and manually. After pressing the softkey, the user is asked what kind of loading to do.

- 1 Press the softkey associated with tool loading.
- 2 Enter the tool number and the magazine position to insert it. By default, it offers the position selected on the list with the cursor.
- 3 Press the [CYCLE START] key to load the tool automatically by means of the spindle or the [ENTER] key to update the positions list after loading manually.
 - Automatic loading (pressing [START]). The CNC loads into the magazine a tool previously defined in the table. It is loaded from ground through the spindle.
 - Manual loading (pressing [ENTER]). The CNC assume that the tool defined in the table has already been manually loaded into the magazine. It updates the magazine list.

18.

TOOL AND MAGAZINE TABLE
Operations with the magazine table

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

18.

TOOL AND MAGAZINE TABLE
Operations with the magazine table**Magazine management using the positions list.**

This type of management is only valid for loading tools manually. It lets update the list of positions after having placed the tools directly in the magazine, without using the CNC.

- 1 Select a magazine position from the list and enter the number of the tool it occupies it.
- 2 Press [ENTER] to update the positions list.

Unloading a tool

The tool is unloaded into the magazine using the vertical softkey menu. After unloading a tool from the magazine, it becomes a ground tool.

Magazine management using the "Unload tool" softkey.

This type of management is valid for unloading tools both automatically and manually. After pressing the softkey, the user is asked what kind of loading to do.

- 1 Press the softkey associated with tool unloading.
- 2 Enter the number of the tool to be unloaded.
- 3 Press the [CYCLE START] key to unload the tool automatically to the spindle or the [ENTER] key to update the positions list after unloading manually.
 - Automatic unload (pressing [START]). The CNC unloads from the magazine a tool defined in the table. It is unloaded to ground through the spindle.
 - Manual loading (pressing [ENTER]). The CNC assumes that the tool defined in the table has already been manually loaded from the magazine. It updates the magazine list.

Magazine management using the positions list.

This type of management is only valid for unloading tools manually. It lets update the list of positions after having removed the tools directly from the magazine, without using the CNC.

- 1 Select a magazine position from the list and delete the number of the tool it occupies it.
- 2 Press [ENTER] to update the positions list.



18.9.2 Load / unload a tool to / from the tool changer arm



A tool is loaded/unloaded to/from the claws (holders) of the changer arm using the vertical softkey menu. To insert a tool in the holders of the changer arm (when available), it must be placed in the magazine. Ground tools cannot be placed in the claws (holders) of the tool changer arm.

Nor can the tools that are in the holders of the tool changer arm be inserted in the spindle.

To load or unload one of these tools:

- 1 Press the softkey associated with the operations with the changer arm.
- 2 Define the tool number in the positions of the tool changer arm.

When using a two-holder changer arm, there may not be tools in the spindle and in the second holder of the changer arm at the same time.

- 3 Press [ENTER] to update the table.

18.

TOOL AND MAGAZINE TABLE
Operations with the magazine table

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

18.

TOOL AND MAGAZINE TABLE

Operations with the magazine table

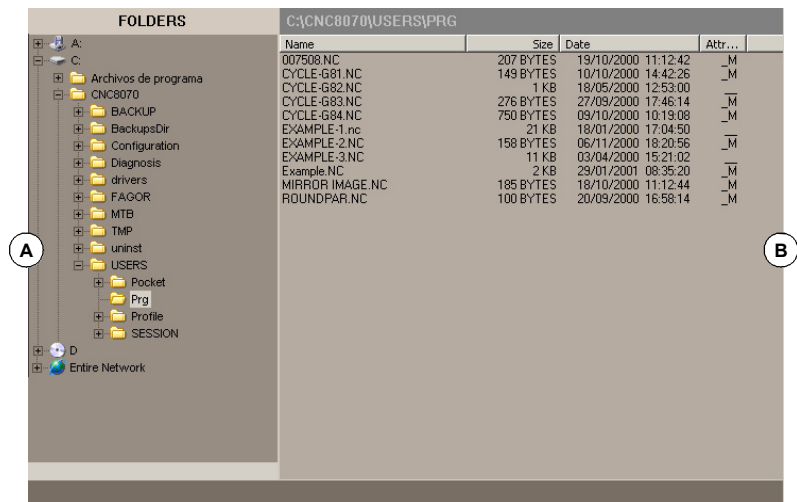


CNCelite
8058 8060
8065 8070

REF: 2508

19.1 Interface description.

The utilities mode screen shows the following information:

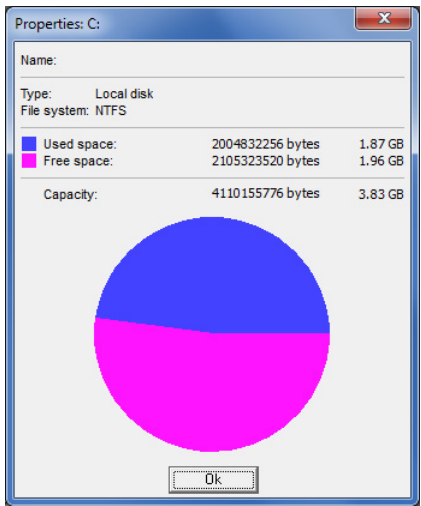


- A Folder tree. The tree shows the folders that may be accessed from the CNC, as well their structure.
- B List of files saved in the selected folder.

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.

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.



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.

19.

UTILITIES MODE
Interface description.

19.1.1 Softkey menus.

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 from 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.
	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.

19.

UTILITIES MODE
Interface description.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

Softkey.	Description.
	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).

19.

UTILITIES MODE

Interface description.

19.2 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.

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.

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.

Reset all

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

Show hidden 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.

19.

UTILITIES MODE

Set how to display the list of programs.

19.3 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 from 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.

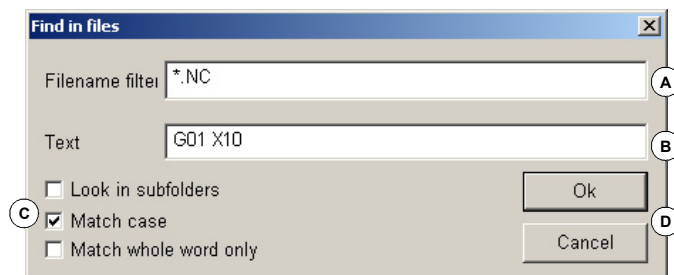
When selecting this option, the CNC shows a dialog box requesting the file name. Write the new name and select one of the buttons to accept it or ignore it. If when defining the folder name, there is already a folder with the same name, it will NOT create the new folder.

19.

UTILITIES MODE
Select files and create folders.

19.4 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.
- B Text included within the files.
- C Defines the search criteria.
- 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.

Wild symbols.

The "*" and "?" wild characters may be used during the search, which have the following meanings:

- * 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".

19.

UTILITIES MODE

Find in files

19.5 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.

General password

It is requested when trying to access this password setting screen.

PLC

It is requested when trying to carry out the following actions at the PLC:

- Editing the PLC program. When entering the wrong password, the PLC program opens as read-only.
- Adding a file to the project.
- Deleting a file.
- Renaming a file.
- Editing PLC messages. When entering the wrong password, the PLC messages may be neither displayed nor edited.
- Generate PLC.
- When accessing to the "Commands" service options.
- In monitoring, when modifying the status of a resource.

Machine parameters

The CNC requests a password when attempting 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.

The CNC requests a password when attempting to carry out the following actions:

- Start the FGUIM application.
- In the file explorer, to show/hide the folder tree.

In user mode, the CNC prompts for a password when attempting to perform the following actions:

- Initialise the magazine.
- Load a magazine from file.
- Loading/unloading tools from the magazine.

19.

UTILITIES MODE
Protection passwords



- Delete a tool from the magazine.

Machine parameters for kinetics

It is requested when trying to carry out the following actions in the machine parameter tables for the kinematics: If not set, it will apply the password used in the rest of the machine parameters.

- Modifying the value of a parameter.
- Initialize a table.
- Loading a table.

Administrator mode

It is requested to start the unit up in administrator mode. The access to the administrator mode is enabled with the validation code. If you don't have this software option, you will not be able to access the administrator mode.

19.

UTILITIES MODE
Protection passwords

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

19.6 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. It will request confirmation before starting the backup.

Restore data from the backup .



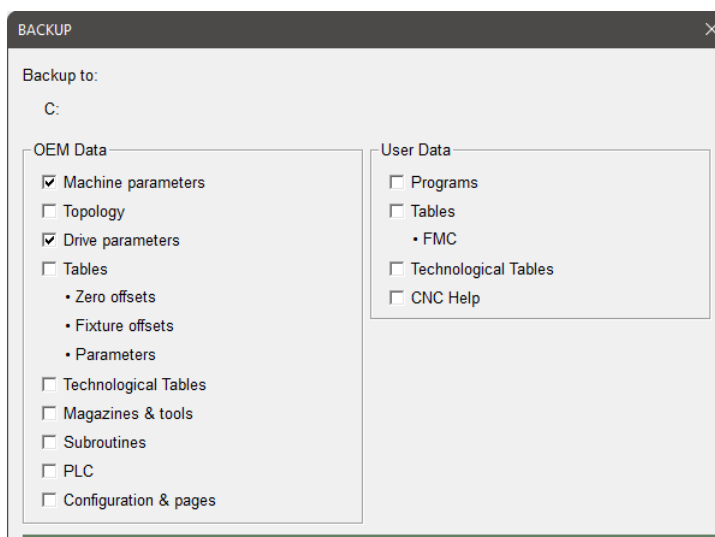
Before restoring the BCSD configuration, they must have the correct node number. This can be done by changing the Pn704 parameter (node number) through the BCSD screen or by using the WinBCSD programme.

OEM data may only be restored when the CNC has been started up in SETUP 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. It will request confirmation before starting the restore.

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

The selected options will be kept until they are modified.



OEM data.

Element.	File.
Machine parameters.	BACKUP_OEM_MP.zip.
Topology.	BACKUP_OEM_TOPOLOGY.zip.
Drive parameters.	BACKUP_OEM_DIGITAL_DRIVE.zip.
Tables. • Zero offsets. • Clamps. • Arithmetic parameters.	BACKUP_OEM_TABLES.zip.
Technology tables.	BACKUP_OEM_TECHTABLES.zip.
Tools and tool magazines.	BACKUP_OEM_MZTOOLS.zip.
Subroutines.	BACKUP_OEM_SUB.zip.
PLC.	BACKUP_OEM_PLC.zip.
Configuration and screens.	BACKUP_OEM_CONFIG.zip.

User data.

Element.	File.
Programmes (includes the following folders). ..\Users\Prg ..\Users\Pocket ..\Users\Profile ..\Users\CycleData	BACKUP_USER_PRG.zip.
Tables • FMC.	BACKUP_USER_TABLES.zip.
Technology tables.	BACKUP_USER_TECHTABLES.zip.
CNC Help.	BACKUP_USER_CNCHHELP.zip.

19.**UTILITIES MODE**

Data safety backup. Backup - Restore

FAGOR
 AUTOMATION 
CNCelite
 8058 8060
 8065 8070

REF: 2508

19.7 Encrypting files.



A file encrypted from the Utilities mode cannot be unencrypted. It is the responsibility of the file creator to keep a copy of the original file.

There are two encryption options, one aimed at the machine manufacturer and the other at third parties who develop applications for the CNC. CNC allows encryption of workpiece programmes, subroutines (including those associated with user cycles) and scripts (scp files). Encryption may be used to protect any file by making it illegible, so it cannot be accessed by anyone else. An encrypted program is not editable and it is not displayed during execution. An encrypted file can be copied, deleted, etc. just like any other file.

Machine manufacturer encryption.

This is the default encryption method. The encryption algorithm uses the general access code (manufacturer's password); if it is not defined, encryption is not possible. If after encrypting a file, the general access code is deleted or changed, the file will no longer be readable by the CNC. To execute encrypted files, the CNC must have the general access code defined, and it must match the code used for the encryption. An encrypted file on one CNC can be executed on another CNC if they have the same general access code.

Third-party encryption.

This encryption method must be enabled in the SysUtil.ini file. The encryption algorithm is generic, it does not use any passwords. There are 2 types of encryption: generic for all CNCs (encrypted files can be used on any CNC) or CNC-specific (encrypted files are associated with a hardware ID). As some folders and files on the CNC are protected by the OEM, it is recommended that the encryption be performed on a simulator and the encrypted files then be copied to the CNC.

Configure third-party encryption.

Third-party encryption is enabled in the utilities mode configuration file (..\Mtb\Mmc\Config\SysUtil.ini). Add the following lines to the file. After adding these lines, the encryption method for the equipment manufacturer will not be available. If this file is modified in the CNC, it is recommended that these lines be deleted when the encryption is completed.

- For general encryption, define the password as "3RDPARTY".
[ENCRYPT]
Password=3RDPARTY
- For CNC-specific encryption, define the password as "3RDPARTY" (without quotation marks) and add the hardware ID of the CNC.
[ENCRYPT]
Password=3RDPARTY-CNC hardware ID.

How to encrypt a 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.

Encrypting the workpiece programmes.

Select the file and press the "Encrypt file" softkey and then select the option to delete the original. The programmes are usually located in the folder ..\Users\Prg\.

Encrypting OEM subroutines.

Select the file and press the "Encrypt file" softkey and then select the option to delete the original. Subroutines are usually found in the ..\MTB\Sub\ folder.

19.

UTILITIES MODE
Encrypting files.



CNCelite
8058 8060
8065 8070

REF: 2508

Encrypting OEM subroutines.

Select the file and press the "Encrypt file" softkey and then select the option to delete the original.

The CNC saves the encrypted subroutines in the folder ..\MTB\SUB\Encrypt . The subroutines are searched for first in ..\MTB\SUB and then in ..\MTB\SUB\Encrypt.

Encrypting scripts.

Select the script associated with the component (NameComponent.scp) and press the "Encrypt file" softkey and then select the option to delete the original. The scripts are found in the folder

..\MTB\MMC\Config\BinaryData\MyComponenet\MyComponent.scp

After encrypting a script, the BinaryDataBackup folder must be deleted before restarting the CNC; otherwise, the original script files (scp files) are restored if this folder has not been deleted.

19.

UTILITIES MODE
Encrypting files.



CNCelite
8058 8060
8065 8070

REF: 2508

19.

UTILITIES MODE

Encrypting files.



CNCelite
8058 8060
8065 8070

REF: 2508

20.1 Appearance of the PLC mode

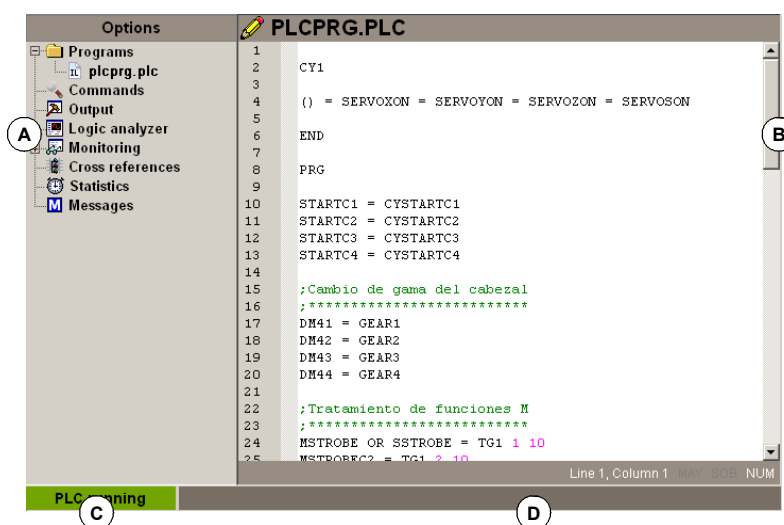
In this operating mode, the PLC may be accessed to check its operation or the status of the different PLC variables. It is also possible to edit and analyze the PLC program as well as the message and error files of the PLC.

At a CNC with a write-protected (read-only) disk, working in user mode, the PLC program is write-protected and any changes made to it are temporary; i.e. it will disappear the next time the unit is turned on. To make the changes permanent, unprotect the PLC program by starting the unit up in setup mode and validating the changes made.

Interface description.



The PLC mode screen shows the following information. To switch windows, press the [FOCUS] key.



- A Service window that shows the list of the services available in the PLC environment.
- B Data window.
- C It shows the PLC status, running or stopped.
- D CNC messages.

Select the PLC services.

This operating mode may be displayed with several screens. Each of them corresponds to a PLC service. The exchange between the different services is carried out with the list of services.



The different services overlap each other. If several services are active, it is possible to switch from one to another using the [NEXT] key, if it has been set accordingly. The services are switched in a rotary manner, in such a way that when pressing on the last active service, it shows the first one. To close a screen, press [ESC].

20.1.1 Icon description (vertical softkeys)

Horizontal softkey menu.

When selecting the PLC mode, the horizontal softkey menu will show the options associated with the PLC service currently selected. The set of options offered by this menu may be changed using the "+" softkey which offers a new set of options.

Vertical softkey menu.

The vertical softkey menu always shows all the options related to this operating mode.

Softkey.	Description.
	Show or hide the service window. This softkey toggles between the shared usage of the area of the PLC environment (that displays both the service window and the data window) and the full work screen (hiding the service window so the data window is expanded covering the whole area of the PLC environment).
	
	Find text. This softkey is used for searching a text in all the files of the project. The result will be shown at the "Outputs" service.
	Go to file. If the cursor is positioned on the result of a text search, a compiling error or a PLC resource, after selecting this option, it opens the corresponding file and the cursor goes to the line it refers to. This softkey only appears with the "Outputs" or "Cross references" service.
	Mnemonic language or contact language. This softkey toggles the display between the mnemonic language and the contact language (ladder diagram). This softkey only appears in the "Programs" service when monitoring a program.

20.

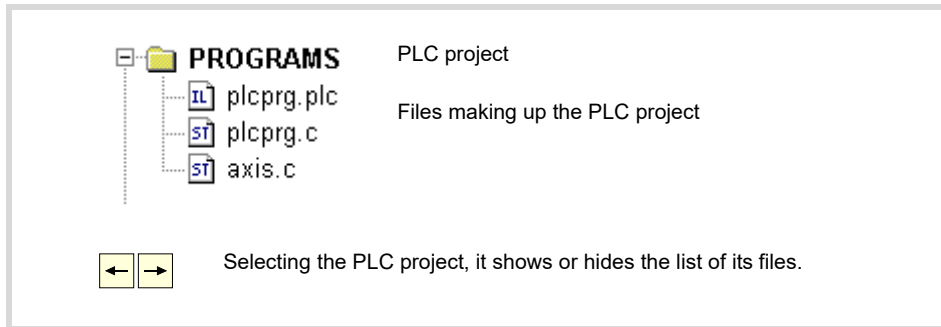
PLC

Appearance of the PLC mode



20.2 "Programs" service

This service is used for managing the PLC project and its files.



PLC project

The PLC project is a set of files that, once compiled, generate the PLC program.

When selecting the PLC project, the softkey menu will show the options to manage that project. Among these options, it is possible to add files to the PLC project and compile it.

Files of the PLC project

The PLC project may consist of several files edited in the following languages:

- Mnemonics or equations language (files with "plc" extension).
- Contac language (ladder diagram) (files with "ld" extension).
- C language (files with "c" extension).

The PLC project may consist of several files in the same language. Two files having the same name and different extension cannot belong to the same project.

When selecting a file, the softkey menu shows the options to manage that file. Among these options, it is possible to edit and monitor the file. These options will show the editing and monitoring screens.

20.

PLC
"Programs" service

20.2.1 Softkey menus. PLC project.

These options are used to create a PLC project and compile it to create the executable PLC program.

Softkey.	Description.
	Compile the PLC project to create the executable program.
	Add a file to the PLC project.
	Add a library to the PLC project.

PLC project "Generate PLC"

This option compiles and loads the PLC program based on the files that make up the PLC project. If an error occurs during compilation, the program will not be generated and the CNC will display a list of the detected errors.

The proper compilation of a file in mnemonic language or contact (ladder) language generates the equivalent files in both languages. That is why two files having the same name and different extension cannot belong to the same project because, when compiled, they are the same file.

PLC project "Add file"

This option is used to add a file to the PLC project. This file may be a new one or an existing one. When selecting this option, the CNC shows a list of the available files. To add a file to the PLC project.

- 1 Select the file from the list or write its name in the bottom window.
- 2 Press [ENTER] to accept the selection and add the file or [ESC] to cancel the selection and close the file listing.

When accepting the selection, the selected file will appear on the list that make up the PLC project.

20.

PLC
"Programs" service

20.2.2 Encrypting/decrypting from the PLC.



The PLC encryption algorithm is not the same as the one used in Utilities mode. Files encrypted with one algorithm cannot be decrypted by the other.

Once the PLC program has been compiled, the source files are not needed to operate the CNC and so they can be removed. If these files are to be kept, for example for future changes, the PLC can encrypt them so they are unreadable, meaning they cannot be used by a third-party. An encrypted program is not editable or visible during monitoring. If a machine service technician needs to modify the source files, these can be decrypted, modified and re-encrypted.

The encryption algorithm takes into account the PLC password. If this password does not exist when trying to encrypt a file, the CNC will show the relevant warning and will abort the process. After encrypting a file, if the password changes then PLC will not be able to decrypt these files.

Softkey menu.

Softkey.	Meaning.
	The source files cannot be found, have not been encrypted from the PLC or the CNC is not in Setup mode.
	Encrypting a PLC project. The CNC only displays this softkey in Setup mode, if there are PLC source files and there is no file that has been encrypted by the PLC.
	Decrypting a PLC project. The CNC only displays this softkey in Setup mode, if there is a file that has been encrypted by the PLC.

Encrypting files.

The "Encrypt" softkey deletes the files encrypted by the PLC (old files) and encrypts all the programs comprising the PLC project. After the files have been encrypted they cannot be edited or monitored. To encrypt the files, the PLC password must be used and the PLC project must be compiled.

Decrypting files.

The "Decrypt" softkey decrypts the files comprising the PLC project and deletes the encrypted files that have been successfully decrypted. Once decrypted, these files can be modified and monitored.

20.

PLC
"Programs" service

20.2.3 Softkey menus. Files of the PLC project.

When selecting a program of the PLC project at the service window, the softkey menu will offers the following options.

Softkey.	Description.
	Files of the PLC project "Edit". This softkey shows the editing window. See "20.3 Program editing" on page 355.
	Files of the PLC project "Monitoring". This softkey shows the monitoring window. See "20.6 Program monitoring" on page 368.
	Files of the PLC project "Eliminate". This softkey deletes the selected file of the PLC project (the file will still be available in the CNC's hard disk).
	Files of the PLC project "Rename". This softkey may be used to change the name of the selected file.
	Files of the PLC project "Copy". This softkey may be used to make a copy of the selected file. When selecting this option, the CNC shows a list with the programs stored at the CNC. To make a copy of the file: (1) Select the destination folder for the copy. (2) Define the file name at the bottom window. To replace an existing file, select it from the list. (3) Press [ENTER] to copy the file or [ESC] to cancel the selection and close the file listing.
	Files of the PLC project "Move up" When the PLC project contains several files, this softkey may be used to move the selected file up.
	Files of the PLC project "Move down" When the PLC project contains several files, this softkey may be used to move the selected file down.

20.

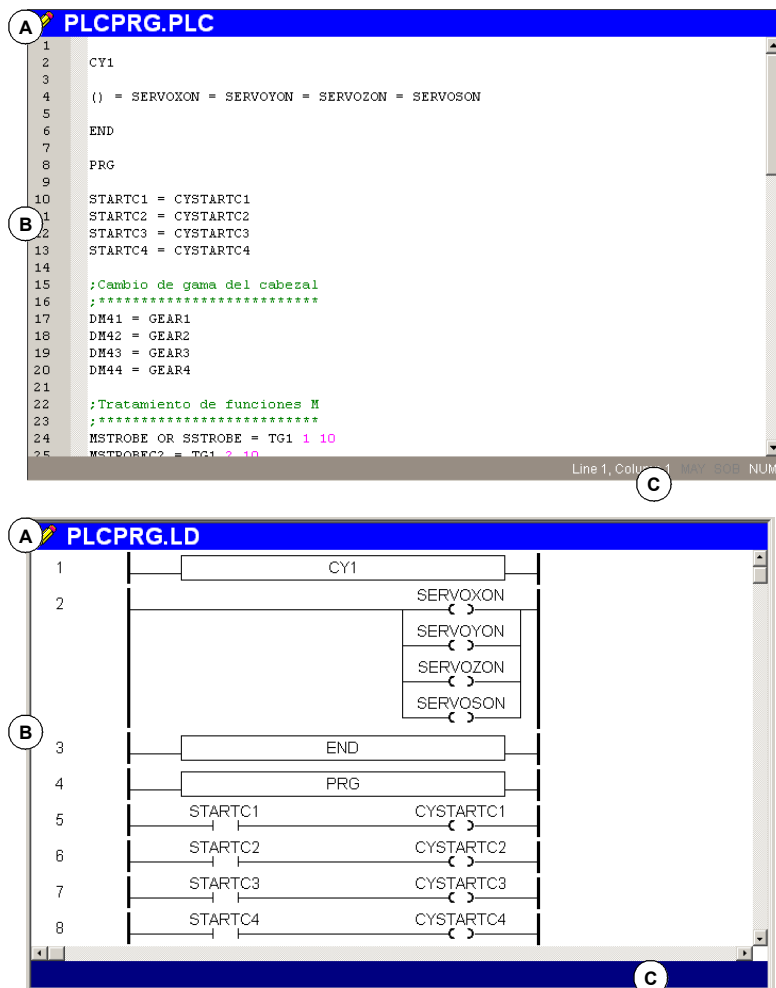
PLC
"Programs" service

20.3 Program editing

To edit a program, select it from the "Programs" service list and select the "Edit" option of the softkey menu. The PLC will show the right editor for the language of the selected program; a text editor if the program is in mnemonic language or C language or a contact (ladder) editor if the program is edited in contact (ladder) language.

Editor description

The text editor window (top image) and that of the contact (ladder) editor (bottom image) show the following information.



A Title bar.

Name of the program selected for editing. An "*" next to the program name means that the changes made to the program have not been saved (the program must be saved so they do not get lost).

B Edit area.

Line number and area for editing the program. A program edited in contact (ladder) language will contain numbered blocks representing the various elements.

C Status bar.

In a program edited in C language or mnemonic language, the bar shows information about the cursor position and the status of the editor options.

CAP Capital letters. When active, the text is always written in capital letters.

OVR Overwrite text. It toggles between overwriting and inserting text. When active, it overwrites the existing text.

NUM Numeric keypad active.

In a program edited in contact (ladder) language, the bar shows the comments of a contact and other messages.

Program editor, Unicode format.

The editor of the CNC admits Unicode characters. When saving a program, the editor respects its original format, ANSI or Unicode, except when editing a Unicode character, in which case the CNC will always save the program in Unicode format.



A third-party text editor (like Windows notepad) may be used to convert Unicode format programs into ANSI format; but special characters that have no ANSI equivalent will be lost in the process.

Functionalities of the editor.

Zoom at the editor.

The editor has the following hotkeys to increase or decrease the size of the editor font. If the CNC has a mouse with a wheel, the [CTRL] key combined with this wheel can also be used to increase and decrease the size of the text font.

[CTRL][+]	Zoom in.
[CTRL][-]	Zoom out.

Multi-line blocks.

```
POCK.)
L
.C.M=0
```



The editor adjusts the long blocks to the size of the window dividing the block into several lines. On the right side of each cut line, the editor shows a symbol to indicate that the block continues in the next line.

Softkey menu

When accessing the PLC program editor, the horizontal softkey menu will show all the options associated with editing a file depending on the editing language.

20.

PLC
Program editing



20.4 Editing in C language or mnemonic language.

Horizontal softkey menu.

Softkey.	Description.
Analyze.	Analyze the program searching for errors.
File.	Recall, save, import or print a file.
Undo.	Undo the last modifications.
Operations with blocks.	Copy text, paste text and export text as an independent file.
Find/replace	Find a line or a text in the program and replace a text with another one.
Customizing.	Set the behavior, properties and appearance of the PLC editor.

Hotkey menu.

The following softkeys may be useful when editing.

Hotkey.	Function.
[CTRL]+[C]	Copy the selected text.
[CTRL]+[X]	Cut the selected text.
[CTRL]+[V]	Paste the selected text.
[CTRL]+[Z]	Undo the last change.
[CTRL]+[Y]	Redo the selected text.
[CTRL]+[G]	Save the program/Recover the original program.
[CTRL]+[+]	Zoom in.
[CTRL]+[-]	Zoom out.
[CTRL]+[HOME]	Move the cursor to the beginning of the program.
[CTRL]+[END]	Move the program to the end of the program.

20.

PLC

Editing in C language or mnemonic language.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

20.4.1 Softkey "Analyze".

This softkey analyzes the program searching for errors. The errors found will be displayed in the "Outputs" service window.



To close this window and return to the editor, press [ESC]. To return to the editor without closing the window, press the [NEXT] key.

20.4.2 Softkey "File".



This softkey is used to recall, save, import or print a file. When selecting this option, the softkey menu shows the available options. To return to the main menu, press the [BACK] key.

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.

This option is only available when the "Save always" option is active. See ["20.4.6 Softkey 'Customize'."](#) on page 361.

File "Save"

This softkey saves the file that is being edited.

This option is only available when the "Save always" option is not active. See ["20.4.6 Softkey 'Customize'."](#) on page 361.

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.

File "Include program"

This softkey is used to import the contents of a file saved at the CNC into the program that is being edited.

File "Print"

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

20.4.3 Softkey "Undo".

This softkey may be used to "undo" the last modifications made. The modifications are undone one by one starting from the most recent one. The CNC offers the following keyboard shortcuts to undo and redo the operations.

[CTRL]+[Z]	Undo the last change.
[CTRL]+[Y]	Redo the selected text.



20.4.4 Softkey "Operations with blocks".

This 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.



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

Operations with blocks "Copy"

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.

20.

PLC
Editing in C language or mnemonic language.

20.4.5 Softkey "Find/Replace".

This 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].

20.

PLC

Editing in C language or mnemonic language.



20.4.6 Softkey "Customize".

This softkey is used for personalizing (customizing) the behavior, properties and appearance of the PLC editor.

Customizing the general options of the editor.

Option.	Meaning.
Save always.	Activating the automatic saving of the program. When this option is active, the CNC will automatically save the program every time the cursor changes blocks. In large programs (more than 200 kB), the CNC saves the program when the user has not modified the program for about 5 seconds. If this option is not active, the program is saved from the softkey menu. See "20.4.2 Softkey "File"." on page 358.
Line adjust.	Adjust the lines to the size of the window.
Show the line number.	Show the line numbers at the editor.
Drop menus.	Organize the horizontal softkey menu in drop menus.

Customize the appearance of the editor.

This option customizes the appearance (color, font, etc.) of the elements that make up the program editor. After defining the new look, press [ENTER] to accept the changes or [ESC] to ignore them.

Customize the syntax coloring.

This option may be used to customize the color of the elements that make up the program. After defining the new look, press [ENTER] to accept the changes or [ESC] to ignore them. In large programs (more than 200 kB), the editor cancels the syntax coloring.

20.

PLC
Editing in C language or mnemonic language.

20.5 Editing in contact (ladder) language (softkeys).

Horizontal softkey menu.

Softkey.	Description.
Analyze.	Analyze the program searching for errors.
File.	Restore, import or export a file.
Editing.	Edit the selected program and copy, cut and paste a block or group of blocks.
View.	Enlarge or reduce the size of the contacts and of the text.
Marks.	Add or remove follow-up marks in the program.
Find.	Find a text or a block by its number.
Customizing.	Set the properties and appearance of the PLC editor.

Hotkey menu.

The following softkeys may be useful when editing.

Hotkey.	Function.
[+] [-]	Window zoom.
[CTRL][+] [CTRL][-]	Cell zoom.
[ENTER]	It edits the element on which the cursor is located. If the cursor is on the left column, it will show the dialog for editing the block comment.
[←] [→] [↑] [↓]	It moves the cursor in all 4 directions.
[SHIFT][←] [SHIFT][→] [SHIFT][↑] [SHIFT][↓]	It changes the selection.

20.

PLC

Editing in contact (ladder) language (softkeys).



20.5.1 Softkey "Analyze".

This softkey analyzes the program searching for errors. The errors found will be displayed in the "Outputs" service window.



To close this window and return to the editor, press [ESC]. To return to the editor without closing the window, press the [NEXT] key.

20.5.2 Softkey "File".



This softkey is used to restore, import or export a file. When selecting this option, the softkey menu shows the available options. To return to the main menu, press the [BACK] key.

File "Save"

This softkey saves the file that is being edited.

File "Save as"

This softkey saves the file, that is being edited, with a different name. After saving the file, one keeps editing the new file.

Softkey "Import file"

This softkey imports the translation of a program edited in mnemonic language into the program that is now being edited. This softkey is useful for converting a mnemonic-language file into a contact (ladder)-language file without having to compile the PLC project.

The following expressions in mnemonic language cannot be translated directly into contact (ladder) language and the CNC will translate them as follows.

- Expressions with the XOR operator.
- Expressions where the NOT operator affects several contacts.

Original expression.	Transformed expression.
a XOR b	(a OR b) AND (NOT a OR NOT b)
NOT (a AND b)	(NOT a OR NOT b)
NOT (a OR b)	(NOT a AND NOT b)

File "Export file"

This softkey exports to a file the translation into mnemonic language of the program that is now being edited.

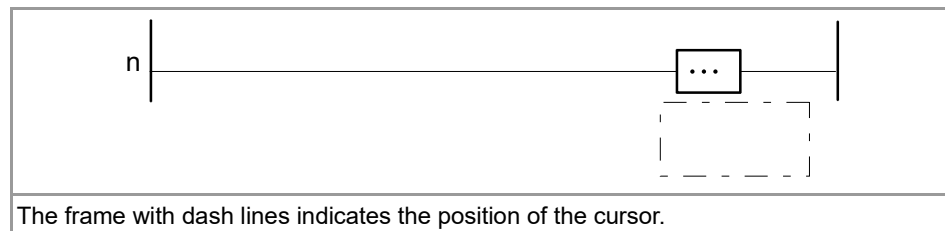
20.5.3 "Edit" softkey.



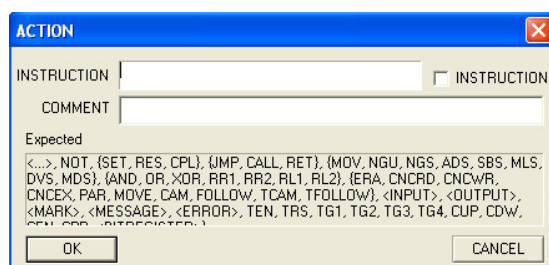
This softkey may be used to edit the selected program and copy, cut and paste a block or group of blocks. When selecting this option, the softkey menu shows the available options. To return to the main menu, press the [BACK] key.

Edit "New block".

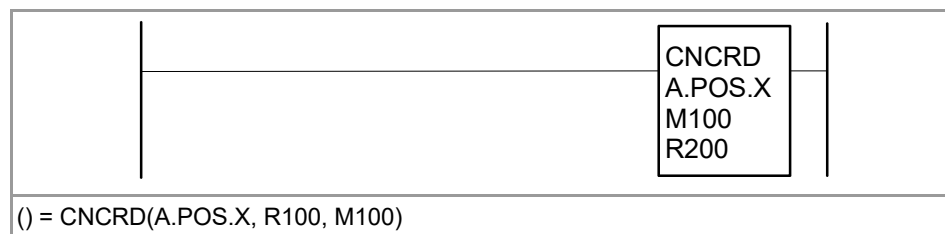
This softkey adds a new block with an empty contact. This contact will be the action for a consultation and "n" the number identifying the block. The new block is inserted above the block where the cursor is located.



Positioning the cursor on the action_contact and pressing [ENTER] displays a dialog box for associating an instruction and a comment with it. The "Expected" field offers a context help with valid instructions.



The instruction must be written in mnemonic language. Placing the cursor on action_contact and using the "Parallel" softkey, it is possible to add other action_contact in parallel.



To define a directing instruction, activate the "Directing" field in the dialog box and then write an instruction. Directing instructions can only be programmed in a block with their action_contact empty and without consultation_contact.



Edit "Cut"

Cut the selected contact or block.

Edit "Copy"

Copy the selected contacts or blocks.

Edit "Paste"

Paste the contacts or blocks that have been previously cut or copied. It pastes them in parallel with the selection.

Edit "Delete"

Delete the selected contacts or blocks.

20.

PLC

Editing in contact (ladder) language (softkeys).

Edit "Undo"

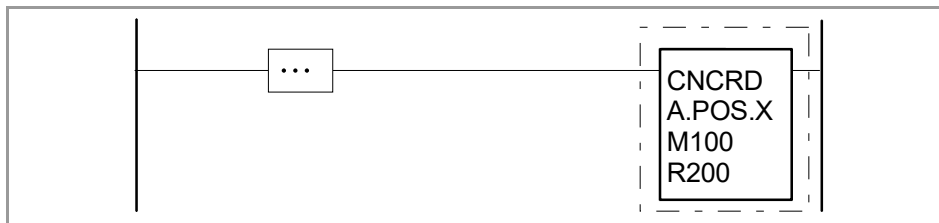
This softkey may be used to "undo" the last modifications made. The modifications are undone one by one starting from the most recent one.

Edit "Redo"

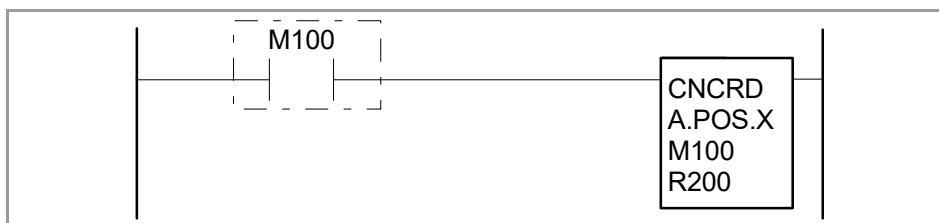
This softkey is active after using "Undo" and restores one by one the previous status of the program, before "Undo" was used.

Edit "Left series"

This softkey adds a consultation_contact to the left of the contact selected with the cursor.



Positioning the cursor on the consultation_contact and pressing [ENTER] displays a dialog box for associating an instruction and a comment with it. The "Expected" field offers a context help with valid instructions.

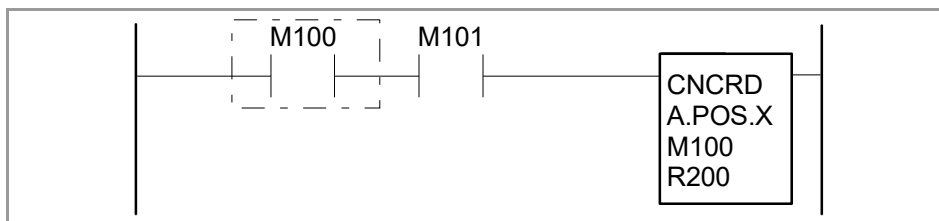


For example, the user can make it to be a contact for comparing two registers with the instruction "CPS R1 EQ R2" or a simple contact of the mark "M100"

In mnemonics, it is the same as $M100 = \text{CNCRD}(\text{A.POS.X}, R100, M100)$

Edit "Right series"

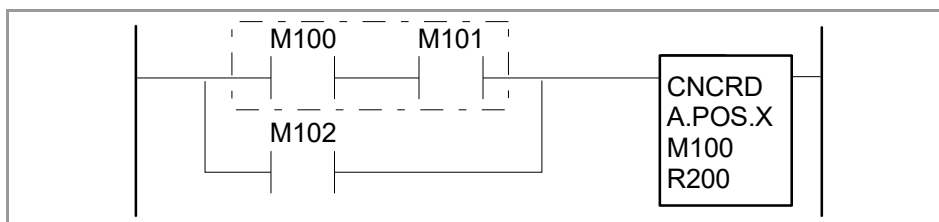
This softkey adds a consultation_contact to the right of the contact selected with the cursor.



$M100 \text{ AND } M101 = \text{CNCRD}(\text{A.POS.X}, R100, M100)$

Edit "Parallel"

This softkey adds a consultation_contact in parallel to the contact selected with the cursor.



$(M100 \text{ AND } M101) \text{ OR } M102 = \text{CNCRD}(\text{A.POS.X}, R100, M100)$

20.**PLC**

Editing in contact (ladder) language (softkeys).

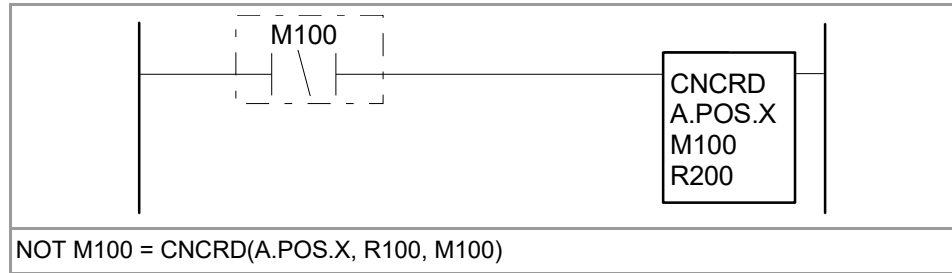
FAGOR
 AUTOMATION

CNCelite
 8058 8060
 8065 8070

REF: 2508

Edit "Negate"

This softkey negates the consultation_contact selected with the cursor.

**20.5.4 Softkey "View".**

This softkey may be used to enlarge or reduce the size of the contacts and of the text. When selecting this option, the softkey menu shows the available options. To return to the main menu, press the [BACK] key.

View "Zoom +"

Increase the size of the font as well as the height and width of the contacts.

View "Zoom -"

Reduce the size of the font as well as the height and width of the contacts.

View "Cell +"

Increases the width of the contacts.

View "Cell -"

Reduce the width of the contacts.

20.5.5 Softkey "Marks".

This softkey may be used to add or remove follow-up marks in the program. When selecting this option, the softkey menu shows the available options. To return to the main menu, press the [BACK] key.

Marks "Set / remove"

Place a mark or remove the mark of the block where the cursor is located.

Marks "Next mark"

Place the cursor on the next marked block.

Marks "Previous mark"

Place the cursor on the previous marked block.

Marks "Remove all"

Remove all the marks.

20.**PLC**

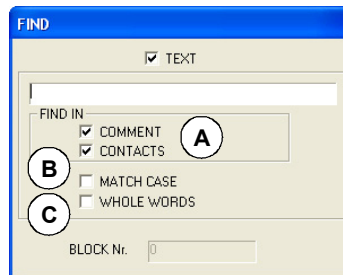
Editing in contact (ladder) language (softkeys).

20.5.6 Softkey "Find".

This softkey is used to find a text or a block by its number.

Find "Find"

When selecting this option, the CNC shows a dialog box requesting the block number or the text to look for. When defining a text search, certain options may also be defining that allow:



- A Select the types of fields in which to search.
- B Ignore uppercase and lowercase.
- C Consider the text to find as a whole word.

After defining the search options, press [ENTER] to do the search or [ESC] to cancel it. Pressing [ENTER] positions the cursor in the first field that matches the search parameters.

Find "Find next"

It positions the cursor in the next field that matches the search parameters.

Find "Find previous"

It positions the cursor in the previous field that matches the search parameters.

20.5.7 Softkey "Customize".

It is used for personalizing the appearance (color, font, etc.) and properties of the editing window.

After defining the new look, use the cursor to select one of the buttons here below to accept or ignore the changes made and press [ENTER]. The dialog box may also be closed directly without accepting the changes by pressing [ESC].

20.

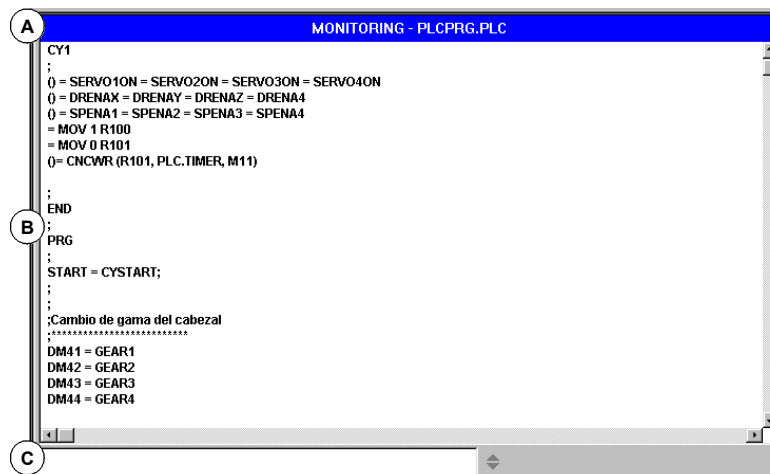
PLC
Editing in contact (ladder) language (softkeys).

20.6 Program monitoring

To monitor a program, select it from the list of the service window and select the "Monitoring" option of the softkey menu. The CNC will display the monitoring window and it will show the instructions of the selected program. To close the monitoring window, press [ESC].

Description of the monitoring window

The editing window shows the following information:



- A** Title bar. It shows the name of the program that is being monitored.
- B** Monitoring area. It shows the status of the instructions being executed. It is a real-time monitoring. Only the instructions that are being executed are monitored.
The PLC will show, in a different color, the instructions that are not being executed such as the first cycle or those subroutines whose call is not active. The default colors are:
 Red Active variables.
 Green Instruction that is not being executed.
 Black Inactive variable or comment.
- C** In a program edited in C language or in mnemonic language, it is the data entry zone. This zone is used to modify the values of the PLC resources.
In a program edited in contact (ladder) language, it is the zone that shows the comments of a contact and other messages. In the "Monitoring" service, it is not possible to change the names or comments associated with the contacts.

20.6.1 Softkey menus. Monitoring in C language or mnemonic language.



This softkey toggles the display between the mnemonic language and the contact language (ladder diagram). The softkeys of the horizontal menu change according to the selected language.

In a program edited in C language or in mnemonic language, the softkey menu shows the following options.

Monitoring "Size +"

Every time this softkey is pressed, the size of the text will increase.

Monitoring "Size -"

Every time this softkey is pressed, the size of the text will decrease.

Monitoring "Bold"

When selecting this option, the text of the program appears in bold.

Monitoring "Find"

This softkey is used to find text in the program. When selecting this option, the CNC shows a dialog box requesting the text to be found. Key in the text and press [ENTER] to begin the search.

As the text is being searched, the cursor will position on the match found. To end the search, press [ESC].

Monitoring "Customize"

It is used to personalize certain functions of the monitoring window. When selecting this option, the CNC shows a dialog box with the options available for customizing.

20.

PLC
Program monitoring

20.6.2 Softkey menus. Monitoring in contact (ladder) language.



This softkey toggles the display between the mnemonic language and the contact language (ladder diagram). The softkeys of the horizontal menu change according to the selected language.

In a program edited in contact (ladder) language, the softkey menu shows the following options.

View.



This softkey may be used to enlarge or reduce the size of the contacts and of the text. When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

View "Zoom +"

Increase the size of the font as well as the height and width of the contacts.

View "Zoom -"

Reduce the size of the font as well as the height and width of the contacts.

View "Cell +"

Increases the width of the contacts.

View "Cell -"

Reduce the width of the contacts.

Marks.



This softkey may be used to add or remove follow-up marks in the program. When selecting this option, the softkey menu shows the available options. To return to the main menu, press the key for the previous menu.

Marks "Set / remove"

Place or remove a mark on the block where the cursor is located.

Marks "Next mark"

Place the cursor on the next marked block.

Marks "Previous mark"

Place the cursor on the previous marked block.

Marks "Remove all"

Remove all the marks.

Find

This softkey is used to find a text or a block by its number.

Find "Find"

When selecting this option, the CNC shows a dialog box requesting the block number or the text to look for.

Find "Find next"

This option positions the cursor in the next field that matches the search parameters.

20.

PLC
Program monitoring

FAGOR
AUTOMATION



CNCelite
8058 8060
8065 8070

REF: 2508

Find "Find previous"

This option positions the cursor in the previous field that matches the search parameters.

Customizing

This softkey is used for personalizing the appearance and properties of the editing window. When selecting this option, the CNC displays a dialog box showing the elements that may be set.

Activate

This softkey is used to turn monitoring on or off. The program is not monitored until the "Activate" option is pressed.

20.

PLC

Program monitoring

20.7 "Commands" service

The "Commands" service is used to debug the PLC program execution by taking advantage of the possibility to execute the various parts of the program separately (first cycle, main module and periodic module). This service also offers the possibility of starting up and interrupting the execution of the PLC program.

When selecting this service, the horizontal softkey menu offers all the options associated with this service.

20.7.1 Softkey menus. Options of the "Commands" service.

Commands "Run"

This softkey is used to start the execution of the PLC program. The PLC executes the first cycle (CY1) once and continues with the cyclic execution of the main program (PRG) and the periodic module (PE). The main program is executed according to the frequency defined by machine parameter. The periodic module is executed according to the frequency defined in the program.

Commands "Stop"

This softkey is used to stop the execution of the PLC program.

Commands "CY1"

This softkey is used to execute the part of the program corresponding to the first cycle (CY1). The CNC does not execute this option when the PLC program is running.

Commands "Cycle"

This softkey is used to execute the main program (PRG) once. The CNC does not execute this option when the PLC program is running.

Commands "Resume"

This softkey is used to resume the cyclic execution of the PLC program. The CNC does not execute this option when the PLC program is running.

Commands "PE module"

This softkey is used to execute the periodic module (PE) once. The CNC does not execute this option when the PLC program is running.

20.

PLC
"Commands" service

20.8 "Outputs" service

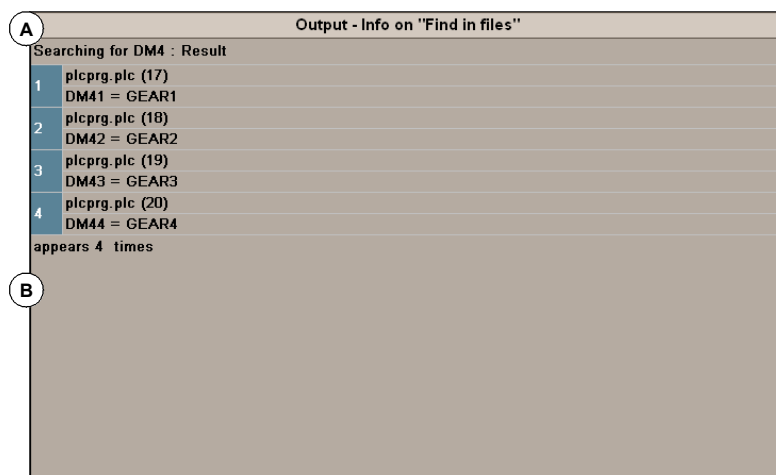
The "Outputs" service shows the following information:

- The result of generating (compiling) the PLC program.
- The result of analyzing a file of mnemonics.
- The result of text search in the files making up the PLC project.

Pressing the "Show" softkey accesses the screen of this service that shows this information.
To close the window, press [ESC].

Description of the outputs service

The screen of this service looks like this:



A Title bar.

It shows the type of information displayed in the window (information about the compilation, file analysis or a search).

B Requested information.

It shows the requested information.

20.8.1 Softkey menus. Options of the "Outputs" service.

When accessing the "Outputs" service, the softkeys will show the following option.

Outputs "Compiling info"



This softkey shows the result of compiling (generating) the PLC program. The screen shows a list of warning or errors detected when compiling the PLC program. After selecting a warning or error from the list, it is possible to go to the erroneous line by pressing the relevant icon.

Outputs "Checking info"

This softkey shows the result of analyzing the file of mnemonics. The screen shows a list of the error detected when analyzing the program. After selecting a warning or error from the list, it is possible to go to the erroneous line by pressing the relevant icon.

Outputs "Search info"

This softkey shows the result of searching text in the programs making up the PLC project. After selecting an element from the list, it is possible to go to its line by pressing the relevant icon.

20.

PLC
"Outputs" service

20.9 "Logic analyzer" service

The "Logic analyzer" service is used to analyze the behavior of the logic PLC signals according to a time base and certain trigger conditions set by the user.

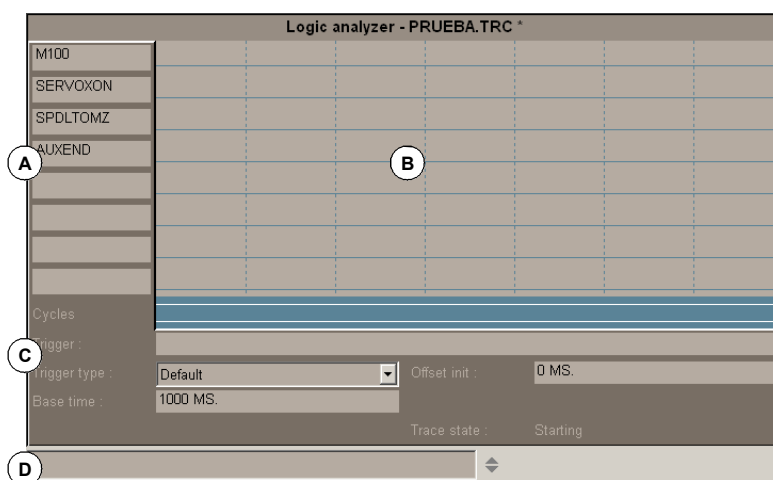
Up to 8 PLC variables or expressions may be analyzed at the same time and the result of the analysis (the traces) will appear on a graphic interface that facilitates the interpretation of the data. Once the done capturing data, the user may:

- Modify the time base to display different zooms of the traces.
- Move along the traces to display points, times, time differences, etc.

To close the window, press [ESC].

Description of the logic analyzer

The logic analyzer looks like this:



A The data area is used to define the variables or expressions to be analyzed. Up to 8 PLC variables or expressions may be defined.

B The graphic area shows the traces for the defined PLC variables or expressions and a trace of the PLC cycles with the indicated conditions.

Also, a vertical red line is displayed to indicate the trigger point (if any) and a green line to indicate the cursor position.

C The information area is used to define the data that conditions the display of the traces (trigger, type of trigger, time base and trace status).

D The data entry area is used to modify the PLC resources.

Softkey menus

When selecting the logic analyzer, the horizontal softkey menu offers all the options associated with this service. The set of options offered by this menu may be changed using the "+" softkey which offers a new set of options.

20.9.1 Editing logic analyzer data

In order to capture trace data, the user must enter the variables or expressions to be analyzed, the trigger type and conditions as well as the time based to be used to display the captured values. To edit this data, select the graphic area of the logic analyzer and press the "Edit view" softkey to place the cursor on the data editing area.

Key.	Meaning.
	Move the cursor through the data.
	Move the cursor within a data.
	Open/close pop-up menus.
	Validate the defined data.
	Cancel edit.

Definition of variables

Up to 8 PLC variables or expressions may be defined to obtain their trace. The definition of the expression cannot exceed 80 characters.

When modifying a variable, if there was already a trace of that expression, the trace will be deleted when validating the new edition.

Trigger condition

A trigger condition is the one used to capture data and may be defined with a PLC variable or expression. The definition of the expression cannot exceed 80 characters.

If the trigger condition is modified after a data capture, all the traces will be deleted when validating the new condition.

Trigger type

It sets whether the data is to be captured "before", "after" or "before and after" the selected trigger condition is met.

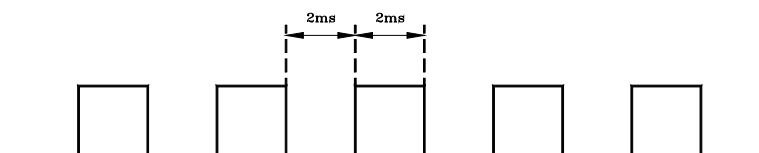
By default	The data capture starts and ends when the operator selects the option to execute and stop the trace.
Before	The data capture starts when the trigger condition is met and ends when the operator selects the option to stop the trace. Once the trace has been executed, the trigger signal will be shown at the beginning of the trace.
After	The data capture starts when the operator selects the option to execute the trace and ends when the trigger condition is met. Once the trace has been executed, the trigger signal will appear at the end of the trace.
In the middle	The data capture starts and ends when the operator selects the option to execute and stop the trace. Once the trace has been executed, the trigger signal will be shown at the center of the trace.

Time base

The operator may use this parameter to assign the timeframe to each vertical division. The size of these divisions, therefore the resolution of these signals will be determined by this time base. Consequently, the smaller the time base, the greater the signal resolution.

The value is given in milliseconds or microseconds according to the active units and the selected value will be displayed in the information area.

There is a mark that changes states every 2ms.



20.

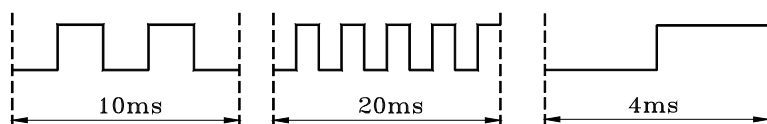
PLC
"Logic analyzer" service

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

With a time base of 10ms, 20ms and 4ms it will look like this.



While the graphic window is selected, use the "+" and "-" keys to divide or multiply this time base by two.

Start distance / Trigger distance

This data is only shown when there is a trace.

Start distance	It represents the time difference between the indicator cursor and the base point of the trace (beginning of the trace if there is no trigger point or it has not been reached).
Trigger distance	It represents the time difference between the indicator cursor and the trigger point (if it has taken place).

The value is updated when moving the indicator cursor of the graphic area. Its value may also be edited, thus updating the cursor position in the graphic area.

Reference distance

This data is only shown when there is a trace already and the operator has put a reference signal. It represents the time difference between the indicator cursor and the reference signal.

The value is updated when moving the indicator cursor of the graphic area. Its value may also be edited, thus updating the cursor position in the graphic area.

Trace status

This element cannot be edited, it automatically reflects the status of the trace. The possible messages are:

- Empty.
- Beginning.
- Executing.
- Stopping.
- Full.

20.

PLC

"Logic analyzer" service

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

20.9.2 Save, load and reset the analyzer configuration

Save the configuration

This option is used to save the current configuration of the logic analyzer (PLC variables and expressions, trigger conditions, graphic traces) into a file. When selecting this option, the CNC shows a list of trace files (TRC) already saved. To save the current configuration:

- 1 Define the file name at the bottom window. To replace an existing file, select it from the list.
- 2 Press [ENTER] to save the configuration or [ESC] to return to the logic analyzer without saving the configuration.

Load the configuration

This option is used to recover a logic analyzer configuration that has been stored earlier. When selecting this option, the CNC shows a list of trace files (TRC) already saved. To load one of these files:

- 1 Define the file name at the bottom window or select it from the list.
- 2 Press [ENTER] to save the configuration or [ESC] to return to the logic analyzer without loading the configuration.

Initialize the configuration

This option is used to reset all the analyzer data eliminating the defined variables or expressions as well as the trigger condition and trigger type. The traces are also deleted because there are no variables to be analyzed.

20.

PLC
"Logic analyzer" service

20.9.3 Execute and analyze trace

Execute trace / Stop trace

This softkey is used to start or stop the data capture to display the traces.

Analyze trace



This softkey is used to analyze the different graphic aspects of the trace. When selecting this option, the softkey menu shows the options available for analyzing the trace. To return to the main menu, press the [BACK] key.

- Go to the beginning
It moves the indicator cursor and the current graphic view to the beginning of the trace.
- Go to the end
It moves the indicator cursor and the current graphic view to the end of the trace.
- Go to the time
It moves the indicator cursor and the current graphic view to the time value set by the operator with respect to the base point of the trace (starting point of the trace if there is no trigger or to the trigger point if there is one).
- Go to trigger
It moves the indicator cursor and the current graphic view to the trigger point.
- Go to reference
It moves the indicator cursor and the current graphic view to the reference point defined earlier.
- Set reference
It sets a reference point at the current cursor position for time difference calculation.
- Remove reference
Removes the reference point defined earlier.

Trace view / Edit view

This softkey toggles between the graphic window and the data and display conditions area.

MS (milliseconds) / US (microseconds)

This softkey toggles between milliseconds and microseconds.

Print

This softkey may be used to print the graphics in a printer connected to the CNC or as a file (*.BMP format) at the CNC. When printing to a file, it is saved in the folder:

"C:\Cnc8070\Mtb\Plc\Watch*.bmp"

20.9.4 Customize the appearance of the logic analyzer

It is used to customize the looks of the different elements of the logic analyzer. When selecting this option, the CNC shows a dialog box with the options available for customizing.

20.

PLC

"Logic analyzer" service

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

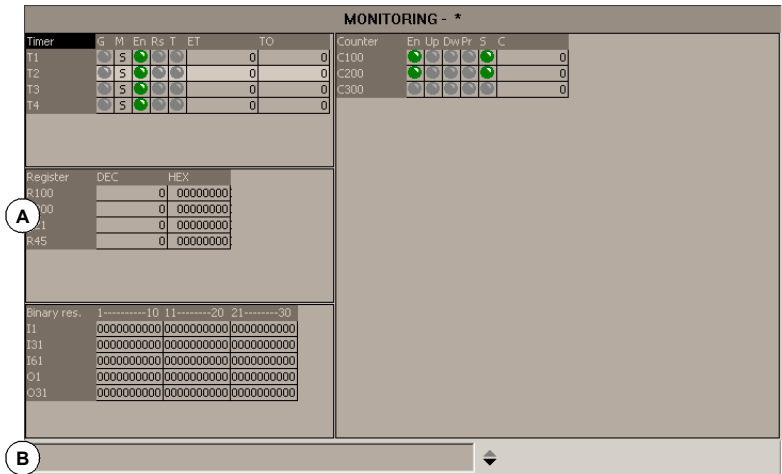
20.10 "Monitoring" service

This service is used to analyze the status of the various PLC variables and resources. When selecting the "Monitoring" option at the service window, it will show a list with the three resource files used last. This will let you recover one of them without having to define it again.

Pressing the "Show" softkey access the service screen. If a file was selected from the list, it will show the resources defined in it. To close the monitoring window, press [ESC].

Description of the monitoring service

This service screen is divided into several windows that may be accessed with the window switching key. The monitoring screen looks like this:



A Resources selected for monitoring. This area shows the user resources and symbols being analyzed. The resources and symbols are shown grouped in the following tables, each of them shows information on the status of those resources.

Timers

Counters

Registers

Binary resources

B Data entry area. This area is used to define the user resources and symbols to be analyzed and modify their values by directly assigning a value to them.

Softkey menus

When selecting this service, the horizontal softkey menu will show all the options associated with resource monitoring. The set of options offered by this menu may be changed using the "+" softkey which offers a new set of options.

20.10.1 Description of resource tables

As mentioned earlier, the user resources and symbols defined are displayed in different tables. Each table shows the following information.

Timers

This table shows the timers and user defined symbols for the timers and their status. This table has the following fields:

Field.	Meaning.
G	It indicates whether any of the trigger inputs of the timer is active (TG1, TG2, TG3 or TG4).
M	Timer status: (S) Stopped / (T) Timing / (D) Disabled.
E	Status of the Enable input (TEN).
R	Status of the Reset input (TRS)
T	Status output.
ET	Elapsed time.
TO	Remaining time.

The inputs and outputs that are high will be indicated with a green symbol.

Registers

This table shows the registers and user symbols defined for the registers as well as their values.

The values may be shown in decimal, hexadecimal or binary format.

Binary resources

This table shows the binary resources (inputs, outputs, messages, errors) and the user defined symbols for the binary resources as well as information on their status.

Counters

This table shows the counters and user symbols defined for the counters as well as their status. This table has the following fields:

Field.	Meaning.
E	Status of the enable input (CEN).
U	Status of the Count-up input (CUP).
D	Status of the Countdown input (CDW).
P	Status of the preset input (CPR).
S	Status output.
C	Value of the internal count of the counter.

The inputs and outputs that are high will be indicated with a green symbol.

20.

PLC
"Monitoring" service

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

20.10.2 Definition of the table resources

Each table shows the following resources and user symbols:

Timer table

- Timers (T1...T512).
- User symbols defined for the timers.

Counter table

- Counters (C1...C256).
- User symbols defined for the counters.

Register table

- Registers (R1...R1024).
- User symbols defined for the registers.
- PLC-CNC register variables.

Binary resource table

- Inputs (I1...I1024) and outputs (O1...O1024).
- Marks (M1...M8192).
- Messages (MSG1...MSG1024) and errors (ERR1...ERR1024).
- User symbols defined for the binary resources.
- CNC-PLC boolean variables.

When defining a resource or user symbol, it will be added to the corresponding table.

Data entry

The resources and user symbols of the tables are defined at the data entry area. Their values may be changed by directly assigning a value to them (M110=1, R300=34). If it is a hexadecimal value, it must be preceded by the "\$" sign (M10=\$1, R200=\$20).

The window saves the last "N" assignments so they can be recalled later on. In order to make selecting easier, it is possible to display a window with the list of the assignments already made.

Key.	Meaning.
	Move the cursor through the list of expressions.
	Select an expression from the list.
	Show the list of expressions.
	Close the list of expressions.

To remove a resource or user symbol from a table, select that element and press [SUP].

Key.	Meaning.
	Select a table.
	Move the cursor.
	Move the cursor page by page.
	Move the cursor to the beginning / end of the table.
	Delete the selected resource from the table.

20.

PLC
"Monitoring" service

FAGOR
AUTOMATION



CNCelite
8058 8060
8065 8070

REF: 2508

20.10.3 Options of the "Monitoring" service (softkeys).

When accessing the "Monitoring" option of the softkey menu, the following options will be available.

Save set

This softkey is used to save the set of defined resources into a file. When selecting this option, the CNC displays a window with the list of the files (.MON) currently saved. To save a set of defined resources:

- 1 Define the file name at the bottom window. To replace an existing one, select it from the list.
- 2 Press [ENTER] to save the program or [ESC] to cancel the selection and close the program listing.

Load set

This softkey is used to recover the set of resources previously saved into a file. When selecting this option, the CNC shows a window with the list of available files. To load one of these files:

- 1 Define the file name at the bottom window or select it from the list.
- 2 Press [ENTER] to load the selected file or [ESC] to cancel the operation and close the program listing.

Binary

This softkey toggles between displaying the register values in decimal and hexadecimal format or only in binary.

Visibility

This softkey shows or hides certain resource tables.

Add row

This softkey increases the size of the active window (where the cursor is) by adding a row to it.

Remove row

This softkey decreases the size of the active window (where the cursor is) by removing a row from it.

Data input

This softkey selects the data entry area.

Clear all

Delete all the resources from the tables.

20.

PLC
"Monitoring" service

20.11 "Cross references" service

The "Cross references" service may be used to obtain information about the PLC resources being used in the PLC project. Pressing the "Show" softkey accesses the window of this service. To close the cross-reference window, press [ESC].

Description

The screen of this service looks like this:

Cross references - Marks			
Name	File	Line	Use
AUXEND	plcprg.plc	31	=
AUXENDC2	plcprg.plc	32	=
AUXENDC3	plcprg.plc	33	=
AUXENDC4	plcprg.plc	34	=
CYSTARTC1	plcprg.plc	10	=
CYSTARTC2	plcprg.plc	11	=
CYSTARTC3	plcprg.plc	12	=
CYSTARTC4	plcprg.plc	13	=
DM41	plcprg.plc	17	?
DM42	plcprg.plc	18	?

- A Name of the resources being used by the PLC project.
- B File of the PLC project where the resource is being used.
- C Line, within the file, containing the resource.
- D How the resource is being used.

For the inputs, outputs, marks and registers:

? The resource is used as a consultation.

= The resource is used as an action.

For the timers.

TG Timer trigger.

T Status consultation.

TEN Enable input.

TRS Reset input.

For the counters.

C Status consultation.

CPR Counter preset.

CEN Enable input.

CUP Count-up input.

CDW Countdown input.

20.11.1 Softkey menus. Options of the "Cross reference" service.

When accessing the "Cross reference" service, the softkey menu will offer the following options:

Inputs

Show information about the inputs.

Outputs

Show information about the outputs.

Marks

Show information about the marks.

Registers

Show information about the registers.

Timers

Show information about the timers.

Counters

Show information about the counters.

Print

It is used to print the cross reference tables out to a printer connected to the CNC or as a "*.PRN" file at the CNC. When printing to a file, it is saved in the folder:

"C:\Cnc8070\Users\Reports*.prn"

When selecting this option, the CNC shows a dialog box requesting the destination for the file (printer or file). When printing to a file, it is possible to select the name and location of the file. After selecting the destination, press [ENTER] to print the table or [ESC] to cancel it.

20.

PLC

"Cross references" service

20.12 "Statistics" service

The "Statistics" service may be used to obtain information about the execution times of the PLC and the files making up the PLC project. Pressing the "Show" softkey access the service screen. To close the statistics screen, press [ESC].

Description

The screen of this service looks like this:

Statistics				
Times	Minimum	Maximum	Average	Frequency
CY1	---	---	0 us	---
PRG	0 us	1000000 us	21 us	8000 us
PE	---	---	---	---
PLC Process	0 us	1000000 us	21 us	4000 us
Files	Size	Type	Modified	
PLCPRG.PLC	1 KB	IL source file	21/01/2010 14:16:40	

A Execution time-table.

This table shows the following data (from left to right):

- Modules making up the PLC program.
- Minimum module execution time.
- Maximum module execution time.
- Average module execution time.
- Module execution frequency.

B PLC file table.

This table shows the following data (from left to right):

- Files making up the PLC project.
- Size of each file.
- File type.
- Date last modified.

20.12.1 Softkey menus. Options of the "Statistics" service.

Softkey.	Description.
	This softkey refreshes the table information.
	This softkey may be used to print the table in the pre-determined printer or save it as a file (prn format) at the CNC. When selecting the "File" option, it will be saved in the folder "C:\Cnc8070\Users\Reports\". When selecting this option, the CNC will show a dialog box requesting the print destination (printer or file). After selecting the destination, press [ENTER] to print it or [ESC] to cancel it.

PLC

"Statistics" service

20.

20.13 "Messages" service

The "Messages" service may be used to edit the texts associated with PLC messages and errors. These files may also be displayed and edited (and thus translated into other languages) using any text editor.

The message and error files are stored in the folder: "C:\CNC8070\MTB\PLC\LANG\<language>" corresponding to the language active at the CNC. To use messages and errors in other languages, copy the files to the folder of the desired language. On CNC power-up, the messages and errors are loaded from the folder corresponding to the language active at the CNC.

Pressing the "Show" softkey access the service screen. To close the message window, press [ESC].

Description

This screen shows the messages (MSG) and errors (ERR) defined at the PLC. The screen shows the following table.

A	B	C	D	E
ERR1	☐			ERROR-1-
ERR2	☒			ERROR-2-
ERR3	☐			ERROR-3-
ID		3		
Show	☐			
Message		ERROR-3-		
Related file		C:\CNC8070\MTB\PLC\Lang\msg3.jpg		
Emergen	☒			
MSG1	☐			MSG-1-
MSG2	☐			MSG-2-
MSG3	☐			MSG-3-
MSG4	☐			MSG-4-

A Message or error identifier.

Number, between 1 and 1024, that identifies the message (MSG) or the error (ERR). The numbering needs not be sequential; i.e. number skipping is allowed.

B Display of the additional information files. This field indicates whether to show or not the file for additional information when displaying the message or the error.

C Address of the additional data file which may be a text file (*.txt), image file (*.bmp, *.jpg), a combination of both (*.htm, *.html) or a video (*.avi).

D Opening the CNC or emergency relay or not. This field appears selected by default when editing a new error. When editing messages, this field does not appear because PLC messages do not activate the emergency signal.

E Text of the message or error displayed for the user.

20.13.1 Softkey menus. Options of the "Messages" service.

Softkey.	Description.
	Save the message and error table into a file in ASCII format (*.MEF).
	Recover the table values saved earlier in the hard disk of the CNC.
	Print the message and error table out to a printer connected to the CNC or as a file (*.PRN format) at the CNC.
	Define a new message in the table.
	Define a new error in the table.
	Select the file for additional information.
	Delete the selected message or error from the table.

PLC

"Messages" service

20.

20.13.2 Editing the message and error table

To add a new message or error to the table, press the "New message" or "New error" softkey. The CNC adds a new row to the table and it identifies it with the label -MSG- or -ERR-, indicating that it corresponds to a message or to an error. To delete a message or an error from the table, select it with the cursor and press the "Delete" softkey.

To edit or modify the table data, use the cursor to select the field whose value is to be changed and define the data as follows:

- The number and text of the message or error is entered directly from the keyboard.
- The "show" option is turned on and off with the [SPACE] key.
- The "EMERGEN" option is turned on and off with the [SPACE] key. This option only appears when editing errors.
- The file for additional information may be entered directly or may be selected with the "Associate file" softkey.

Press [ENTER] every time a data is edited to accept the new value or [ESC] to cancel it.

20.

PLC

"Messages" service

20.13.3 Displaying PLC messages

When activating a message (marks "MSG1" through "MSG1024"), the PLC message window (top right) shows the message number and its associated text. If the message has been defined so it shows a file with additional information, it will be displayed at full screen (if the file does not exist, a blue screen will be displayed).

If there are several active messages, it will display the one with highest priority (the one with lowest number). It will also show the "+" sign next to the PLC message window to indicate that there are more messages activated by the PLC.

Message window

In order to expand the PLC message window and display the active message list, press the key combination [CTRL]+[M]. The list will show a symbol next to each message to indicate that it has a file with additional information associated with it.



It does not have a file with additional information.



It has a file with additional information.

To display a message, select it with the cursor and press [ENTER]. If the message has an additional information file, it will be displayed on the screen. To close the additional information window, press [ESC].

File with additional information

The files with additional information, defined by the machine manufacturer, may be displayed automatically when activating the message (if so defined in the table for messages and errors) or when requested by the user (at the message window). The following keys are available to move around in the additional information window depending on the type of file displayed (TXT, BMP, JPG, HTM, HTML or AVI):

TXT, BMP, JPG, HTM, HTML file.

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.

20.

PLC
"Messages" service

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

20.13.4 Displaying PLC errors

When activating an error (marks "ERR1" through "ERR1024"), part-program execution is interrupted and the center of the screen displays a window with the error number and its associated text. If the error has been defined so it shows a file with additional information, it will be displayed at full screen. If the error has the "Emergen" field selected, the error will open the emergency relay of the CNC.

Error window.



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

When there is an active error, no other action but eliminating the error state is allowed. Although the window displaying the errors may be closed by pressing [ESC], it does not mean that the error status has been taken care of. To do that, press [RESET]. Pressing the [RESET] key assumes the initial conditions.

File with additional information

The files with additional information, defined by the machine manufacturer, may be displayed automatically when activating the 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.

20.13.5 Grouping the additional information text files in a single file.

PLC messages and errors can show an additional information file in text format. The PLC allows grouping several or all these files into a single file as follows.

Defining the additional information file.

It must be a text file (extension txt) and may have any name. The information of each message and error must be structured in the following format:

[<id>]

<text>

The <id> field, keeping the brackets, will be the code that identifies the help text inside the file, which needs not be the same as the number of the error or message it will be associated with. The <text> field will be the information text that may have up to 500 characters including line feeds.

For example, the OEM.txt file will have the following structure.

```
[10]
Help text.
[27]
Help text.
[33]
Help text.
```

Calling the texts from the PLC message or from the PLC error.

To associate the help message with a PLC message or PLC error, the "associated file" field must be defined like <file>#<id>. The <file> field will be the path and the name of the file. The <id> field will be the code that identifies the help text inside the file.

For example, the "associated file" will be defined as follows.

```
C:\CNC8070\MTB\PLC\LANG\OEM.txt#27
```

20.

PLC
"Messages" service

20.13.6 Save, load and print a message and error table

Save table

To save the table data, press the "Save" softkey and the CNC will show a list with the files saved at the CNC.

To save the table data:

- 1 Select the destination folder.
- 2 Define the file name at the bottom window. To replace an existing file, select it from the list.
- 3 Press [ENTER] to save the file or [ESC] to cancel the operation.

The file will be saved with the extension: *.MEF.

Load table

To recover the table data, press the "Load" softkey, the CNC will show a list with the files available at the CNC.

To load the table data:

- 1 Select the folder where the file is.
- 2 Select the file or write its name at the bottom window.
- 3 Press [ENTER] to accept the selection or [ESC] to cancel it and close the file listing.

Print table

This softkey may be used to print the table of messages and errors in the pre-determined printer or save it as a file (prn format) at the CNC. When selecting the "File" option, it will be saved in the folder "C:\Cnc8070\Users\Reports\".

When selecting this option, the CNC will show a dialog box requesting the print destination (printer or file). After selecting the destination, press [ENTER] to print it or [ESC] to cancel it.

20.

PLC
"Messages" service

The CNC must receive specific machine data in order for the machine tool to properly execute the programmed instructions, such as for feedrates, accelerations, feedback, automatic tool change, etc. This data is set by the machine manufacturer and must be defined in the machine parameter table.



Refer to the installation manual for more information on these parameters.

Recommended order for editing tables.

- 1 Check that there is a proper bus detection.
- 2 Define the connections (motors, feedback, etc.).
- 3 In the connection table, associate each connection to a physical connector.
- 4 Define the general axis and spindle parameters.
- 5 Define the tandem and gantry axis parameters.
- 6 Define the filter and compensation parameters and the associate them to the axes and spindles.
- 7 Define the other machine parameters (magazine, M functions, etc.).
- 8 Save the data for the tables. It is recommended to save the data in the tables so as to have a backup copy of them.

21.1 Interface description.

The composition of the machine parameter tables (right window) is dependent on the selected table. Most tables have the same appearance, with a single editable column. Occasionally (for example, the M function table), there are several columns of data. Examples of some parameter tables of shown below.

21.

MACHINE PARAMETERS. Interface description.

21.1.1 Connection and machine parameter table.

Machine parameters tables.

The screenshot shows the 'Machine parameters' window. On the left is a tree view (A) with categories like General, Axes, Spindles, Manual, M Functions, Kinetics, Filters, Gantry axes, Tandem axes, Compensations, OEM, Magazine, and HMI. The 'General' category is selected. On the right is the 'GENERAL PARAMETERS' table (B). The table has columns for 'Value', 'Stored value', and 'Unit'. It lists various parameters like NCHANNEL, NAXIS, AXISNAME, NSPOL, SPDLNAME, CNCTIME, SYSTEMTIME, PRGFREQ, CANOPENFREQ, RSTYPE, INCHES, MAXLOCP, MINLOCP, and MAXGLBP. Callouts point to specific features: (C) status icons (lock, power, save), (D) parameter names, (E) user-edited values (green), (F) saved values, (G) units, (H) limits and default values, and (I) parameter descriptions at the bottom.

Parameter	Value	Stored value	Unit
NCHANNEL	1	1	
NAXIS	3	3	
AXISNAME			
NSPOL	1	1	
SPDLNAME			
CNCTIME	2.0000	4.0000	ms
SYSTEMTIME	0.2500	0.2500	ms
PRGFREQ	2	2	cycles
CANOPENFREQ	1 Mbps	1 Mbps	
RSTYPE	RS232	RS232	
INCHES	mm	mm	
MAXLOCP	25	25	
MINLOCP	0	0	
MAXGLBP	299	299	

Limits = [1..2] Default Value = 1
Number of channels

(A) Connection tree and parameters.
(B) Parameter table.
(C) Table status icons (validated and saved).
(D) Mnemonic or parameter name.
(E) Value edited by the user (the table shows non-validated values in green).
(F) Saved or validated value of the parameter, selected using the horizontal softkeys.
(G) Parameter units.
(H) Parameter limit (maximum and minimum) and default values.
(I) Parameter description.

This is the most common layout for the machine parameter tables. Only the "Value" column is editable in this table, where the non-validated values are shown in green.

21.1.2 M function customization table.

M function table.

The screenshot shows the 'M FUNCTION TABLE : M FUNCTION VALUES' window. On the left, a tree view (A) lists machine parameters: General, Axes, Spindles, Manual, M Functions, Kinetics, Filters, Gantry axes, Tandem axes, Compensations, OEM, Magazine, and HMI. The 'M Functions' parameter is selected. The main table (B) displays the following data:

MNUM	SYNCHTYPE	MTIME	MPROGNAME	MPLC
12	Without Synch	10	sub_M12.nc	Yes
15	Before-Before	10	sub_M15.nc	Yes
16	Before-After	10	sub_M16.nc	No
80	After-After	10		Yes
81	Before-After	10		Yes
82	Before-After	10		No
83	Before-After	10		No
84	Before-Before	10		Yes
85	Before-Before	10		Yes
86	Before-Before	10		Yes

At the bottom, a status bar (F) shows 'Limits = [0..32767] Default Value = <Empty>' (E).

- (A) Connection tree and parameters.
 (B) Parameter table.
 (C) Table status icons (validated and saved).
 (D) Mnemonic or parameter name.
 MFUN: M function number.
 SYNCHTYPE: Type of synchronism.
 MTIME: Estimated execution time.
 MPROGNAME: Associated subroutine.
 MPLC: Send the M function to the PLC during block search.
 COMMENT: Comment associated with the function.
 (E) Parameter limit (maximum and minimum) and default values.
 (F) Parameter description.

This table is accessed using the DATA parameter. All columns can be edited in this table, where the non-validated values are not highlighted. Comments are saved in the MComments.txt file (one per language), in the c:\FagorCnc\Mtb\Data\Lang folder.

21.

MACHINE PARAMETERS.
Interface description.

21.1.3 Kinematics configuration table.

21.

MACHINE PARAMETERS.
Interface description.

Kinematics configuration table.

	VALUE	MAX OFFSET	OFFSET
TDATA 1	0	0	0
TDATA 2	0	0	0
TDATA 3	0	0	0
TDATA 4	0	0	0
TDATA 5	0	0	0
TDATA 6	0	0	0
TDATA 7	0	0	0
TDATA 8	0	0	0
TDATA 9	0	0	0
TDATA 10	0	0	0
TDATA 11	0	0	0
TDATA 12	0	0	0
TDATA 13	0	0	0
TDATA 14	0	0	0
TDATA 15	0	0	0

Limits = [+999999999] Default Value = 0

- (A) Connection tree and parameters.
 (B) Parameter table.
 (C) Table status icons (validated and saved).
 (D) Mnemonic or parameter name.
 VALUE: Value of the parameter.
 MAX OFFSET: Maximum offset that may be defined by the user.
 OFFSET: Offset of the parameter.
 (E) Parameter limit (maximum and minimum) and default values.
 (F) Parameter description.

This table is accessed through the TDATA or TDATA_I parameters. All columns can be edited in this table, where the non-validated values are not highlighted.

21.1.4 OEM parameter table.

OEM parameter table.

Machine parameters

- General
- Axes
- Spindles
- Manual
- M Functions
- Kinetics
- Filters
- Gantry axes
- Tandem axes
- Compensations
- OEM
 - MTBPAR
 - CAMTABLE

MTBPAR : DATA

	VALUE	MTBPARMODE	INCHES	COMMENT
P0	0.0000	Read	No	
P1	0.0000	Read	No	
P2	0.0000	Read	No	
P3	0.0000	Read	No	
P4	0.0000	Read	No	
P5	0.0000	Read	No	
P6	0.0000	Read	No	
P7	0.0000	Read	No	
P8	0.0000	Read	No	
P9	0.0000	Read	No	

Limits = [±99999.9999] Default Value = 0.0000

- (A) Connection tree and parameters.
 (B) Parameter table.
 (C) Table status icons (validated and saved).
 (D) Mnemonic or parameter name.
 VALUE: Value of the parameter.
 MTBPARMODE: Mode for accessing the parameter from its variable.
 INCHES: The parameter is affected by the change of units
 COMMENT: Comment associated with the parameter.
 (E) Parameter limit (maximum and minimum) and default values.
 (F) Parameter description.

This table is accessed using the DATA parameter. All columns can be edited in this table, where the non-validated values are not highlighted. Comments are saved in the MTBComments.txt file (one per language), in the c:\FagorCnc\Mtb\Data\Lang folder. The CNC assumes the edited values immediately, without having to validate them.

The simulation environment has a copy of this table. On CNC power-up, the values of the parameters of the real table are copied into the simulation table and from there on, they become different in the writing of the variables of both tables. In the simulation table, only the parameter values may be modified, not the rest of the permissions. The values of the simulation table can only be read or modified through their variable.

21.

MACHINE PARAMETERS.
Interface description.

21.1.5 Horizontal softkey menu.

Softkey.	Description.
	Soft key to display the options available for the selected table. The softkey only offers the options available for each specific case. • Initialize table. • Display mnemonics. • Display alias. • Renaming a table. • Add a table. • Clone a table. • Delete a table. • Print a table. • Export data. • Import data. • Teach-in.
	Soft key to show validated or saved values.
	Softkey to select the previous parameter.
	Softkey to select the next parameter.
	Soft key to find text or a value in the active table.

21.1.6 Vertical softkey menu.

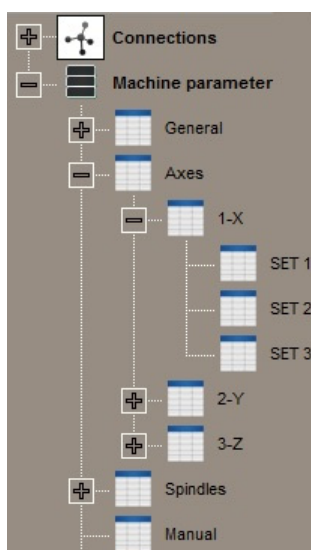
Softkey.	Description.
	Softkey to toggle the parameter units (coordinates, feedrate, etc) between millimeters and inches. The softkey highlights the selected units in color. The CNC will display this softkey or not depending on how parameter MMINCHSOFTKEY has been set.
	Soft key to save the selected table or all tables in a file (xml format).
	Soft key to retrieve the selected table or all tables from a file (xml format).
	Soft key to validate the parameters of the selected table and for the CNC to assume the edited values as the new active values. Any changes made to the tables must be validated or discarded before changing work mode. This table is necessary after changing a data. In certain cases, the CNC will have to be turned off and back on in order for the data to be validated.
	Soft key to validate the selected connection table and for the CNC to assume the edited values as new active values. Any changes made to the connection table must be validated or discarded before changing work mode. This table is necessary after changing a data. In certain cases, the CNC will have to be turned off and back on in order for the data to be validated.

21.

MACHINE PARAMETERS.
Interface description.

21.2 Operations with tables.

21.2.1 Navigating the tables.



The parameter tables are organized in a tree to the left side of the window. The branches can be expanded and collapsed by clicking on the + and - symbols respectively (or the [➡][⬅] keys). The window displays the corresponding table when a branch is selected.

To add elements to a branch (for example, more axes), move the cursor onto the branch and press the [INS] key. Move the cursor onto one of the elements and press the [DEL] key to delete it. Both actions can also be carried out by using the horizontal "Options" softkey.

21.

MACHINE PARAMETERS.

Operations with tables.

Connection tree.

The connection tree shows the buses and connections (motors, handwheels, etc.) of the CNC. Within the "Buses" branch, only the simulated bus nodes can be modified. The nodes for the other buses are defined by the CNC depending on the configuration detected at startup. All other connections (motors, feedbacks, etc.) are defined manually. Only connections are defined in this branch. The axis/spindle parameters associates each motor with its axis or spindle. Also, the connection table associates each connection with a physical drive or CNC input, etc.

Machine parameter tree.

The parameter tree adjusts the CNC to the mechanical characteristics of the machine, such as the number of axes, feedrate, accelerations, magazine type, etc. These parameters also associate the previous connections to the axes and spindles (motors, feedback, etc.).

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

21.2.2 Machine parameter editing.



In user mode, the machine parameters are write-protected and any changes made are temporary; this means that they will be removed the next time the unit is turned on. To make the changes permanent, unprotect the machine parameters by starting the unit up in setup mode and validating the changes made.

When the desired table is selected in the parameter tree, the CNC will display the parameter list. For each parameter, the table shows the edited value (non-validated values are in green), saved or validated value (selected from the horizontal softkeys) and the units (the vertical menu is used to toggle between millimeters and inches). When selecting a parameter, the lower part of the tool table displays the limits (minimum and maximum value) and default value for the parameter.

Limit (minimum and maximum value) and default value of the MAXGLP parameter.

MINLOCP	0	0	
MAXGLBP	299	299	
Límites = [100..9999] Valor por defecto = 299			

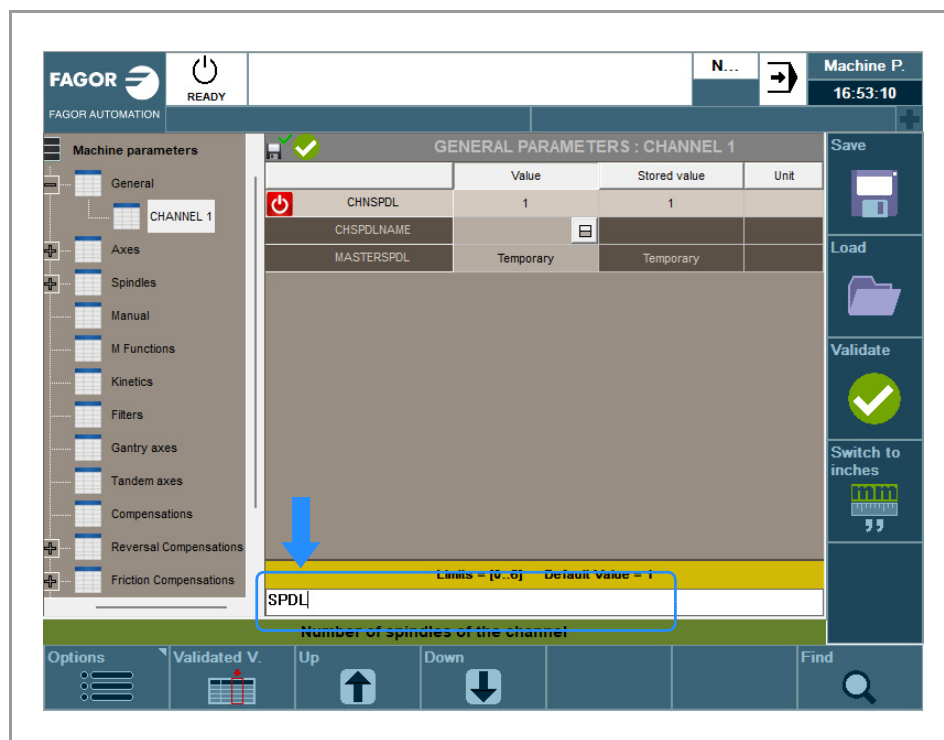
Mnemonic of machine parameters in Chinese.

When the Chinese language is selected, the machine parameters are shown in that language. You can switch between Fagor and personalized mnemonics using the [CTRL]+[N] keys or from the [F1] menu. The variables associated with the machine parameters are always accessed with the name Fagor.

Filter for machine parameters.

The [CTRL][F] keys display a filter to show only the parameters in the table whose name fully or partially matches the filter. The filter remains active as long as it is visible; that is, when switching tables, the filter stays active.

- To apply the filter, type in the text and press [ENTER].
- To cancel the filter, hide the filter bar ([CTRL][F]) or clear the text and press [ENTER].



21.

MACHINE PARAMETERS.
Operations with tables.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Non-editable machine parameters.

Icon.	Meaning.
	This icon indicates that the parameter cannot be modified directly in the table. The parameter updates its value automatically; for example, when the parameter tree (number of axes, sets, etc.) is modified or when certain elements (motor, encoder, etc) are selected.

Machine parameter editing.

The [ESC] key cancels the changes and retrieves the previous parameter value.

Icon.	Meaning.
No icon.	For those parameters that do not have an icon, a value must be edited within the indicated limits. Press the [ENTER] key to assume the entered value. If the value exceeds the accepted limits, the parameter will maintain its value.
	The parameter has a list of values; press the [ENTER] key to display the menu and select one of the options using the [↑] [↓] keys. After selecting a value, press [ENTER] again to accept it.
	The parameter accesses a data table. Press the [ENTER] key to access the table.
	The parameter accesses a parameter table. Press the [ENTER] key to access the table.
	The parameter refers to a file. Press the [ENTER] key to display the file explorer. Select a file and press [ENTER].

Selecting the default value.

If [SUP] is pressed, the parameter will assume the default value.

Renaming a table.

To rename a table, first select it in the tree and use the “Rename” option from the horizontal softkey “Options”.

Add a table.

To add a table, first select a branch (for example, “Axes”) and use the “Add” option from the “Options” horizontal softkey or the [INS] key.

Clone a table.

To clone a table, select it and use the “Clone” option from the “Options” horizontal softkey. When cloning a table, the CNC creates a copy of the selected table.

Delete a table.

To delete a table, select it in the tree and use the “Delete” option from the “Options” horizontal softkey or the [DEL] key.

Activate the Teach-in mode.

The “Teach-in” option is available from the “Options” horizontal softkey. Teach-in mode allows the machine axes to be moved manually and to assign the axis position to a cell. The teach-in mode is displayed at the bottom of the table.

21.

MACHINE PARAMETERS.

Operations with tables.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

21.2.3 Import and export cross compensation and ballscrew data (position and error).

The ballscrew and cross compensation tables are saved and retrieved with the other tables by using vertical "Save table" and "Retrieve table" softkeys. Compensation data (position and error) can also be individually exported and imported using the "Export data" and "Import data" options from the horizontal softkey "Options". The data is saved in a text file with the extension .mp.

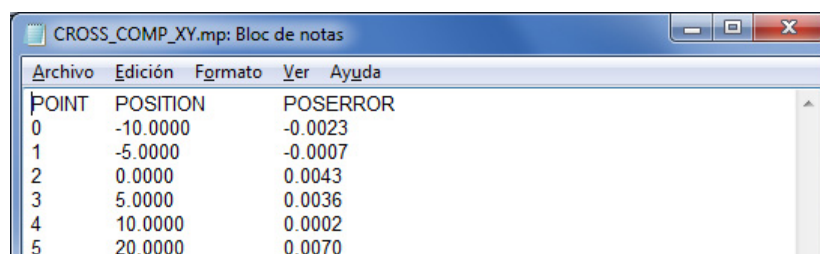
When saving compensation data (position and error), it is recommended to include the axis name in the file name, so as to identify it more easily; for example, the LSCRW_X.mp for ballscrew compensation tables and CROSS_XY.mp for cross compensation tables.

Generating the ASCII file for cross compensation and leadscrew error compensation manually.

In order to save time and eliminate transcription errors, instead of entering the data manually, it is possible to adapt the format of a text file that contains the result of the measurement and import it later on. The file of points must have the extension mp and meet the following requirements.

- The points of the table must be ordered by their position on the axis and the table must begin by the most negative point (or least positive) to be compensated.
- The machine reference point, whose position is indicated in parameter REFVALUE, must have ·0· error.
- For the leadscrew error compensation table, the maximum slope allowed is ·1·; i.e. the compensation steps must not exceed the position step between two consecutive points.

The format of the file must be like in the following example. The first column, as well as the titles of the columns, are optional and do not appear in the mp file when exporting the tables. The data of the tables may be separated by a blank space or by a tabulator. When using bi-directional compensation, a new column must be inserted in the table with the amount of error to be compensated when moving in the negative direction.



POINT	POSITION	POSERROR
0	-10.0000	-0.0023
1	-5.0000	-0.0007
2	0.0000	0.0043
3	5.0000	0.0036
4	10.0000	0.0002
5	20.0000	0.0070

21.2.4 Validating machine parameters.



The HMI parameter table uses the *MPMANAGEMENT* parameter to determine if the CNC must validate and save the parameters automatically when exiting the tables.

After editing or modifying the data in the table, the new values must be validated for the CNC to assume them. Any changes made to the tables must be validated or discarded before changing work mode. The icons at the top of the table indicate their status (validated/saved).

Icon.	Meaning.
	The table is validated.
	The table is not validated.
	The table is saved.
	The table is not saved.

Validate the table data (connections).



Softkey to validate the connections and for the CNC to assume the modified values as new values. Any changes made to the connection table must be validated or discarded before changing work mode.

Validate the table data (parameters).



Soft key to validate the parameters and for the CNC to assume the modified values as the new active values. Any changes made to the tables must be validated or discarded before changing work mode.

Restart the CNC to validate some of the parameters.

It is necessary to restart the CNC to validate some parameters. These parameters will be identified with the following symbol next to their name.

	WIDTH	800	800	
	HEIGHT	600	600	

The icon next to the WIDTH parameter indicates that the CNC must be restarted to validate it.

21.

MACHINE PARAMETERS.

Operations with tables.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

21.2.5 Saving and retrieving machine parameters from a file.



The HMI parameter table uses the MPMANAGEMENT parameter to determine if the CNC must validate and save the parameters automatically when exiting the tables.

Saving the tables to a file.



Soft key to save the selected table or all tables in a file (xml format).

- 1 From the connections and parameters tree, select the table to be saved or any table if all are to be saved.
- 2 Press the "Save" softkey.
- 3 Select between saving the selected table or all tables.
- 4 To save the selected table, type the name of the file. To save all the tables, write the name of a folder. To replace an existing file or folder, select it from the list.
- 5 Press the [ENTER] key to accept or the [ESC] key to cancel.

Loading tables from a file.



Soft key to load the selected table or all tables from a file (xml format).

- 1 From the connections and parameters tree, select the table to be loaded or any table if all are to be loaded.
- 2 Press the "Load" softkey.
- 3 Select between loading the selected table or all tables. The CNC will display the list of stored tables.
- 4 To load the selected table, select it from the list. To load all the tables, select the folder that contains them.
- 5 Press the [ENTER] key to accept or the [ESC] key to cancel.

21.

MACHINE PARAMETERS.
Operations with tables.



21.2.6 Print a table.

To print a table, first select it in the tree and use the "Print" option from the horizontal softkey "Options". The CNC can print the table in the pre-determined printer or save it as a file (prn format) at the CNC. When selecting this option, the CNC shows a dialog box requesting the destination for the file (printer or file). When printing to a file, it is possible to select the name and location of the file. After selecting the destination, press [ENTER] to print the table or [ESC] to cancel it.

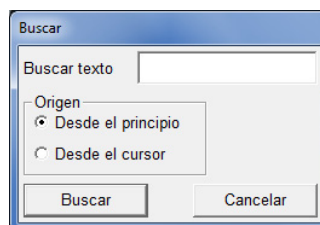
21.2.7 Initialize a table.

To initialize a table, move the cursor on it (right window) and use the "Initialize" option from the "Options" horizontal softkey. When initializing a table, all parameters assume their default value. The CNC will request confirmation of the command.

21.2.8 Search for text or a value in the active table.



Soft key to find text or a value in the active 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.



Tras definir las opciones de búsqueda, pulsar la tecla [ENTER] para realizar la búsqueda o la tecla [ESC] para cancelarla. Tras pulsar [ENTER] el cursor se posiciona en el primer campo que coincida con los parámetros de búsqueda.

21.

MACHINE PARAMETERS.

Operations with tables.

21.

MACHINE PARAMETERS.

Operations with tables.



CNCelite
8058 8060
8065 8070

REF: 2508

The set of utilities of the setup assistance is meant to speed up and simplify the machine setup procedure. These utilities are help tools that only show the response of the system to different settings; it is always up to the technician to decide which is the optimum setting. The different tools are accessed from the horizontal softkey menu.

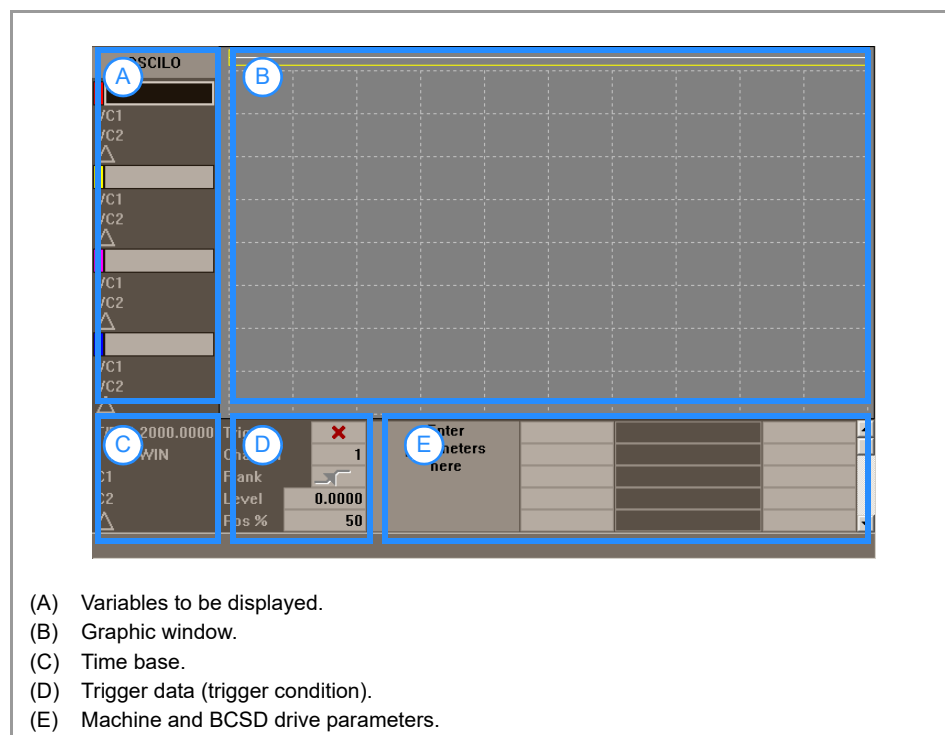
Softkey.	Meaning.
	Oscilloscope.
	Bode diagram.
	Circularity (roundness) test.

22.1 Oscilloscope.

The oscilloscope makes it possible to plot 4 variables to see the response of the system when the CNC machine parameters and/or BCSD drives that affect the setting are modified.

22.1.1 Interface description.

The oscilloscope looks as follows. Some of this data may also be defined at the configuration screen. See ["22.1.3 Configuration screen"](#) on page 414.



Variables to be displayed.

The visibility and color associated with each variable may be defined in the configuration page.

Graphic window.

Graphical representation of the selected variables. For each variable, it shows the scale used to draw it.

Time base.

The focus on "WIN" allows you to move the oscilloscope window. Cursors "C1" and "C2" may be used to analyze each signal of the last data capture. These two cursors may be used to obtain the position in milliseconds or Hertz of each signal and the time difference between them "Δ".

Trigger data (trigger condition).

The options offered by this trigger data area are:

Data.	Meaning.
Trigger	The trigger indicates when the data capture begins. • If selected, the trigger condition must be specified with the "Flank", "Level" and "Position" data. • If not selected, the data capture begins as soon as the user gives the command to begin. The "Flank", "Level" and "Position" data are ignored.
Channel	The channel indicates which variable or channel (CH1, CH2, CH3, CH4) is to be used as reference or trigger condition.
Flank	It may be up-flank (leading edge) or down-flank (trailing edge). It is taken into account when trigger has been selected.
Level	It sets the value to be assumed by the variable to start capturing data. It is taken into account when trigger has been selected.
Position (%)	Number of samples taken before trigger (between 0 and 100 %). For example, a 10% position indicates that 10% of the total number of scheduled samples will be taken before trigger and the remaining 90% after. The trigger condition (edge and level) is evaluated after the indicated % of samples are available. If the position is defined as 50% and the Trigger condition takes place after taking 10% of the samples, it is ignored until capturing 50% of the samples.

Machine and BCSD drive parameters.

Area to define the parameters and their value. Use the keys [←][→][↑][↓] to move the focus and [ENTER] to accept the value. See ["22.1.5 Machine parameter editing."](#) on page 416.

22.

SETUP ASSISTANCE
Oscilloscope.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

22.1.2 Softkey menus.

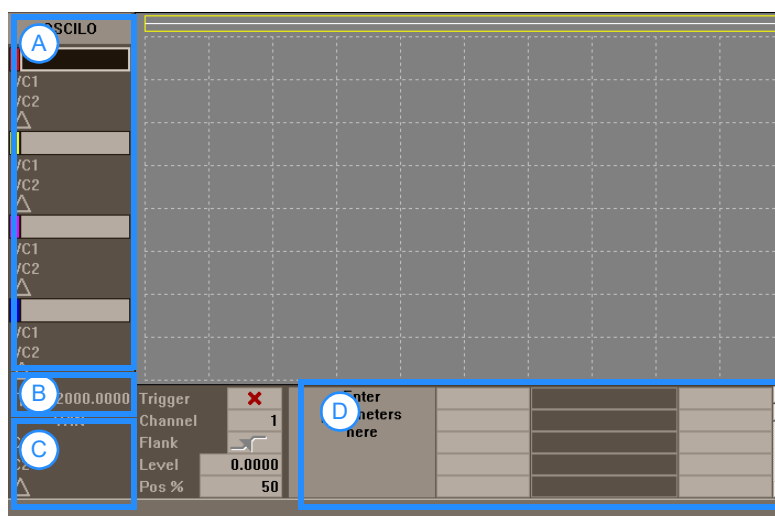
Horizontal softkey menu.

The horizontal softkey menu offers the following options:

Softkey.	Description.
Softkey "Config".	This softkey accesses the configuration screen. It may be used to define which variables will be displayed, their scale, colors, time base, the Trigger, the trigger condition, the parameters to act upon, etc.
Softkey "Data".	It accesses the data processing submenu. This submenu serves to save and load the captured trace as well as the configuration defined for it. It may also be used to send data of the captured trace out to printer or to a file.
Softkey "Enlarge screen". 	This option allows you to enlarge the graphic window or the variable definition area.
Softkey "Overlap channels".	With this option, several channels may be overlapped.
Softkey "Autoscale".	When autoscaling a channel, the system determines the proper vertical scale and offset so the signal appears as enlarged as possible inside its graphic area.
Softkey "Begin capture". 	Start taking data to show it graphically.
Softkey "End capture". 	Finish the data capture and stop the graphic representation.
Softkey "Freeze screen". 	Freeze the display without interrupting the data capture. This softkey only appears on the horizontal softkey menu if the capture is in continuous mode. To set the capture in continuous mode, change the "mode" option of the configuration screen.
Softkey "Unfreeze capture". 	Unfreeze the display.

Vertical softkey menu.

Depending on which data has the focus, the vertical softkeys show different options.



22.

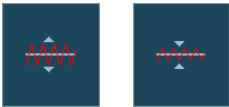
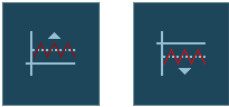
SETUP ASSISTANCE
Oscilloscope.

FAGOR
AUTOMATION

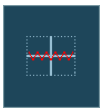
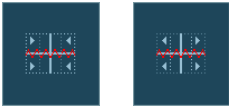
CNCelite
8058 8060
8065 8070

REF: 2508

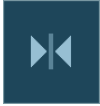
Zona "A". Focus on a variable of a channel.

Softkey.	Description.
	Softkeys to modify the scale of the variable. The new scale value is shown on the graph next to the variable.
	Softkeys to move the variable up or down.
	Softkey to center the variable.

Zone "B". Focus on the "T/Div" field.

Softkey.	Description.
	Softkey to autoscale the graph. The oscilloscope selects the appropriate time scale so that the signal is magnified as much as possible within the corresponding graphical range.
	Softkey to increase or decrease the time scale.

Zone C. Focus on the "WIN", "C1" and "C2" field.

Softkey.	Description.
	Softkey to bring "C1" or "C2" to the beginning or end of the capture.
	Softkey to bring "C1" or "C2" to the trigger; if there is no trigger, to the center of the window.
	Softkey to move the window, "C1" or "C2" to the left or right.

Zone "D". Focus on the "parameter name" field.

Softkey.	Description.
	Softkey to increase the parameter value by the amount indicated in the configuration screen.
	Softkey to decrease the parameter value by the amount indicated in the configuration screen.
	Softkey to return the default value to the parameter.
	Softkey to validate the parameter value.

22.

SETUP ASSISTANCE
Oscilloscope.

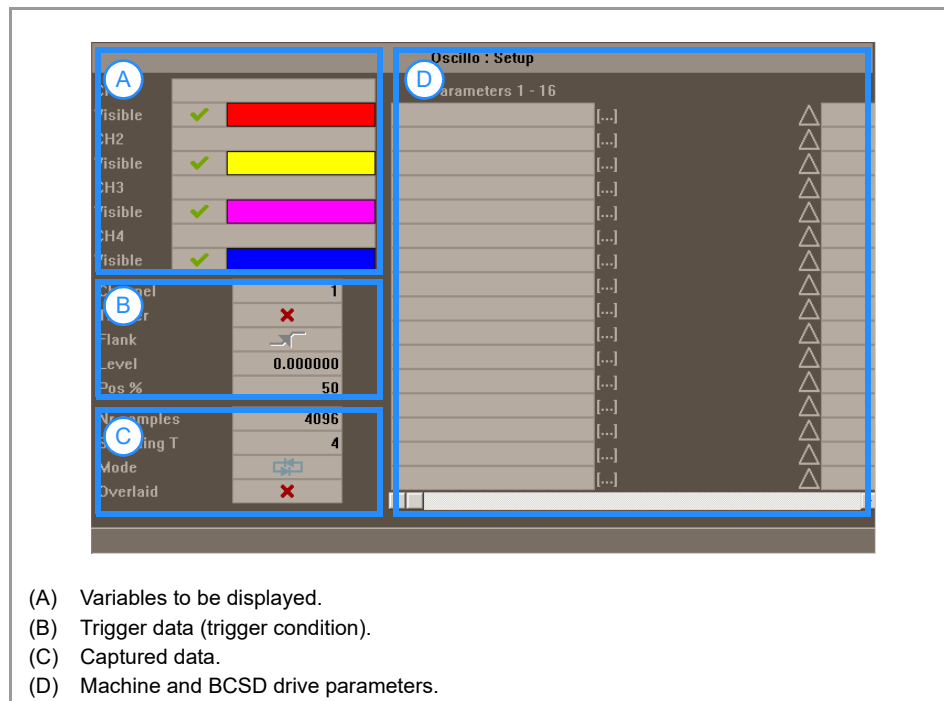
FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

22.1.3 Configuration screen

The configuration screen may be accessed with the "Config" softkey.



Variables to be displayed.

Variables to be represented in the graphic window, color and visibility.

Trigger data (trigger condition).

Same data as described in the oscilloscope window.

Captured data.

- Number of samples.
- Sampling period.
- Single or continuous capture mode.
- Overlap channels.

The sampling period must be multiple of the machine parameter CNCTIME; if a wrong value is entered, the CNC will set it to the right one.

Machine and BCSD drive parameters.

Machine and BCSD drive parameters to be modified, initial value and increase/decrease of the value when pressing the softkeys. By default, the increase/decrease value of a parameter is initialized to 1.

22.

SETUP ASSISTANCE
Oscilloscope.

22.1.4 Configure and execute the oscilloscope function

In oscilloscope mode, from a single screen you can select the variables to be analyzed, the trigger conditions and the machine parameters of the CNC or BCSD drive to be modified.

How to operate the oscilloscope.

- 1 Execute a part-program to move the axis or axes to be adjusted. Once the program is running, go into the adjustment mode and access the oscilloscope function.
- 2 In the oscilloscope, define the variables to be analyzed, the trigger conditions and the machine parameters of the CNC or BCSD drive that can be modified.
- 3 Data capture and later analysis of the data.
- 4 Once the data capture has been interrupted or completed, it is possible to analyze the signals and modify the parameters that have been previously selected in order to improve the machining conditions.
- 5 Repeat the capture, the analysis and the modification of parameters until obtaining the best machining conditions.

Recommendations

Execute endless repetitive movements. After adjusting the axes individually, readjust the interpolating axes together. The user must determine which is the best adjustment, the oscilloscope function is just an assistance tool.

Data editing.

To select the editable data, use the keys [↑][↓][→][←]. To scroll a list of values (e.g. the list of variables), press [SPACE]. To select a value from a list, use the keys [↑][↓]. To modify an icon, press [SPACE]. To accept a value, press [ENTER].

22.

SETUP ASSISTANCE
Oscilloscope.

22.1.5 Machine parameter editing.



During start-up, the values accepted with the [ENTER] key are valid only until switch-off. Once the setup has been completed, the values must be validated using the corresponding softkey to save the values in the parameter table and/or the BCSD drive, as appropriate.

Machine parameters that may be modified.

The table indicates the parameters that will be assumed at the beginning of the next block. For this to be so, the instruction "#SYNC POS" must be programmed after each motion block.

CNC	Associated variable	Update
PRELOAD	MPTDM.PRELOAD[tandem]	Immediate
PRELFITI	MPTDM.PRELFITI[tandem]	Immediate
TPROGAIN	MPTDM.TPROGAIN[tandem]	Immediate
TINTIME	MPTDM.TINTIME[tandem]	Immediate
TCOMPLIM	MPTDM.TCOMPLIM[set]	Immediate
INPOSW	MPA.INPOSW.[set].eje	Immediate
BACKLASH	MPFB.BACKLASH.[set].eje	Immediate
G00FEED	MPA.G00FEED.[set].eje	Beginning of next block
MAXVOLT	MPAD.MAXVOLT.[set].eje	Immediate
PROGAIN	MPA.PROGAIN.[set].eje	Immediate
FFGAIN	MPA.FFGAIN.[set].eje	Immediate
MANFFGAIN	MPA.MANFFGAIN.[set].eje	Immediate
ACFWFACTOR	MPA.ACFWFACTOR.[set].eje	Immediate
ACFGAIN	MPA.ACFGAIN.[set].eje	Immediate
MANACFGAIN	MPA.MANACFGAIN.[set].eje	Immediate
LACC1	MPA.LACC1.[set].eje	Beginning of next block
LACC2	MPA.LACC2.[set].eje	Beginning of next block
LFEED	MPA.LFEED.[set].eje	Beginning of next block
ACCEL	MPA.ACCEL.[set].eje	Beginning of next block
DECEL	MPA.DECEL.[set].eje	Beginning of next block
ACCJERK	MPA.ACCJERK.[set].eje	Beginning of next block
DECJERK	MPA.DECJERK.[set].eje	Beginning of next block
SERVOOFF	MPAD.SERVOOFF.[set].eje	Immediate
MINANOUT	MPAD.MINAOUT.[set].eje	Immediate
BAKANOUT	MPA.BAKANOUT.[set].eje	Immediate
BAKTIME	MPA.BAKTIME.[set].eje	Immediate
PEAKDISP	MPA.PEAKDISP[set].eje	Immediate
REVEHYST	MPA.REVEHYST[set].eje	Immediate
FBMIXTIME	MPA.FBMIXTIME[set].eje	Immediate
FLIMIT	MPA.FLIMIT.eje	Immediate
SLIMIT	MPA.SLIMIT.eje	Immediate
MAXACCEL	[channel].G.MAXACCEL	Beginning of next block
MAXJERK	[channel].G.MAXJERK	Beginning of next block

If the machine parameters are protected, when accessing any of them, the CNC will display a window requesting a password.

22.

SETUP ASSISTANCE
Oscilloscope.

FAGOR
AUTOMATION



CNCelite
8058 8060
8065 8070

REF: 2508

BCSD drive parameters.

The oscilloscope can be used to modify the parameters of BCSD drives. Refer to the related documentation for parameter names.

Saving, loading and resetting the configurations

When defining a new configuration of variables to be displayed and parameters modified at the oscilloscope, the system allows saving that configuration into a file. Likewise, the system allows loading a configuration that was saved previously or resetting the current configuration.

Save

To save the current configuration, go to the configuration screen and press the "Save" softkey. After this, a new screen appears showing the available configurations with the focus on the last one. Also, it is possible to rename the new configuration whose extension must be ".osc".

Load

To load a previously saved configuration, go to the configuration screen and press the "Load" softkey. Then, a new screen appears showing a list of previously saved configurations, being possible to select any of them.

The oscilloscope has predefined configurations for the X, Y, Z, A, B, C axes with typical trace variables and parameters.

Reset

Pressing the "Reset" softkey of the configuration screen deletes or initializes the current configuration. There are neither variables nor parameters selected and the rest of conditions (colors, trigger, etc.) assume the values assigned by default.

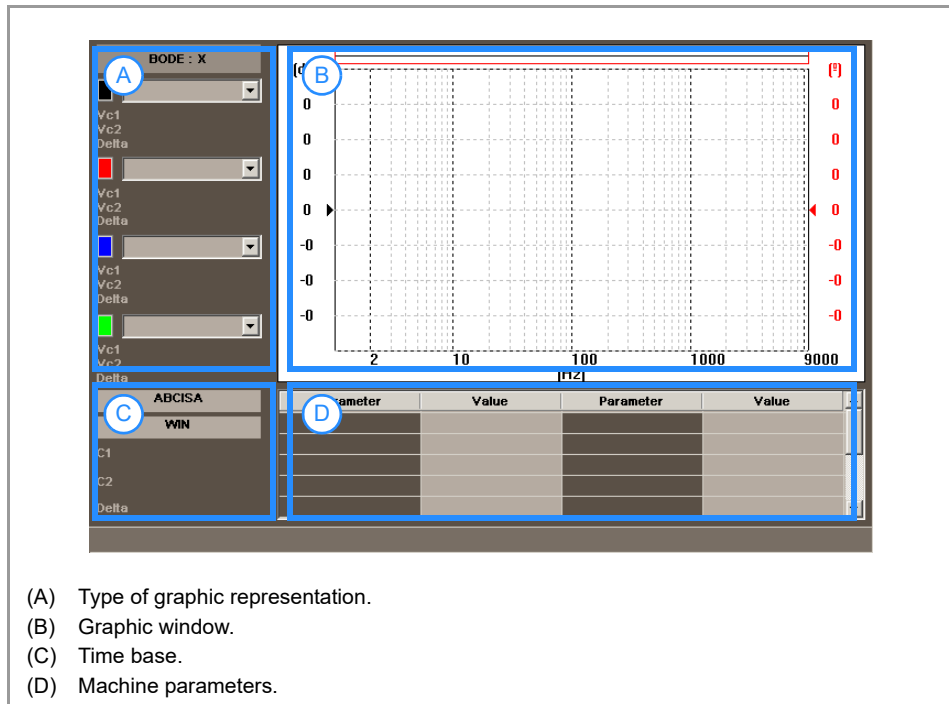
22.**SETUP ASSISTANCE**
Oscilloscope.

22.2 Bode diagram.

The Bode diagram makes it possible to obtain the frequency response of the system. This tool introduces a PRBS signal into an axis, so that by varying the frequency and analyzing the response, the behavior of the axis for different frequencies is obtained, which will give the user an idea of its behavior to any signal. This diagram may be used to check the system's gain, the bandwidth and the mechanical resonance. All this helps achieve the proper adjustment of the loops, analyze mechanical problems and check the final features.

22.2.1 Interface description.

The Bode looks as follows. Some of this data may also be defined at the configuration screen. See ["22.2.4 Configuration screen"](#) on page 423.



Type of graphic representation.

Up to 4 graphs can be plotted at the same time. The user can change the type of graph without having to do a new capture. The color associated with each graph can be defined in the configuration page.

Time base.

Cursors "C1" and "C2" may be used to analyze each signal of the last data capture. These two cursors may be used to obtain the position in milliseconds or Hertz of each signal and the time difference between them (Delta).

Machine parameters.

Area to define the parameters and their value. Use the keys [←][→][↑][↓] to move the focus and [ENTER] to accept the value. See ["22.2.3 Machine parameter editing."](#) on page 422.

22.

SETUP ASSISTANCE
Bode diagram.

22.2.2 Softkey menus.

Horizontal softkey menu.

The horizontal softkey menu offers the following options:

Softkey.	Description.
Softkey "Config".	This softkey accesses the configuration screen. It may be used to define the axis to obtain its Bode, the two variables (IN/OUT) that will be displayed, the types of graph to show the variables, its colors, the capture configuration and the movement configuration. The configuration screen offers the options to save, load or reset the current configuration and also load the reference Bode configuration.
Softkey "Data".	It accesses the data processing submenu. This submenu serves to save and load the captured trace as well as the configuration defined for it. It can also print the data about the captured trace.
Softkey "Views".	It accesses the submenu to display the graph.
Softkey "Scales".	It accesses the submenu to configure the abscissa and ordinate axes. This submenu may be used to activate the logarithmic scale on the abscissa axis.
Softkey "Begin capture". 	Start taking data to show it graphically.
Softkey "End capture". 	Finish the data capture and stop the graphic representation.

Treatment of the data and the captured trace.

The following options appear in the "Data" menu.

Softkey.	Description.
Save	Save the captured trace.
Load	Load a trace.
Print. 	Print the captured trace.
Save reference	Save the captured trace as a reference bode.
Load reference	Load a trace as a reference bode.
Convert into reference capture. 	It may be used to convert the data of the capture channels into a reference Bode with which to compare the capture just done or another one that is located in a file.

Treatment of the data and the captured trace.

The following options appear in the "Views" menu.

Softkey.	Description.
Enlarge screen. 	With this option, it is possible to enlarge the graphics window to the left, using the area that shows the data of the graphs to be displayed. Pressing the same softkey returns the screen to the previous size.
Overlap channels.	With this option, several channels may be overlapped.
Activate reference capture. 	It may be used to activate or cancel the reference Bode with which to compare the capture just done or another one that is located in a file.

22.

SETUP ASSISTANCE
Bode diagram.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

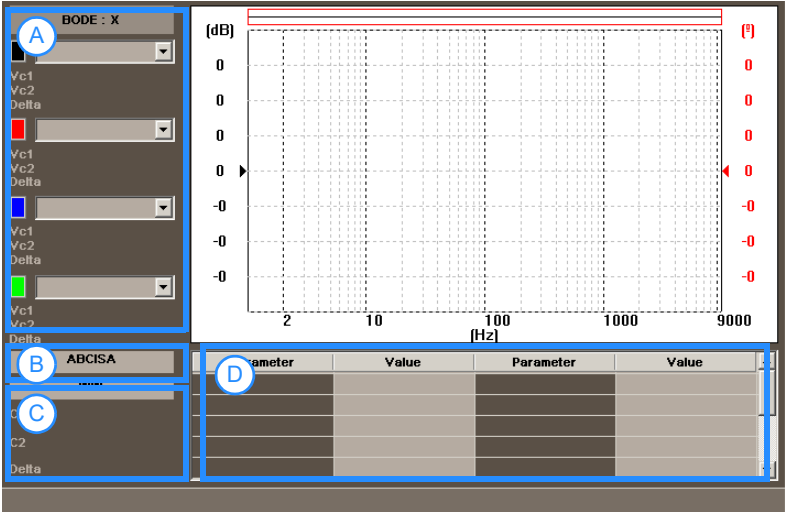
Vertical softkey menu.

Depending on which data has the focus, the vertical softkeys show different options.

22.

SETUP ASSISTANCE

Bode diagram.



Zona "A". Focus on a type of graph.

Softkey.	Meaning.
	Softkeys to modify the scale of the graph. The new scale value is shown on the graph.
	Softkeys to scroll the graph up or down.
	Softkey to center the graph.

Zone "B". Focus on the "ABSCISSA" field

Softkey.	Description.
	Softkey to autoscale the graph. The Bode selects the appropriate time or frequency scale so that the signal is as large as possible within the corresponding graphic band.
	Softkey to increase or decrease the time or frequency scale.

Zone C. Focus on the "WIN", "C1" and "C2" field.

Softkey.	Description.
	Softkey to bring "C1" or "C2" to the beginning or end of the capture.
	Softkey to bring "C1" or "C2" to the trigger; if there is no trigger, to the center of the window.
	Softkey to move the window, "C1" or "C2" to the left or right.

Zone "D". Focus on the "parameter name" field.

Softkey.	Description.
	Softkey to increase the parameter value by the amount indicated in the configuration screen.
	Softkey to decrease the parameter value by the amount indicated in the configuration screen.
	Softkey to return the default value to the parameter.
	Softkey to validate the parameter value.

22.

SETUP ASSISTANCE
Bode diagram.

22.2.3 Machine parameter editing.



During start-up, the values accepted with the [ENTER] key are valid only until switch-off. Once the setup has been completed, the values must be validated using the corresponding softkey to save the values in the parameter table.

Machine parameters that may be modified.

The table indicates the parameters that will be assumed at the beginning of the next block. For this to be so, the instruction "#SYNC POS" must be programmed after each motion block.

CNC	Associated variable	Update
PRELOAD	MPTDM.PRELOAD[tandem]	Immediate
PRELFITI	MPTDM.PRELFITI[tandem]	Immediate
TPROGAIN	MPTDM.TPROGAIN[tandem]	Immediate
TINTIME	MPTDM.TINTIME[tandem]	Immediate
TCOMPLIM	MPTDM.TCOMPLIM[tandem]	Immediate
INPOSW	MPA.INPOSW.[set].eje	Immediate
BACKLASH	MPFB.BACKLASH.[set].eje	Immediate
G00FEED	MPA.G00FEED.[set].eje	Beginning of next block
PROGAIN	MPA.PROGAIN.[set].eje	Immediate
FFGAIN	MPA.FFGAIN.[set].eje	Immediate
MANFFGAIN	MPA.MANFFGAIN.[set].eje	Immediate
ACFWFACTOR	MPA.ACFWFACTOR.[set].eje	Immediate
MANACFGAIN	MPA.MANACFGAIN.[set].eje	Immediate
LACC1	MPA.LACC1.[set].eje	Beginning of next block
LACC2	MPA.LACC2.[set].eje	Beginning of next block
LFEED	MPA.LFEED.[set].eje	Beginning of next block
ACCEL	MPA.ACCEL.[set].eje	Beginning of next block
DECEL	MPA.DECEL.[set].eje	Beginning of next block
ACCJERK	MPA.ACCJERK.[set].eje	Beginning of next block
DECJERK	MPA.DECJERK.[set].eje	Beginning of next block
SERVOOFF	MPAD.SERVOOFF.[set].eje	Immediate
BAKTIME	MPA.BAKTIME.[set].eje	Immediate

If the machine parameters are protected, when accessing any of them, the CNC will display a window requesting a password.

22.

SETUP ASSISTANCE
Bode diagram.



22.2.4 Configuration screen

The configuration screen may be accessed with the "Config" softkey. This screen shows four different areas.

- The selected axis and the two variables to be displayed.
- The selected types of graph.
- The capture configuration data.
- The data of the movement and of the excitation signal.

22.

SETUP ASSISTANCE
Bode diagram.

Axis

Axis whose Bode is to be obtained.

Type of capture

The variables that are captured (In/Out) are preset in all types of capture except in the "ADVANCED" type where they are set by the user.

Variable.	Meaning.
ADVANCED	"In/Out" variables that may be set by the user.
CUR_CL	Closed current loop. Current command / current feedback.
VEL_CL	Closed velocity loop. Velocity command / velocity feedback.
POS_CL	Closed position loop. Position command / position feedback.
CUR_OL	Open current loop.
VEL_OL	Open velocity loop.
POS_OL	Open position loop.
TF_CURCOM_MOTSPE	Transfer function. Current command / motor speed.
TF_MOTSPE_LOASPE	Transfer function. Motor speed / load speed.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Channels

Different ways to display the graph of the obtained data. The selected channels must have the same units in the abscissa axis, "time" graphs and "frequency" graphs cannot be displayed at the same time.

- "IN, OUT"

The captured signals themselves. They are "time" graphs; in other words, the scale of the abscissa axis is given in time units.

- "MAG, PHASE"

Gain and phase after transferring the "In/Out" variables. They are "frequency" graphs; in other words, the scale of the abscissa axis is given in frequency units (Hz).

Channel.	Meaning.
MAG_AVG PHASE_AVG	Average of the captures done so far.
MAG_DB PHASE_DB	Data in decibels.
MAG_OL PHASE_OL	Data in closed loop to which a mathematical transformation is applied to represent the open loop system.

- "COHERENCE"

Coherence after transferring the "In/Out" variables. It is a "frequency" graph; in other words, the scale of the abscissa axis is given in frequency units (Hz).

Captured data.

Except the "Number of averages" field that can always be edited, the rest of the fields can only be edited by the user when the type of capture is "ADVANCED".

- "FINAL FREQUENCY"

Maximum frequency displayed. It is directly related to the sampling period ($1 / 2 * \text{SamplingPeriod}$)).

- "FREQUENCY INCREMENT"

Step between frequencies. It is directly related to the final frequency and the number of samples ($\text{FinalFrequency} / \text{NumberOfSamples}$).

- "SAMPLING PERIOD"

- "NUMBER OF SAMPLES"

- "NUMBER OF AVERAGES"

Number of consecutive captures performed automatically. Each capture involves the movement of the axis, the application of the PRBS and the capture itself.

Movement

- "FEEDRATE (F/S)"

Moving speed.



It changes the axis moving direction, positive or negative.

- "INITIAL POSITION"

Position where each capture begins.

- "PRBS AMPLITUDE"

Amplitude of the excitation signal.



It changes the units of the amplitude of the excitation signal.

- ENTRY POINT

It may be used to select the point where to apply the excitation signal, only when the type of capture is "ADVANCED".

22.

SETUP ASSISTANCE
Bode diagram.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Horizontal softkey menu.

When defining a new graphic configuration and modified parameters, the system permits saving that configuration in a file. Likewise, the system allows loading a configuration that was saved previously or resetting the current configuration.

- Save the current configuration.

After pressing the "Save" softkey, the CNC shows the list of available configurations and lets you save the current configuration. The file extension must be bod.

- It loads a previously saved configuration.

After pressing the "Load" softkey, the CNC shows the list of previously saved configurations and lets you select any of the them.

- Reset or initialize the current configuration.

Pressing the "Reset" softkey of the configuration screen deletes or initializes the current configuration. There are neither variables nor parameters selected and the rest of conditions assume the values assigned by default.

- Activate/Cancel reference capture.

It may be used to activate or cancel the configuration of the reference Bode.

22.

SETUP ASSISTANCE

Bode diagram.



CNCelite
8058 8060
8065 8070

REF: 2508

22.3 Circularity (roundness) test.

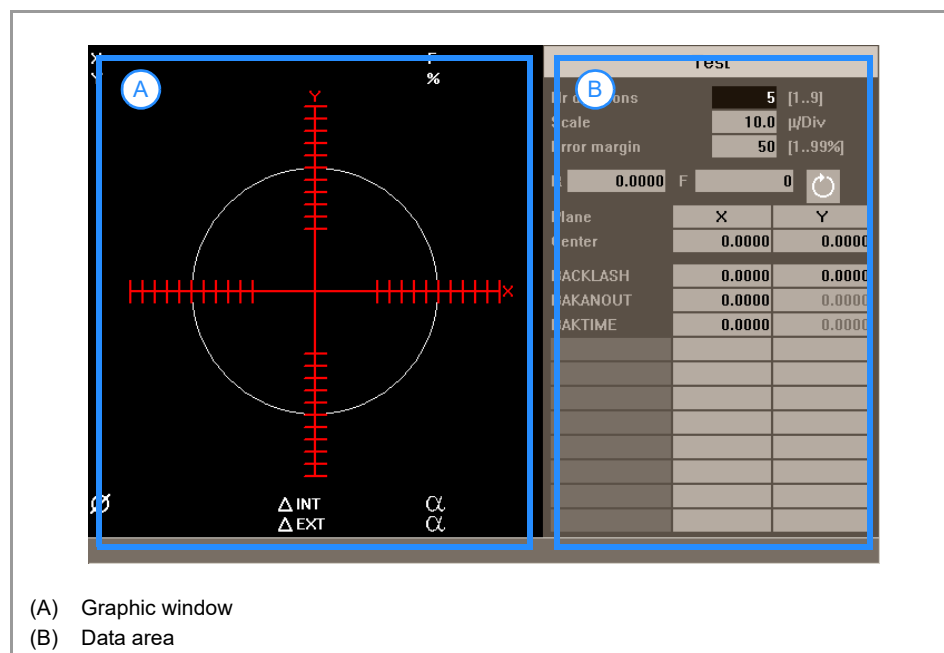
The circularity test makes it possible to run a circle with the axes and to graphically represent the difference between the real feedback elevation and the theoretical elevation. This makes it possible to know the reversal peak of the axes, and to adjust it by modifying the machine parameters that affect the reversal of the motion. These parameters may be modified while running the test, thus being possible to evaluate the response of the system to these changes and optimize the adjustment on the go.

22.

SETUP ASSISTANCE
Circularity (roundness) test.

22.3.1 Interface description.

The screen of the circularity test looks like this, with two clearly different parts. A graphics window that shows the result of the test and a data area for interacting with the system.



Graphic window.

It is the area that shows, graphically, the result of the test. The graphic shows the two moving axes and the theoretical circle of the interpolation that will be carried out. As the test is being run, the positioning error at each point is drawn on the circle. This error is shown projected radially.

Superimposed on the graphics, it shows the following additional data that is updated by the CNC.

- The real coordinates of the axes.
- Programmed feedrate and % applied.
- Diameter of the displayed circle.
- Maximum and minimum error over the theoretical radius and angular position where it has been detected.

Data area.

It is the area where the user interacts with the system. It offers a set of data for defining the graphic environment, the subroutine that will be used to generate the machine movement and the machine parameters involved in the adjustment.

Graphic environment data to be configured

- Number of divisions on both sides of the theoretical circle.
- Scale or value in microns of each division.
- Error margin. Percentage of the area that is occupied by the error margin (divisions area).

Data for the motion subroutine

The circular interpolation is carried out using a subroutine. See ["22.3.5 Define and execute the movement subroutine"](#) on page 431.

The following data is taken into consideration when executing the subroutine.

- Plane where the circle is executed.
- Circle center coordinates.
- Circle radius.
- Programmed feedrate.
- Turning direction.

Parameters to be set

The CNC machine parameters involved can be modified to perform the adjustment. See ["22.3.7 Adjustment of the machine parameters involved"](#) on page 433.

It is possible to interact with up to 11 different machine parameters. Some of these parameters are always visible, but some may be defined at will. For each parameter, it shows its value on each axis of the work plane.

To see the whole list of parameters that may be defined, see section ["22.3.9 Machine parameters that may be modified"](#).

22.

SETUP ASSISTANCE
Circularity (roundness) test.

22.3.2 Softkey menus.

Horizontal softkey menu.



The "Stop" softkey only stops the data capture. It does not stop the movement of the axes. To stop the movement of the axes, press the [STOP] key of the operator panel.

The horizontal softkey menu offers the following options.

Softkey.	Description.
Softkey "Simple".	It captures the data of a full circle.
Softkey "Continuous".	It captures data continuously.
Softkey "Stop".	It stops the data capture. This softkey is shown when data capture is running.
Softkey "Clear".	It deletes the graphic representation. It may be done while the capture is running; in that case, it goes on with the graphic representation.
Softkey "Load".	It loads a previously saved configuration.
Softkey "Save".	It saves the configuration defined.

Vertical softkey menu.

The vertical softkey menu offers the following options.

Softkey.	Meaning.
	The "Validate" softkey saves the values of the machine parameters in the CNC tables.
	The "Initialize" softkey initializes the window data to their default values.

22.

SETUP ASSISTANCE
Circularity (roundness) test.

22.3.3 Configuring and executing the circularity (roundness) test

The process consists in executing a circle with the axes of the machine and verifying it on its graphic representation. The graphic shows the difference between the actual (real) coordinate obtained from the feedback device and the theoretical coordinate calculated in that point. This difference is shown projected radially.

The interpolation of the machine axes and the point capture for the graphics are initiated separately. The data capture is handled from the softkey menu whereas the movement of the axes is controlled from the operator panel.

The adjustment process is repeated until obtaining the best adjustment of the axes. The circularity (roundness) test diagram is an assistance tool that only shows the response of the system to the various settings (adjustments); it is up to the technician to choose the best adjustment.

How to execute the circularity (roundness) test

Basically it is done as follows. Each of these points is described in more detail in the indicated chapter. The configuration used may be saved in a file for later.

- 1 Before starting the process, some aspects of the process must be configured, such as the graphical environment, the circular interpolation to be performed, the values of the parameters involved, etc. A previously saved configuration may be used. See ["22.3.3 Configuring and executing the circularity \(roundness\) test"](#) on page 429.
- 2 Execute a circular interpolation of the machine axes. The axes must be moving before the data capture starts. See ["22.3.5 Define and execute the movement subroutine"](#) on page 431.
- 3 Initiate the data capture and the graphic simulation. Once the axes are moving, the data capture for the graphics may begin. While capturing data, the position error at each point is drawn on the graphics. See ["22.3.6 Data capture for the graphic"](#) on page 432.
- 4 Optimize the adjustment by modifying certain CNC machine parameters. This makes it possible to evaluate the system response to these changes and optimize the adjustment. See ["22.3.7 Adjustment of the machine parameters involved"](#) on page 433.
- 5 Save the modified values and the configuration used. See ["22.3.8 Validate the changes and save the configuration used"](#) on page 434.

Move through the data

To enter or modify a data, it must be selected; i.e. it must have the editing focus on it. To select a data, use the [←][→][↑][↓] keys. The data are accepted with the [ENTER] key. Before accepting a data, it is checked out and if the data is not correct, it is not accepted.

22.

SETUP ASSISTANCE
Circularity (roundness) test.

22.3.4 Configure the graphic environment

This operation may be carried out before or during the test. If they are modified during the test, the screen is cleared and the graphic representation goes on.

The following graphic characteristics may be defined from the data window.

- Number of divisions on both sides of the theoretical circle.
- Scale or value in microns of each division.
- Error margin. Percentage of the area that is occupied by the error margin (divisions area).

Initialize the data.



Bear in mind that the softkey "Initialize" initializes all the data of the window, including the values of the machine parameters.

When accessing the circularity test, it assumes the values used last. Pressing the "Initialize" softkey restores the default values.

- | | |
|------------------------|---------------------|
| • Number of divisions: | 5 |
| • Scale: | 10 microns/division |
| • Error margin: | 50 % |

22.

SETUP ASSISTANCE
Circularity (roundness) test.

22.3.5 Define and execute the movement subroutine



The movement of the axes is managed from the operator panel. The softkey menu only controls the data capture for the graphics.

In order to run the test, the axes of the machine must be executing a circular interpolation. The axes must be moving before initiating the point capture and they must keep moving during the whole process. To achieve this, a repetitive movement must be executed.

Subroutine associated with the circularity test

The circular interpolation is executed with the subroutine associated with the circularity test. This subroutine is located in the folder "..\MTB \SUB \testcirc_vx.nc", where vx indicates the subroutine version and neither its name nor its location must be changed. If the subroutine does not exist, the CNC creates a predefined one the first time you try to run it. This subroutine may be modified by the OEM to adapt it to his needs.

Some data of the subroutine are defined in this window.

- Plane where the circle is executed.
- Circle center coordinates.
- Circle radius.
- Programmed feedrate.
- Turning direction of the axes. The turning direction is given by an icon. To change the turning direction, place the focus on the icon and press [SPACE].

Execute the associated subroutine

Press the [START] key to execute the subroutine. The subroutine data cannot be modified while it is being executed. The movement must be stopped before editing any of the data. The movement is stopped with the [STOP] key of the operator panel.

22.

SETUP ASSISTANCE
Circularity (roundness) test.



CNCelite
8058 8060
8065 8070

REF: 2508

22.3.6 Data capture for the graphic



When capturing points in the circularity test, they are not drawn on the graphics of the CNC.

After defining the graphical representation and with the subroutine running, you can start capturing points for the graph. The point capture is initiated from the softkey menu. The point capture may be either simple, where the graph is made once (a single whole circle), or continuous where the graph is redrawn for every interpolation of the machine (after every whole circle). The machine parameters may be modified while capturing points. See ["22.3.7 Adjustment of the machine parameters involved"](#) on page 433.

Simple data capture

Data capture is initiated with the "Simple" softkey. This softkey deletes the graph and starts drawing the circularity error enlarged to the defined scale. The capture ends after a whole circle or when the "Stop" softkey is pressed.

Continuous point capture

Data capture is initiated with the "Continuous" softkey. This softkey deletes the graph and starts drawing the circularity error enlarged to the defined scale. The capture ends when the "Stop" softkey is pressed.

When modifying the machine parameters, it is possible to see the new graph over the previous one or press the "Clear" key to only see the new one.

Stop the point capture



This softkey only stops the data capture. It does not stop the movement of the axes. To stop the movement of the axes, press the [STOP] key of the operator panel.

Press the "Stop" softkey to stop the data capture. This softkey stops the simple or continuous data capture at any time. Once the data capture is completed, two lines are drawn on the graph indicating the angular position of the maximum and minimum error.

Delete the graphic representation

Use the "Clear" softkey to delete the graph. This softkey deletes the graph and the data shown over it. It may be pressed at any time, even while it is being drawn.

22.

SETUP ASSISTANCE
Circularity (roundness) test.



22.3.7 Adjustment of the machine parameters involved

The best adjustment of the backlash peak may require modifying the value of certain machine parameters. This may be done directly in this window, thus without having to go to the machine parameter table. The parameters may be modified before or during the test.

When the CNC is turned on, the values defined in the tables are assumed. When changing these values, the changes are effective immediately and are assumed by the CNC until it is turned off. Once the setup is completed, the changes must be validated so they can be effective next time the CNC is turned on.

Selection of the parameters to be displayed

It is possible to interact with up to 11 different machine parameters. Some of these parameters are always visible, but some may be defined at will. However, the CNC will only admit the valid parameters for this type of adjustment. See ["22.3.9 Machine parameters that may be modified"](#) on page 435.

BACKLASH	Backlash
BAKANOUT	Additional velocity command pulse
BAKTIME	Duration of the additional velocity command pulse

How to select the parameters

The parameter setting area shows three data columns. The parameters are defined in the first column. The rest of the fields show the parameter value for each axis.

The list of valid parameters is shown when editing a parameter or pressing [SPACE]. Use the [▲][▼] keys to move through the list and [ENTER] to select one of them. Once selected, the rest of the fields show the parameter value for each axis.

Save the value of the parameters.



During start-up, the values accepted with the [ENTER] key are valid only until switch-off. Once the setup has been completed, the values must be validated using the corresponding softkey to save the values in the parameter table.

Password-protected parameters

If the machine parameters are protected, a password will be requested when trying to modify them. If entered correctly, it stores it and it does not request it again unless the CNC is turned off. If the password is wrong, the values cannot be modified and it requests it again every time.

The test may be executed even when not knowing the access password, but the machine parameters cannot be changed. The access passwords are determined from the utilities mode.

Initialize the value of the parameters



The Initialize softkey initializes all data in the window.

When the "Initialize" softkey is pressed, the parameters assume their default value, and not those defined by the OEM.

22.

SETUP ASSISTANCE
Circularity (roundness) test.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

22.3.8 Validate the changes and save the configuration used

Validate the parameters

The new data is assumed by the CNC until it is turned off. Once the setup is completed, the changes must be saved so they can be effective next time the CNC is turned on.



Use the "Validate" softkey to save the data. The CNC saves the new values in the CNC tables. When quitting the circularity test without having saved the parameters, the CNC will show a message warning about it and will give a chance to save them.

Save and load the configuration



The configuration saving option does not update the machine parameter table. To do that, use the Validate softkey.

The system permits saving the current configuration into a file in ASCII format (extension "TST"). This file only contains the configuration. It contains neither the graphics nor the values of the machine parameters. When loading a configuration, the parameters assume the value they have at the time.

Save the configuration

Press the "Save" softkey to save the current configuration. Select the folder and the file name and press [ENTER]. If there is a configuration already saved with the same name, it will ask whether it is to be replaced or not.

By default, the configuration is saved in the folder "..\MTB\DATA" or in the last folder selected by the user.

Load the configuration

Press the "Load" softkey to load a previously saved configuration. Select the folder and the file name and press [ENTER].

22.

SETUP ASSISTANCE
Circularity (roundness) test.



22.3.9 Machine parameters that may be modified

General machine parameters.

Parameter	Units	Update
PRELFIT	ms	Immediate
PRELOAD	%	Immediate
TCOMPLIM	%	Immediate
TINTIME	ms	Immediate
TPROGAIN	%	Immediate

Axis machine parameter.

Parameter	Units	Update
BACKLASH	mm(inch) or degrees	Immediate
BAKANOUT	rpm	Immediate
BAKTIME	ms	Immediate
ACCEL	mm(inch)/s ² or degrees/s ²	Beginning of the next block
ACCJERK	mm(inch)/s ³ or degrees/s ³	Beginning of the next block
ACFGAIN	%	Immediate
ACFWFACTOR	ms	Immediate
DECEL	mm(inch)/s ² or degrees/s ²	Beginning of the next block
DECJERK	mm(inch)/s ³ or degrees/s ³	Beginning of the next block
FFGAIN	%	Immediate
G00FEED	mm(inch) or degrees	Beginning of the next block
INPOSW	mm(inch) or degrees	Immediate
LACC1	mm(inch)/s ² or degrees/s ²	Beginning of the next block
LACC2	mm(inch)/s ² or degrees/s ²	Beginning of the next block
LFEED	mm(inch) or degrees	Beginning of the next block
MANACFGAIN	%	Immediate
MANFFGAIN	%	Immediate
MAXVOLT	mV	Immediate
PROGAIN	1000/min	Immediate
SERVOOFF		Immediate

22.

SETUP ASSISTANCE
Circularity (roundness) test.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

22.

SETUP ASSISTANCE

Circularity (roundness) test.



CNCelite
8058 8060
8065 8070

REF: 2508

23.1 Appearance of the diagnosis mode.

The diagnosis is a testing and displaying tool; it cannot be used to modify the displayed values. In this operating mode, the configuration of the CNC hardware and software can be tested.

The diagnosis screen shows the following information.

Configuration

System

Disk

Software

CNC8070 - 0

A

System

Info	Value
CNC Serial nr	#ERROR#
CNC Version	V02.21.00
Platform	V1.38
Image	<?> <?> <?> 0.00
ID1C	Not Available
ID3P	Not Available
ID4H	Not Available
ID Password	E761-4765-6C79-8B7A-4C00-0A74-4B69
Val. Code	No val. Code
Windows lic. n°	Terminal Server
Windows Version	6.2
User	Fagor Automation S. Coop
Physical Memory	7.92 Gbytes
Total Virtual Memory	2.00 Gbytes
Free Virtual Memory	1.53 Gbytes
Free disk space	104.35 Gbytes

B

(A) System configuration.

(B) Diagnostic information.

System diagnosis.

Information on the system elements; CNC version, user name, microprocessor(s) used, status of the various system memories, etc.

Disc diagnostics.

Hard disk monitoring via S.M.A.R.T. data.

Software diagnosis.

Information about the modules that make up the CNC software and the software options installed.

Hardware diagnosis.

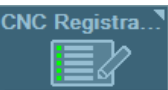
Information about the modules connected to the CNC via the buses.



CNCelite
8058 8060
8065 8070

23.1.1 Softkey menus.

Horizontal softkey menu.

Softkey.	Description.
	Generate the reportfagor.zip file with all the relevant information for proper error diagnosis and, if necessary, the end user can send it to Fagor Automation.
	View the history of errors and warnings issued by the CNC.
	User log.
	The diagnostic mode creates a file with the serial numbers of the CNC and the elements connected to it. ..\Users\Session\Reports\Serials.csv
	Access particular utilities of the operative system and other applications.
	Diagnosis mode allows you to generate one or more QR codes to register the following hardware on the Fagor Automation website. • CNC. • Quercus Drives. • EtherCAT regulators. • Motors.

Vertical softkey menu.

Softkey.	Description.
	Update (refresh) the data of the diagnosis window.
	Print the configuration in the pre-determined printer or save it as a file (prn format) at the CNC.

23.

DIAGNOSIS

Appearance of the diagnosis mode.

23.2 Configuration diagnosis

23.2.1 System diagnosis

When selecting this configuration element, the diagnosis window shows the list of the system elements and their values.

Configuration	System																																		
System Disk Software CNC8070 - 0	<table> <tr> <th>Info</th><th>Value</th></tr> <tr> <td>CNC Serial nr</td><td>#ERROR#</td></tr> <tr> <td>CNC Version</td><td>V02.21.00</td></tr> <tr> <td>Platform</td><td>V1.38</td></tr> <tr> <td>Image</td><td><?> <?> <?> 0.00</td></tr> <tr> <td>ID1C</td><td>Not Available</td></tr> <tr> <td>ID3P</td><td>Not Available</td></tr> <tr> <td>ID4H</td><td>Not Available</td></tr> <tr> <td>ID Password</td><td>E761-4765-6C79-8B7A-4C00-0A74-4B69</td></tr> <tr> <td>Val. Code</td><td>No val. Code</td></tr> <tr> <td>Windows lic. n°</td><td>Terminal Server</td></tr> <tr> <td>Windows Version</td><td>6.2</td></tr> <tr> <td>User</td><td>Fagor Automation S. Coop</td></tr> <tr> <td>Physical Memory</td><td>7.92 Gbytes</td></tr> <tr> <td>Total Virtual Memory</td><td>2.00 Gbytes</td></tr> <tr> <td>Free Virtual Memory</td><td>1.53 Gbytes</td></tr> <tr> <td>Free disk space</td><td>104.35 Gbytes</td></tr> </table>	Info	Value	CNC Serial nr	#ERROR#	CNC Version	V02.21.00	Platform	V1.38	Image	<?> <?> <?> 0.00	ID1C	Not Available	ID3P	Not Available	ID4H	Not Available	ID Password	E761-4765-6C79-8B7A-4C00-0A74-4B69	Val. Code	No val. Code	Windows lic. n°	Terminal Server	Windows Version	6.2	User	Fagor Automation S. Coop	Physical Memory	7.92 Gbytes	Total Virtual Memory	2.00 Gbytes	Free Virtual Memory	1.53 Gbytes	Free disk space	104.35 Gbytes
Info	Value																																		
CNC Serial nr	#ERROR#																																		
CNC Version	V02.21.00																																		
Platform	V1.38																																		
Image	<?> <?> <?> 0.00																																		
ID1C	Not Available																																		
ID3P	Not Available																																		
ID4H	Not Available																																		
ID Password	E761-4765-6C79-8B7A-4C00-0A74-4B69																																		
Val. Code	No val. Code																																		
Windows lic. n°	Terminal Server																																		
Windows Version	6.2																																		
User	Fagor Automation S. Coop																																		
Physical Memory	7.92 Gbytes																																		
Total Virtual Memory	2.00 Gbytes																																		
Free Virtual Memory	1.53 Gbytes																																		
Free disk space	104.35 Gbytes																																		

23.

DIAGNOSIS
Configuration diagnosis

23.2.2 Hard disk monitoring via S.M.A.R.T. data.

Hard disk status via S.M.A.R.T. data. (Self-Monitoring, Analysis, and Reporting Technology). Monitoring allows for early detection of the end of life of the hard drive. If the hard drive has more than 75% of life, the CNC will display a warning, and if it has more than 90%, it will display a fatal error that will force the CNC to shut down.

Configuration	Disk																		
System Disk Software CNC8070 - 0	<table> <tr> <th>Info</th><th>Value</th></tr> <tr> <td>Device model</td><td>KINGSTON SA400S37480G</td></tr> <tr> <td>Serial nr</td><td>50026B7783219897</td></tr> <tr> <td>Firmware Version</td><td>SBFKK1B3</td></tr> <tr> <td>User capacity</td><td>480.103.981.056 bytes [480 GB]</td></tr> <tr> <td>Unexpected power loss count</td><td>11</td></tr> <tr> <td>Temperature</td><td>31 (Min/Max 16/41)</td></tr> <tr> <td>Lifetime left</td><td>97%</td></tr> <tr> <td>Total sectors of write</td><td>6182</td></tr> </table>	Info	Value	Device model	KINGSTON SA400S37480G	Serial nr	50026B7783219897	Firmware Version	SBFKK1B3	User capacity	480.103.981.056 bytes [480 GB]	Unexpected power loss count	11	Temperature	31 (Min/Max 16/41)	Lifetime left	97%	Total sectors of write	6182
Info	Value																		
Device model	KINGSTON SA400S37480G																		
Serial nr	50026B7783219897																		
Firmware Version	SBFKK1B3																		
User capacity	480.103.981.056 bytes [480 GB]																		
Unexpected power loss count	11																		
Temperature	31 (Min/Max 16/41)																		
Lifetime left	97%																		
Total sectors of write	6182																		

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

23.2.3 Software diagnosis

This option may be used to analyze the modules that make up the CNC software and the software options installed.

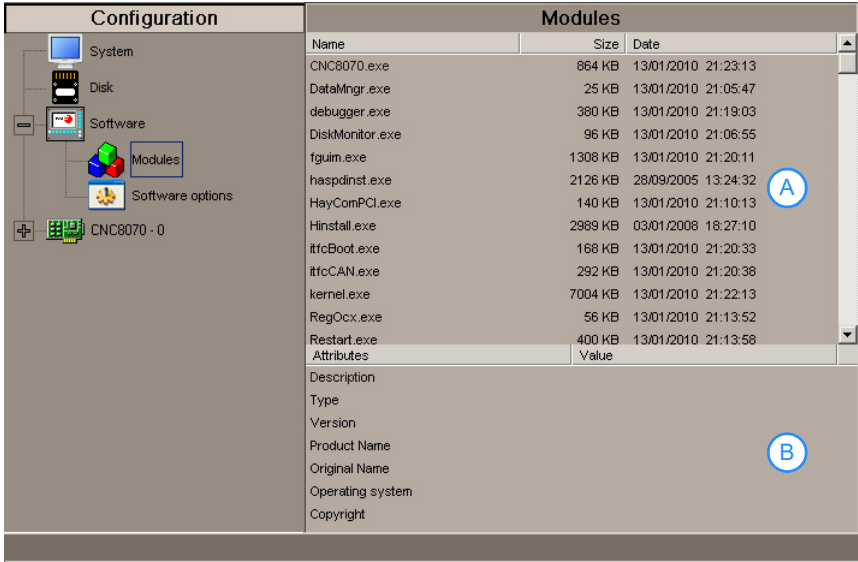
Module information

When selecting this configuration element, the diagnosis window shows the list of the modules that make up the CNC software. It shows the size of each module (in bytes) and the date it was created. When selecting a module from the list, the bottom of the screen shows more detailed information.

23.

DIAGNOSIS

Configuration diagnosis



(A) Modules listing.

(B) Detailed information of the selected module.

Name	Size	Date
CNC8070.exe	864 KB	13/01/2010 21:23:13
DataMgr.exe	25 KB	13/01/2010 21:05:47
debugger.exe	380 KB	13/01/2010 21:19:03
DiskMonitor.exe	96 KB	13/01/2010 21:06:55
fguim.exe	1308 KB	13/01/2010 21:20:11
haspdinst.exe	2126 KB	28/09/2005 13:24:32
HayComPCI.exe	140 KB	13/01/2010 21:10:13
Hinstall.exe	2989 KB	03/01/2008 18:27:10
itfcBoot.exe	168 KB	13/01/2010 21:20:33
itfcCAN.exe	292 KB	13/01/2010 21:20:38
kernel.exe	7004 KB	13/01/2010 21:22:13
RegOcx.exe	56 KB	13/01/2010 21:13:52
Restart.exe	400 KB	13/01/2010 21:13:58
Attributes	Value	

Description

Type

Version

Product Name

Original Name

Operating system

Copyright

Software options

When selecting this element of the configuration, the diagnosis window shows the software options currently installed. The "Validation code" horizontal softkey is also displayed, which allows the validation code associated with the CNC to be entered.

23.2.4 Hardware diagnosis.

When selecting this element of the configuration, the diagnosis window shows the software identification of the CNC's communications board (version, boot, checksum) and the type of buses connected to it. Likewise, it monitors the different voltage and temperature alarms.

Configuration

System

Software

Modules

Software options

CNC8070 - 0

Bus 1 - CAN

Group 1

Group 2

Node 0

Node 1

Node 2

Bus 2 - SERCOS

Modules

Info

Value

Version

7.3

Piggyback

Type

Bus 1

CAN

Bus 2

SERCOS

Hardware Test

Value

VBattery

No Battery

+5V

5.14 V

+12V

11.48 V

-12V

-12.09 V

Temperature

33 °C

23.

DIAGNOSIS

Configuration diagnosis

23.2.5 CAN bus diagnostics.

- Resources connected and their status.
- Groups connected and number of nodes that make up each group (detected and active).

Detailed information of the selected group.

- Group resources and their status.
- Information about the nodes that make up the group.

Detailed information of the selected node.

- Software version, boot and checksum of the code.
- Node status.
- Resources of each module.

The symbol next to the node indicates its status and its possible association.

Symbol.	Meaning.
	Inactive node.
	Node active, but in error status.
	Inactive node.

23.

DIAGNOSIS

Configuration diagnosis

23.2.6 Sercos bus diagnostics.

Listing of nodes and drive associated with each one of them. For each drive, it indicates its software version, type of drive and the motor connected to it.

General information about the bus

General purpose, it shows the following information about the Sercos bus.

Field.	Meaning.
Version	SERCON chip version.
Cycle time	Time between two synchronization messages (MST). It is the same as general parameter LOOPTIME.
T2	Time from the MST until the CNC starts sending the telegram with the position commands (MDT).
T3	Time from the MST until the drives have the position commands.
T4	Time from the MST when the drives read the feedback values.
TL	Time from the MST until the RT IT takes place at the CNC.
Distortion errors	Distortion error counter at the CNC.
Ring errors	Sercos ring error counter.
Phase	Current Sercos phase.
Phase status	Phase status (0 = OK).
Logic ID and name	Name and driveID (number of the rotary switch) of the Sercos axes.

23.

DIAGNOSIS
Configuration diagnosis

General information about the bus

It shows the following information for each drive.

General information

Field.	Meaning.
T1	Time from the MST until the drive starts sending its telegram (AT).
Control Word	Contents of the drive's control word.
Status Word	Contents of the drive's status word.
Name	Name of the Sercos axis.

Error information

Field.	Meaning.
"Reset Hard" counter	Number of times that the drive has been reset.
Distortion errors	Counter of distortion errors detected at the drive.
AT errors	Counters of AT reception errors of each drive.

Information on the last operation carried out by the service channel

Field.	Meaning.
ID	Processed Sercos ID.
Element	Processed data (Name, Units, min/max Value, Status, Value, Attributes).
Process	Operation carried out (Read, Write).
Command status	In processes with commands, it indicates its status.
Error code	Operation error code (0 = OK).

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

23.2.7 EtherCAT bus diagnostics.

General information about the bus.

23.

DIA GNOSIS

Configuration diagnosis

Configuration

System

Software

Hardware

Local Bus

Bus 1 - CANopen

ethercat

Slave 1

Slave 2

Bus 1 Ethercat	
Info	Value
busID	5
BusType	ETHERCAT
Bus state	OPERATIONAL
Alignment Error Counter	0
FCS Error Counter	0
Transmitted Frame Counter	28557
Received Frame Counter	28557

Information of the slaves.

Configuration

System

Software

Hardware

Local Bus

Bus 1 - CANopen

ethercat

Slave 1

Slave 2

Slave 1	
Info	Value
DeviceName	F-IL EC BK-PAC
DeviceType	5001
HardVersion	00
NodeID	1
ProductCode	33000060(0x325ADC)
Revision	1
SerialNumber	1358900810
SoftVersion	1.11
nModules	10

Module information.

Configuration

Bus 1 - CANopen

ethercat

Slave 1

Module 1

Module 2

Module 3

Module 4

Module 5

Module 6

Module 7

Module 8

Module 9

Bus 1 Node 1 Module 2	
Info	Value
ModuleTypeeld	702(0x2BE)
ModuleType	IB IL 24 DI/32, Inline terminal with 32 digital In...



CNCelite
8058 8060
8065 8070

REF: 2508

23.3 Report generation

Automatic reports generated when accessing the diagnosis mode

Every time when accessing the diagnosis mode, the CNC generates the following reports. These reports are saved in the folder C:\CNC8070 \Diagnosis\Blackbox.

SystemInfo.txt	Validation code.
SercosInfo.txt	Version of drives, motors and feedback.

Generate a report with the system configuration

A report with all the information offered by the diagnosis mode may be obtained from any section of the diagnosis mode. This produces a document with all the information about the FAGOR system installed on a machine such as the software version, status of the remote modules, installed drives and motors, etc.



To print the information, use the "Print" softkey. Pressing this softkey, the CNC will request the data it must include on the report. The CNC can create a report with the data shown on the screen or with the whole configuration of the system.

This information may be printed or saved to a file. When printing to a file, it is possible to select the name and location of the file.

Generating the warranty registration report

The CNC can generate a template so the end user can send to Fagor the documentation with the serial numbers of all the elements in order to obtain the corresponding warranty. This template, one printed out, contains the fields that the user must fill out and then send it to Fagor Automation.



To print the warranty registration form, use the "Print" softkey. After pressing this softkey, select "warranty registration report".

The report may be printed or saved to a file. When printing to a file, it is possible to select the name and location of the file.

23.

DIAGNOSIS
Report generation

23.4 View the history of errors and warnings issued by the CNC.



The softkey menu shows a window with the history of errors and warnings issued by the CNC. Pressing this softkey displays the following window.

Error LOG				
E6002	The PLC program is not running.	320-558	25/01/10 9:58:08	
E6002	The PLC program is not running.	320-558	25/01/10 9:58:08	
E4	Checksum error in PLC data.	694-359	25/01/10 9:58:06	

(A) Error or warning number.
 (B) Number of the channel where the error or warning occurred; this cell does not show any value when it is a system error or system warning.
 (C) Text of the error or warning.
 (D) Internal code of the error or warning.
 (E) Date when the error or warning occurred.

23.5 Generate the Fagor file for error diagnosis.



The softkey menu offers the possibility to generate the reportfagor.zip file with all the relevant information for proper error diagnosis and, if necessary, the end user can send it to Fagor Automation.

After pressing this softkey, the CNC creates the reportfagor.zip file and saves it in the "C:\Cnc8070\Diagnosis" folder. When pressing this softkey, the CNC also generates the following files in the "C:\Cnc8070\Users\Reports" folder and includes them in the reportfagor.zip file.

Diagcnc.txt	CNC diagnosis report.
Hardware.txt	Hardware configuration report.
Times.txt	Time statistics report.

23.6 Access particular utilities of the operative system and other applications.



When the "User" mode to access Windows is not available, this softkey may be used to access certain utilities of the operative system (Windows date and time, Task Manager, etc.) from the CNC itself. See chapter "24 APPS."

The utilities and applications that may be accessed with this softkey are preset by the OEM (file C:\...\MTB\MMC\Config\Apps.ini).

23.

DIAGNOSIS

View the history of errors and warnings issued by the CNC.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

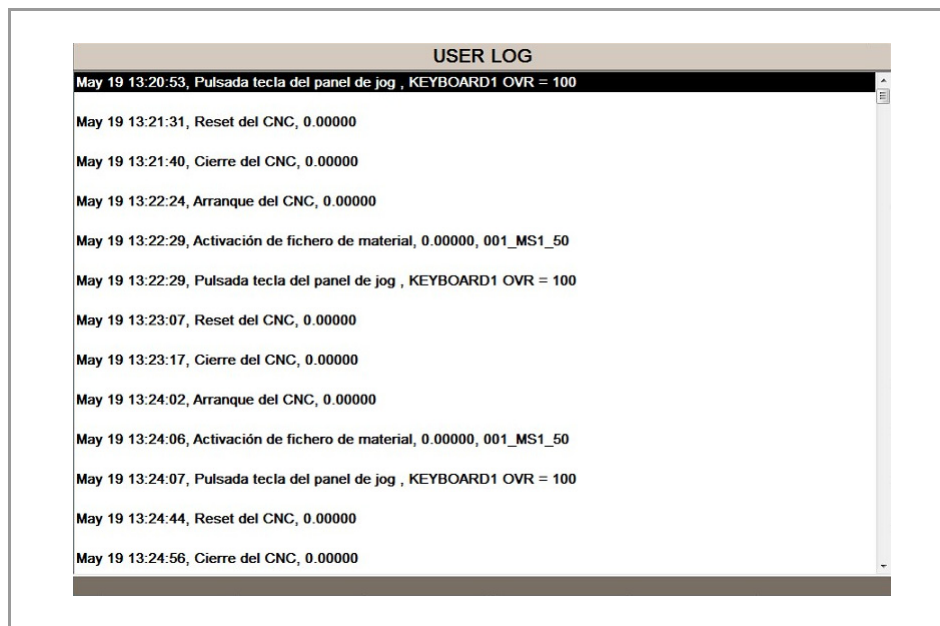
REF: 2508

23.7 User log.



Log that chronologically saves events that have occurred at the CNC and that could be important for the user. For example, the user log can be used to detect where and when an error has occurred. The user log shows the following events.

- Errors (but not warnings).
- Pressing keys on the jog panel.
- Predefined events; moving to Manual, program selection, etc.
- Events predefined by the OEM.



23.7.1 OEM configuration of the user log.

The CNC writes and encodes the events in the userlog.txt file. The manufacturer can modify the displayed information by decoding the file with the help of a script (WriteUserEventFile.js) to obtain the userEvent.txt file with the full log information.

File locations.

File.	Description.
userLog.txt.	Encoded user log. Path: ..\DIAGNOSIS\BLACKBOX\userLog.txt.
userEvent.txt	Decoded user log. Path: ..\Diagnosis\Blackbox\userEvent.txt.
WriteUserEventFile.js	Script to decode the user log. Path: The CNC first looks for it in the folder.. \MTB\Utilities\WriteUserEventFile.js and if not found, in ..\Fagor\Utilities\WriteUserEventFile.js
cncUserLog.txt	Texts associated with user events. Path: ..\MTB\Data\Lang\{idioma}\cncUserLog.txt

23.7.2 Maximum size of the userLog1.txt file

The maximum size of the userLog1.txt file is 200 kB. After this limit, the next event generates a copy such as userLog2.txt, deletes the contents of userLog1.txt and begins writing to userLog1.txt again. When the CNC encounters a problem renaming or deleting files, it creates new files called userLog3.txt and userLog4.txt.

User events. cncUserLog.txt file.

The cncUserLog.txt file contains the texts associated with user events and there may be one file for each language. This file can be modified by the OEM, which can add its own events. The first positions are reserved for events defined by Fagor and their meaning should not be modified. The manufacturer may include the variable (V.)E.USERLOG in its subroutines for writing the file events in the user log.

Event ID.	Description.
1	Pressing a user key or jog key on the operator panel.
2	Pressing a key on the operator panel; spindle keys, [START], [STOP], [ZERO], [SBLOCK], [RESET], override switch, feedrate mode switch.
3	Pressing a key on the operator panel; spindle override.
4	Going to manual mode.
5	Going to automatic mode.
6	Executing an MDI block.
7	Exiting MDI mode.
8	Starting a program in automatic mode.
9	CNC Reset.
10	Machine parameter validation.
11	Entering tool inspection mode.
12	Repositioning axes in tool inspection mode.
13	Exiting tool inspection mode.
14	CNC shut down.
15	Turning the CNC on.
16	Sending messages (#MSG).
17	Executing an interruption subroutine.
18	Activating material file.

Variables associated with log events.

Variable.	Meaning.
(V.)E.USERLOG	- Reading of the variable indicates the id of the last occurring user event. - The writing of the variable adds a line to the user log and defines the user event.
(V.)E.USERLOGVALUE	Additional value to add to the user event.

V.E.USERLOG=35

- The variable adds id 35 to the userLog1.txt file.
- The variable adds the following line to userLog1.txt: (DateAndTime),35,20,0,0

V.E.USERLOGVALUE = 12.12345

V.E.USERLOG = 35

- The variable adds the following line to userLog1.txt: (DateAndTime),35,20,0,0,12.12345

23.

DIAGNOSIS
User log.

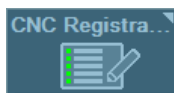
FAGOR
AUTOMATION



CNCelite
8058 8060
8065 8070

REF: 2508

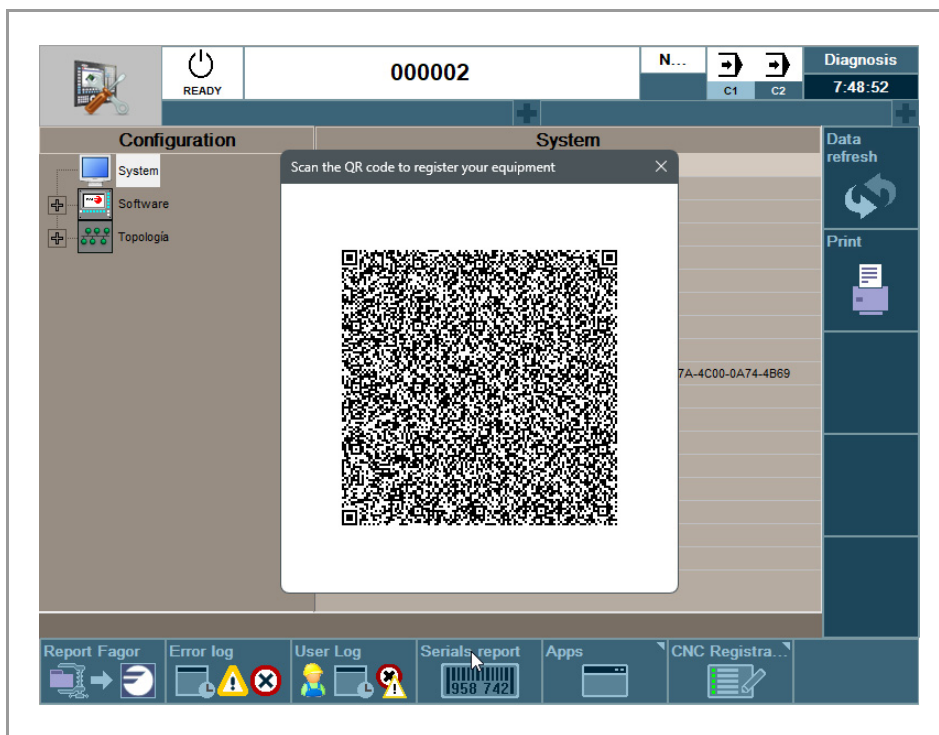
23.8 Hardware registration using QR codes.



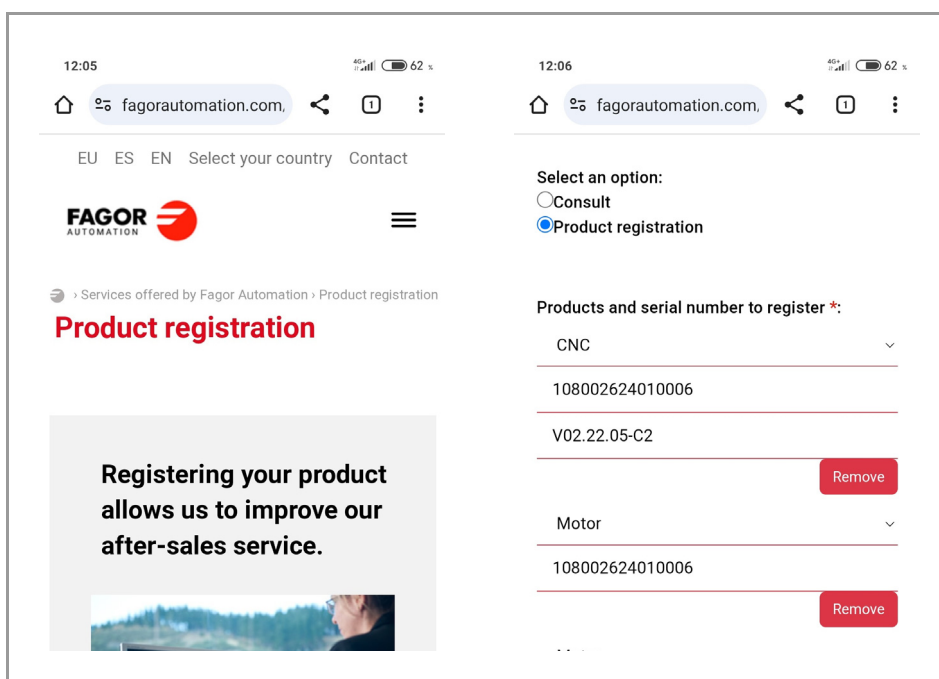
Diagnosis mode allows you to generate one or more QR codes to register the following hardware on the Fagor Automation website.

- CNC.
- Quercus Drives.
- EtherCAT regulators.
- Motors.

For registration, the CNC identifies the hardware type (CNC, drive or motor), serial number and software version. The CNC can register a maximum of 80 elements (CNC + 79 drives or motors). Each QR code can contain the information of 40 elements. If there are more than 40 elements, the CNC generates 2 QR codes.



Use a smartphone to read the QR codes and access the Fagor Automation website to register the devices.



23.

DIAGNOSIS

Hardware registration using QR codes.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

23.9 Operating Terms.

23.

DIAGNOSIS
Operating Terms.

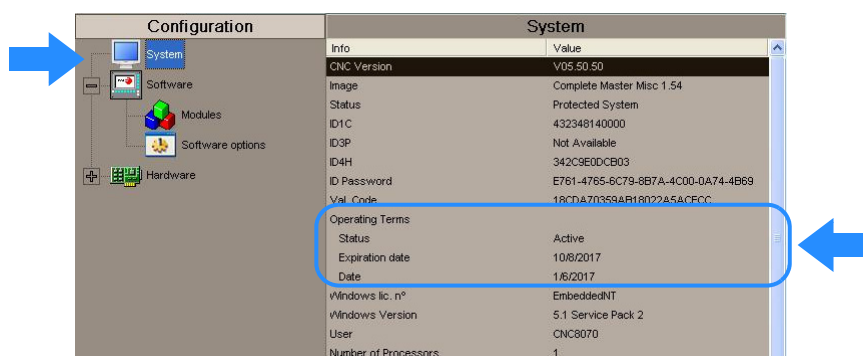
The "Operating Terms" option activates a temporary user license for the CNC, which is valid until the date determined by the OEM. While the license is valid, the CNC will be fully operational (according to the purchased software options). When the final validity of the license has expired, the CNC will not accept the [START] key and, consequently, it will not execute any programs. Seven days before the expiry of the temporary license, the CNC will warn the user with a message. The CNC will repeat this message at each start-up or when a day has passed since the last warning. The OEM can modify or cancel the final license date by providing a code to the user, who must enter it in the CNC.

This feature depends on the "Operating Terms" software option.

- If this option is active, before a valid code has been entered, the temporary user license will not be active.
- Once the "Operating Terms" option is canceled, it cannot be reactivated.
- When sending the code to the machine user, the expiration date and the hardware ID should also be sent so the user can check that the data are correct.
- The "Operating Terms" will block the CNC if the user sets back the date more than 24 hours, sets back the date more than once or the system date is prior to the date of the drivers on the CNC.

Status of the "Operating Terms" option.

The status of the "Operating Terms" option can be consulted in the diagnosis mode in the "System diagnosis" section.



This window displays the following information related to the "Operating Terms".

Status.

- Inactive: The "Operating Terms" option has not been activated.
- Active: The "Operating Terms" option is active.
- About to expire: There are less than 7 days left before the expiration date.

During the CNC start-up, and once 24 hours have passed since the last warning, the CNC displays a warning indicating that the period granted by temporary user license. The warning shows the CNC's hardware ID, which the user must send to the OEM to obtain the code that extends or cancels the expiration date.

- Expired: The temporary user license has reached the expiration date. During the start-up of the CNC, when the [START] key is pressed and once 24 hours have passed since the last warning, the CNC displays an error indicating that the user must obtain the code from the OEM to extend or cancel the temporary user license.
- Canceled: The "Operating Terms" option is canceled.
- Locked up: The CNC is blocked because an invalid date change has been detected.

Due date.

Expiration date of the temporary user license.

Date.

Current date on the CNC.

23.9.1 Generate the "Operating Terms" codes (for OEM only).

The "Operating Terms" program (independent of the CNC), governs the generation of the code to activate, modify or cancel the temporary user license. This program does not create any files, rather it creates an alphanumeric code that must be entered in the CNC. There is one version of the program for Windows and another for Android. With the latter, the codes may be generated on a smartphone using Android OS. To install the "Operating Terms" app on an Android device, execute the file "AppOpertingterms.apk" on that device.

1. Execute the "Operating Terms" program (which generates the codes) and enter the following data:

Windows version.

Android version.

- Hardware ID of the CNC. The code to activate or cancel the temporary user license will only be associated with this hardware ID. The CNC displays the hardware ID in the main window in diagnosis mode (identified as ID4H) and in the warnings and errors associated with the "Operating Terms".
- OEM KEY. This key is determined by the OEM and is only for the "Operating Terms" program; it must not be confused with the OEM password for the CNC. All operations with a hardware ID on the CNC must have the same OEM KEY. Once the temporary user license is active on a CNC, the OEM KEY cannot be changed for that CNC. The OEM KEY can have between 4 and 12 characters (letters and numbers).
- Due date. Expiration date of the temporary user license.
- Cancellation code. Mark this option to generate the code to cancel the temporary user license. If this option is selected, the code generated will have the final date of 31/12/2077.



- To cancel the temporary user license, the OEM must have a record of the hardware ID of the CNC and have a register of the OEM KEY used to generate the code.
- To change the expiration date of the temporary user license, the hardware ID and the OEM KEY contained in the new code must match those of the CNC. In addition, the final date in the new code must be later than the date of the active code on the CNC and later than the current date.

2. The program will display a screen with code of 14 numbers and letters.

Windows version.

Android version.

23.

DIAGNOSIS
Operating Terms.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508



This button enables the code to be copied to the clipboard, and then pasted into an email and sent. The button is only available in the Windows version.



This button enables the code to be sent by email, SMS, etc. The button is only available in the Android version.

23.**DIAGNOSIS**

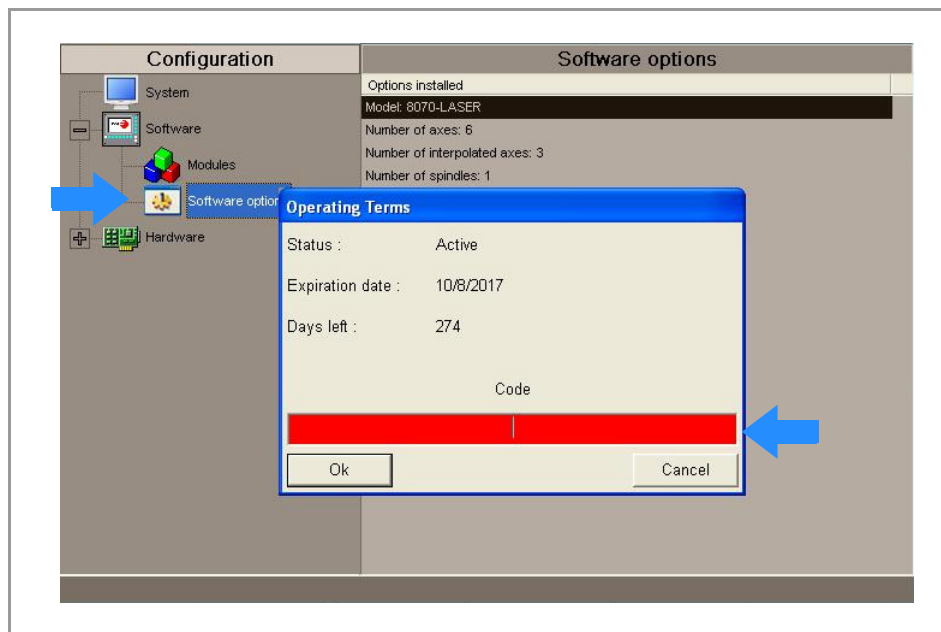
Operating Terms.

**CNCelite****8058 8060****8065 8070**

REF: 2508

23.9.2 Enter the CNC code to activate or cancel the temporary user license.

The code to activate or cancel the temporary user license is entered during diagnosis mode, in the "Software options", "Operating Terms" softkey section. If the CNC has the OEM password, the CNC will request this password the first time that someone tries to enter the code. The code can be entered with or without spaces.



DIAGNOSIS
Operating Terms.

23.



- When activating the "Operating Terms" feature and entering the first code, the date on the CNC must be correct.
- It is not possible to enter a final date older than the current one at the CNC.

One of the following two cases will occur when entering the new code:

- The code extends the period of the temporary user license. The code changes the final license date and the remaining days.
- The code cancels the temporary user license.

If the user makes a mistake when entering the code, the CNC will report this and will request the code again. The CNC will only allow three tries to enter the code per start-up; after the three tries, the CNC must be turned off and turned back on again. The CNC will display a message with the cause of the error.

- Wrong code. Type it again.
The user has entered the code incorrectly. Re-enter the code.
- Code generated wrong. Hardware ID does not match.
The hardware ID of the new code is not that of the machine; request a new code with the correct hardware ID from the manufacturer.
- Code generated wrong. The OEM KEY does not match.
The OEM KEY of the new code is not the same as that used by the manufacturer when "Operating Terms" were activated; request a new code with the correct OEM KEY from the manufacturer.
- Code generated wrong. It must be a later date.
The new expiration date is not later than the current expiration date; request a new code with the correct expiration date from the manufacturer.



When the OEM sends the "Operating Terms" code, it should also send the expiration date and the hardware ID in order to check that the data are correct.

FAGOR
AUTOMATION 

CNCelite
8058 8060
8065 8070

REF: 2508

23.

DIAGNOSIS
Operating Terms.



CNCelite
8058 8060
8065 8070

REF: 2508



The applications are available from the Diagnosis mode, softkey Apps. This softkey may be used to access particular utilities of the operating system (date and time, Task Manager, etc.) and applications of the CNC itself.



The utilities and applications that may be accessed with this softkey are preset by the OEM (file ..\MTB\MMC\Config\Apps.ini).

24.1 DiskMonitor application. Changing the work mode and device registration.

The DiskMonitor window allows for the CNC work and protection mode to be changed (administrator, setup and user mode). When the CNC is in setup or user mode, this window also allows for devices connected to the CNC to be registered. For both cases, this process implies having to restart the unit and, if the manufacturer has defined it this way, it will also be necessary to enter the corresponding password.



Accessing the DiskMonitor window.

- With the CNC application turned off, press the [ALT][D] hotkey or the active work mode icon on the task bar of the operative system (alongside the clock).
- With the CNC application open, press the [ALT] [D] hotkey or in diagnosis mode ("Apps" softkey).

Changing work mode.

This window displays a button for each work mode. Each of these buttons closes the application and restarts it in the selected mode. To start the CNC in administrator and setup modes, first enter the corresponding password (if the manufacturer has defined it in this way).

Permanently registering devices in setup and user modes.

When devices are connected physically to the device (for example, a pendrive) in setup or user modes, the device identifies and loads the drivers during each startup. The equipment does not register these devices permanently because the disk is protected. To force the registration of these devices you must use the DiskMonitor application.

By pressing the "Device recognition" button, the CNC does the following. Before beginning the process, the CNC prompts for confirmation and explains the implications of this process.

- 1 Closes the CNC application and restarts the computer in administrator mode.
- 2 Registers detected devices.
- 3 Restarts the device in the first mode (setup or user).

24.

APPS.

DiskMonitor application. Changing the work mode and device registration.



Protecting or unprotecting the folders.

Fagor delivers the unit with the 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.

Administrator mode allows folders to be protected or unprotected. The [i] button at the top right of the DiskMonitor displays the list of unprotected files and folders. With this list visible, pressing [CTRL]+[ALT]+[SHIFT]+[TAB] protects or unprotect any system folder or file that is not essential for the proper control operation.

24.

APPS.

DiskMonitor application. Changing the work mode and device registration.

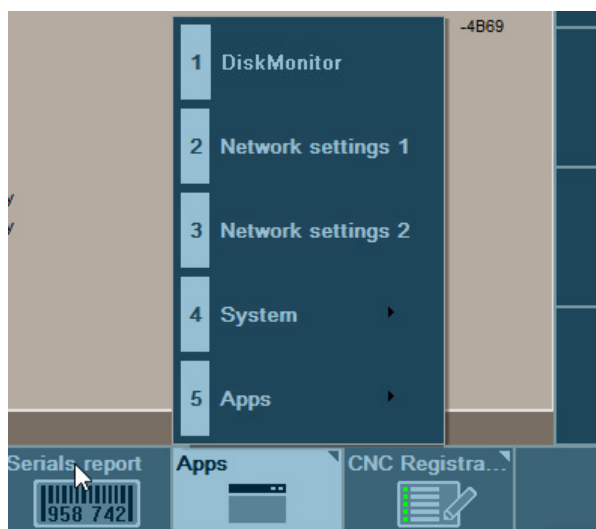


CNCelite
8058 8060
8065 8070

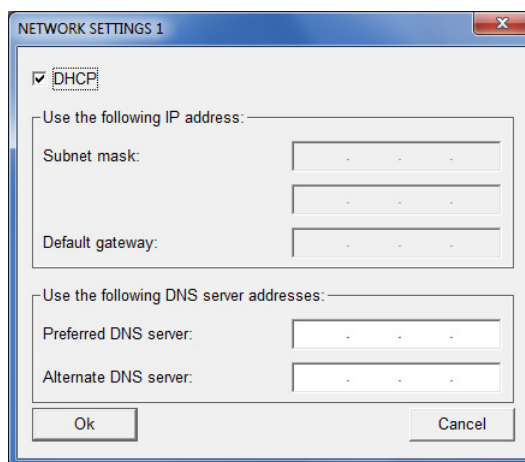
REF: 2508

24.2 Network Settings. Configuring network properties.

Integrated models have one network connector (H31B) and modular models have two (H31A and H31B). Configure the network properties by selecting the Apps > Network Settings soft key in the Diagnostic mode. The menu will display one or two network connections, depending on the central unit. In models with two network connections, the first connection corresponds to the H31A connector and the second to the H31B connector.

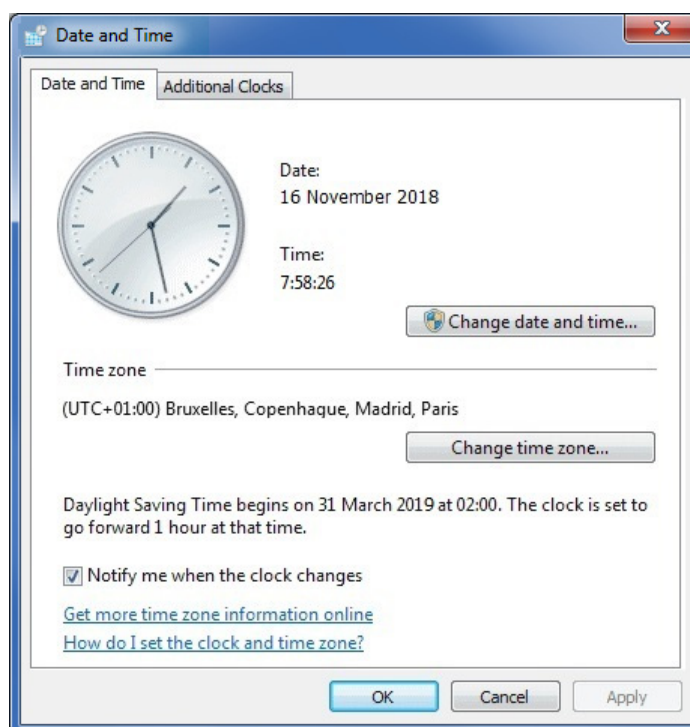


After selecting a network configuration, the CNC displays the following window. The data for defining the network configuration are only visible when there is an active connection (cable connected); otherwise, the window does not display any data.



24.3 System. Set the date and time.

In Diagnosis mode, open softkey Apps > Apps > FAEmailSender.



24.

APPS.

System. Set the date and time.

FAGOR
AUTOMATION



CNCelite
8058 8060
8065 8070

REF: 2508

24.4 APPS. Send emails.

The purpose of this application is to allow unattended machining, sending emails to the user with information on the events occurring at the CNC. The application allows selecting the type of event that generates the email (error, warning, etc.) and filtering them so only a few messages are sent and not others. The available events are the following:

- CNC errors (they may be filtered by number).
- Warnings (they may be filtered by number).
- CNC messages (all of them or none are sent).
- PLC messages (they may be filtered by number).

The application supports the Unicode character set and is available in several languages.

Server type; external (Internet) or internal (local network).

Emails may be sent in 2 ways; having the CNC connected to a local SMTP server or to an external SMTP server (like hotmail, gmail, yahoo, etc.).

- When the CNC is connected to an external SMTP server, make sure that the IP of the CNC does not filter the communications for the ports 25 (hotmail) and 465 (gmail, yahoo); otherwise, this email sending feature will not work. A valid email address is required on that server.
- When the CNC is connected to a local SMTP server, the notifications may be received without having access to Internet and even without authenticating at the server (depending on configuration).

In either case, to confirm that the CNC has access to the local SMTP server, execute the instruction "ping mail-server" (the name of the "mail-server" will vary from one company to another). See ["24.2 Network Settings. Configuring network properties."](#) on page 458.

In either case, to send emails to hotmail, gmail or yahoo email accounts, the communications must be enabled for the ports mentioned earlier. For example, to use the gmail server, check the access to it by doing ping to its "ping smtp.gmail.com" address.

Start up the application.

The application is pre-installed at the unit. The user must start up the application (it does not start up with the CNC) and configure it.

- 1 Start the unit up in user mode.
- 2 Start the CNC up, because the application communicates with it and, therefore, the CNC must be running in order to use it.
- 3 Start the email sending application. In Diagnosis mode, expand the softkey Apps > Apps > FAEmailSender. Optionally, a direct access may be created on the desktop to the FAEmailSender.exe to start the application (in administrator mode).
- 4 Configure the email sending application by entering the desired email accounts, network configuration and filters. Use the test message to confirm that the configuration is correct and that the message reaches its destination.
- 5 Enable the email sending service. The application is minimized automatically.
- 6 When one of the selected events occurs, the application will send the corresponding notification.



Files that make up the application.

The application consists of the following files, installed at the CNC in the following folders.

File.	Address and description.
FAEmailSender.exe	(8060) C:\FAGORCNC\Fagor\Release (8065) C:\CNC8070\Fagor\Release Application for configuring and enabling the email sending service.
FAemailsender.config	C:\Program Files\Fagor Automation\FagorApps Text file containing the configuration of the application.
FAfilters.config	C:\Program Files\Fagor Automation\FagorApps Text file containing the configuration of the filters applicable to the messages, errors and warnings. This file is created automatically the first time the application is executed.
FAemailsender.js	C:\Program Files\Fagor Automation\FagorApps. Javascript for sending the emails.
FAemailsenderLOG.txt	C:\Program Files\Fagor Automation\FagorApps. Text file that logs the application (sent messages, errors that came up, etc.). This file is created automatically the first time the application is executed.
FAEmailSender.ini	C:\Program Files\Fagor Automation\FagorApps. File that indicates the selected language.
*.lng	C:\Program Files\Fagor Automation\FagorApps\Language. Texts in various languages.

24.

APPS.

APPS. Send emails.

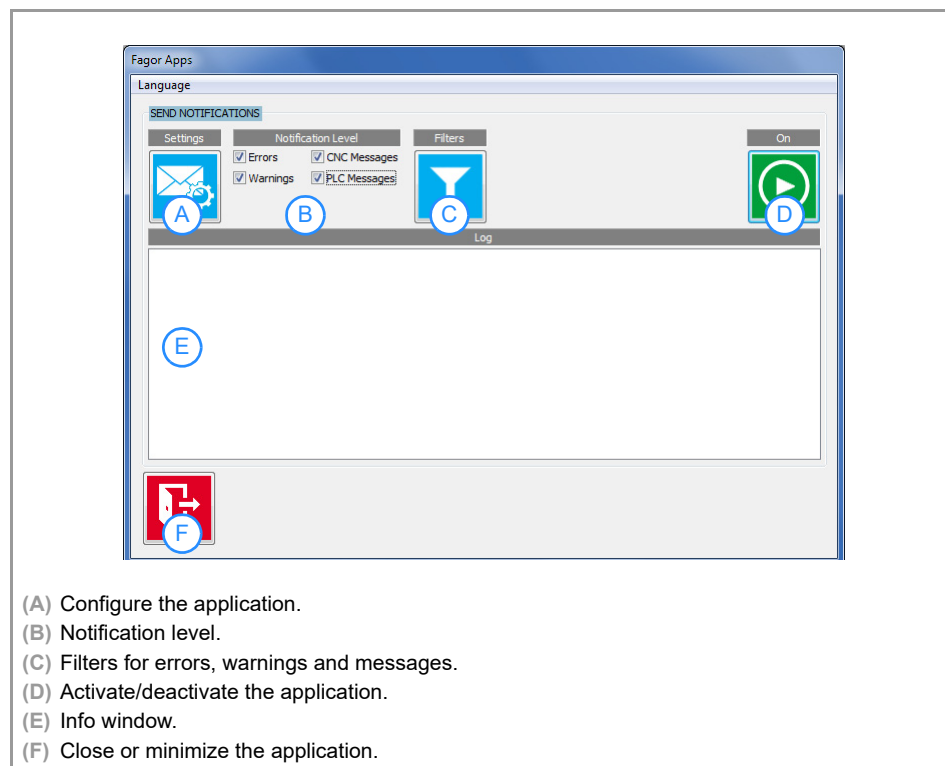


CNCelite
8058 8060
8065 8070

REF: 2508

24.4.1 Configure and activate the application.

The main window of the application may be used to configure the email service (configuration of network, email accounts and filters), activate the service, close the window and it also shows a log of the events occurring at the CNC.

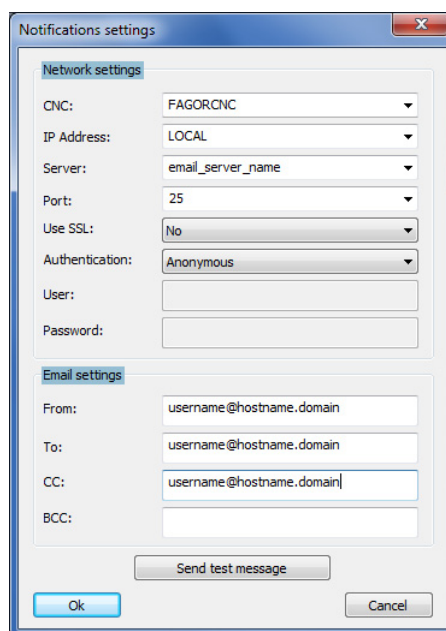


The application logs in the file FAemailsenderLOG.txt all the emails sent and errors coming up during the execution of the application.

Configure the application.



This icon is used to access the application configuration window. The architecture of the network must be known before carrying out this configuration. The configuration is saved in the file FAemailsender.config.



- As mail server, it is possible to use one of the predefined external servers (requires Internet connection) or the internal server of the company by entering its name.

24.

APPS.

Send emails.

FAGOR
AUTOMATION



CNCelite
8058 8060
8065 8070

REF: 2508

- When using the company mail server, the SMTP port, the SSL protocol being used and the authentication protocol depend on the configuration of the network. The user and the password also depend on the configuration of the network.

Notification level.

Select the type of event that generates an email; at least one of them must be selected for the application to work.

Filters for errors, warnings and messages.



This icon gives access to the window for selecting the errors and messages that the user wants to receive or not. The configuration is saved in the file FFilters.config.

Activate/deactivate the application.



After configuring the application, it must be activated so it can send the emails. When activating the application, it is minimized automatically. Closing the application will stop it (deactivate it), it must be active in the background.

Info window.

This window shows the status of the application and the list of actions that it carries out.

Close or minimize the application.



If the application is active, pressing this icon will minimize it.

If the application is not active, it asks whether to close the application or not; selecting "No" will minimize it on the task bar.

24.

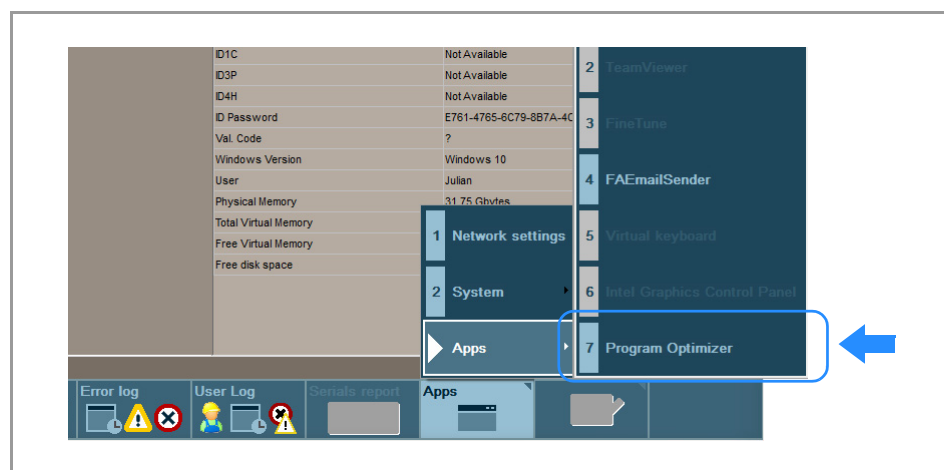
APPS.
APPS. Send emails.

24.5 Configuring the brightness and contrast of the monitors.

The CNC comes with the “Intel Graphics Control Panel” application to adjust monitor brightness and contrast.

24.6 Program Optimizer. Preprocessing of CAM programs.

Fagor uses an application to preprocess programs coming from CAD/CAM, which allows smoothing and correcting potential profile discontinuities to optimize machining and radius compensation. This application is available in Diagnosis mode, under softkey Applications > Apps (once installed).



Install the “Program Optimizer” application.

- 1 It is a prerequisite that the CNC is installed and that an MTB folder exists.
- 2 With the CNC in Administrator mode, run one of the application installers. Always use the latest version.

32-bit system.

- Use the setup “FagorProgramOptimizerInstaller_v1.00_x86.exe”.
- Installation folder: c:\Program Files (x86)\Fagor Automation\ProgramOptimizer

64-bit system.

- Use the setup “FagorProgramOptimizerInstaller_v1.00_x64.exe”.
- Installation folder: c:\Program Files (x64)\Fagor Automation\ProgramOptimizer

Usage mode:

- 1 Simulating the part-program. If the simulation is completed correctly, it is not necessary to use the “Program Optimizer” application.
- 2 If the simulation detects any errors in the paths, select the “Program Optimizer” application in Diagnosis mode. This application will optimize the last simulated program.
- 3 The application will notify which program it is going to correct and will ask for the tool radius to be used. The application creates a new program with the same name and the extension “NCX”.

The kinematic calibration allows a 5-axis system to be automatically adjusted, either during set-up or on a regular basis, by means of a measuring probe and a calibrated ball. With these elements, and with the data defined in the cycle, the CNC performs the necessary movements to calibrate the kinematics and displays the results so that the user can decide what actions to take. The cycle generates a file where you can see the adjustments made and possible mechanical problems.

The cycle offers two levels of calibration. The offsets calibration, which allows you to calculate the dimensions of the kinematics, and the rotational calibration, which makes it possible to offset non-linearities in a set of positions of both rotational axes. The proper procedure is to run the offsets calibration first and then the rotational calibration. After a calibration of the offsets, the existing rotational calibration is no longer valid and has to be repeated.

Requirements for the kinematics.

- The first three axes of the channel must be linear.
- The axes of the kinematics can be Hirth axes.
- The CNC allows the calibration of kinematics 1 to 16 and vector kinematics 50 to 52 (parameter TYPE). For kinematics types 1 to 16, if the cycle does not correctly calibrate any vector, it is recommended to re-define the kinematics as vectorial and repeat the calibration.
- The kinematics must be defined using values that are as close to the real ones as possible, by taking the tool base as a reference.
- The rest position of the rotary axes must be 0°. The rest position is when the tool is perpendicular to the work plane (parallel to the longitudinal axis).

Requirements for the probe.

- The probe must be accurately calibrated in radius and length, and defined in the tool table with zero length wear.
- The possible delay from probing until the CNC receives the signal (infrared communication, etc.) must be defined in the PROBEDELAY parameter of each axis involved in probing. The calibration cycle of the touch probe can calculate this parameter.
- If the CNC cannot orient the spindle (it cannot execute M19), the probe must be properly calibrated mechanically for X and Y (without any deviations).

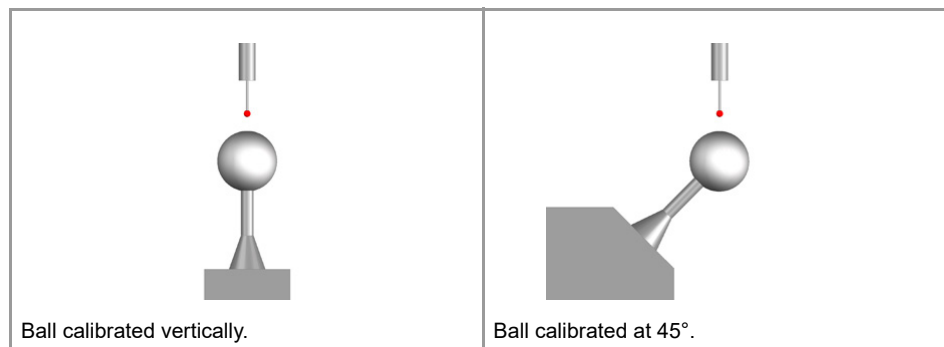
Calibration of the stylus in length.

It is recommended to calibrate the stylus length as follows:

- 1 Using a master part, touch the tip of the tool to the surface and make a dimension preset G92 Z{height}, where Z is the height of the master part.
- 2 Define the probe to an approximate length, place it in the spindle and run the surface measurement cycle.
- 3 At the end of the cycle, parameter P299 shows the difference between the approximate pr length and the actual length. The actual length of the touch probe is the length of the tool plus the value of P299.

Requirements for the calibrated ball.

To calibrate the kinematics, a calibrated ball of any size is required (the radius of the ball is defined by the cycle). The calibrated ball may be set vertically or at 45°.



Subroutines associated with the calibration of kinematics.

Subroutine.	Meaning.
KinCal_Begin.nc	The CNC executes this subroutine before the cycle.
KinCal_End.nc	The CNC executes this subroutine after the cycle.
Kin_G_Move.nc	This subroutine allows you to customise the movement of the rotary axes.

The kinematics calibration cycle has two associated subroutines (KinCal_Begin.nc and KinCal_End.nc), which the CNC executes before and after the cycle. The OEM can add initial and/or final conditions to the calibration cycle using these subroutines. Also, the OEM can customize the movement of the rotary axes in the subroutine Kin_G_Move.nc. Calling this subroutine is defined in the GMOVE macro of the KinCal_Begin.nc subroutine, as shown in the subroutine provided as an example.

Subroutine location.

The subroutines KinCal_Begin.nc and KinCal_End.nc, as well as the special subroutines defined by the manufacturer, must be saved in the folder ..\Mtb\Sub. The subroutine Kin_G_Move.nc must be saved in the folder ..\Fagor\Sub. All channels use the same subroutines.

25.

KINEMATICS CALIBRATION.

25.1 Kinematics calibration (Kinematic offsets).

The calibration of the offsets calculates the dimensions of a kinematics for the first time on the basis of approximate data, and also recalibrates it from time to time in order to correct possible deviations that may arise in the daily work of the machine. After the calibration has been completed, the CNC will display on the results page the kinematics parameters with the original and calculated values, so the user can select which values to save.

Files generated from the kinematics calibration.

File.	Meaning.
KinCalReport_{KINID}_{DATE}.m	This file shows the calibration data, where you can see the adjustments made and deduce possible mechanical problems. {KINID} shall be the number of the kinematic and {DATE} shall be the date.

Calibration generates the file in the folder ..\Users\Session\Reports.

25.1.1 Page 1. Cycle data.

On this page, the necessary data to define the kinematics offsets must be defined. Use the horizontal softkey menu to select the axes to be calibrated; it is recommended to calibrate all the axes of the kinematics.

The screenshot displays the 'Kinematics calibration' interface. It includes a 3D model of a machine tool with axes X, Y, Z, and a rotary axis C. The interface is divided into several sections:

- Top Left:** A diagram showing the tool's geometry with dimensions D_s , R_s , D_r , F_a , F , and F_s .
- Top Right:** A 3D model of the machine tool with a coordinate system (X, Y, Z).
- Bottom Left:** A table of 'Actual' values for various axes.
- Bottom Center:** A table of 'Cycle parameters'.
- Bottom Right:** A table of 'Rotative Axis Parameters'.

Labels A through E are placed over specific elements in the interface:

- A:** Points to the 'Actual' values table.
- B:** Points to the 'Number and type of active kinematics' (D 1, Type 4).
- C:** Points to the 'Probe data' (T 1, D 1, L 181.906, R 2.968).
- D:** Points to the 'Rotative Axis Parameters' table.
- E:** Points to the 'Data for calibration' (Fs, F, F) table.

Axis	Value
X	-3.215
Y	27.659
Z	-59.268
C	0.000
B	0.000
V	27.659
Z1	-66.803

Parameter	Value	Parameter	Value
Fs	500.00	Ds	10.00000
F	200.00	Dr	3.00000
F	2500.00	Rs	11.11200

Parameter	C	B
Pi	0.00	-90.00
Pf	300.00	90.00
Np	5	5
Pos	45.00	0.00

- A Current position of the axes (not editable).
- B Number and type of active kinematics (not editable).
- C Probe data (not editable).
- D Rotary axis parameters (editable).
- E Data for calibration (editable).

Number and type of active kinematics.

The cycle shows the active kinematics. The kinematics can be changed from MDI mode.

25.

KINEMATICS CALIBRATION.
Kinematics calibration (Kinematic offsets).

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

25.

KINEMATICS CALIBRATION.
Kinematics calibration (Kinematic offsets).**Probe data.**

T	Number of the tool associated with the probe.
D	Number of the tool offset associated with the probe; if not programmed, the cycle will take the first offset associated with the tool set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.
L	Probe length.
R	Probe radius.

Parameters of the rotary axes.

Pi	Starting position of the rotary axes.
Pf	Final position of the rotary axes.
Np	Number of points to calculate.
Pos	Initial position of the second rotary axis for the movements of the first rotary axis.

Initial and final position of rotary axes.

For each rotary axis to be calibrated, a starting point (Pi) and a final point (Pf) of its movement must be defined, bearing in mind that all the axes must stay within the limits. The cycle makes it possible to start the calibration of the main rotary axis at any position of the secondary rotary axis. If the rotary axis is of the "module" type, the cycle will only allow part of the travel; the part of the itinerary to be calibrated; the not calibrated part is not compensated.

Number of points to calculate.

Three points must be measured, but the result is best with more than four points. If the rotary axis is of the "module" type and you want to calibrate the whole range from 0° to 360°, as both points are the same, you have to define a minimum of four points.

Check the position of the axes with respect to the itinerary limits.

To check that the axes remain within their limits, move the probe over the ball, at short distance of around 10 mm (0.4 inches). In that position, activate RTCP and, in jog mode, check that the linear axes have enough room to move within the programmed positions. Bear in mind that in each position of the rotary axis, the linear axes must move the programmed Ds distance in order to be able to calculate the center of the ball.

Parameters of the cycle.**Feedrate.**

Fa	Positioning feedrate
Fs	Rapid probing feedrate. Then, the CNC will repeat the probing movement at feedrate · F.
F	Accurate probing feedrate.
Ssp	The speed and turning direction for spindle orientation; the sign means the turning direction. The CNC only offers this parameter when the spindle may be oriented (M19), so the probe always touches the ball with the same surface.



It is up to the user to make sure that the probe can rotate without danger. If the probe has a cable, check that the cable does not hinder the rotation of the probe.



Safety distance and withdrawal.

- Ds Safety distance between the probe and the calibrated ball. This parameter only admits positive values greater than 0 (zero).
- Dr Distance the probe withdraws after the first probing movement. Once it withdraws this distance, it makes a second probing movement. This parameter only admits positive values greater than 0 (zero).

Ball size.

- Rs Calibrated ball radius. See the OEM characteristics to obtain this value.

Adjusting the probe.

Icon.	Meaning.
	The cycle verifies the probe adjustment and will display a warning if it detects a non-adjusted PROBEDELAY value.
	The cycles does not verify the probe adjustment.

Points to probe for calculating the center of the ball.

Icon.	Meaning.
	The points to be measured are on the axes.
	The points to be measured are 45° with respect to the axes.

25.

KINEMATICS CALIBRATION.
Kinematics calibration (Kinematic offsets).

25.1.2 Page 2. Calibration progress.

The CNC displays this page during the calibration process. The calibration process allows toggling between pages 2 and 3 with the "View" softkey in the horizontal menu.

Calibration positions

C	-0.00	75.00	150.00	225.00	300.00
B	-90.00	-45.00	0.00	45.00	90.00

Measured deviations

	C	B
YZ plane(°)		0.050484
ZX plane(°)		-
XY plane(°)		0.001471
Positioning(mm)		0.250176
Circularity (mm)		0.091946

M41

G7 G17 G92 G100 CS K

F real	200.08 mm / min
F prog	200.00 mm / min
Dyn	100 % 100 %
S real	0.0 RPM
S prog	0.0 RPM
S pos	0.0

	Command	Actual	To Go	Machine Base
X	0.000	14.708	- 14.717	1550.514
Y	0.000	14.708	- 14.717	-1185.052
Z	5.556	5.556	0.000	-934.160
C	300.000	300.000	0.000	300.000
B	45.000	45.000	- 0.000	45.000
V	-1202.856	-1185.543	- 17.313	-1185.052
Z1	-66.803	-66.803	0.000	-66.803

A Position of the rotary axes at each calibration point. The cycle shows the position being calibrated in yellow and the completed positions in green.

B Measured deviations:

Plane YZ: Angular deviation of the rotary axis from the theoretical axis, in each plane.
 Plane ZX:
 Plane XY:

Positioning: Maximum axis positioning error; difference between the first probing and the probing after calculating the center of the sphere.

Circularity: The calibration cycle adjusts the measured points to a circle. The circularity shows the error of the point that is most deviated from the circle fit.

C Active functions.

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

"F real" Real (actual) feedrate of the axes.

"F prog" Programmed feedrate.

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

"Dyn" Percentage of dynamics being applied.

E Information related to the spindle speed "S".

F Programmed coordinate and current position of the axes.

25.

KINEMATICS CALIBRATION.
 Kinematics calibration (Kinematic offsets).

25.1.3 Page 3. Calibration result.

The CNC displays this page during the calibration process. This page shows the probed coordinates and the calculated data. The calibration process allows toggling between pages 2 and 3 with the "View" softkey in the horizontal menu.

Measured centers				Calculated centers		M41	
	C1	C2	C		B	G0 G7 G17 G92 KIN	
X	1660.2250...	1475.3751...	1155.0533...	Xc	1410.9420...		
Y	-973.310600	-732.577200	-848.5679...	Yc	-973.342129		
Z	-917.316100	-917.089400	-916.8917...	Zc	-917.100510		
Ang	-0.000100	74.999900	150.00010	R	249.283142		
Error	0.178654	0.468541	0.559732	nx	0.000819		
ECir	-0.000138	-0.045577	0.069264	ny	-0.000245		
				nz	1.000000		
				Error	0.390330		
				ECir	0.052474		

	Command	Actual	To Go	Machine Base
X	-128.033	-128.033	0.000	1411.085
Y	222.120	222.120	0.000	-973.115
Z	148.500	148.500	0.000	-804.045
C	0.000	0.000	0.000	0.000
B	0.000	-0.000	0.000	-0.000
V	222.120	222.120	0.000	-973.115
Z1	-66.803	-66.803	0.000	-66.803

C:\CNC8070\USERS\SESSION\REPORTS\KinCalReport_ID1_[05.07.2019]_[11h 38m 08s].m

A Ball centers, measured in each position.

B Calculated centers.

Xc, Yc, Zc: Axis rotation center in base machine coordinates.

nx, ny, nz: Rotation vector.

Error: Error due to machine parameters and geometry. The better the kinematics parameters are set, the smaller the error should be.

ECir: Circularity (roundness) error due to machine geometry.

C Active functions.

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

"F real" Real (actual) feedrate of the axes.

"F prog" Programmed feedrate.

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

"Dyn" Percentage of dynamics being applied.

E Information related to the spindle speed "S".

F Programmed coordinate and current position of the axes.

25.

KINEMATICS CALIBRATION.
Kinematics calibration (Kinematic offsets).

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

25.1.4 Page 4. Display of calculated data.

The CNC displays this page when the calibration process ends. This page shows the TDATA values of the kinematics, the values of the machine parameters and the new calculated values. To take over the calculated values (Calculated value or Calculated Offset), select them and press the Validate softkey.

MNEMONIC	Initial value	Calculated value	Initial offset	Calculated offset	Maximum offset
TDATA 1	159.560332820088	✓ 159.556456296640	0.000000000000	☐ -0.003876523448	1.000000000000
TDATA 2	0.057972957087	✓ -0.114191621005	0.000000000000	☐ -0.172164578092	0.000000000000
TDATA 3	-0.065217295465	✓ -0.008688370141	0.000000000000	☐ 0.056528925324	0.000000000000
TDATA 4	0.000000000000	☐ 0.000000000000	0.000000000000	☐ 0.000000000000	0.000000000000
TDATA 5	0.11212581	✓ 0.12178855	0.000000000000	☐ 0.01186274	0.000000000000
TDATA 6	0.000000000000	☐ 0.000000000000	0.000000000000	☐ 0.000000000000	0.000000000000
TDATA 7	0.000000000000	☐ 0.000000000000	0.000000000000	☐ 0.000000000000	0.000000000000
TDATA 8	0.000000000000	☐ 0.000000000000	0.000000000000	☐ 0.000000000000	0.000000000000
TDATA 9	0.000000000000	☐ 0.000000000000	0.000000000000	☐ 0.000000000000	0.000000000000
TDATA 10	0.000000000000	☐ 0.000000000000	0.000000000000	☐ 0.000000000000	0.000000000000
TDATA 11	0.000000000000	☐ 0.000000000000	0.000000000000	☐ 0.000000000000	0.000000000000
TDATA 12	0.000000000000	☐ 0.000000000000	0.000000000000	☐ 0.000000000000	0.000000000000
TDATA 13	-0.170680649668	✓ -0.012443057850	0.000000000000	☐ 0.158237591818	0.000000000000
TDATA 14	0.065217295465	✓ 0.008688370141	0.000000000000	☐ -0.056528925324	0.000000000000
TDATA 15	-159.560332820088	✓ -159.556456296640	0.000000000000	☐ 0.003876523448	0.000000000000
TDATA 16	0.000000000000	☐ 0.000000000000	0.000000000000	☐ 0.000000000000	0.000000000000
TDATA 17	0.000000000000	☐ 0.000000000000	0.000000000000	☐ 0.000000000000	0.000000000000
TDATA 18	0.000000000000	☐ 0.000000000000	0.000000000000	☐ 0.000000000000	0.000000000000
TDATA 19	0.000000000000	☐ 0.000000000000	0.000000000000	☐ 0.000000000000	0.000000000000
TDATA 20	0.000000000000	☐ 0.000000000000	0.000000000000	☐ 0.000000000000	0.000000000000

- A Parameters of the kinematics.
- B Initial value of the parameter, defined in the machine parameters.
- C New parameter value, calculated during calibration.
- D Initial offset of the parameter, defined in the machine parameters.
- E New parameter offset, calculated during calibration.
- F Maximum offset allowed for the parameter, set at the machine parameters.

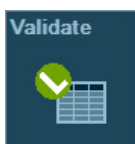
25.1.5 How to calibrate a kinematics.

To calibrate the kinematics, the user must first define a series of data, such as the dimensions of the calibration ball, the amplitude of the movements of the rotary axes, etc. The CNC then makes the necessary movements to calibrate the kinematics. Once the movements are completed, the CNC shows the data obtained and it suggests new values for the kinematics. It is up to the user to update the corresponding machine parameters.

How to calibrate a kinematics.

- 1 Calibrate the stylus and measure its length.
- 2 Activate the kinematics to be calibrated.
- 3 Go into jog mode. Place the axes of the kinematics in their resting position. Place the probe about 10 mm (0.4 inch) over the ball.
- 4 Access the kinematics calibration mode. This mode is only available from the task window ([CTRL][A]).
- 5 When accessing this kinematics calibration mode, the CNC shows the data setting page. On this page the data must be defined that is required for calibration, as well as the dimensions of the ball, the path of the rotary axes, the probe movement, etc. The horizontal softkey menu allows the axes to be selected for calibration; it is recommended to calibrate all the axes of the kinematics.
- 6 Press the [START] key to start the calibration. The CNC displays the progress of the calibration on two pages, accessible from the "View" softkey of the horizontal menu.
- 7 In the first phase, the CNC measures the calibrated ball by touching the stylus at 4 different points (at 90°). The CNC then executes the calibration phase, rotating the rotary axes as defined in the data page.
- 8 When the data capture is completed, the CNC switches to the page of calculated data. This page shows, for each TDATA parameter of the kinematics, the values and offsets defined in the machine parameters and the new calculated values. On this page, it is possible to select whether to update the OEM defined offset values on the machine parameter table or to keep the original values.

Updating the machine parameter tables.



Depending on the protection mode the CNC is in, it is possible to modify the field "Value" (SETUP mode) or the field "Offset" (SETUP or USER mode) of machine parameter table. To do this, select the data from the results page to be modified and press the "Validate" softkey.

KINEMATICS TABLE : KINEMATIC 2			
Mnemonic	Value	Max Offset	Offset
TYPE	2		
COMPID	5		
HEADREF	No ▼		
TDATA 1	153.238983089554	1	0
TDATA 2	-0.172823553469	1	0
TDATA 3	-0.764892736303	1	0

Machine parameter tables for the kinematics.

25.

KINEMATICS CALIBRATION.

Kinematics calibration (Kinematic offsets).

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

25.1.6 File with the results of the calibration.

This file contains the calibration data, where you can see the adjustments made and deduce possible mechanical problems. The data displayed in the file depends on the type of kinematics.

- Kinematic data.
- Rotary axis parameters, defined in the cycle.
- Cycle and probe data, defined in the cycle.
- TDATA and OFFSETS parameter values. Original values (those defined in the parameter table) and those calculated by the calibration cycle.
- Measured data.

Data.	Meaning.
Meas1	Center of the sphere, measured by the cycle.
Palp1	First stylus measurement at each point.
SphereRadius	Radius of the sphere, measured by the cycle.
AngleDiff	Difference in the measured angle.
Error1	The first column shows the difference between the first palpation (Palp) and the palpation after calculating the center of the sphere (Meas). The second column shows how much the above difference has deviated from the circle adjustment made.

- Calculated results.

Result.	Meaning.
Center	Center of the calculated circle.
Normal	Vector calculated for the rotary axis. During execution, the cycle shows the projection of this vector on the YZ, ZX, XY planes.
R	Radius of the calculated circle.
AD	Average angle difference.
Error	Average positioning error of the rotary axis; difference between the first probing and the probing after calculating the center of the sphere.
ECir	Circularity error. The calibration cycle adjusts the measured points to a circle. This figure shows the average deviation of the circle.

25.

KINEMATICS CALIBRATION.
Kinematics calibration (Kinematic offsets).

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Result.	Meaning.
SphereRadius Theoreticall	Theoretical radius of the sphere.
Rotational deviation	Deviation from the mean value of the positioning error.

25.

KINEMATICS CALIBRATION.

Kinematics calibration (Kinematic offsets).



CNCelite
8058 8060
8065 8070

REF: 2508

25.2 Rotational kinematics calibration.

Rotational calibration of the kinematics allows you to calculate and offset non-linearities in a set of positions of both rotational axes, by means of an advanced calibration cycle. This calibration allows you to correct deviations in the position of the linear axes X Y Z, for different positions of the rotary axes, in order to achieve a more accurate adjustment of the measured points. At the end of the calibration, the CNC displays a page with the results and offers the option of validating them, in which case it updates the COMPID machine parameter of the kinematics.

Files generated from the kinematics calibration.

File.	Meaning.
Kin{KINID}RotationalData{COMPID}.csv Kin{KINID}Rotational{COMPID}.mp	The csv file contains the calculated errors and the mp file contains the data for volumetric offset. {KINID} shall be the kinematics number and {COMPID} shall be the volumetric offset number.

After validating the results, the calibration generates the files in the ..\Users\Session\Data folder.

25.2.1 Page 1. Cycle data.

The screenshot displays the 'Rotational calibration' interface. It includes a 3D model of a machine tool, a table of 'Actual' positions for X, Y, Z, A, and B axes, and several parameter sections: 'Tool data', 'Rotative Axis Parameters', and 'Cycle parameters'. The 'Actual' table shows X: 0.000, Y: 0.000, Z: 15.123, A: 0.000, and B: 0.000. The 'Tool data' section shows T: 14, D: 1, L: 110, and R: 9.875. The 'Rotative Axis Parameters' section shows A and B parameters for Pi, Pf, and Np. The 'Cycle parameters' section shows Fs, F, Fa, Ssp, Ds, Dr, and Rs. The 'Comp ID' is 1.

- A Kinematic number (non-editable) and CompID (editable).
- B Probe data (not editable).
- C Rotary axis parameters (editable).
- D Current position of the axes (not editable).
- E Data for calibration (editable).

Number of the active kinematics.

The cycle shows the active kinematics. The kinematics can be changed from MDI mode.

25.

KINEMATICS CALIBRATION.
Rotational kinematics calibration.

Volumetric offset number.



The CNC has four generic volumetric offsets (VOLCOMP parameter) and six associated with kinematics (COMPID parameter). The latter are activated and deactivated together with the kinematics, they cannot be managed from the PLC.

The Comp ID defines the volumetric offset number associated with the kinematics, where the cycle will wait for the data necessary for the offset. By default, Comp ID takes the value defined in the machine parameters (COMPID parameter). Each Comp ID can take the values 1 to 6, but no two kinematics with the same value are allowed. To deactivate the offset of a kinematics, the COMPID parameter of the kinematics table must be set to 0, and the machine parameters must be validated. If a previously calibrated Comp ID is selected, the cycle will offer the same intervals and number of points of the previous calibration, which can be modified before launching the cycle.

Probe data.

T	Number of the tool associated with the probe.
D	Number of the tool offset associated with the probe; if not programmed, the cycle will take the first offset associated with the tool set in the tool table. The cycle shows an icon of the tool type; this icon cannot be modified from the cycle.
L	Probe length.
R	Probe radius.

Parameters of the rotary axes.

Pi	Starting position of the rotary axes.
Pf	Final position of the rotary axes.
Np	Number of points to measure.

Initial and final position of rotary axes.

For each rotary axis to be calibrated, a starting point (Pi) and a final point (Pf) of its movement must be defined, bearing in mind that all the axes must stay within the limits. If the rotary axis is of the "module" type, the cycle will only allow part of the travel; the part of the itinerary to be calibrated; the not calibrated part is not compensated.

Check the position of the axes with respect to the itinerary limits.

To check that the axes remain within their limits, move the probe over the ball, at short distance of around 10 mm (0.4 inches). In that position, activate RTCP and, in jog mode, check that the linear axes have enough room to move within the programmed positions. Bear in mind that in each position of the rotary axis, the linear axes must move the programmed Ds distance in order to be able to calculate the center of the ball. To perform these movements the parameters TDATA of the kinematics must be defined with approximate values.

Parameters of the cycle.

Feedrate.

Fa	Positioning feedrate
Fs	Rapid probing feedrate. Then, the CNC will repeat the probing movement at feedrate $\cdot F$.
F	Accurate probing feedrate.

25.

KINEMATICS CALIBRATION.
Rotational kinematics calibration.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

Spindle orientation.

It is up to the user to make sure that the probe can rotate without danger. If the probe has a cable, check that the cable does not hinder the rotation of the probe.

Ssp The speed and turning direction for spindle orientation; the sign means the turning direction. The CNC only offers this parameter when the spindle may be oriented (M19), so the probe always touches the ball with the same surface.

Safety distance and withdrawal.

Ds Safety distance between the probe and the calibrated ball. This parameter only admits positive values greater than 0 (zero).

Dr Distance the probe withdrawals after the first probing movement. Once it withdraws this distance, it makes a second probing movement. This parameter only admits positive values greater than 0 (zero).

Ball size.

Rs Calibrated ball radius. See the OEM characteristics to obtain this value.

Points to probe for calculating the center of the ball.

Icon.	Meaning.
	The points to be measured are on the axes.
	The points to be measured are 45° with respect to the axes.

Adjusting the probe.

Icon.	Meaning.
	The cycle verifies the probe adjustment and will display a warning if it detects a non-adjusted PROBEDELAY value.
	The cycles does not verify the probe adjustment.

New calibration or check active calibration.

Icon.	Meaning.
	Perform a new calibration.
	Check the calibration, without overwriting the active values.

25.

KINEMATICS CALIBRATION.
Rotational kinematics calibration.



25.2.2 Page 2. Display of the progress of the rotational calibration.

The CNC shows the calculated offsets for each position of the rotary axes. At the end of the rotational calibration cycle, the table of calculated deviations can be activated. This will change the COMPID value of the corresponding kinematics table. At the end of the rotational calibration cycle, the table of calculated deviations can be activated by pressing the Validate softkey.

KIN ID 2 A Comp ID 1		M41	
Results table		G0 G7 G17 G92 KIN C	
	X	Y	Z
C0 B-90	-0.03240	-0.34790	0.23590
C50 B-90	-0.40450	0.09840	0.34180
C100 B-90	-0.86170	0.25920	0.55320
C150 B-90	-1.19530	0.07910	0.73750
C200 B-90	-1.24400 B	-0.39280	0.71150
C250 B-90	-0.77470	-0.89560	0.51320
C300 B-90	-0.19350	-0.90650	0.29640
C0 B-45	0.02830	-0.15160	0.02930
C50 B-45	-0.33290	0.36940	0.12430
C100 B-45	-0.90090	0.53720	0.26920
Command Actual To Go Machine Base			
X	-177.386	-177.386	0.000 1410.862
Y	308.286	308.286	0.000 -973.480
Z	383.797	383.797	0.000 -810.828
C	0.000	0.000	0.000 0.000
B	0.000	-0.000	0.000 -0.000 F
V	308.286	308.286	0.000 -973.480
Z1	-66.803	-66.803	0.000 -66.803

A Number of kinematics and COMPID.

B Measured offset in X, Y, Z for each position of the rotary axes.

C Active functions.

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

"F real" Real (actual) feedrate of the axes.

"F prog" Programmed feedrate.

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

"Dyn" Percentage of dynamics being applied.

E Information related to the spindle speed "S".

F Programmed coordinate and current position of the axes.

25.

KINEMATICS CALIBRATION.
Rotational kinematics calibration.

FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

25.2.3 Page 3. Display and activate the different offset tables calculated.

The offset table shows the list of calculated tables. By selecting one of them, the results table shows the calculated deviations. Once a table is selected, the "Activate" softkey allows you to activate it, and if this table is already active, the "Deactivate" softkey will be displayed to deactivate it. The CNC activates the table for the kinematics with which it was calculated (KIN in the Volumetric Table), not for the active kinematics.

KIN ID2

A

omp ID1

Volumetric table

KIN	COMP ID
2	1
2	3

Results table

	X	Y	Z
C0 B-90	-0.03240	-0.34790	0.23590
C50 B-90	-0.40450	0.09840	0.34180
C100 B-90	-0.86170	0.25920	0.55320
C150 B-90	-1.19530	0.07910	0.73750
C200 B-90	-1.24400	-0.39280	0.71150
C250 B-90	-0.77470	-0.89560	0.51320
C300 B-90	-0.19350	-0.90650	0.29640
C0 B-45	0.02830	-0.15160	0.02930
C50 B-45	-0.33290	0.36940	0.12430
C100 B-45	-0.90090	0.53720	0.26920
C150 B-45	-1.32820	0.26110	0.39490
C200 B-45	-1.36980	-0.30570	0.35800
C250 B-45	-0.88030	-0.81180	0.20250
C300 B-45	-0.21870	-0.79380	0.05360
C0 B0	0	0	0
C50 B0	-0.26130	0.48420	-0.00750
C100 B0	-0.80510	0.61180	-0.02020
C150 B0	-1.23830	0.27510	-0.02710
C200 B0	-1.26670	-0.27250	-0.02560
C250 B0	-0.85720	-0.64980	-0.01720

B

C

Don't check probe

- A Number of kinematics and COMPID.
- B Offset table, where all calculated tables are shown.
- C Table of results, showing the calculated deviations in X, Y, Z for each positioning of the rotary axes.

25.

KINEMATICS CALIBRATION.
Rotational kinematics calibration.

User notes:



FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508

User notes:



CNCelite
8058 8060
8065 8070

REF: 2508

User notes:



FAGOR
AUTOMATION

CNCelite
8058 8060
8065 8070

REF: 2508



Fagor Automation S. Coop.

Bº San Andrés, 19 - Apdo. 144
E-20500 Arrasate-Mondragón, Spain

 +34 943 039 800

contact@fagorautomation.es
www.fagorautomation.com

