CNCelite

8058/8060
8065

Operating manual (MC).

Ref. 2109

FAGOR

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.

FAGOR AUTOMATION

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

The information described in this manual may be 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. 7
About the product. 9
EC Declaration of Conformity, warranty conditions and quality certificates. 15
Safety conditions. 17
Returning conditions. 21
CNC maintenance. 23
New features. 25

CHAPTER 1 GENERAL CONCEPTS

1.1 Accessing the conversational mode 35

CHAPTER 2 OPERATING IN JOG MODE

2.1 Introduction 38
2.1.1 Standard screen of the conversational mode. 38
2.1.2 Auxiliary screen of the conversational mode. 39
2.1.3 Cycle editing. 41
2.1.4 Cycle simulation 42
2.1.5 Cycle execution. 43
2.2 Operations with the axes. 44
2.2.1 Home search. 44
2.2.2 Jog 45
2.2.3 Jogging the axes with handwheels 47
2.2.4 Moving an axis to a particular position (coordinate). 49
2.2.5 Coordinate preset. 49
2.3 Spindle control 50
2.4 Tool selection and tool change 51
2.5 Setting the feedrate and spindle speed. 51
2.6 Setting and activating the zero offsets and the fixture offsets. 52
2.7 Tool calibration. 53
2.7.1 Manual calibration. Calibration without a probe 54
2.7.2 Semi-automatic calibration. Calibration with a probe 57
2.7.3 Automatic calibration with a probe and a canned cycle 60

CHAPTER 3 CYCLE EDITOR

3.1 Configuring the cycle editor. 65
3.2 Configuring el cycle editor (rotary axis cycles). 66
3.3 Teach-in mode. 67
3.4 Selecting data, profiles and icons 68
3.5 Associate a multiple machining operation with a canned cycle 69
3.6 Simulate a canned cycle. 70

CHAPTER 4 WORK PLANES AND SHIFTING OF THE MACHINING OPERATIONS.

CHAPTER 5 CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.

5.1 Center punching. 76
5.1.1 Basic operation. 78
5.2 Drilling. 79
5.2.1 Basic operation. 80
5.3 Deep hole drilling. 81
5.3.1 Basic operation. 83
5.4 Bore milling. 84
5.4.1 Basic operation. 86
5.5 Tapping. 87
5.5.1 Basic operation. 89
5.6 Thread milling. 90
5.6.1 Basic operation. 92
5.7 Reaming. 93
5.7.1 Basic operation. 94
5.8 Boring. 95
5.8.1 Basic operation. 96

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

5.9 Boring with spindle orientation. 97
5.9.1 Basic operation. 99

CHAPTER 6 CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.

6.1 Simple rectangular pocket. 102
6.1.1 Basic operation. 105
6.2 Rectangular pocket with rounding. 106
6.2.1 Basic operation. 111
6.3 Circular pocket. 112
6.3.1 Basic operation. 116
6.4 Pre-emptied circular pocket. 117
6.4.1 Basic operation. 121
6.5 Rectangular boss. 123
6.5.1 Basic operation. 127
6.6 Circular boss. 128
6.6.1 Basic operation. 131

CHAPTER 7 CANNED CYCLES OF THE EDITOR. 2D/3D PROFILE POCKETS.

7.1 2D profile pocket. 134
7.1.1 Executable pocket file 139
7.1.2 Basic operation. 140
7.1.3 Examples of how to define 2D profiles 142
7.2 3D profile pocket with islands. 145
7.2.1 Executable pocket file 151
7.2.2 Basic operation. 152
7.2.3 Examples of how to define 3D profiles 153

CHAPTER 8 CANNED CYCLES OF THE EDITOR. ROUGHING.

8.1 Surface milling. 158
8.1.1 Basic operation. 161
8.2 Point-to-point profile milling. 162
8.2.1 Basic operation. 165
8.3 Free profile milling. 166
8.3.1 Basic operation. 169
8.4 Slot milling. 170
8.4.1 Basic operation. 174

CHAPTER 9 CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.

9.1 Multiple machining in straight line. 178
9.1.1 Programming example. 179
9.2 Multiple machining in arc pattern. 180
9.2.1 Programming example. 181
9.3 Multiple machining in rectangular pattern. 182
9.3.1 Programming example. 183
9.4 Multiple machining in grid pattern. 184
9.4.1 Programming example. 185
9.5 Random multiple machining. 186
9.5.1 Programming example. 186

CHAPTER 10 CANNED CYCLES OF THE EDITOR FOR ROTARY AXES. LONGITUDINAL MACHINING.

10.1 Multiple center-punching. 188
10.1.1 Basic operation. 191
10.2 Multiple drilling. 192
10.2.1 Basic operation. 196
10.3 Multiple tapping. 197
10.3.1 Basic operation. 200
10.4 Multiple reaming. 201
10.4.1 Basic operation. 204
10.5 Multiple boring. 205
10.5.1 Basic operation. 209
10.6 Multiple bore milling. 210
10.6.1 Basic operation. 213
10.7 Multiple thread milling. 214
10.7.1 Basic operation. 218
10.8 Multiple slot milling. 219
10.8.1 Basic operation. 223

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

CHAPTER 11 CANNED CYCLES OF THE EDITOR FOR ROTARY AXES. PROFILES.

11.1 ZC/YZ plane profiling cycle. ... 226
11.1.1 Basic operation. ... 230
11.2 Rectangular pocket cycle in the ZC/YZ plane. ... 231
11.2.1 Basic operation. ... 236
11.3 Circular pocket cycle in the ZC/YZ plane. ... 237
11.3.1 Basic operation. ... 241
11.4 Circular pre-emptied pocket cycle in the ZC/YZ plane. ... 242
11.4.1 Basic operation. ... 246
11.5 2D profile pocket cycle in the ZC/YZ plane. ... 247
11.5.1 Basic operation. ... 252
11.6 XC/XY plane profiling cycle. ... 253
11.6.1 Basic operation. ... 257
11.7 Rectangular pocket cycle in the XC/XY plane. ... 258
11.7.1 Basic operation. ... 263
11.8 Circular pocket cycle in the XC/XY plane. ... 264
11.8.1 Basic operation. ... 268
11.9 Circular pre-emptied pocket in the XC/XY plane. ... 269
11.9.1 Basic operation. ... 273
11.10 2D profile pocket cycle in the XC/XY plane. ... 274
11.10.1 Basic operation. ... 279

CHAPTER 12 SAVING PROGRAMS

12.1 List of saved programs ... 281
12.2 Edit a new part-program ... 282
12.3 Delete a new part program ... 282
12.4 Inserting comments in any part-program. ... 282
12.5 Inserting a cycle in the current part-program. ... 282

CHAPTER 13 EXECUTION AND SIMULATION

13.1 Execute a part-program ... 284
13.1.1 Execute a portion of a part-program ... 284
13.1.2 Graphics screen in execution. ... 285
13.2 Simulating a part-program ... 286
13.2.1 Simulate a portion of a part-program ... 286
13.2.2 Graphics screen in simulation. ... 287
13.2.3 Simulation options. ... 288
13.3 Simulating or executing an operation that has been saved ... 289
13.3.1 Simulating a cycle ... 289
13.3.2 Cycle execution ... 290

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

ABOUT THIS MANUAL.

Title.

Operating manual (MC).

Models.

CNCelite
8058 8060
8065

Type of documentation.

End user manual. This manual describes how to operate in IIP (Interactive Icon-based Pages) or conversational mode for the M model. This mode works with the CNC in a graphical and guided way based on predefined cycles.

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_mc_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.

September, 2021

Manual reference

Ref. 2109

Associated version.

v2.00.01

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: info@fagorautomation.es

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

FAGOR AUTOMATION

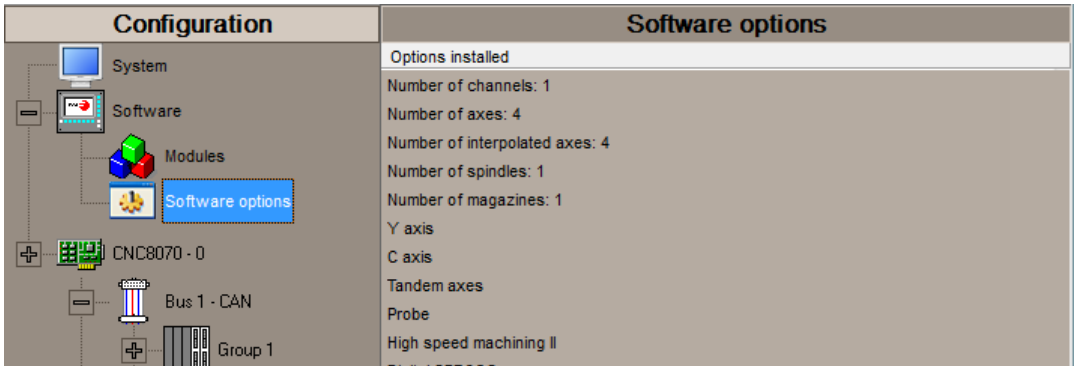
CNCelite
8058 8060
8065

REF. 2109

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.
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 an 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).

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

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.

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

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.

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).

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

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.

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

CNC MAINTENANCE.

CLEANING

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

To clean the operator panel and the monitor, a smooth cloth should be used which has been dipped into de-ionized water and /or non abrasive dish-washer soap (liquid, never powder) or 75º alcohol. 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, venzole, esters and ether which can damage the plastics used to make the unit's front panel.

PRECAUTIONS BEFORE CLEANING THE UNIT

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

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

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

NEW FEATURES.

Manual reference:	Ref. 2109
Date of publication:	September, 2021
Associated software:	v2.00.01

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

List of features.	Manual
OPC-UA Protocol	
New laser model. • CUT_VIEW mode for previewing the part to be cut as well as the piercing points. • Technological tables (piercing, cutting and generic) to define the necessary parameters for cutting (power, speed, etc.). • Material files relating the material characteristics (type, thickness, etc.) to the technology tables. • Manual and automatic nesting. • Radius compensation. • Subroutines to perform the initial and final laser operations. • Change from the part-programme of the status of PLC marks, logic inputs and outputs. • Activation of a material from the part-programme. • Activation of a generic technology table from the part-programme. • Piercing and cutting with associated subroutines. • The CNC makes it possible to run a program with microjoints that is generated without them. • Control of laser power, PWM duty and/or PWM frequency depending on the feed rate. • Gap control. • Leapfrog. • CBDS (Constant Beam Delivery System).	[LSR]
Integration of Lantek Expert Inside software for laser models. From figures drawn in dxf, dwg and parametric files, this software allows these to be distributed on a sheet to optimize their use (nesting), define the input/output strategy (lead-in/lead-out), etc., to ultimately create a part-program in Fagor language. Fagor provides two types of nesting, which are available via software options; manual nesting (the user places the pieces on the sheet as required) and automatic nesting (the software places the pieces on the sheet based on their configuration parameters).	[LSR]
New software option "SOFT 60 Y AXIS". Option to enable lathe Y-axis kinematics and associated canned cycles.	
New software option "SOFT FVC STANDARD". Option to enable standard volumetric compensation. 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.	
New software option "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.	
New software option "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.	
New software option "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.	
New software option "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.	

[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT] ..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

List of features.	Manual
New software option "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.	
New software option "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.	
New software option "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. During manual nesting, the operator distributes the figures on the sheet.	
New software option "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. During automatic nesting, the application distributes the figures on the sheet material and optimizes the spaces.	
This version must be installed in administrator mode, so that the process properly registers the new FCombo.ocx control.	
New languages for the HMI. • Polish and Turkish. - Machine parameters: LANGUAGE	[INST]
HMI. • Define the initial position of the interface on the screen. - Machine parameter: POSX, POSY • The interface can be moved by clicking with the mouse on the icon showing the program status of the active channel. Double click on this icon and the interface is positioned in the upper left corner (POSX=0 POSY=0). • Define the size of the interface. - Machine parameter: RESOLUTION, WIDTH, HEIGHT • Customize the number of softkeys and their size; between 7 and 12 horizontal softkeys and between 5 and 12 vertical softkeys. - Machine parameter: HMENUNSOFTKEY, HMENUNSOFTKEY, SOFTKEYMETRICS	[INST]
Basic volumetric compensation. • Compensation for 15625 points (maximum 1000 points per axis). Quick to calibrate (time) but less accurate than the other two (corrects fewer error components), but sufficient for the desired tolerances. - Machine parameter: VMOVAXIS1, VMOVAXIS2, VMOVAXIS3 NPOINTS1, NPOINTS2, NPOINTS3 INIPOSAX1, INIPOSAX2, INIPOSAX3 INCREAX1, INCREAX2, INCREAX3	[INST]
Volumetric compensation. • Definition of rotary axes as axes of motion and compensation. - Machine parameter: VMOVAXIS, VCOMPAXIS • Definition of 1 or 2-axis offsets. - Machine parameter: VMOVAXIS, VCOMPAXIS	[INST]
Number the digital inputs of the CANopen bus based on logic blocks. - Machine parameter: NDIMOD, DIMODADDR, NODE, BLOCK, ADDRESS, NDI	[INST]
Number the digital outputs of the CANopen bus based on logic blocks. - Machine parameter: NDOMOD, DOMODADDR, NODE, BLOCK, ADDRESS, NDO	[INST]
Number of special analog inputs (non-voltage). - Machine parameters: NSAI, ADDRESS, FACTOR	[INST]
PLC program. • Increases the limit of PDEF symbols. The PDEF directing instruction can define up to 200 symbols and can be used in the PLC program, a PLC subroutine in C language, part program (using variables) or external application. • Subroutines in C language support DEF/PDEF symbols. Every time the PLC compiles the program, it generates the file plc_pdef.h with the #define of all the symbols defined in the PLC program. The file is saved in the folder ..\MTB\PLC\Project.	[INST]
Asynchronous magazine management. • New TMOPERATION=12 operation for asynchronous magazine. Leave the spindle tool in the magazine and take a tool from the same magazine (as TMOPERATION=5). It is an optimised operation, valid when both tools are special and can collide in the arm or when it is not desirable for them to coincide in the changer arm. - PLC registration: TMPOPERATION	[INST]

[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT]..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
Disable the local feedback input alarms. - PLC mark: LCOUNTALARMOFF1	[INST]
Observe the G00 feedrate during synchronized switching. - Machine parameter: SWTG0FEED	[INST]
Configure the axes of a mill to execute the rotary axis cycles. - Machine parameter: LONGAXIS, FACEAXIS, CAXIS	[INST]
Calculate the maximum scanning feed rate. If PROBEFEED is set to 0, the CNC calculates and uses the maximum possible value of this parameter. - Machine parameter: PROBEFEED	[INST]
Defining kinematics. • The kinematics table only indicates the parameters that affect a kinematics type. • Point of origin of TDATA parameters; "Tool base", "Head base" and "Auto tool base". With the new "Auto Tool Base" value, on activating the CNC kinematics, calculate the value of TDATA13/TDATA14/TDATA15 (non-vectorial kinematics) or TDATA1/TDATA2/TDATA3 (vectorial kinematics). - Machine parameter: HEADREF HEADREF=Head base TDATA13(+) TDATA15(+) HEADREF=Tool base TDATA13(+) TDATA15(+)	[INST]
If a movement smaller than the INPOSW parameter is programmed on a dead axis, the CNC does not enable or move the axis. - Machine parameters: INPOSW	[INST]
Changes to read-only MTBPAR parameters using variables in Setup mode. - Machine parameters: MTBPAR - Variables: (V.)MTB.P, (V.)MTB.PF	[INST]
HD graphics. • Configure the HD graphics for the rotary axis cycles (·M· model). • If the HD graphics work with the tool tip coordinates, then they respond to the parameters FACEAXIS and LONGAXIS. If the HD graphics work with the tool base coordinates, then they respond to the parameters FACEAXIS and LONGAXIS. - Machine parameters: FACEAXIS, LONGAXIS	[INST]
Non-modal G02 and G03 functions. - Machine parameter: G2G3MODAL - Function: G2, G3	[INST] [PRG]
Non-modal function G00. - Machine parameter: G0MODAL, IMOVE - Function: G0.	[INST] [PRG]
New axis names; E, E1..E9. • Values of the variable (V.)G.AXISNAMEn for the E axis; E=110, E1=111, etc. • Values of the variable (V.)G.GAXISNAMEn for the E axis; E=110, E1=111, etc.	[INST] [PRG] [VAR]
If the execution of an interruption subroutine (marks INT1/INT4 of the PLC) ends with M30, then the program execution also ends. If the execution of an interruption subroutine ends with #RET, then the program execution continues. - PLC marks: INT1 / INT4	[INST] [PRG]
In programs that include #SET AX (change configuration of axes) and #DEFGRAF (load graphic configuration), the CNC will allow the program to be simulated after executing a #SET AX in MDI different to that of the program.	
New radius compensation algorithm, optimized to solve stepped profiles.	
Subroutine associated with the reset. - Subroutine: PROGRAM_RESET	[INST] [PRG]
Being the CNC in SETUP mode, the OEM subroutines with an fst extension will be loaded into RAM memory during the preparation of the blocks.	[INST] [PRG]
OEM subroutines for kinematics calibration can execute motions in two axes at the same time.	

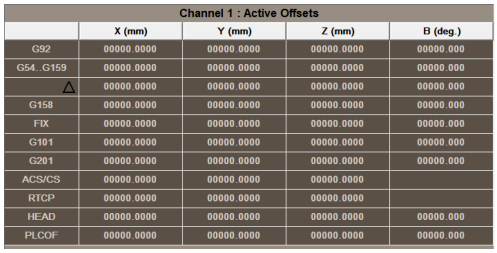
[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT] ..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
New search criteria for help files associated with the subroutines. - File: pcall.txt - Archivo: subroutine_name.txt subroutine_name.bmp	[INST] [PRG]
The subroutines associated with functions G500-599 may also have help files which are displayed during editing. - File: G500.txt, G501.txt, etc. G500.bmp, G501.bmp, etc.	[INST] [PRG]
The subroutines associated with functions G8000-8999 may also have help files which are displayed during editing. - File: G8000.txt, G8001.txt, etc. G8000.bmp, G8001.bmp, etc.	[INST] [PRG]
User subroutines (G500-G599) do not alter the modal/non-modal status of other possible active subroutines (#MCALL). - Function: G500, G501, etc. - Statement: #MCALL	[PRG]
User subroutines (G8000-G8099) do not alter the modal/non-modal status of other possible active subroutines (#MCALL). - Function: G8000, G8001, etc. - Statement: #MCALL	[PRG]
Calls to subroutines with parameter initialization allow the programming of 32 additional parameters (P26 to P57), which can also be defined as "D0= " to "D31=", so that "D0=" is equal to P26, "D1=" to P27 and so forth. - Function: G500, G501, etc. G180, G181, etc. G380, G381, etc. - Statement: #PCALL, #MCALL	[PRG]
Work zones. The command "E" must always be placed after the function G122; otherwise the CNC will interpret it as the name of an axis. - Function: G122.	[PRG]
Scaling factor per axis. - Function: G72.	[PRG]
Defining the path and the name of a program in a macro. - Statement: #DEF.	[PRG]
CNC messages. - Statement: #MSG • Format identifiers (%D, %i, %u, etc). • Define messages using a number. • The OEM or the user can define a text group for the cncMsg.txt file.	[PRG]
Errors and warnings. - Statement: #ERROR, #WARNING, #WARNINGSTOP • Format identifiers (%D, %i, %u, etc). • Change location of cncError.txt file. • Programming a warning with a value of 0 clears all the warnings that are being displayed.	[PRG]
Change the HMI variables from the workpiece programme. - Statement: #MSGVAR. - Variable: (V.)G.CNCMSG[num]	[PRG]
Table+spindle kinematics (type 52). • The CNC does not take into account the axes of the table that are not programmed. - Statement: #SELECT ORI. • When positioning the tool, take into account the position of the table. - Statement: #SELECT ORI. ALL command. • When activating the RTCP, define the part of the kinematics to be used (table or head) and the type of RTCP (static or dynamic). - Statement: #RTCP. HEAD and TABLE commands. • Activation of the RTCP according to the values defined in the machine parameters (TDATA). - Statement: #RTCP. CLEAR command. • Rotate the part coordinate system when rotating the table. - Statement: #RTCP. COROT command. • Working mode with RTCP in spindle+table kinematics. - Variable: (V.)G.RTCPMODE	[PRG]
New mode to define an inclined plane (MODE 7), rotating first on the third axis of the part, then on the second axis of the new plane and finally on the first axis of the new plane. - Statement: #CS / #ACS. MODE 7.	[PRG]
Management of discontinuities in the orientation of rotary axes. Orient the tool in the programmed direction of the main or secondary rotary axis. - Statement: #DEFROT. DIRF, DIRS commands.	[PRG]

[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT]..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
Opening and writing files. • Does not prompt an error for operations using #OPEN, #WRITE y #CLOSE. - Statement: #OPEN. MUTED command. • Verify if the file exists. - Statement: #OPEN. R(READ) command.	[PRG]
If the CNC reaches the end of a program without an M02/M30, then it ends the execution without prompting a warning.	[PRG]
The #CYL instruction admits programming of the CZ plane (#CYL[C,Z]).	
Activate a kinematic displaying the tool tip dimensions. - Statement: #KIN ID. TIP command. - Variable: (V.)G.KINIDMODE	[PRG] [VAR]
The user variables have reading permission via the interface. - Variable: (V.)P.var_name (V.)P.var_name[nb] (V.)P.var_name (V.)P.var_name[nb]	[PRG] [VAR]
Array user variables. - Variable: (V.)P.var_name[nb] (V.)P.var_name[nb]	[PRG] [VAR]
Correcting the values indicated by the hardware. - Variable: (V.)G.HARDTYPE	[VAR]
Status of the power-up and power-down process. - Variable: (V.)E.CNCLOGST	[VAR]
Identifying the installation folder. - Variable: (V.)G.FOLDERID	[VAR]
Identification of the language selected in the CNC. - Variable: (V.)G.LANGUAGEID	[VAR]
Monitoring the precision of rigid tapping. - Variable: (V.)G.RIGIER	[VAR]
Error code for #OPEN, #WRITE or #CLOSE statements. - Variable: (V.)G.FILEERRNO	[VAR]
From the part-program (#MSGVAR) change the HMI variables. - Variable: (V.)G.CNCMSG[num]	[VAR]
Axis logic number, based on its order in the channel. - Variable: (V.)G.NLOGAXIS1 (V.)G.NLOGAXIS2 .. (V.)G.NLOGAXISn	[VAR]
Spindle logic number, based on its order in the channel. - Variable: (V.)G.NLOGSPDL1 (V.)G.NLOGSPDL2 .. (V.)G.NLOGSPDLn	[VAR]
Modify the proportional gain (Kp) of the tandem from the PLC. - Variable: (V.)MPG.TPROGAIN[nb]	[VAR]
Number of teeth on the tool. - Variable: (V.)G.NCUTTERS (V.)TM.NCUTTERS[offset] (V.)TM.NCUTTERST[tool][offset]	[VAR]
Percentage of feedrate in G00. - Variable: (V.)PLC.G00FRO	[VAR]
Operating Terms. Temporary license for use at the CNC. - Variable: G.CNCDISSTAT	[OPT] [VAR]
FMC Function (Fagor Machining Calculator) • A database of materials to be machined and of machining operations (milling and turning) and an interface to choose suitable cutting conditions.	[OPT]
FCAS (Fagor Collision Avoidance System). • 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. - Variable: (V.)G.COLLISIONPERF	[OPT] [VAR]
FCC Function (Fagor Feed Control). • 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.	[OPT]

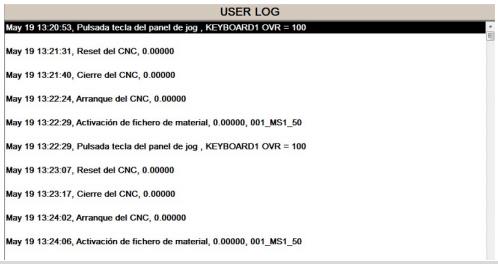
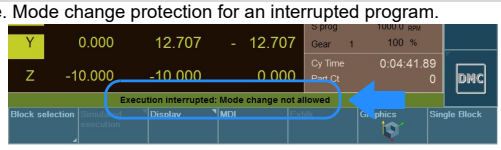
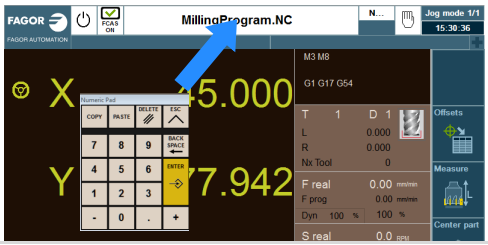
[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT] ..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
DMC Function (Dynamic Machining Control). • DMC adapts the feedrate during machining to maintain the cutting power as close as possible to ideal machining conditions. - Machine parameter: MINDMCOVR MAXDMCOVR DMCPEAKSIZE DMCPWRFACTOR DMCSPDL - PLC mark: DMCPWRPEAK DMCPWRHOLD - Statement: #DMC - Variable: (V.)MPG.MINDMCOVR (V.)MPG.MAXDMCOVR (V.)PLC.DMCON (V.)G.FRO (V.)G.PRGFRO (V.)G.PLC.FRO (V.)G.CNCFRO (V.)G.DMCPWRSP (V.)G.DMCOVRMIN (V.)G.DMCOVRMAX (V.)G.DMCFZMIN (V.)G.DMCFZMAX (V.)G.DMCON (V.)G.LEARNEDPWRSP (V.)G.DMCACTPWR (V.)G.DMCOVR (V.)G.DMCFZ (V.)G.DMCNOLOADPWR (V.)G.DMCSAVEDTIME	[OPT] [PRG] [VAR]
Identification and registration of the devices connected to the CNC, in setup and user modes.	[OPT]
Encrypting/decrypting a PLC project.	[OPT]
Kinematics calibration. • 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 cycle allows the calibration of all types of kinematics (parameter TYPE). • New icon to verify the probe adjustment. • GMOVE macro in the subrouting KinCal_Begin.nc. • The axes may be Hirth axes. • 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. • In user mode, the data calculated by the recalibration of the kinematics are saved in the "Offsets" field of the machine parameter table; the "Value" field defined by the OEM cannot be modified. • When calibrating kinematics in Setup mode, if the calculated value is equal to the sum of TDATA+Offset saved, the CNC will not offer to update TDATA and set the offset to 0. • The cycle creates the KinCalReport.mp file with the results of the calibration.	[OPT]
Tool table. • New Zn data. Number of teeth on the tool.	[OPT]
User tables. • Active offsets table. The user tables offer a new table for showing the active values in different functions; G92, G159, G201, etc. 	[OPT]
Edisimu mode. The program translator written in the 8055 language can define the destination folder for the translated program (by default ..\Users\Prg\PRG_8055_TO_8070).	[OPT]
Translator 8055. • Translation of the G72+G16+G15 sequence. • Translation of conversational cycles.	[OPT]

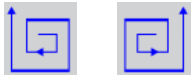
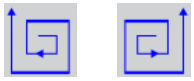
Channel 1 : Active Offsets

	X (mm)	Y (mm)	Z (mm)	B (deg.)
G92	00000.0000	00000.0000	00000.0000	00000.000
G54..G159	00000.0000	00000.0000	00000.0000	00000.000
Δ	00000.0000	00000.0000	00000.0000	00000.000
G158	00000.0000	00000.0000	00000.0000	00000.000
FIX	00000.0000	00000.0000	00000.0000	00000.000
G101	00000.0000	00000.0000	00000.0000	00000.000
G201	00000.0000	00000.0000	00000.0000	00000.000
ACS/CS	00000.0000	00000.0000	00000.0000	
RTCP	00000.0000	00000.0000	00000.0000	
HEAD	00000.0000	00000.0000	00000.0000	00000.000
PLCOF	00000.0000	00000.0000	00000.0000	00000.000

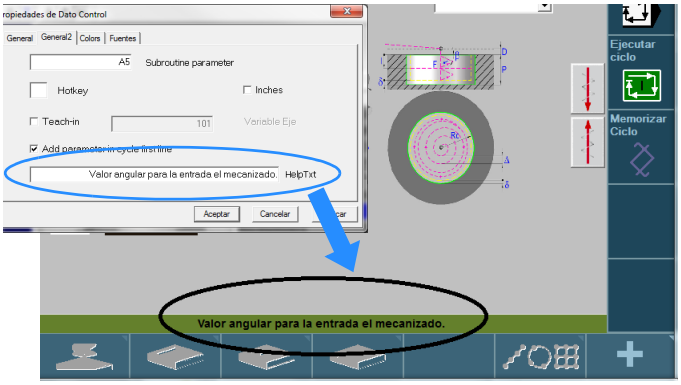
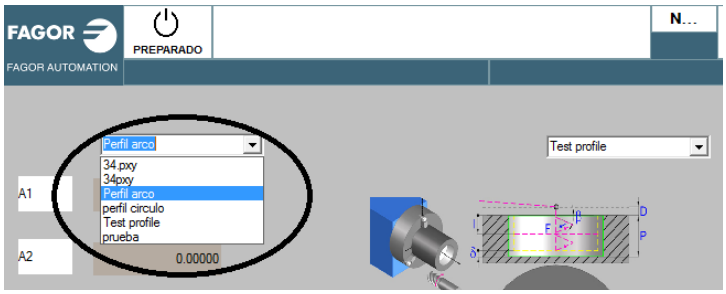
[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT]..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
"Intel Graphics Control Panel" application. Configuring the brightness and contrast of the monitors.	[OPT]
User log, which chronologically stores events that have occurred on the CNC, which may be important to the user, e.g. errors, keystrokes on the operator panel, etc. - Variable: (V.)E.USERLOG 	[OPT]
Utilities mode. The CNC will ask for confirmation before backing up the data and before restoring data from a back up.	[OPT]
Utilities mode. Encryption of scripts (scp files) and subroutines associated with user cycles.	[OPT]
Automatic mode. From this and later versions, after interrupting the execution of the program and canceling it using the "Cancel-Continue" softkey, the CNC allows the machine to be moved using the jog keys.	[OPT]
Automatic mode. Mode change protection for an interrupted program. 	[OPT]
Improved resolution of the help drawings displayed on various CNC screens; for example, machining cycles, geometric aids, etc.	[OPT]
Edisimu mode. The selected simulation options are maintained after the CNC has been restarted.	[OPT]
STD graphics. • In conversational mode, the CNC remembers the type of window, "Simple" (only shows the graphic) or "Full" (shows dimensions, feeds, etc.). • The selected window configuration is maintained after the CNC has been restarted. • Selecting the simulation speed to a minimum will pause the simulation. • The selected simulation speed is maintained after the CNC has been restarted.	[OPT]
HD graphics. • The default configuration shows the axes and tool paths. • Lathe HD graphics take into account offsets in the X axis of part zero.	[OPT]
Error 3753. Absolute encoder. Monitor the coordinate difference during startup (parameter MAXDIFREF).	[OPT]
The number of softkeys in the menus can be configured. 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.	[OPT]
Mini-numeric keypad. 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. 	[OPT]
The profile editor shows the CZ plane oriented correctly.	
Cycle editor (model M). • New configurator for the cycles associated with the rotary axes. • Softkey to hide Fagor cycles (machining and probe cycles). The softkey is also available in conversational mode. 	[CYC-M] [OPT-MC]

[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT] ..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
Editor cycles (M model). New canned cycles associated with the rotary axes. • Multiple center-punching. • Multiple reaming. • Multiple boring. • Multiple bore milling. • Multiple thread milling. • Multiple drilling. • Multiple tapping. • Multiple slot milling. • Profile on the cylindrical surface of the part. • Rectangular pocket on the cylindrical surface of the part. • Circular pocket on the cylindrical surface of the part. • Pre-emptied circular pocket on the cylindrical surface of the part. • 2D pocket profile on the cylindrical surface of the part. • Profile on the front surface of the part. • Rectangular pocket on the front surface of the part. • Circular pocket on the front surface of the part. • Pre-emptied circular pocket on the front surface of the part. • 2D pocket profile on the front surface of the part.	[CYC-M] [OPT-MC]
Editor cycles (M model). 2D profile pocket cycle. • Define the machining strategy; from the inside out or vice versa. • Defining the direction of the machining; clockwise or counterclockwise. 	[CYC-M] [OPT-MC]
Cycle editor (T model). • Softkey to hide the Fagor cycles from the editor (machining and probe cycles). The softkey is also available in conversational mode. 	[CYC-T] [OPT-TC]
Editor cycles (T model). 2D profile pocket in the ZC/YZ plane. • Define the machining strategy; from the inside out or vice versa. • Defining the direction of the machining; clockwise or counterclockwise. 	[CYC-T] [OPT-TC]
Editor cycles (T model). Profile turning cycle. • New operation to remove the residual material during roughing operations.	[CYC-T] [OPT-TC]
Editor cycles (T model). Grooving cycles (except cutting cycles). • Penetration with chip removal in all passes. 	[CYC-T] [OPT-TC]
Cycle editor (T model). Profiles • The canned cycles may be defined by the CZ plane. 	[CYC-T] [OPT-TC]

[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT]..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

List of features.	Manual
Simulation options in conversational mode. • Radius compensation. • Conditional stop during simulation. • Software and work zone limits. • Block skip. • Cancel channel synchronization. • Synchronizing spindles. • Assume the active origins for execution. • Assume the active zones in execution.	[OPT-MC] [OPT-TC]
ProGTL3 language. Generic user subroutines (G8000-G8999).	[PROGTL3]
ProGTL3 language. Programming cycles (macros). • Milling of a surface defined by a plane profile and one or more section profiles. Machine only the bottom of the pocket. - G8736. Parameter D0 (D0=3). • Invert the machining direction of the profile - G8754. Start of profile. - G8753. End of the profile. • Circular polygon pocket defined by a profile. - G8777. Define the cycle and its parameters. - G8778. Activate the cycle. - G8701. Profile pocket. Start of the profile and internal island profiles. - G8702. Profile pocket. Profile section of the pocket and islands (optional).	[PROGTL3]
Customisation of the user cycle header. The “Data” control has a new property called “Add parameter in cycle first line”. If this property is checked, the cycle adds the data to the cycle header as “Variable=Value”. The variable will take on the name of the property “Subroutine parameter”. +#CYCLE BEGIN (1-Simple screw) (V.C.TOOL=1 V.C.SPEED=100) (CW_V_1_0)	[FGUIM]
Association of a help text to user cycle data. The “Data” control has the new “HelpTxt” property that defines the help text appearing on the CNC when selecting the data.  	[FGUIM]
New FCombo control to manage profiles in user cycles.  	[FGUIM]
Fguim allows changes to be made to the interface while the CNC is running. Any changes made by Fguim are updated in the CNC during the next startup. For the changes to be assumed at the CNC, they must be previously saved from the Fguim.	[FGUIM]

[CYC-M]..... Machining canned cycles (·M· model).
[CYC-T]..... Machining canned cycles (·T· model).
[EXA-T]..... Examples manual (·T· model).
[FGUIM]..... Customizing guide (FGUIM & API).
[INST]..... Installation manual.
[LSR]..... Operating and programming manual (·L· model).
[OPT] ..... Operating manual.
[OPT-MC]..... Operating manual (MC option).
[OPT-TC]..... Operating manual (TC option).
[PRG]..... Programming manual.
[PROGTL3]..... ProGTL3 language.
[VAR]..... CNC variables.

FAGOR AUTOMATION

CNCelite
8058 8060
8065

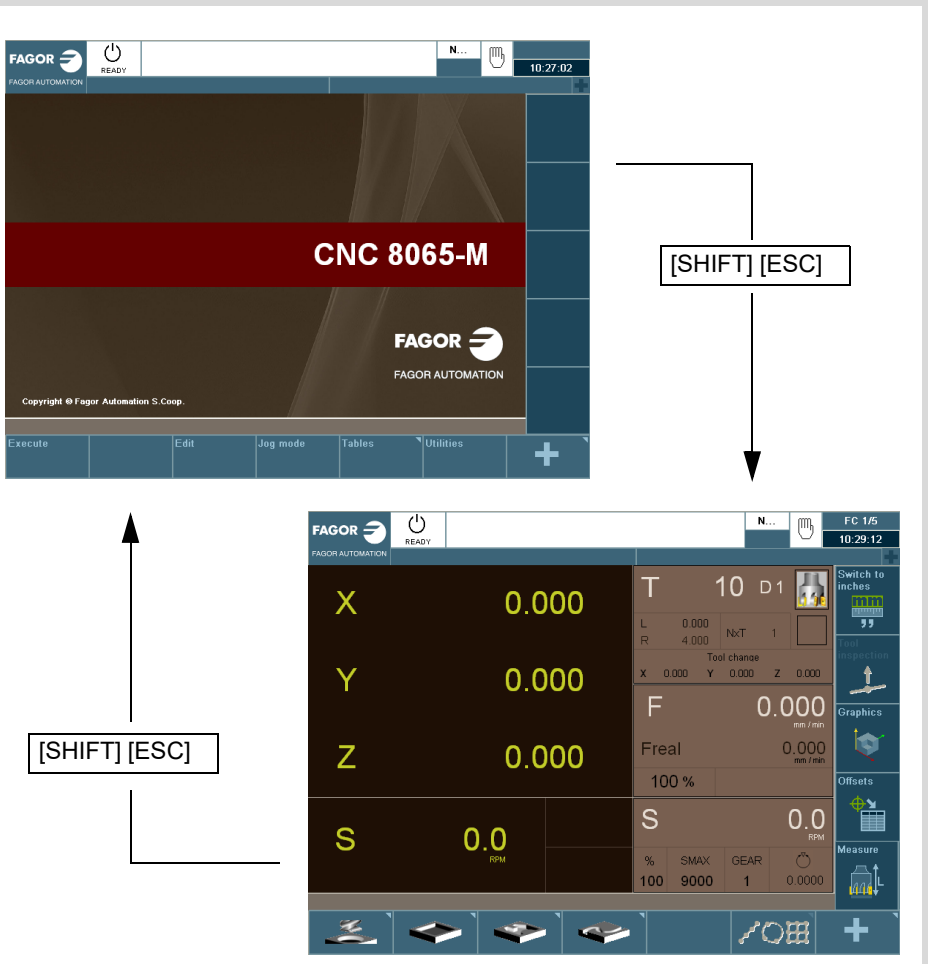
REF. 2109

GENERAL CONCEPTS

1

1.1 Accessing the conversational mode

Once the CNC has been started up, press the key sequence [SHIFT] [ESC] to switch to conversational mode. Press [SHIFT] + [ESC] again to return to M mode.



The CNC setup must be done in M mode. Likewise, some errors must be eliminated in M mode.

1.

GENERAL CONCEPTS

Accessing the conversational mode

FAGOR

FAGOR AUTOMATION

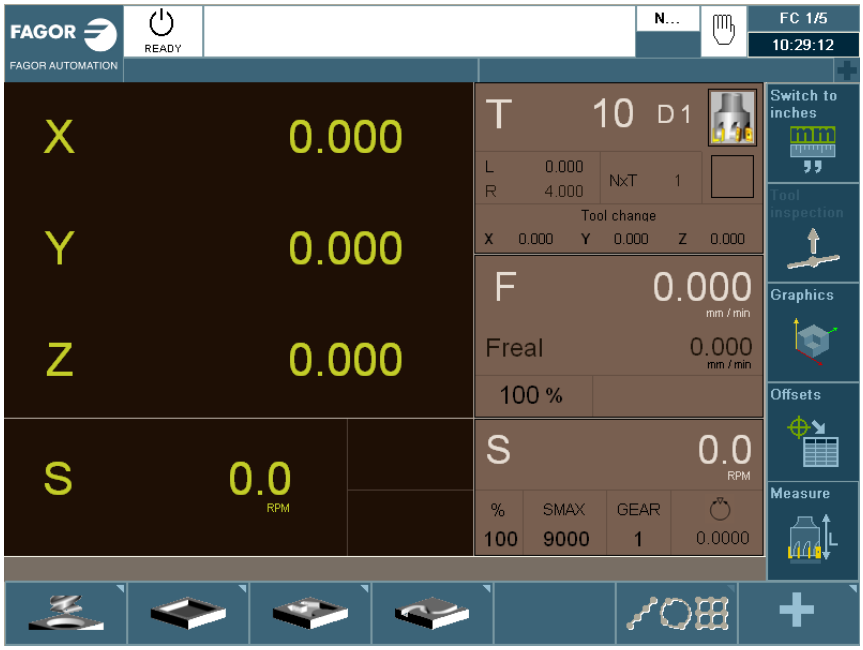
CNCelite
8058 8060
8065

REF. 2109

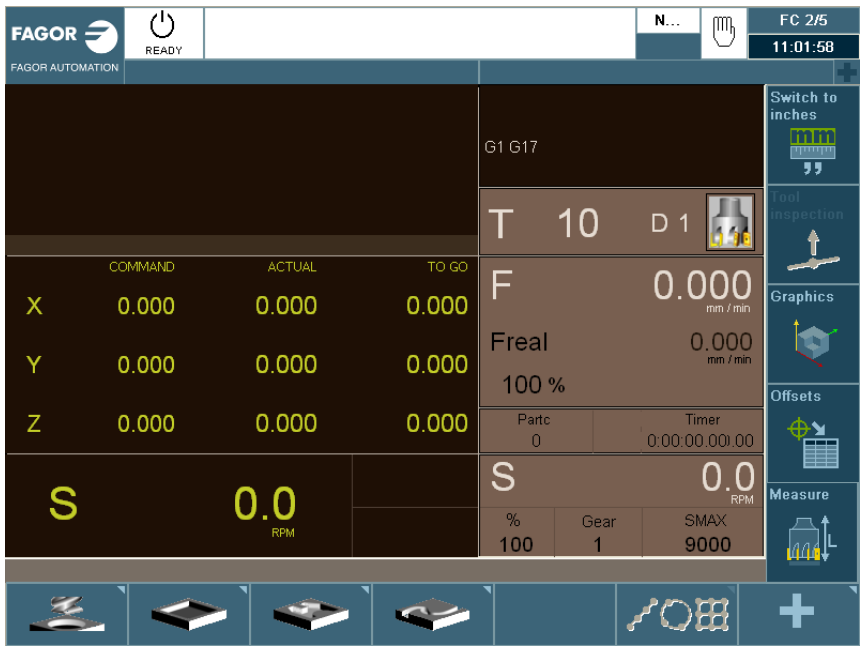
OPERATING IN JOG MODE

2

The standard screen of the MC mode is the following:

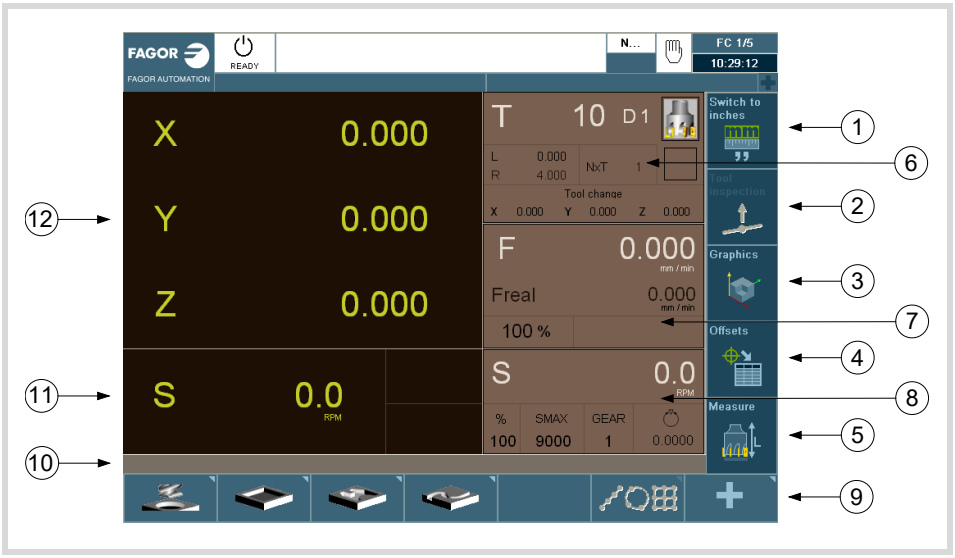


When pressing the two-colored key, the CNC shows the auxiliary screen of the MC mode:



2.1 Introduction

2.1.1 Standard screen of the conversational mode



1 Softkey for selecting units mm/inches.

2 Softkey to go into tool inspection.

3 Softkey to access the graphics in execution mode.

4 Softkey for selecting OFFSETS.

5 Softkey for tool calibration.

6 Window that shows:

- The selected tool (T).
- Graphic representation of the location code (shape).
- The (D) offset number associated with the selected tool.
- The offset defined for the tool.
- The position values (coordinates) of the tool change point referred to machine reference zero. If one of these coordinates is selected, it may be assigned the value of the current position of that axis by pressing [RECALL].

7 Window showing the axis feedrate F currently selected, the % of F being applied and the real F value. When selecting an incremental jog or a handwheel, this window will also show the selected % with the corresponding icon and the selected %.

8 Window showing spindle related information:

- The selected theoretical turning speed. S value when working in rpm and CSS value when working at constant surface speed.
- Spindle status. It is represented with an icon and may be turning clockwise, counterclockwise or stopped.
- The % of spindle speed being applied.
- Maximum spindle rpm (Smax).
- Active spindle range (gear).

9 Softkeys for cycle editing.

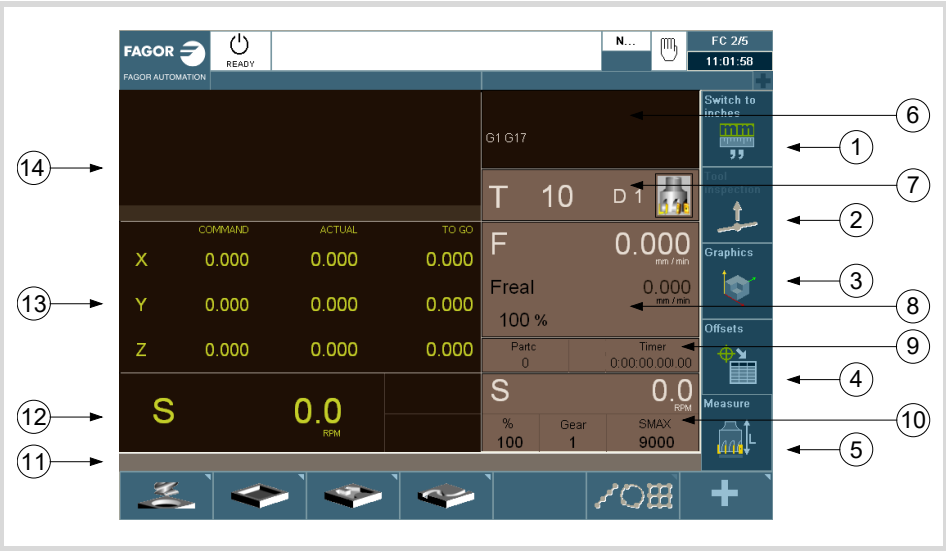
10 Message bar.

11 Real spindle rpm.

12 Position (coordinates) of the axes. The f symbol indicates that the axis is working in diameter.

If there are more than one spindle in the active channel, S may be pressed repeatedly to select the spindle whose data is being displayed. If the cell for programmed turning speed is already selected, every time S is pressed, it will show the data of the next spindle.

2.1.2 Auxiliary screen of the conversational mode



1 Softkey for selecting units mm/inches.

2 Softkey to go into tool inspection.

3 Softkey to access the graphics in execution mode.

4 Softkey for selecting OFFSETS.

5 Softkey for tool calibration.

6 Window that shows the status of the G, F, T, D, M functions.

7 Window that shows:

- The selected tool (T).
- Graphic representation of the location code (shape).
- The (D) offset number associated with the selected tool.

8 Window showing the axis feedrate F currently selected, the % of F being applied and the real F value.

9 Window that shows the value of the variables:

- Partc: It indicates the number of consecutive parts executed with the same part-program. Every time a new program is selected, this variable is reset to "0".
- CyTime: It indicates the time elapsed while executing the part. It is given in "hours : minutes : seconds : hundredths of a second" format. Every time a part-program execution starts, even when repetitive, this variable is reset to "0".
- Timer: It indicates the count of the timer enabled by PLC. It is given in "hours : minutes : seconds" format.

10 Window with spindle related information:

- The selected theoretical turning speed. S value when working in rpm and CSS value when working at constant surface speed.
- The % of spindle speed being applied.
- Maximum spindle rpm (Smax).
- Active spindle range (gear).

11 Message bar.

12 Window with spindle related information:

- Theoretical speed.
- Speed in RPM.
- Speed in m/min.

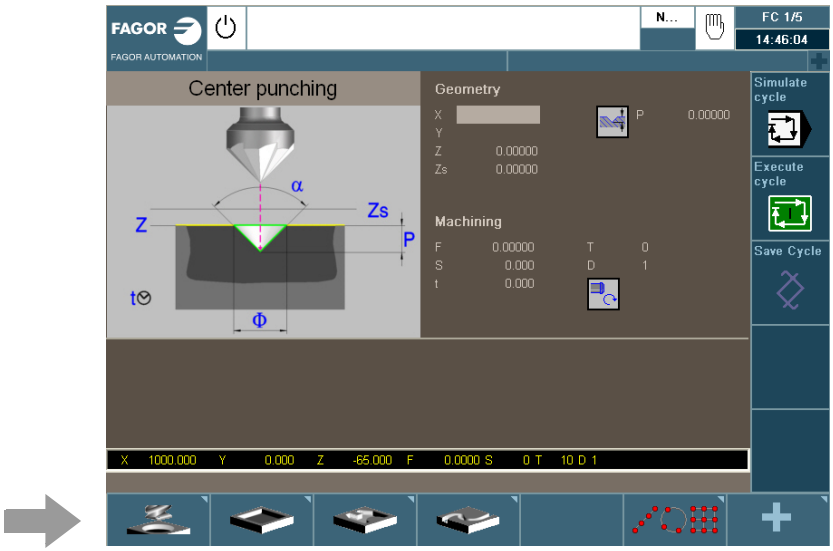
13 Window with axis related information:

- COMMAD: It indicates the programmed coordinate or position that the axis must reach.
- ACTUAL: It indicates the actual (current) position of the axis.
- TO GO: It indicates the distance which is left to run to the programmed coordinate.

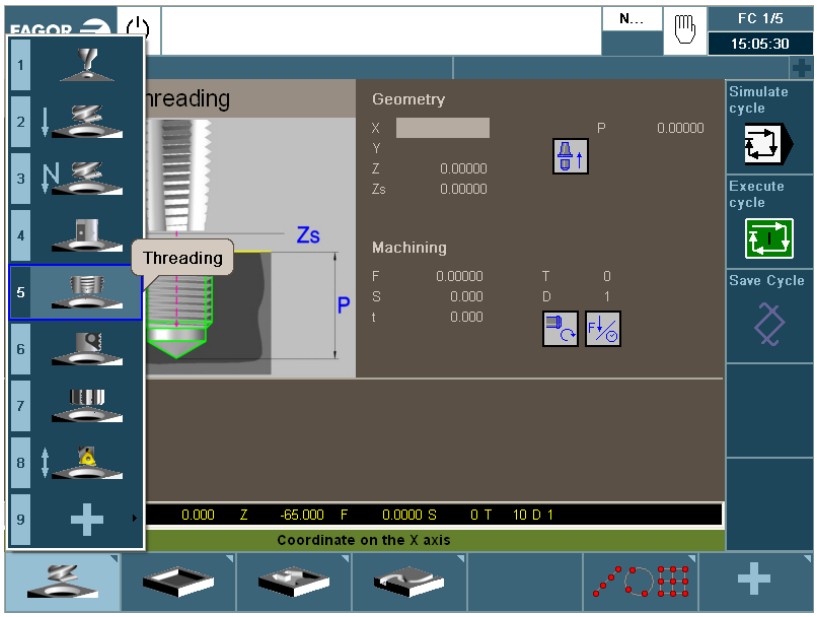
14 Window that shows the lines of the program being executed.

2.1.3 Cycle editing

To edit a cycle, press the softkey for the desired cycle.



To select another cycle of the same family as the one selected, press the softkey again to drop the menu with the available cycles.



Once the cycle to be edited has been selected, enter the data in the windows corresponding to each parameter of that cycle. To validate each parameter and go on to the next one, [ENTER].

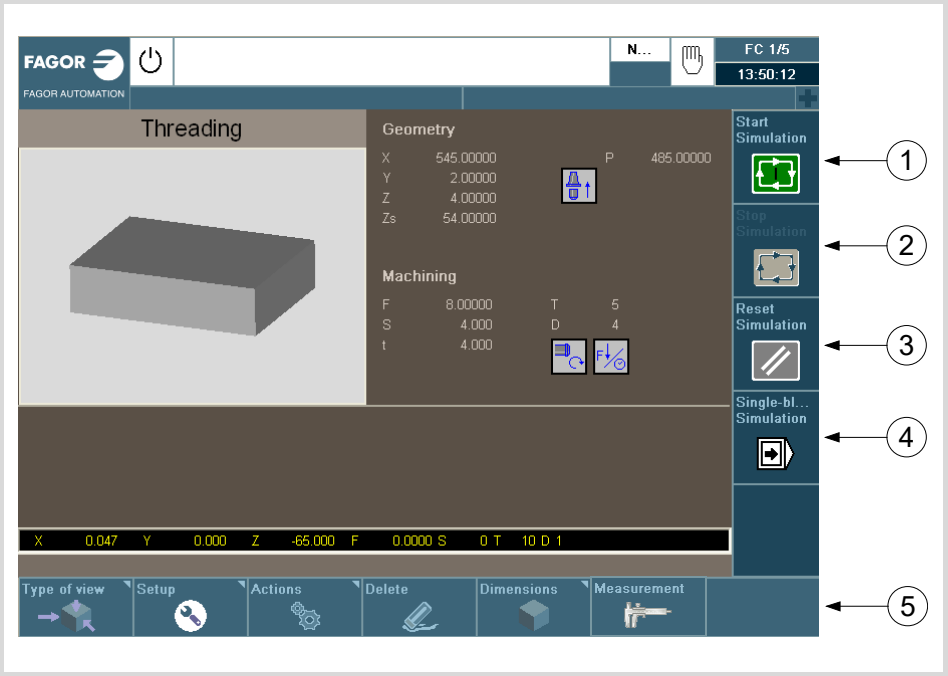
For further information on editing cycles, see the chapter *"3 Cycle editor"*.

After editing a cycle, it may be simulated, executed or saved using the vertical softkey menu.

For further information on saving cycles, see the chapter *"12 Saving programs"*.

2.1.4 Cycle simulation

To simulate the edited cycle, press the vertical softkey [Simulate cycle].



1 Softkey to start cycle simulation.

2 Softkey to stop cycle simulation.

3 Softkey to reset the simulation.

4 Softkey to simulate the cycle block by block.

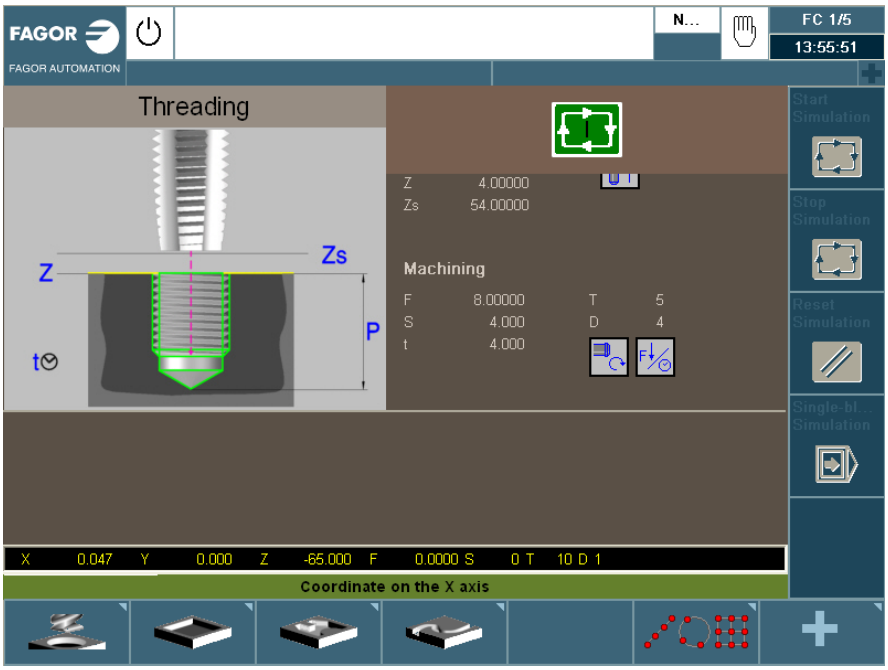
5 The horizontal softkeys may be used to configure how to display the simulated cycle.

- Type of view.
- Configuration.
- Actions.
- Delete.
- Dimensions.
- Measurement.

2.1.5 Cycle execution

To execute an edited cycle, press the vertical softkey [Execute cycle]. An icon will then appear with the start symbol to warn the user that it is going to execute the cycle.

To execute the cycle, press [START]. Otherwise, press [ESC].



2.2 Operations with the axes.

2.2.1 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.

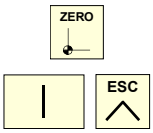
The axes may be homed manually (axis by axis from the operator panel) or automatically (using a subroutine).

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.

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.

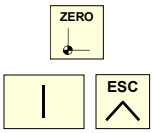


2 Press the homing key [ZERO]. The CNC will display the "1" symbol in the numeric area indicating that a home search will take place.

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

Automatic home search (with subroutine).

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

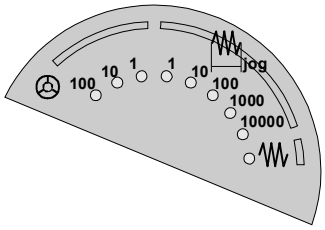


1 Press the homing key [ZERO]. The CNC will display a dialog box requesting confirmation to execute the home search.

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

2.2.2 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

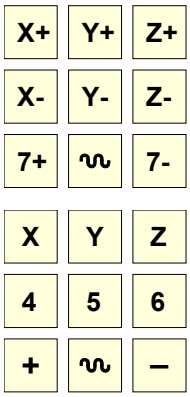
Incremental jog

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.

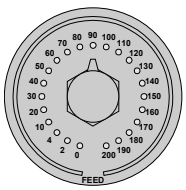
To jog an axis requires activating both the axis key and the moving direction. There are two options, depending on how the jog keyboard has been configured.

- 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.

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, the new one will move at the same time and under the same conditions.

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.

2.2.3 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.

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.

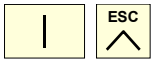
This feature must be managed from the PLC. Usually, this feature is turned on and off using an external push button or key configured for that purpose.

2.2.4 Moving an axis to a particular position (coordinate)

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

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

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.

2.2.5 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".

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.

2.3 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

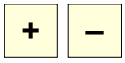
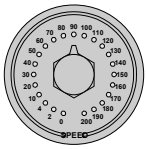
The spindle may be controlled manually using the following keys of the operator panel. The keys always refer to the master spindle of the active channel.

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%.

2.4 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.

2.5 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.

2.6 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.

2.7 Tool calibration

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

	Tool calibration in a mill 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.

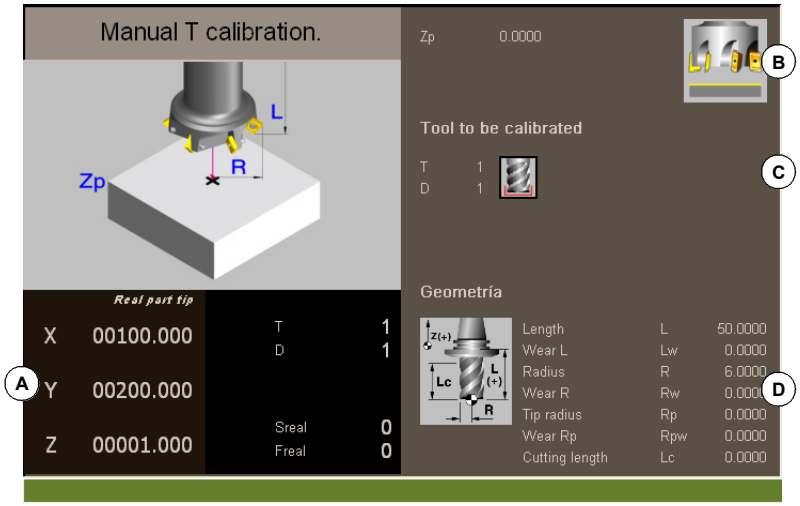
Probe selection

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.
```

2.7.1 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.



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.

Tool calibration

- 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.

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.

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.

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

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 54.

3 Calibrate the tool. Approach the tool manually until touching the part and then validate the calibration using the softkey menu.

   After validating the calibration, it updates the values and initializes the wear value to zero. Them, the new values are saved in the tool table.

4 Press [START] for the CNC to assume the new values of the offset.

To calibrate another tool, repeat steps 2 and 3.

Considerations for the offsets and their wear.

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

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

Sign criteria for the offsets and their wear.

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

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

Entering wear values incrementally or absolutely.

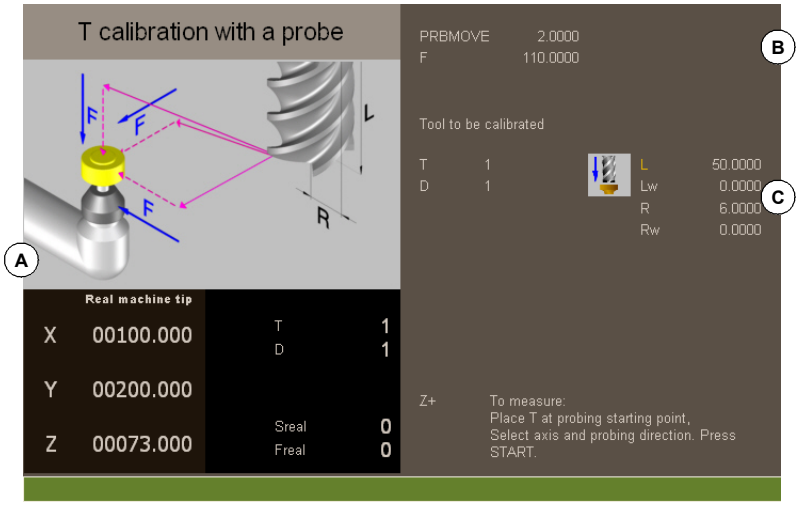
In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute.

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

2.7.2 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.



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.

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.

Tool calibration

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.

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.

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.
Off X Off Z	Tool offsets on each axis.

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 57.
3. Manually approach the tool to the probe until it is placed on the path that will be used for probing. To calibrate the radius with a cylindrical probe, the path must coincide with the probe's center point; if not, the radius will be calculated wrong.
4. Calibrate the tool. Select the axis and the probing direction on the softkey menu and press [START].

   The probe moves in parallel to the axis and in the selected direction until touching the probe. It updates the measured value and resets the wear value to zero. The data is stored in the tool table.
5. Press [START] again for the CNC to assume the new values of the offset. For the new values to be assumed, press [START] while the bottom message is displayed; otherwise, it executes the probing movement again.

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.

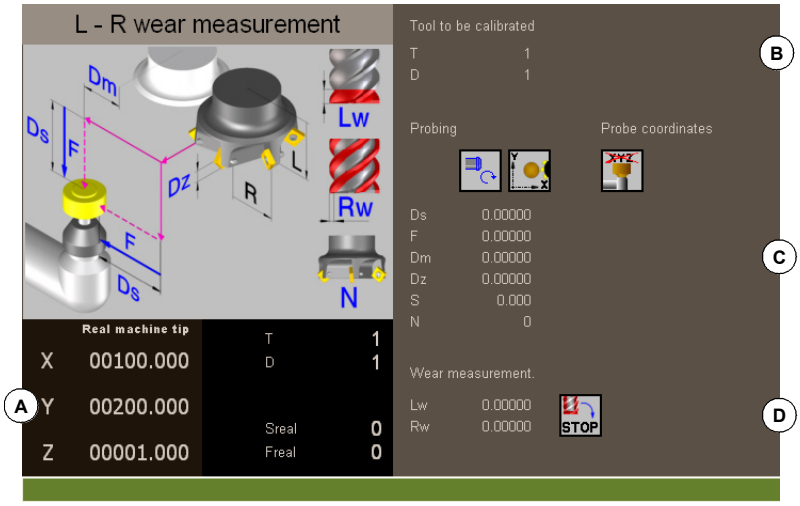
In the tool table, it is possible to define whether the wear value being entered must be incremental or absolute.

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

2.7.3 Automatic calibration with a probe and a canned cycle

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.

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.

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.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

CYCLE EDITOR

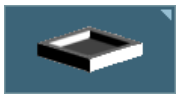
3

Select the machining cycles.

The machining cycles integrated into the cycle editor are grouped as follows. When pressing one of these softkeys, the editor shows the cycle of that group used last. When pressing the same softkey again, the menu shows all the cycles of the group.

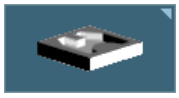
Z axis machining.

Center punching, drilling, deep hole drilling, bore milling, tapping, thread milling, reaming, boring, boring with spindle orientation.



Pockets / Bosses.

Simple rectangular pocket, rectangular pocket with rounding, circular pocket and pre-emptied circular pocket, rectangular boss and circular boss.

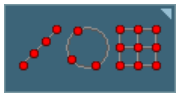


2D/3D profile pockets.

2D profile pocket and 3D profile pocket with islands.

Roughing.

Point-to-point profile milling, free profile milling, surface milling and slot milling.



Multiple machining.

Points in line, points in arc, points in a rectangular pattern, points in a grid pattern, points in a random pattern (several points defined by the user).

Activate the teach-in mode.

The "+" softkey shows the softkey to activate the teach-in mode that allows jogging the axes of the machine and entering the actual (real) position of the axes into the data. See *"3.3 Teach-in mode."* on page 67.

Configuring the cycle editor.

The "+" softkey shows the softkey to configure some of the options of the cycles of the editor.

Accessing the probe cycles (if available).

The "+" softkey displays the softkey to access the probe cycles of the milling model.

Accessing the milling cycles.

The "+" softkey displays the softkey to access the milling machining cycles.

Accessing the rotary axis cycles (if available).

The "+" softkey displays the softkey to access the machining cycles associated with the rotary axes. These cycles are only available if the “C axis” or “Y Axis” software options are active.

Accessing the machining cycles of the lathe model (if available).

The "+" softkey displays the softkey to access the machining cycles of the lathe model. These cycles are only available if “Combined Machine” software option is active.

Accessing the probe cycles of the lathe model (if available).

The "+" softkey displays the softkey to access the probe cycles of the lathe model. These cycles are only available if “Combined Machine” software option is active.

Hide the Fagor cycles.

The "+" softkey shows the softkey for configuring which Fagor cycles (machining and probe cycles) are visible or not. The menu showing the softkey can also remove the softkeys from the main menu.

3.1 Configuring the cycle editor.

The "+" softkey shows the softkey to configure some of the options of the cycles of the editor.

Programming M functions in each operation.

Enable the programming of M functions in the canned cycles, to execute them before each machining operation. This permits, for example, to execute subroutines associated with M functions before the various operations.

Being this option active, the editor offers in each operation of the cycle the option to edit up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

On the cycle screens, the display must be activated in order to see and define the M function data; otherwise, the data will not be displayed.

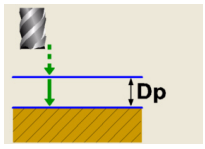
Programming the next tool.

Enable the capability to program the next tool (the one that will be executed after the cycle) in the cycles. When it is a random magazine, it prepares the tool while the cycle is being executed, hence reducing the machining time.

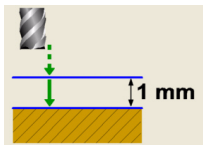
On the cycle screens, the display must be activated in order to see and define the next tool; otherwise, the data will not be displayed.

Programming the part surface approach distance.

Activate the possibility to program the part surface approach distance. This option is available for the center punching, drilling, threading, reaming and boring cycles.

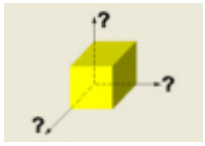


This option is enabled. The cycles show parameter ·Dp· for programming the part surface approach distance.



This option is disabled. The cycles assume an approach distance of 1 mm.

Select axis configuration.



Setting an axis configuration for the cycle editor. The defined axis configuration is only good for making it easier to edit the cycle, because it shows the data related to coordinates and plans according to the selected axis configuration.

The canned cycles have no work plane associated with them, they are executed in the current active work plane.

3.2 Configuring el cycle editor (rotary axis cycles).

When they cycles are selected for the rotary axes, the "+" softkey displays the softkey to configure some of the cycle options of the editor.

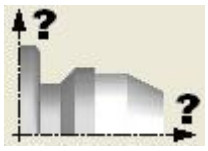
Programming M functions in each operation.

Enable the programming of M functions in the canned cycles, to execute them before each machining operation. This permits, for example, to execute subroutines associated with M functions before the various operations.

When this option is active, the editor offers the option to edit up to four M functions for each cycle operation. To execute only some of these, first define them and leave the rest unprogrammed.

On the cycle screens, the display must be activated in order to see and define the M function data; otherwise, the data will not be displayed.

Select axis configuration.



Setting an axis configuration for the cycle editor. The defined axis configuration is only valid for making it easier to edit the cycle because it shows the data related to the coordinates according to the chosen axis configuration.

The canned cycles have no work plane associated with them, they are executed in the current active work plane.

Selecting the axis associated with the rotary axis cycles.

Setting the configuration of the rotary axes for the cycle editor; A (rotary axis on the X axis), B (on the Y axis) or C (on the Z axis). The defined axis configuration is only good for making it easier to edit the cycle, because the editor shows the data related to coordinates and plans, according to the selected axis configuration.

Selecting the positioning axis.

Position the Y axis (or whichever one is missing from the trihedron) before executing the cycle. In the cycles for the rotary axes in which the Y axis is not involved, it is sometimes necessary to position this axis before executing the cycle; for example, after a tool change. By activating this option, the cycles will display the positioning axis (the one missing from the trihedron) of the shape Yi, Xi or Zi.

If this parameter is programmed, the cycle positions the axis (movement in G00) before executing the cycle.

3.3 Teach-in mode.

The "+" softkey shows the softkey to activate the teach-in mode that allows jogging the axes of the machine and entering the actual (real) position of the axes into the data. The reset of the data must be edited manually.

Being this mode active, the bottom of the cycle editor shows a window with the actual axis position and the active machining conditions. The information in the window cannot be configured, it is not conditioned by the configuration made in the EDISIMU mode for the teach-in mode.

When teach-in mode is active, it is possible to keep editing the data of the axes directly from the keyboard or they may be assigned the actual position of the axes. Both editing methods may be used indistinctly, even while defining a cycle. To assign the position of its axis to a data, proceed as follows:

1. Select one of the data with the cursor.
2. Move the axes to the desired position using the jog keys, the handwheels or the MDI/MDA mode.
3. Press the [RECALL] key. The editor enters the actual position of the corresponding axis into the selected data.

3.4 Selecting data, profiles and icons

Data selection.

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 [⬅] [➡] [⬆] [⬇] keys or with the direct access keys. The first data of each group may also be selected by pressing the page-up and page-down keys.

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

Data entry.

Place the cursor in the corresponding window, key in the desired value and press [ENTER]. If [ENTER] is not pressed, the new value will not be assumed.

If the Teach-in mode is selected, the current position of the machine may be associated with a coordinate. Place the cursor in the relevant window and press the [RECALL] key.

For the X axis parameters, it will take the coordinate of the first axis of the channel where the edit-simulation mode is active. For the Y axis parameters, the coordinate of the second axis and for the Z axis parameters, the coordinate of the third one.

Changing the state of an icon.

Place the cursor on the desired icon and press the space bar.

Select - Define a profile.

To select or modify a profile, the corresponding data must be selected; i.e. it must have the editing focus on it.

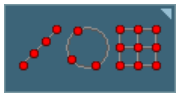
- To select an existing profile, press the [⬇] key to expand the list of defined profiles and select one or type its name.
- To define a new profile, write the desired name and press the [RECALL] key to access the profile editor.
- To modify an existing profile, select it from the list or write its desired name and press the [RECALL] key to access the profile editor.
- To delete a profile, press the [⬇] key to expand (drop) the list of profiles and select one. Press the [DEL] key to delete it.

3.5 Associate a multiple machining operation with a canned cycle

At the cycle editor, a multiple machining operation may be associated with the following cycles:

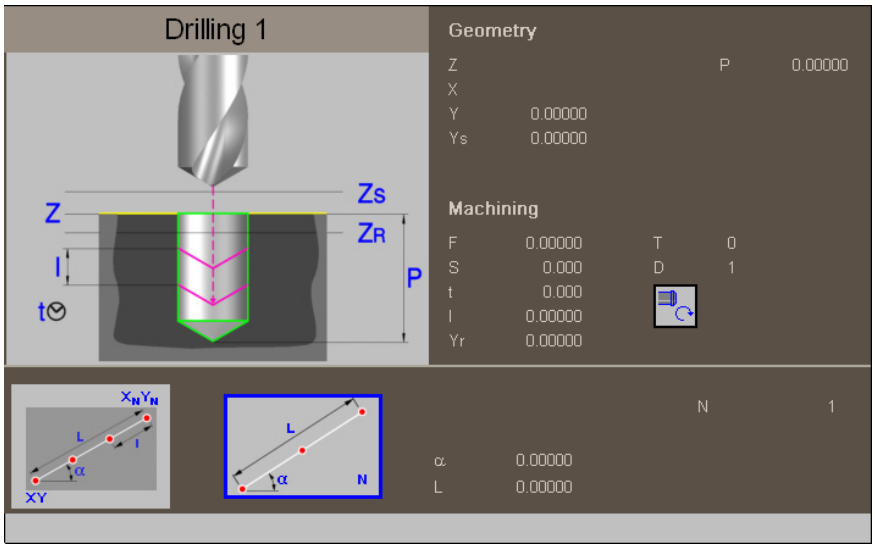
- Center punching, drilling, deep hole drilling, bore milling, tapping, thread milling, reaming, boring, boring with spindle orientation.
- Simple rectangular pocket, rectangular pocket with rounding, circular pocket and pre-emptied circular pocket and 2D profile pocket.
- Rectangular boss and circular boss.

How to select multiple machining.



To associate multiple machining to a cycle, first select and define a machining cycle from those allowed. Then, without quitting the editing of the cycle, press the softkey associated with multiple machining and select one of them.

The next figure shows the drilling cycle (top) with a multiple linear machining operation associated with it (bottom). To edit the data of the canned cycle or of the multiple machining operation, select the relevant window using the [FOCUS] key.



When the canned cycle takes up the whole screen, the multiple machining operation is superimposed on it as shown in the figure. In these cases, while editing the cycle data, the top window is shifted automatically to show the data.

3.6 Simulate a canned cycle.

With the canned cycle editor, it is possible to simulate the cycle being edited without having to simulate the whole part-program. During simulation, the editor allows viewing and editing another canned cycle and also returning to the program editor.

If the cycle editor is included in the automatic operating mode, it will not be possible to simulate a cycle.

Simulating a cycle

Pressing the [START] icon begins the simulation of the cycle that is being edited. The simulation may be interrupted with the [STOP] icon or canceled with the [RESET] icon.

The simulation graphics is always superimposed on the help graphics of the main cycle. If the cycle has a positioning associated with it, the graphics is superimposed on the main cycle; in the case of a 2D pocket with drilling, on the pocket.

Once the simulation has started, it is maintained until the cycle is over or the [RESET] icon is pressed. Even when changing cycles or returning to the program editor during simulation, the previous cycle is still in effect during the simulation.

Cycle simulation window

The graphics window (in simulation) is activated by pressing the [START] icon and is canceled by pressing the [RESET] icon. This window is placed over the cycle help graphics; it may be expanded to full screen (or shrunk again) using the key combination [CTRL]+[G].

The lower left corner of the window indicates the name of the cycle and the simulation channel, which will be the channel of the program editor from which the cycle editor has been called.

Configuring the graphic environment

When activating or selecting the graphics window, the horizontal softkey menu shows the available graphic options. For further information on the graphic options, see the chapter on the edit-simulation mode of the operation manual.

Some graphic options can also be edited manually. The editing area is only shown when the window is expanded ([CTRL]+[G]).

The simulated graphics are maintained until erased; i.e. starting to simulate a new cycle does not erase the previous graphics.

Best area for displaying the graphics

The display area may be established from the softkey menu associated with the simulation graphics window or may be left up to the CNC to periodically calculate the best area.

While the graphics window is visible, the key combination [CTRL]+[D] activates the calculation of the best area. From that moment on and until quitting the cycle editor, the CNC periodically calculates the best display are for the graphics. When quitting the graphics, it will assume as the new display area the one calculated last.

Window for simulation and data editing

While the graphics window is selected, it may be switched to the cycle parameter area using the direct access keys. If the parameter belongs to a positioning cycle, firs press [CTRL]+[F2] (window change)

If the cycle is simulated at full screen, the cycle editor may also be accessed by pressing the [ESC] key. To select the graphics window again, use the key combination [CTRL]+[G] or [SHIFT]+[G] or [G].

The horizontal softkey menu will show the graphic options when the graphics window has the focus and those of the cycle editor if otherwise.

The simulation in progress is not interrupted while editing data. If the cycle data is changed during simulation, they will be assumed for the next simulation of the cycle; i.e. after RESETting the simulation in progress once it has finished or after a STOP and RESET to abort it.

Summary of the hotkeys while simulating a cycle.

[CTRL] + [F2]	In the parameter window, it toggles between the cycle parameters and the positioning parameters.
[CTRL]+[G]	It selects the graphics window. It shrinks or expands the graphics window. It shows the dialog area for the graphics data.
[CTRL]+[D]	It activates the periodic calculation of the best display area.
[SHIFT]+[G] [G]	It shows the graphics window when a simulation is running and the parameter editing window is active.
[ESC]	If the graphics are shown at full screen, it shows the cycle editor screen.

3.

CYCLE EDITOR

Simulate a canned cycle.

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

WORK PLANES AND SHIFTING OF THE MACHINING OPERATIONS.

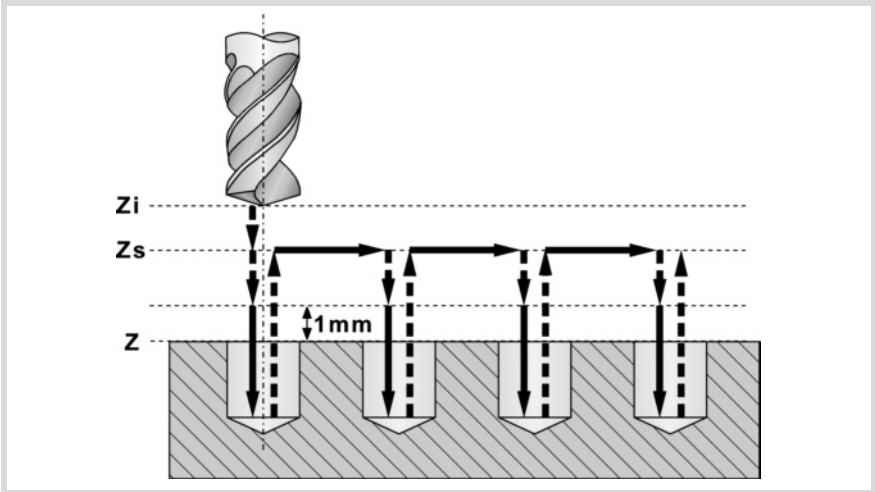
4

The canned cycles have no work plane associated with them, they are executed in the current active work plane. To make defining the cycle easier, the editor may be set with an axis configuration that will be good to show the data of the editor. Depending on this configuration, the planes along the longitudinal axis may be called, for example, Xs, Ys or Zs.

Work planes along the longitudinal axis.

These are the four work planes available in all operations (assuming the Z axis as longitudinal).

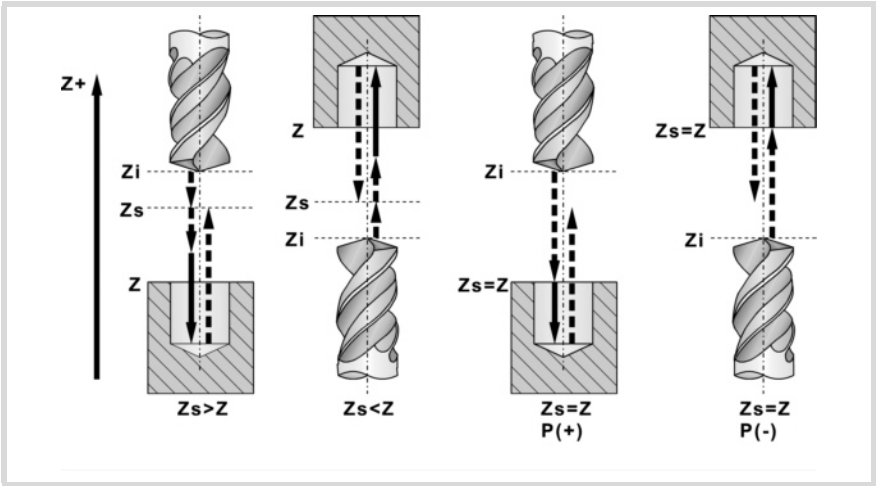
- Starting plane or tool position when calling the cycle Zi). It is not necessary to define this plane.
- Safety plane, for the first approach to the part and to move the tool between machining operations. This plane is defined by parameter Zs of the cycle.
- Plane to approach to the part, for a rapid approach to the part before starting machining. The cycle sets this plane 1 mm off the part. Depending on the configuration of the editor, in Z axis machining operations (center punching, drilling, etc.) this plane may be defined with parameter Dp.
- Part surface. The part surface is defined by parameter Z of the cycle.



Machining direction.

The machining direction is set by the position of the part surface (Z) and of the safety plane (Zs). If they are both the same, the direction is set by the sign of total machining depth

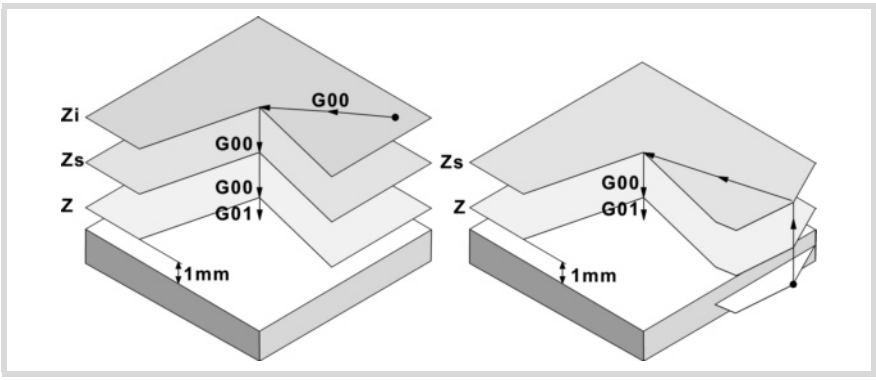
(parameter P). If Z=Zs and P>0 machining in the negative direction of the longitudinal axis (Z-), if Z=Zs and P<0 machining in the positive direction (Z+).



Movments in the work planes.

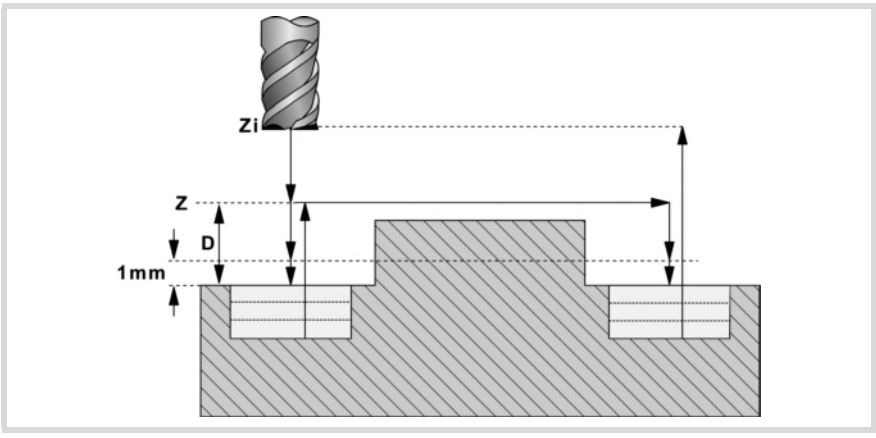
When beginning to execute the cycle, the tool moves in rapid (G0) from the starting plane (Zi) to the safety plane (Zs).

- If the starting plane is above the safety plane (left figure), it first moves in the plane and then along the longitudinal axis Z.
- If the starting plane is under the safety plane (right figure), it first moves along the longitudinal axis and then in the plane.



Then, the tool moves in rapid (G0) to the approach plane and finally at working feedrate to carry out the machining operation. Once the machining operation has concluded, the tool returns to the safety plane (Zs). If the cycle has a multiple machining associated to it, the tool moves along the safety plane (Zs), up to the next point to be machined.

The approach plane permits, as in the case of the figure, a fast approach to the machining surface when the safety plane (Zs) is far away from the part surface.

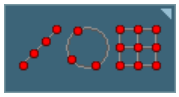


CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.

5

When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.

- Center punching.
- Drilling.
- Deep hole drilling.
- Bore milling.
- Tapping.
- Thread milling.
- Reaming.
- Boring.
- Boring with spindle orientation.

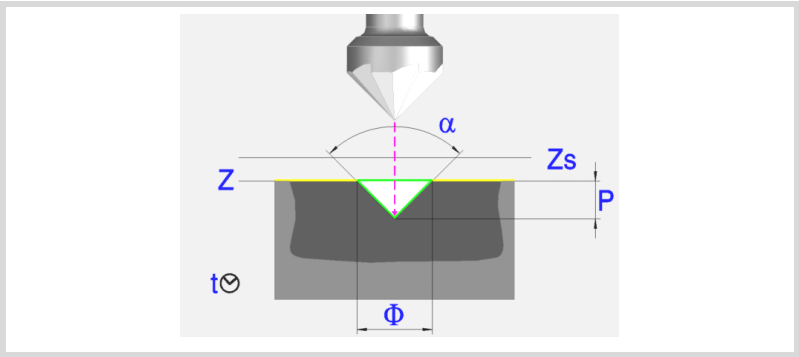


Multiple machining may be added to pocket cycles so the cycle is repeated in several points. The available multiple machining operations are the following. See chapter *"9 Canned cycles of the editor. Multiple positioning."*.

- Points in line.
- Points in arc.
- Points in rectangle.
- Points in grid.
- Random points (several points defined by the user).

To associate multiple machining to a cycle, first select and define a machining cycle from those allowed. Then, without quitting the editing of the cycle, press the softkey associated with multiple machining and select one of them.

5.1 Center punching.



Geometric parameters:

X, Y	Machining point.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
Dp	Part surface approach distance. The editor will only show this data if the user has configured the editor to allow it. If this option is not enabled, the cycle assumes an approach distance of 1 mm.

Depth programming type (icon).

Programming the total depth.

Programming the angle and the diameter.

P	Total depth.
α	Center-punching angle.
ϕ	Center-punching diameter.
	With Z=Zs the machining direction is always towards Z(-)

Machining parameters:

F	Feedrate.
S	Spindle speed.
T	Tool.
D	Tool offset.
t	Dwell at the bottom, in seconds.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

Activating or deactivating the preparation of the next tool.

The editor will only show this option if the user has configured the editor to allow programming the next tool.

FAGOR AUTOMATION

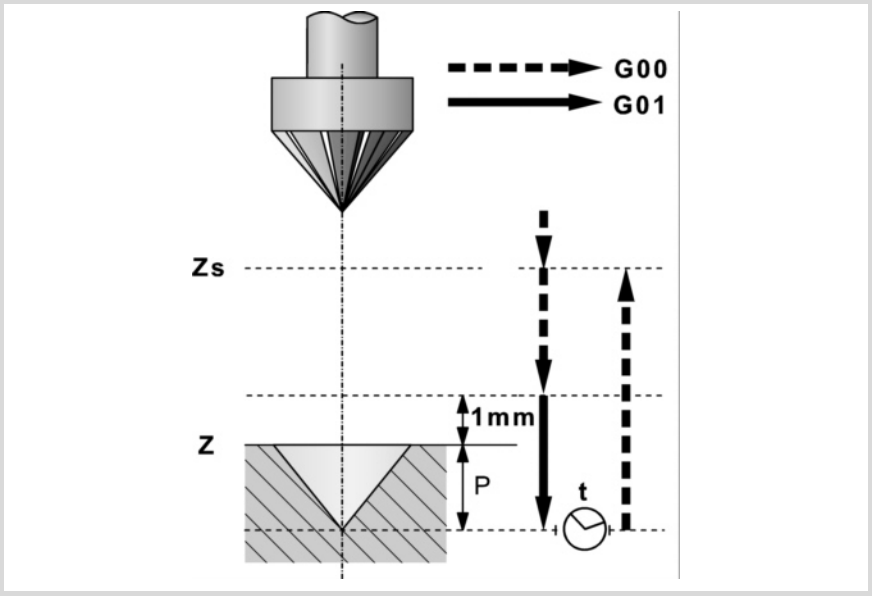
CNCelite
8058 8060
8065

REF. 2109

5.1.1 Basic operation.

1 It starts the spindle in the requested direction.

2 Rapid movement (G0), up to the XY point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.



3 Rapid movement (G0) up to the approach plane.

4 Penetration at feedrate "F".

5 Dwell "t".

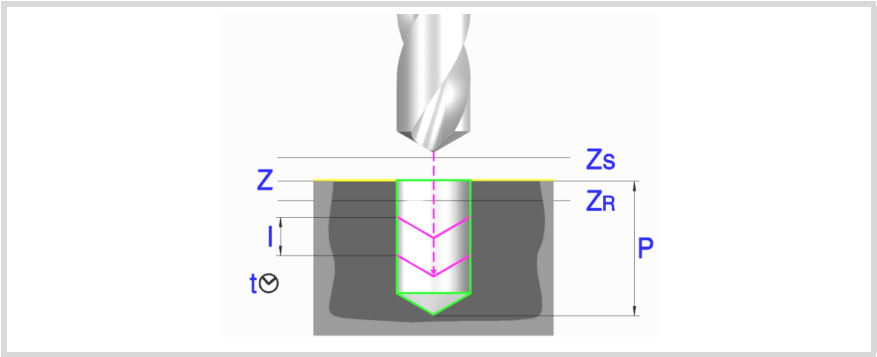
6 Rapid withdrawal (G0) up to the safety plane (Zs).

If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

7 Rapid movement (G0) to the next point.

8 Repeats steps 3, 4, 5, 6.

5.2 Drilling.



Geometric parameters:

X, Y	Machining point.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
Dp	Part surface approach distance. The editor will only show this data if the user has configured the editor to allow it. If this option is not enabled, the cycle assumes an approach distance of 1 mm.
P	Total depth.

Machining parameters:

I	Penetration step. The drilling takes place with the given step, except the last step that machines the rest. If programmed with a 0 value, the CNC will display the corresponding error message.
Zr	Relief coordinate it returns to, in rapid (G0), after each drilling step. If it has not reached the "Zr" coordinate, it returns to the approach plane.
F	Feedrate.
S	Spindle speed.
T	Tool.
D	Tool offset.
t	Dwell at the bottom, in seconds.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

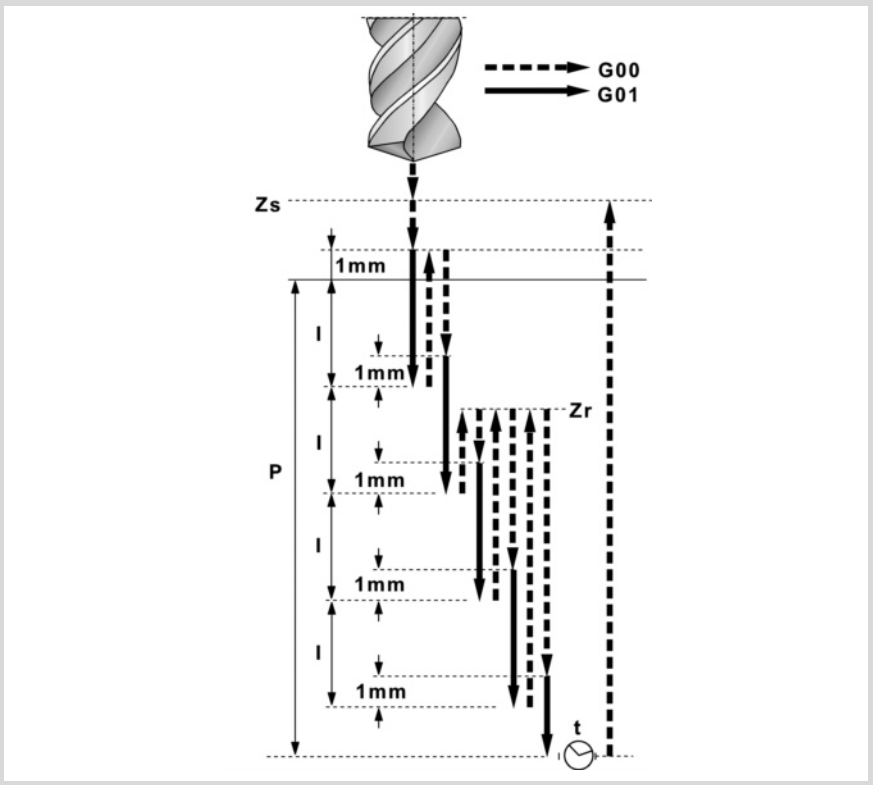
Next tool.

Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

5.2.1 Basic operation.

1 It starts the spindle in the requested direction.

2 Rapid movement (G0), up to the XY point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.



3 Rapid movement (G0) up to the approach plane.

4 It penetrates the distance "I" at the feedrate "F".

5 Drilling loop until reaching the total depth "P".

First, rapid withdrawal (G0) up to the relief coordinate Zr. If it has not reached the "Zr" coordinate yet, the tool returns to the approach plane. Then, rapid approach (G0) up to 1 mm from the previous drilling step (peck). Finally, "I" distance penetration at feedrate "F".

6 Dwell "t".

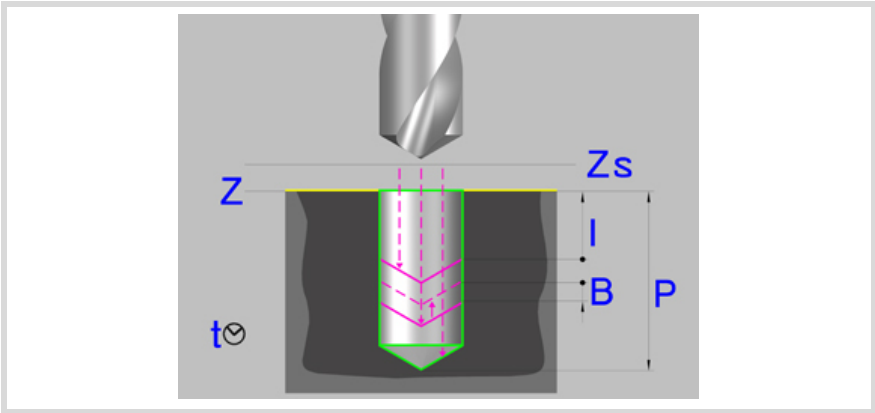
7 Rapid withdrawal (G0) up to the safety plane (Zs).

If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

8 Rapid movement (G0) to the next point.

9 Drills a new hole, steps 3, 4, 5, 6, 7.

5.3 Deep hole drilling.



Geometric parameters:

X, Y	Machining point.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
Dp	Part surface approach distance. The editor will only show this data if the user has configured the editor to allow it. If this option is not enabled, the cycle assumes an approach distance of 1 mm.
P	Total depth.

Machining parameters:

I	Penetration step. The drilling takes place with the given step, except the last step that machines the rest. If programmed with a 0 value, the CNC will display the corresponding error message.
B	Relief distance (it withdraws), in rapid (G0), after each drilling step. If set to ·0· (zero), the tool returns to the approach plane located 1 mm off the surface.
KI	Factor that reduces the drilling peck (step) "I". The first peck will be "I", the second "KI*I", the third "KI(KI*I)" and so on. If it is not programmed or "KI=0" is programmed, it assumes "KI=1". With "KI=1", all the drilling pecks will have the value of "I".
δI	Minimum drilling peck (step). Minimum value for the drilling peck. It is used with "KI" values other than 1. If not programmed, it assumes a value of 0 mm.
F	Feedrate.
S	Spindle speed.
T	Tool.
D	Tool offset.
t	Dwell at the bottom, in seconds.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

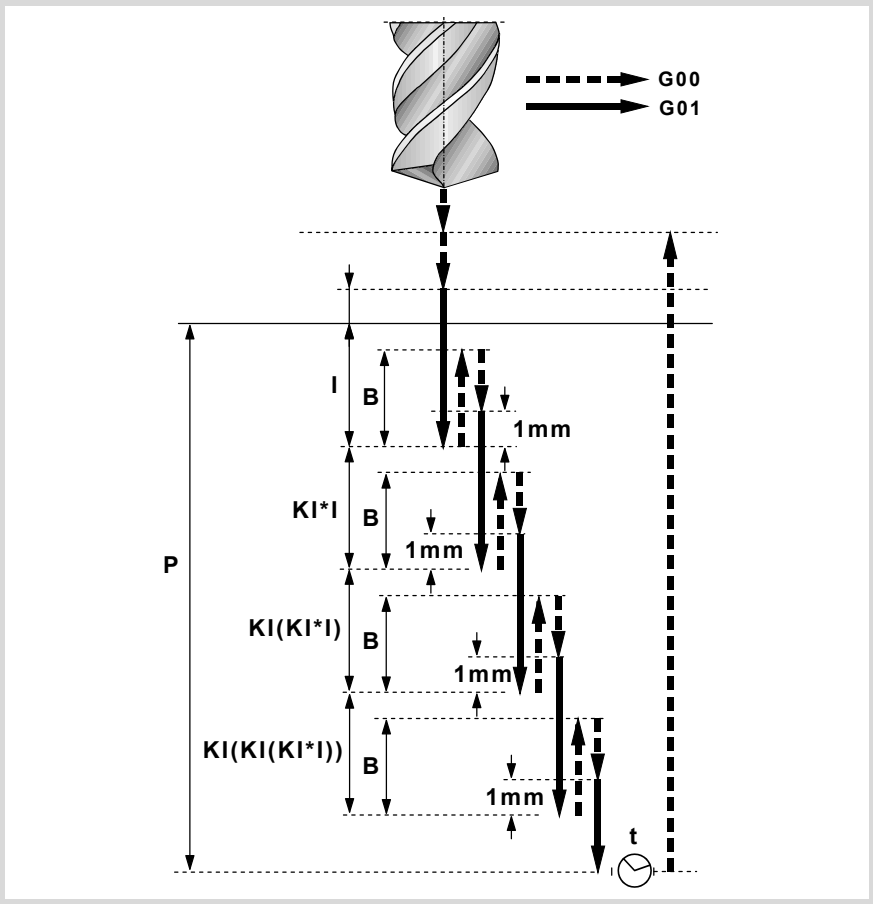
Next tool.

Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

5.3.1 Basic operation.

1 It starts the spindle in the requested direction.

2 Rapid movement (G0), up to the XY point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.



3 Rapid movement (G0) up to the approach plane.

4 It penetrates the distance "I" at the feedrate "F".

5 Drilling loop until reaching the total depth "P".

First, rapid withdrawal (G0) a relief distance "B". If B=0, return to the approach plane located 1 mm off the surface. Then, rapid approach (G0) up to 1 mm from the previous drilling step (peck). If B=0, approach to the previous machining step. Finally, "I" and "K" distance penetration at feed rate "F".

6 Dwell "t".

7 Rapid withdrawal (G0) up to the safety plane (Zs).

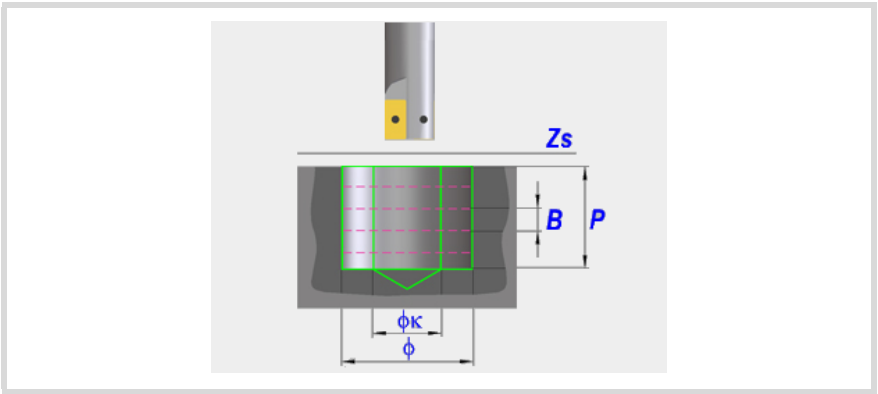
If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

8 Rapid movement (G0) to the next point.

9 Repeats steps 3, 4, 5, 6, 7.

5.4 Bore milling.

This cycle may be used to increase the diameter of a hole through a helical movement of the tool. Besides this, if the tool allows it, it is also possible to mill a hole without having to drill it first.

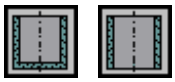


Geometric parameters:

Machining direction.

Defines the direction of the helical drilling path.

Bottom milling.



It defines whether the bottom of the hole (blind hole) is milled or not (through hole).

X, Y	Machining point.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
P	Total depth.
Dp	Part surface approach distance. The editor will only show this data if the user has configured the editor to allow it. If this option is not enabled, the cycle assumes an approach distance of 1 mm.
ϕ	Hole diameter.
ϕK	Pre-hole diameter. Starting with a hole previously drilled, this parameter defines the diameter of that hole. If not programmed or programmed with a 0 value, it means that no hole has been previously drilled. The tool must meet the following conditions: • The tool radius must be smaller than J/2. • The tool radius must be equal to or greater than (J-K)/4. If these two conditions are not met, the CNC issues the corresponding error message.
B	Helical penetration step.

Machining parameters:

F	Feedrate.
S	Spindle speed.
T	Tool.
D	Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

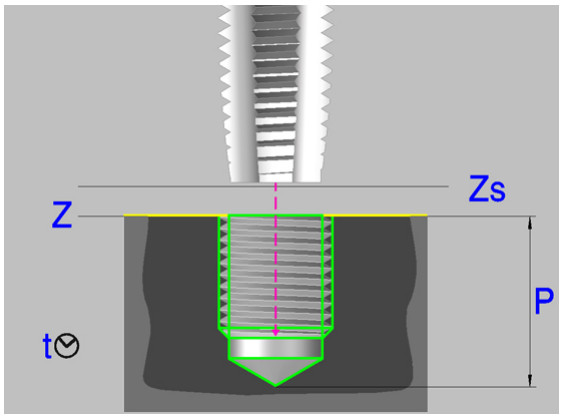
Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

5.4.1 Basic operation.

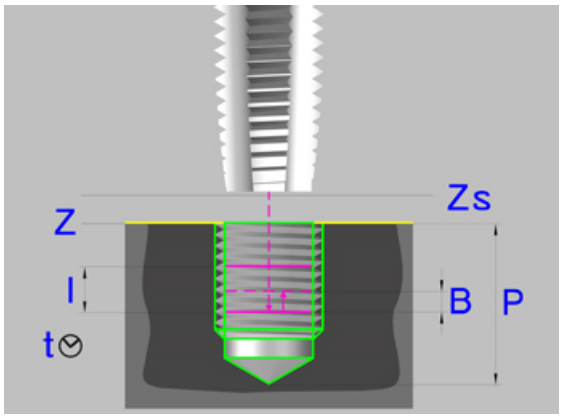
1. Rapid movement to the center of the hole (X, Y).
2. Rapid movement to the reference plane (Z).
3. Rapid movement to the tangential entry coordinate along the longitudinal axis.
4. Tangential entry to the helical path of the drilling.
5. Helical movement, with the pitch given by parameter B and in the direction given by the icon, down to the bottom of the hole.
6. Milling of the bottom of the hole (this step is only carried out if parameter B has a positive sign).
7. Tangential exit movement to the helical path of the drilling to the center of the hole.
8. Rapid movement to the reference plane (G99) or to the starting plane (G98).

5.5 Tapping.

Tapping with a clutch.



Rigid tapping.



Geometric parameters:

X, Y	Machining point.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
Dp	Part surface approach distance. The editor will only show this data if the user has configured the editor to allow it. If this option is not enabled, the cycle assumes an approach distance of 1 mm.
P	Total depth.
Kf	Feedrate factor for the exit. Rigid tapping allows a rapid exit from the tap maintaining always the synchronism between the feedrate and the speed. The withdrawal feedrate is multiplied by this factor (Kf) and the speed adapts to the new feedrate.
I	Penetration step when tapping with chip removal. If not programmed, the CNC executes the cycle in a single pass. If programmed with a 0 value, the CNC will display the corresponding error message.
B	Withdrawal distance after each penetration step. If not programmed or programmed a value of 0, the tool withdraws up to the reference plane Z. The cycle does each withdrawal taking the feedrate factor into account (parameter Kf).

Type of tapping (icon).

Tapping with a clutch.

Rigid tapping.

Machining parameters:

F	Feedrate.
S	Spindle speed.
T	Tool.
D	Tool offset.
t	Dwell at the bottom, in seconds.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Type of feedrate (icon).

In mm/min or (inch/min)

In mm/turn.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

Activating or deactivating the preparation of the next tool.

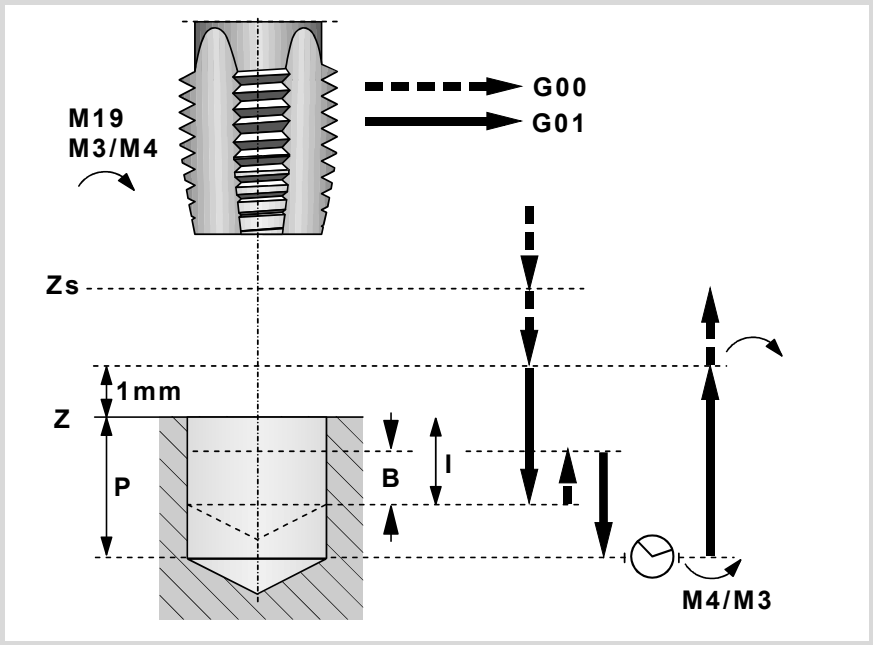
The editor will only show this option if the user has configured the editor to allow programming the next tool.

5.5.1 Basic operation.

1 If rigid tapping, it orients the spindle (M19).

If tapping with clutch, it starts the spindle in the requested direction.

2 Rapid movement (G0), up to the XY point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.



3 Rapid movement (G0) up to the approach plane.

4 Tapping, at 100% of the feedrate "F" and spindle speed "S" programmed.

Tapping with a clutch cannot be interrupted.

If "I" is not programmed, rigid tapping will be assumed. The cycle will execute the tap in a single pass.
If "I" is programmed, rigid tapping with chip removal will be assumed. The cycle executes the tap in several passes "I" deep withdrawing an "B" distance after each pass.
In rigid tapping, the feedrate override percentage may be changed and even stopped (0% override).

5 If "t" other than 0, spindle stop (M05) and dwell.

6 The spindle reverses turning direction. Withdrawal and exit the tap, to the approach plane. The cycle does the withdrawal taking the feedrate factor into account (parameter Kf). In regular tapping (with a clutch) the thread exit cannot be interrupted. In rigid tapping, the feedrate override percentage may be changed and even stopped (0% override).

7 If tapping with a clutch, it reverses the spindle turning direction (restores the initial one).

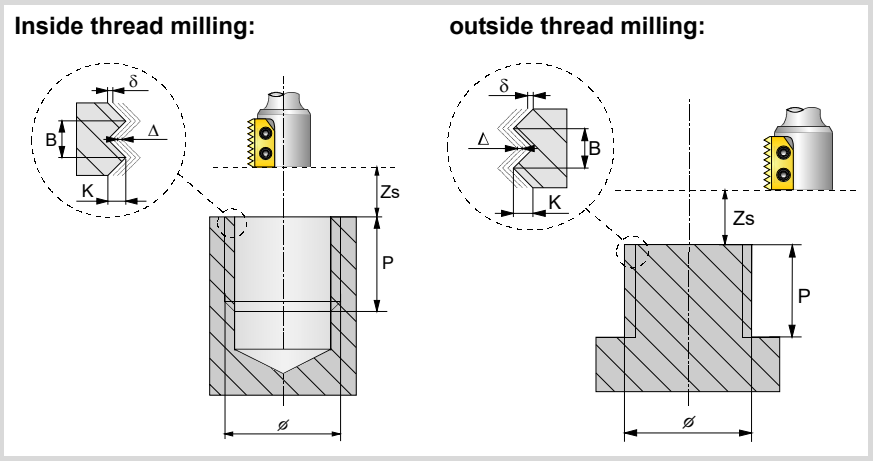
8 Rapid withdrawal (G0) up to the safety plane (Zs).

If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

9 Rapid movement (G0) to the next point.

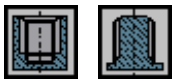
10 Repeats steps 3, 4, 5, 6, 7, 8, 9.

5.6 Thread milling.



Geometric parameters:

Type of tapping.

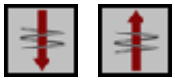


Defines the type of threading to be carried out (inside or outside).

Machining direction.

Defines the direction of the helical drilling path.

Thread machining direction.



It defines the thread cutting direction (from the part surface down to the bottom of the thread or from the bottom up to the part surface).

Type of tapping.

It depends on the type of tool being used.

X, Y	Machining point.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
P	Total depth.
Dp	Part surface approach distance. The editor will only show this data if the user has configured the editor to allow it. If this option is not enabled, the cycle assumes an approach distance of 1 mm.
ϕ	Thread diameter.
K	Thread depth.
B	Thread pitch.
Ds	Approach distance.
α	Thread entry (start) angle. Angle (in degrees) of the segment formed by the center of the hole and the thread entry point with respect to the abscissa axis.
N	Number of cutter edges It defines the number of cutting edges of the cutter (only when machining with a cutter of n edges).

Machining parameters:

Δ	Thread penetration step.
δ	Finishing stock.
F	Feedrate.
S	Spindle speed.
T	Tool.
D	Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

5.6.1 Basic operation.

1 Rapid movement to the center of the hole (X, Y).

2 Rapid movement to the reference plane (Z).

3 Rapid movement of the plane axes to the thread entry point.

4 Rapid movement to the thread entry point coordinate along the longitudinal axis.

5 Thread entry with a helical movement tangent to the first helical threading path.

6 Making the thread according to the selected tool type:

(1) Helical movement, in the direction indicated, to the bottom of the thread (the movement will only be one revolution).
(2) Helical thread exiting movement, tangent to the previous helical path.
It must be borne in mind that in the exit tangent to the helical path, the exit point will exceed the coordinate of the bottom of the thread along the longitudinal axis.

(1) Helical movement, with the pitch and direction given, to the bottom of the thread.
(2) Helical thread exiting movement, tangent to the previous helical path.
It must be borne in mind that in the exit tangent to the helical path, the exit point will exceed the coordinate of the bottom of the thread along the longitudinal axis.

(1) Helical movement, with the pitch and direction given (the movement will be one revolution).
(2) Helical thread exiting movement, tangent to the previous helical path.
(3) Rapid movement to the thread entry point of the next threading path.
(4) Rapid movement to the Z coordinate of the thread entry point of the next threading path.
(5) Repetition of the previous 3 steps until reaching the bottom of the thread. It must be borne in mind that in the last helical exit, the exit point will exceed the coordinate of the bottom of the thread along the longitudinal axis.

7 Rapid movement to the center of the hole (X, Y).

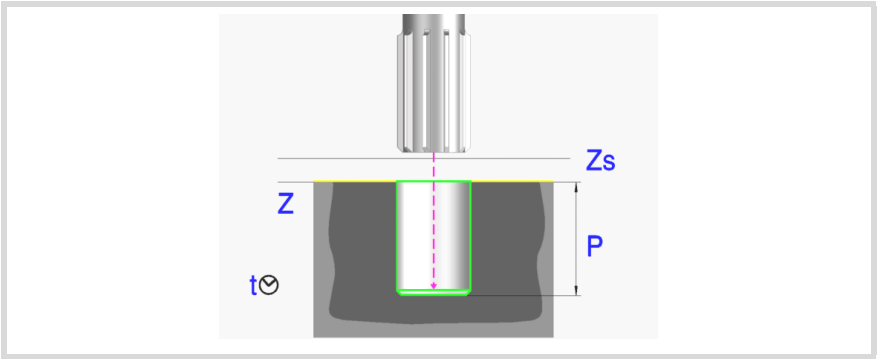
8 Rapid movement to the thread entry coordinate along the longitudinal axis.

9 Repetition of steps 3 to 8 until reaching the depth of the finishing stock.

10 Repetition of steps 3 to 8 until reaching the bottom of the thread.

11 Rapid movement to the reference plane (G99) or to the starting plane (G98).

5.7 Reaming



Geometric parameters:

X, Y	Machining point.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
Dp	Part surface approach distance. The editor will only show this data if the user has configured the editor to allow it. If this option is not enabled, the cycle assumes an approach distance of 1 mm.
P	Total depth.

Machining parameters:

F	Feedrate.
S	Spindle speed.
T	Tool.
D	Tool offset.
t	Dwell at the bottom, in seconds.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

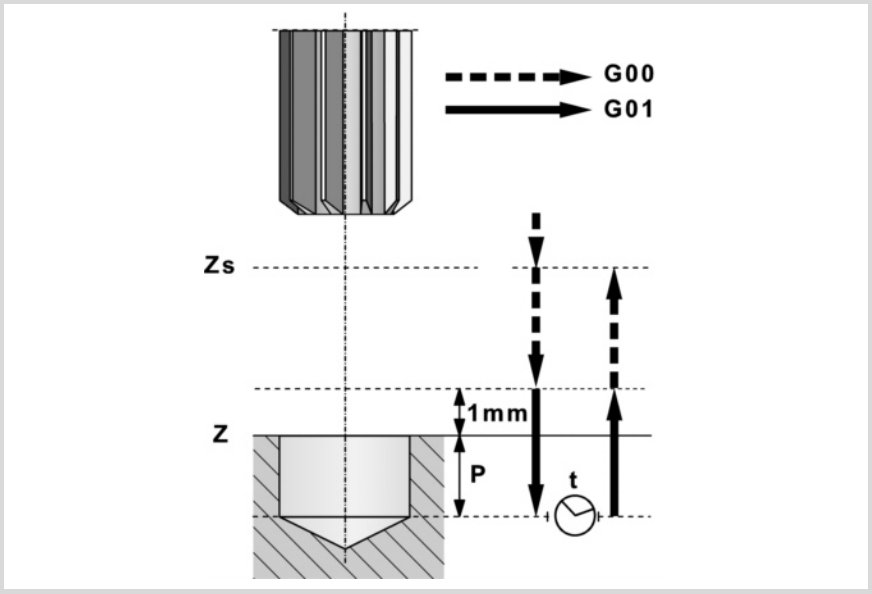
Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

5.7.1 Basic operation.

1 It starts the spindle in the requested direction.

2 Rapid movement (G0), up to the XY point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.



3 Rapid movement (G0) up to the approach plane.

4 Penetration at feedrate "F".

5 Dwell "t".

6 Withdrawal, at feedrate "F", to the approach plane.

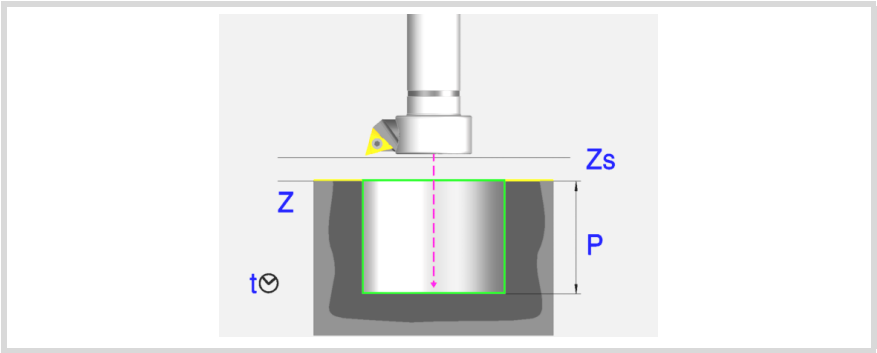
7 Rapid movement (G0) up to the safety plane (Zs).

If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

8 Rapid movement (G0) to the next point.

9 Repeats steps 3, 4, 5, 6, 7.

5.8 Boring.



Geometric parameters:

X, Y	Machining point.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
Dp	Part surface approach distance. The editor will only show this data if the user has configured the editor to allow it. If this option is not enabled, the cycle assumes an approach distance of 1 mm.
P	Total depth.

Machining parameters:

F	Feedrate.
S	Spindle speed.
T	Tool.
D	Tool offset.
t	Dwell at the bottom, in seconds.

Withdrawal direction (icon).

At feedrate "F" and the spindle turning.

In rapid (G0) with the spindle stopped.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

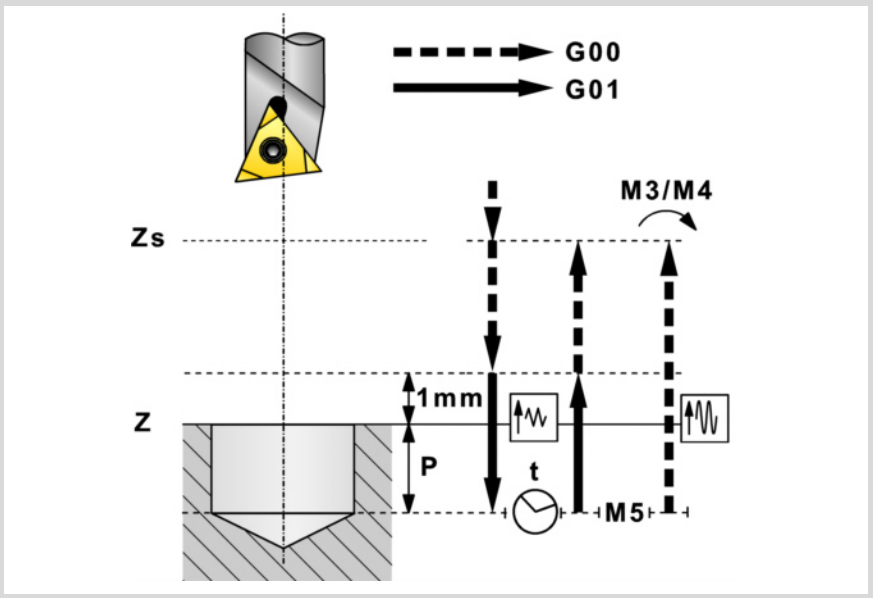
Next tool.

Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

5.8.1 Basic operation.

1 It starts the spindle in the requested direction.

2 Rapid movement (G0), up to the XY point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.

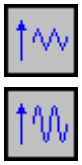


3 Rapid movement (G0) up to the approach plane.

4 Penetration at feedrate "F".

5 Dwell "t".

6 Withdrawal.



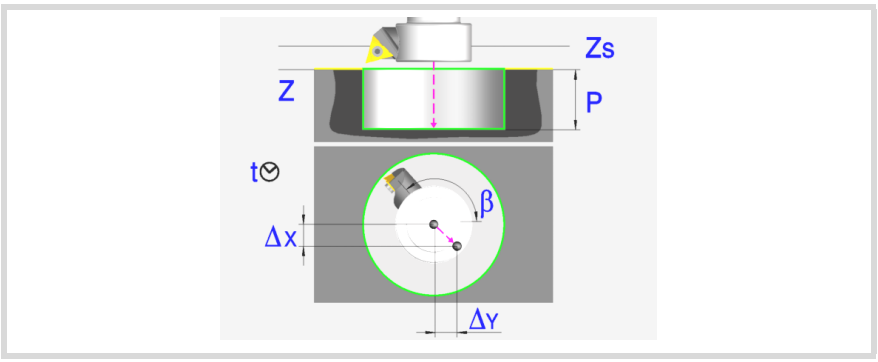
- It withdraws at feedrate "F" to the approach plane (at 1 mm above the surface "Z") and then in rapid (G0) to the safety plane Zs.
- Spindle stop. It withdraws in rapid (G0) to the safety plane Zs and then starts the spindle in the direction it was turning.

If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

7 Rapid movement (G0) to the next point.

8 Repeats steps 3, 4, 5, 6, 7.

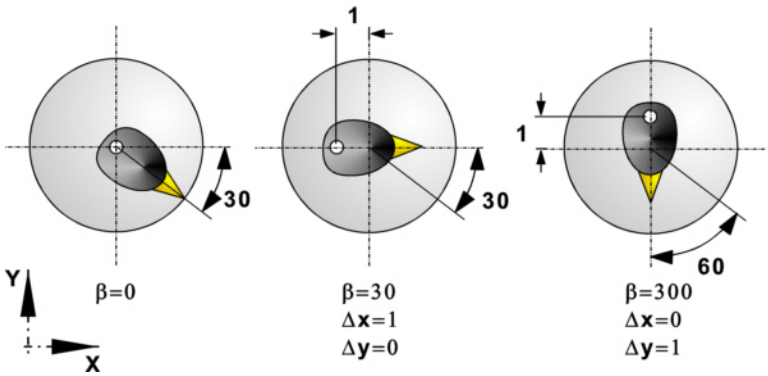
5.9 Boring with spindle orientation.



Geometric parameters:

X, Y	Machining point.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
Dp	Part surface approach distance. The editor will only show this data if the user has configured the editor to allow it. If this option is not enabled, the cycle assumes an approach distance of 1 mm.
P	Total depth.
β	Spindle position, in degrees, for the withdrawal.
Δx, Δy	Distance the tool must move to get the cutter off the wall before withdrawing.

The following example shows how to use parameters β, Δx and Δy. The spindle rest position (I0 position) is at -30º with respect to the X axis.



Machining parameters:

F	Feedrate.
S	Spindle speed.
T	Tool.
D	Tool offset.
t	Dwell at the bottom, in seconds.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

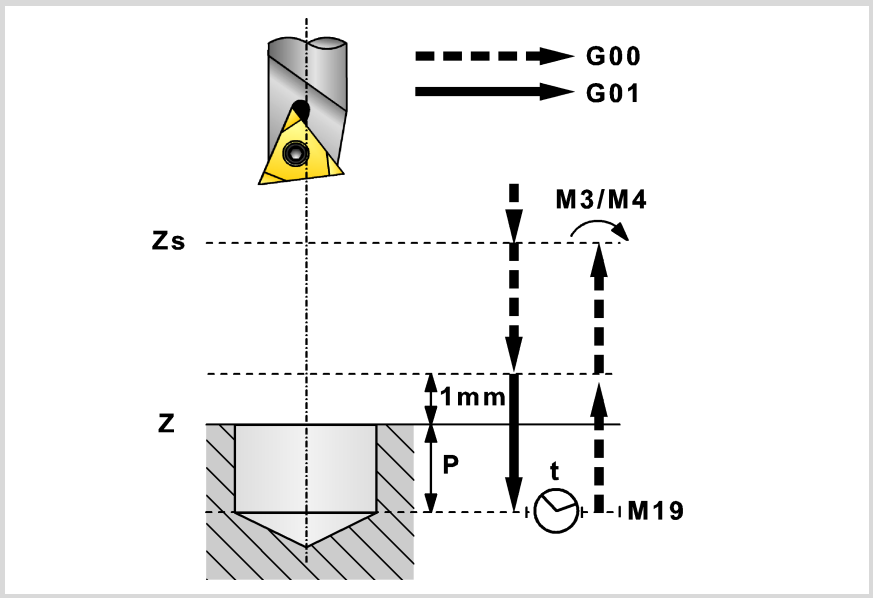
Next tool.

Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

5.9.1 Basic operation.

1 It starts the spindle in the requested direction.

2 Rapid movement (G0), up to the XY point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.



3 Rapid movement (G0) up to the approach plane.

4 Penetration at feedrate "F".

5 Dwell "t".

6 The spindle stops and the tool is oriented in the "β" position (M19).

7 It gets the cutter off the wall. It moves the distance indicated by "Δx, Δy".

8 Rapid withdrawal (G0) up to the approach plane.

9 The tool returns to its position (XY) and starts the spindle in the direction it was turning.

10 Rapid movement (G0) up to the safety plane (Zs).

If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

11 Rapid movement (G0) to the next point.

12 Repeats steps 3, 4, 5, 6, 7, 8, 9, 10.

5.

CANNED CYCLES OF THE EDITOR. Z AXIS MACHINING.

Boring with spindle orientation.

FAGOR

FAGOR AUTOMATION

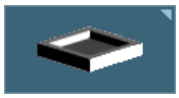
CNCelite
8058 8060
8065

REF. 2109

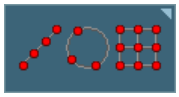
CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.

6

When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



- Simple rectangular pocket.
- Rectangular pocket rounding.
- Circular pocket.
- Pre-emptied circular pocket.
- Rectangular boss.
- Circular boss.



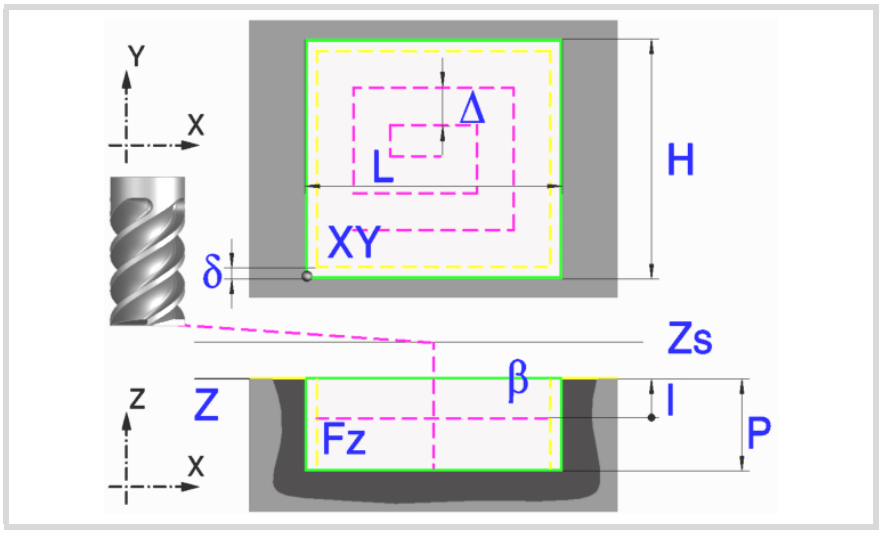
Multiple machining may be added to pocket cycles so the cycle is repeated in several points. The available multiple machining operations are the following. See chapter *"6 Canned cycles of the editor. Pockets / Bosses."*.

- Points in line.
- Points in arc.
- Points in rectangle.
- Points in grid.
- Random points (several points defined by the user).

To associate multiple machining to a cycle, first select and define a machining cycle from those allowed. Then, without quitting the editing of the cycle, press the softkey associated with multiple machining and select one of them.

6.1 Simple rectangular pocket.

The way the roughing and finishing blocks of this cycle are joined will be the one previously set by the user with the instructions #HSC, G5, G50 or G7. We recommend to use #HSC or G5 controlling the shape of the corner with the instruction #ROUNDPAR.



Geometric parameters:

X, Y Coordinates of the starting point of the pocket.

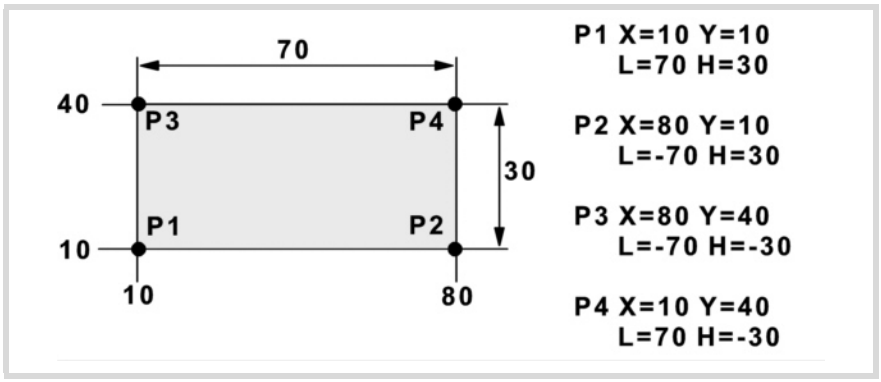
Starting point of the pocket (icon).

Starting point at one corner of the pocket.

Starting point at the center of the pocket.

L, H Pocket dimensions.

When the starting point of the pocket is one of its corners, the sign indicates the orientation with respect to the XY point.



Z Part surface coordinate.

Zs Safety plane coordinate.

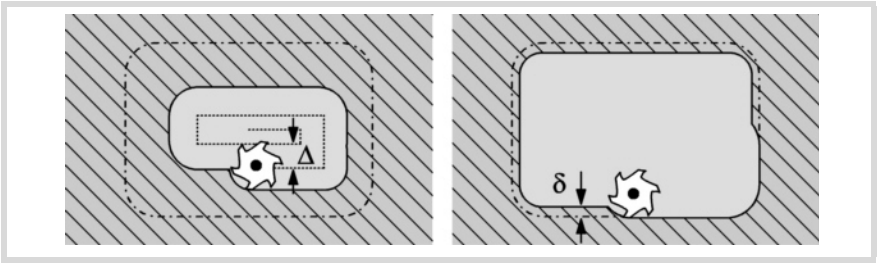
P Total depth.

Machining parameters:

Δ Maximum milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

δ Finishing stock on the side walls.

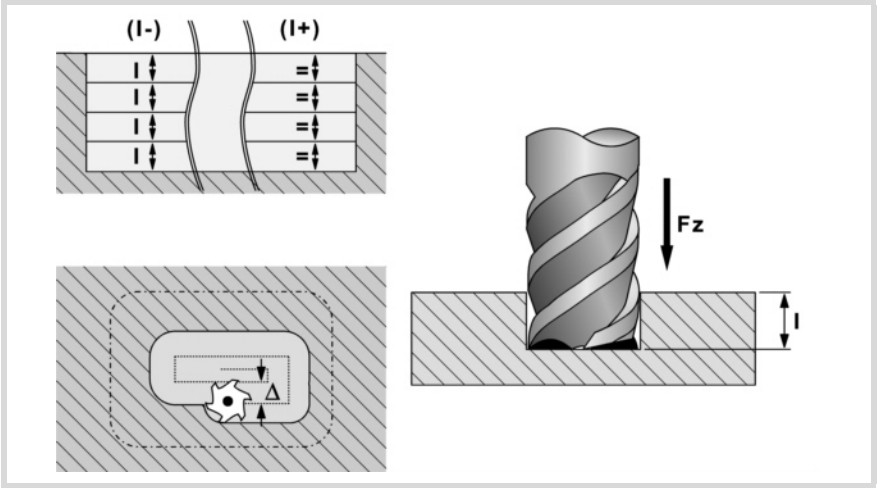


I Penetration step.

- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the pocket is machined with the given pass (step) except the last pass that machines the rest.
- When programming with a 0 value, the cycle assumes the step as the cutting length assigned to the tool in the tool table (Lc data). If the cutting length defined by the tool table is also set at 0, the cycle executes the pocket in a single penetration.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

Fz Penetration feedrate. If programmed with a 0 value, the cycle uses half of the programmed F feedrate.



F Surface milling feedrate.

S Spindle speed.

T Tool.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

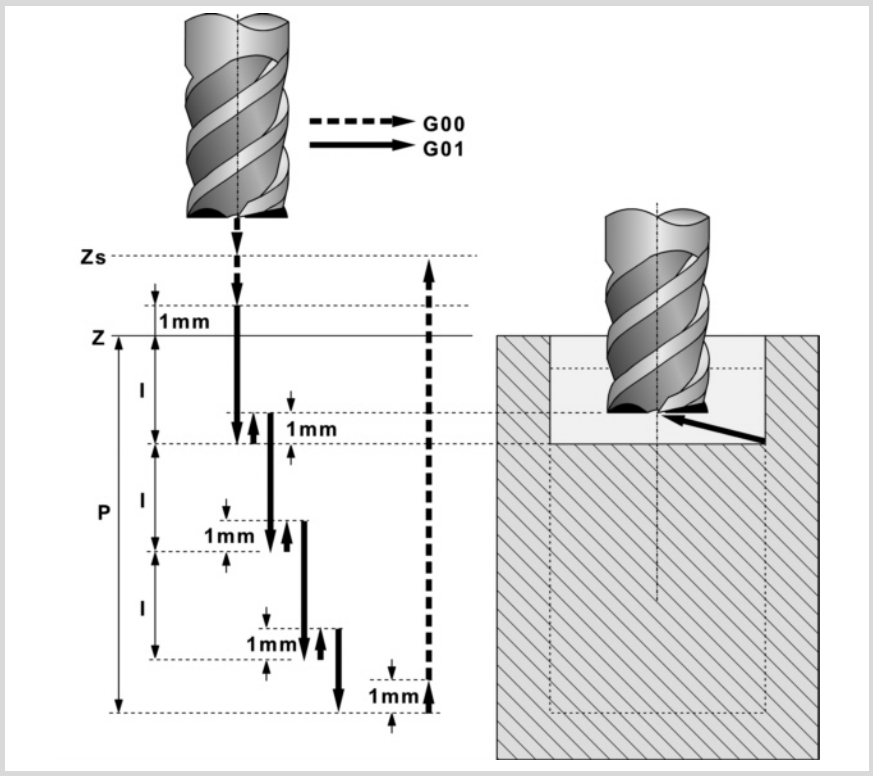
Activating or deactivating the preparation of the next tool.

The editor will only show this option if the user has configured the editor to allow programming the next tool.

6.1.1 Basic operation.

1 It starts the spindle in the requested direction.

2 Rapid movement (G0) to the center of the pocket and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.



3 Rapid movement (G0) up to the approach plane.

4 First penetration, the "Fz" feedrate, the amount "I".

5 Milling of the pocket surface.

Roughing is carried out at feedrate "F" with the passes defined by "Δ" and up to a distance "δ" from the pocket wall. The finishing pass "δ" is carried out with tangential entry and exit and at feedrate "F".

6 Rapid withdrawal (G0) to the center of the pocket in the approach plane.

7 New milling surfaces until reaching the total depth of the pocket.

Penetration, at the feedrate indicated in "Fz" up to a distance "I" from the previous surface. Milling of the new surface following the steps indicated in points 5 and 6.

8 Rapid withdrawal (G0) up to the safety plane (Zs).

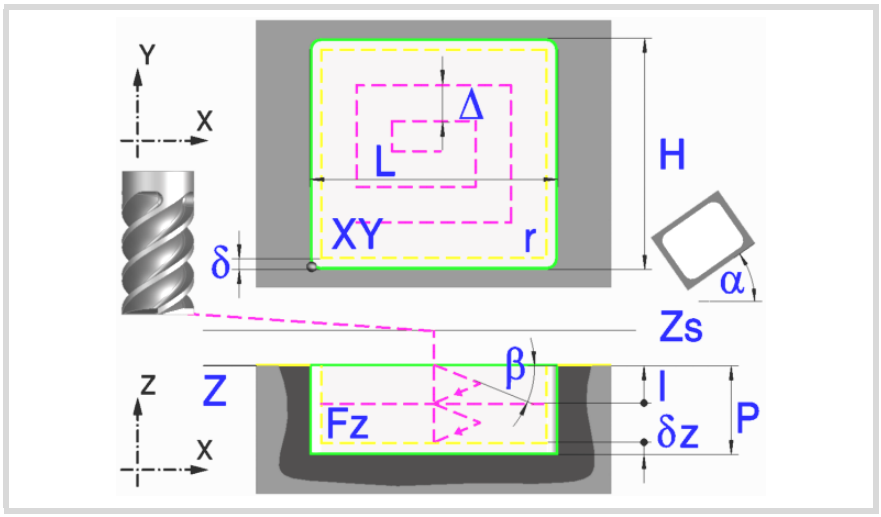
If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

9 Rapid movement (G0) to the next point.

10 Repeats steps 3, 4, 5, 6, 7, 8.

6.2 Rectangular pocket with rounding.

The way the roughing and finishing blocks of this cycle are joined will be the one previously set by the user with the instructions #HSC, G5, G50 or G7. We recommend to use #HSC or G5 controlling the shape of the corner with the instruction #ROUNDPAR.



Geometric parameters:

X, Y Coordinates of the starting point of the pocket.

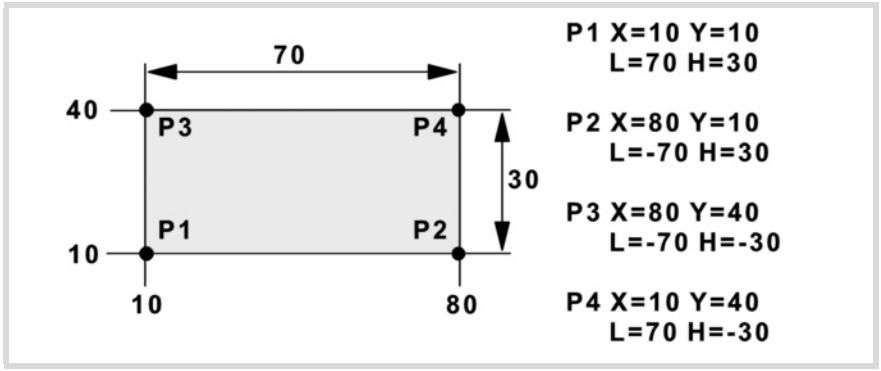
Starting point of the pocket (icon).

Starting point at one corner of the pocket.

Starting point at the center of the pocket.

L, H Pocket dimensions.

When the starting point of the pocket is one of its corners, the sign indicates the orientation with respect to the XY point.



Z Part surface coordinate.

Zs Safety plane coordinate.

P Total depth.

α Angle, in degrees, between the pocket and the abscissa axis. The turn is carried out on the defined corner, X,Y point.

Type of corner (icon).

Square corner with icon.

Rounded corner with icon.

Chamfered corner with icon.

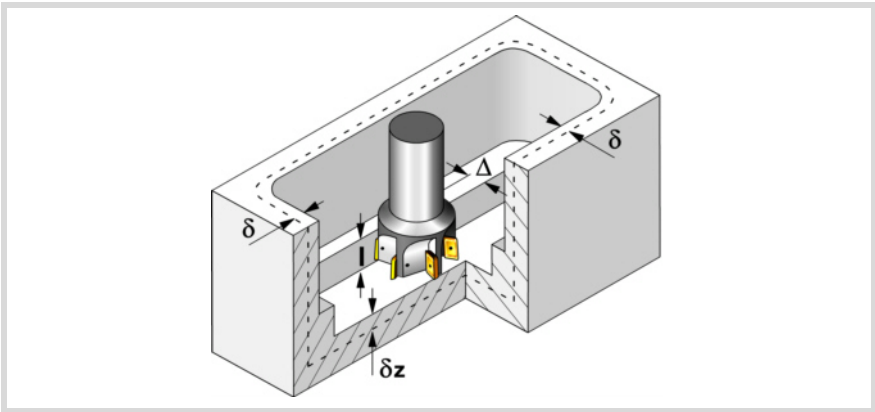
r Rounding radius or chamfer size.

Roughing parameters:

The roughing operation empties the pocket leaving the following finishing stocks. Both stocks are defined as finishing parameters.

δ Finishing stock on the side walls.

δz Finishing stock at the bottom of the pocket.



The roughing operation defining parameters are:

Δ Maximum milling pass or width.

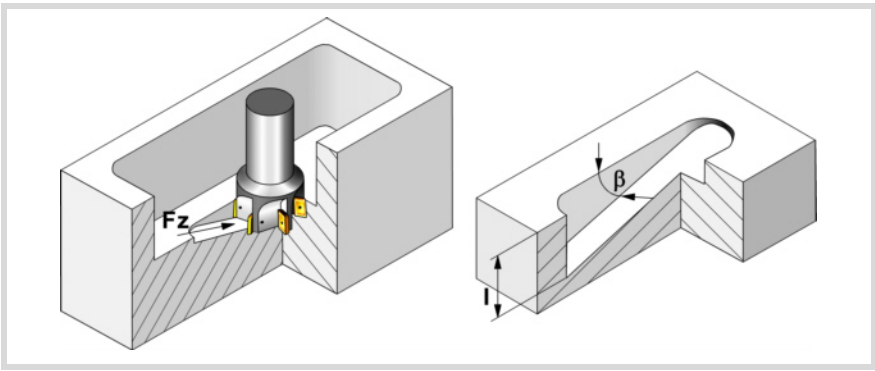
The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

I Penetration step.

- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the pocket is machined with the given pass (step) except the last pass that machines the rest.
- When programming with a 0 value, the cycle assumes the step as the cutting length assigned to the tool in the tool table (Lc data). If the cutting length defined by the tool table is also set at 0, the cycle executes the pocket in a single penetration.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

Fz Penetration feedrate. If programmed with a 0 value, the cycle uses half of the programmed F feedrate.



β Penetrating angle.

The penetration is carried out in zigzag, starting and ending at the center of the pocket.
If defined with a value greater than the one assigned to the tool in the tool table, it assumes the table value. If programmed with a 0 value, the cycle assumes the value assigned to the tool in the tool table (Ae data). If the table value is also 0, they cycle will perform a vertical penetration, without any inclination (90° angle).

F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

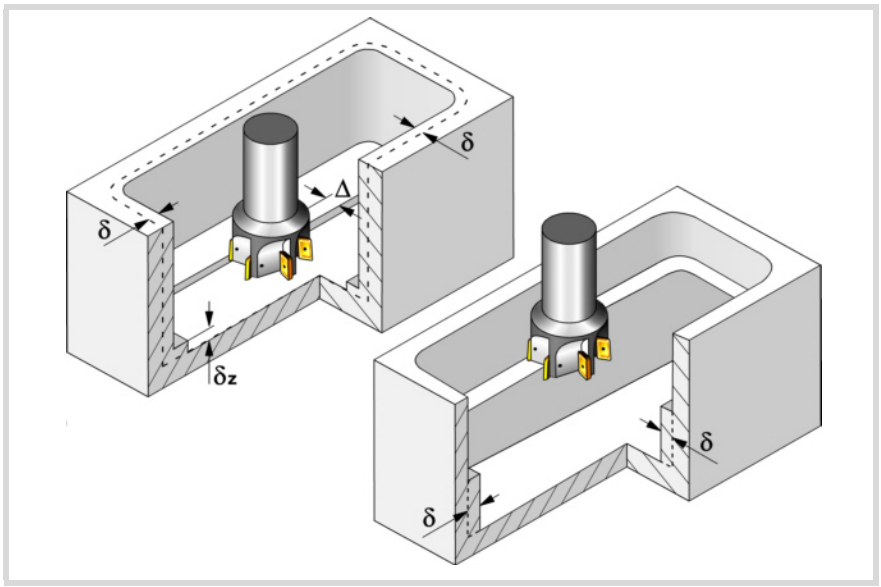
Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Finishing parameters:

The finishing operation is carried out in two stages. First, it machines the bottom of the pocket and then the side walls, with tangential entry and exit.



The finishing operation defining parameters are:

δ Finishing stock on the side walls.

δz Finishing stock at the bottom of the pocket.

Δ Milling pass or width at the bottom of the pocket.

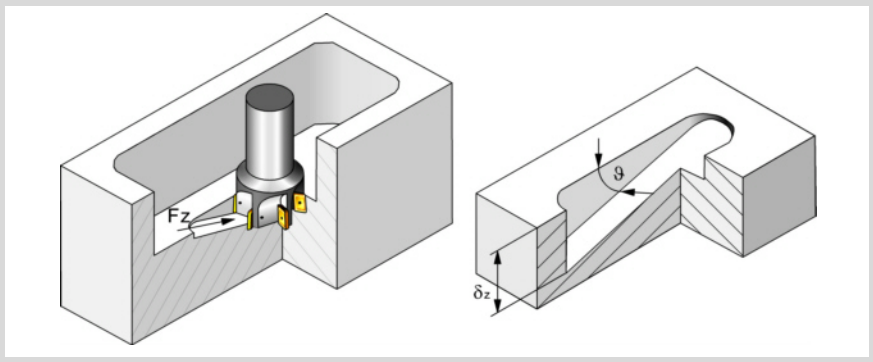
The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

N Number of penetration passes (steps) for the side finishing. If the resulting step is greater than the cutting length assigned to the table in the tool table, the step will be limited to that value. When programming with a 0 value, it carries out the least amount of passes possible, considering the cutting length assigned to the tool in the tool table (Lc data). If the table value is also 0, they cycle checks the roughing and finishing tools.

- If both tools are the same, the wall finishing is carried out with tangential entry and exit at each penetration after the roughing operation.
- If both tools are different, the CNC will prompt the corresponding error message.

θ Penetrating angle.

The penetration is carried out at the feedrate set by roughing parameter "Fz" starting and ending at the center of the pocket. If defined with a value greater than the one assigned to the tool in the tool table, it assumes the table value. If programmed with a 0 value, the cycle assumes the value assigned to the tool in the tool table (Ae data). If the table value is also 0, they cycle will perform a vertical penetration, without any inclination (90° angle).



F Surface and side milling feedrate.

S Spindle speed.

T Finishing tool.

If programmed T=0, there is no finishing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

Activating or deactivating the preparation of the next tool.

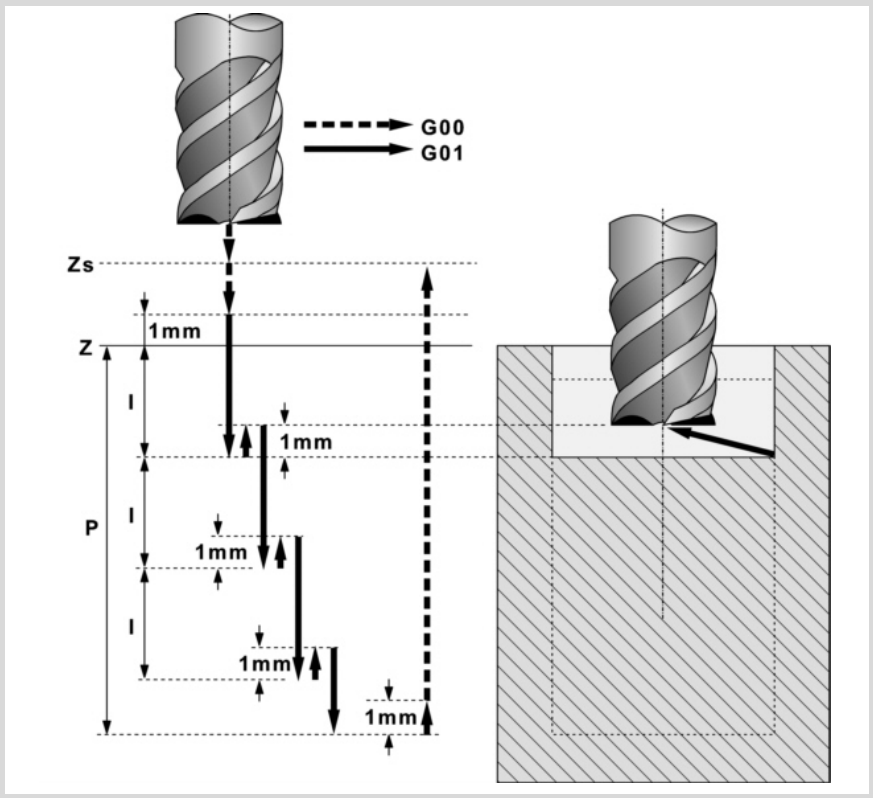
The editor will only show this option if the user has configured the editor to allow programming the next tool.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

6.2.1 Basic operation.

1 It selects the roughing tool and starts the spindle in the requested direction.

2 Rapid movement (G0) up to the safety plane (Zs) positioning at the center of the pocket. Depending on the tool position, it first moves in XY and then in Z or vice versa.



3 Rapid movement (G0) up to the approach plane.

4 Roughing operation. It is carried out in layers, until reaching the total depth minus the finishing stock at the bottom "δz".

First, penetration "I" at feedrate "Fz" at an angle "β". Then, milling of the pocket surface up to a distance "δ" from the pocket wall. It is carried out at feedrate "F" and, if necessary, it recalculates the pass (Δ) so all the passes are identical. And last, rapid withdrawal (G0) to the center of the pocket, 1 mm off the machined surface.

5 Rapid withdrawal (G0) up to the safety plane (Zs).

6 It selects the finishing tool and it approaches in rapid (G0) down to 1 mm from the roughed-out bottom.

7 Finishing of the bottom of the pocket.

Penetration at feedrate "Fz" at an angle "θ". Milling of the bottom of the pocket up to a distance "δ" from the pocket wall. It is carried out at finishing feedrate "F" and, if necessary, it recalculates the finishing pass (Δ) so all the passes are identical.

8 Withdrawal, in rapid (G0), to the center of the pocket in the approach plane (1 mm off the "Z" surface).

9 Finishing of the side walls. The finishing operation is carried out in "N" passes at the finishing feedrate "F" and with tangential entry and exit.

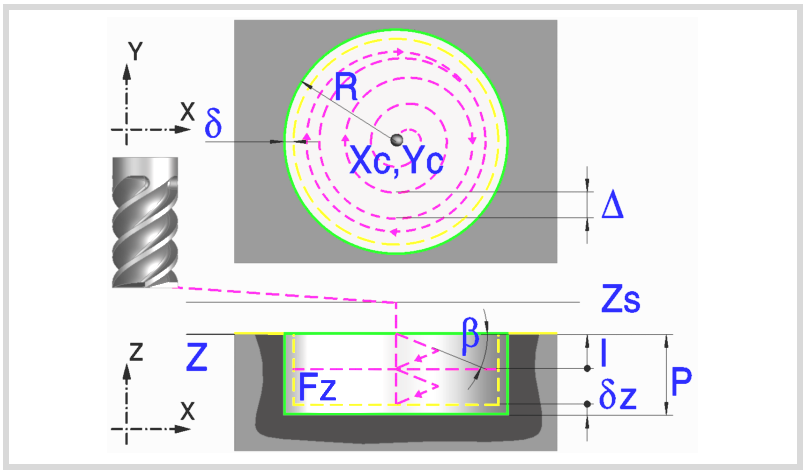
10 Rapid withdrawal (G0) to the center of the pocket in the safety plane (Zs).

If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

11 Rapid movement (G0) to the next point.

12 Repeats steps 3, 4, 5, 6, 7, 8, 9, 10.

6.3 Circular pocket.



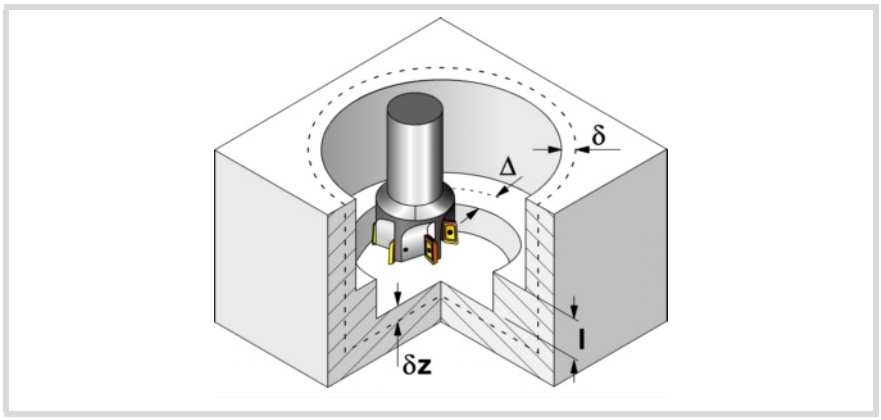
Geometric parameters:

Xc, Yc	Center of the pocket.
R	Pocket radius.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
P	Total depth.

Roughing parameters:

The roughing operation empties the pocket leaving the following finishing stocks. Both stocks are defined as finishing parameters.

δ	Finishing stock on the side walls.
δz	Finishing stock at the bottom of the pocket.



The roughing operation defining parameters are:

Δ Maximum milling pass or width.

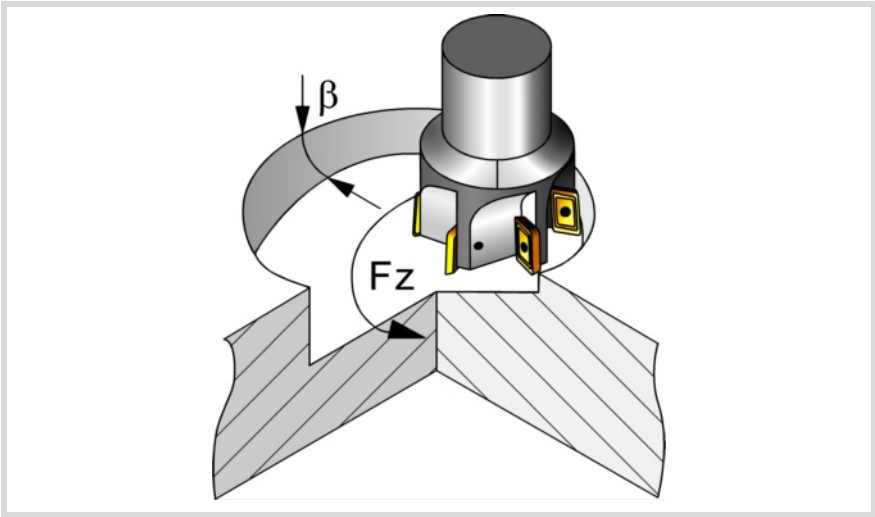
The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

I Penetration step.

- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the pocket is machined with the given pass (step) except the last pass that machines the rest.
- When programming with a 0 value, the cycle assumes the step as the cutting length assigned to the tool in the tool table (Lc data). If the cutting length defined by the tool table is also set at 0, the cycle executes the pocket in a single penetration.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

Fz Penetration feedrate. If programmed with a 0 value, the cycle uses half of the programmed F feedrate.



β Penetrating angle.

The penetration is carried out along a helical path, starting and ending at the center of the pocket. If defined with a value greater than the one assigned to the tool in the tool table, it assumes the table value. If programmed with a 0 value, the cycle assumes the value assigned to the tool in the tool table (Ae data). If the table value is also 0, they cycle will perform a vertical penetration, without any inclination (90° angle).

F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

Counterclockwise.

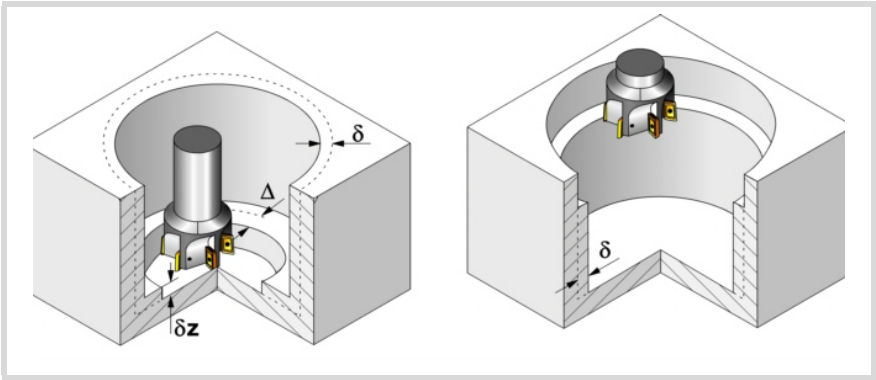
Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Finishing parameters:

The finishing operation is carried out in two stages. First, it machines the bottom of the pocket and then the side walls, with tangential entry and exit.



The finishing operation defining parameters are:

δ Finishing stock on the side walls.

δz Finishing stock at the bottom of the pocket.

Δ Milling pass or width at the bottom of the pocket.

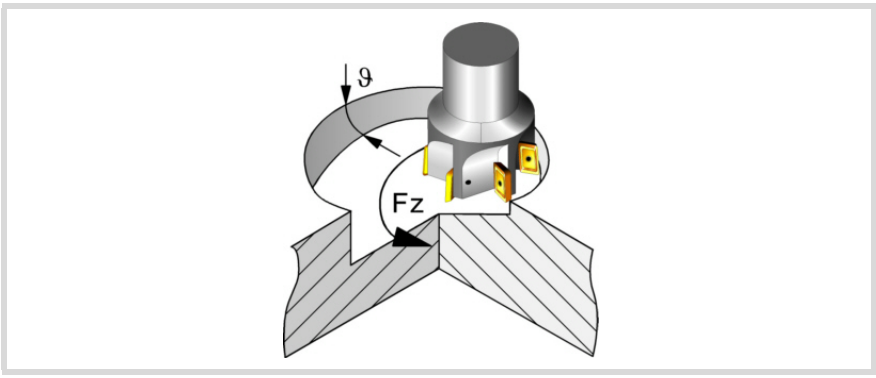
The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

N Number of penetration passes (steps) for the side finishing. If the resulting step is greater than the cutting length assigned to the table in the tool table, the step will be limited to that value. When programming with a 0 value, it carries out the least amount of passes possible, considering the cutting length assigned to the tool in the tool table (Lc data). If the table value is also 0, they cycle checks the roughing and finishing tools.

- If both tools are the same, the wall finishing is carried out with tangential entry and exit at each penetration after the roughing operation.
- If both tools are different, the CNC will prompt the corresponding error message.

θ Penetrating angle.

The penetration is carried out along a helical path at the feedrate set by roughing parameter "Fz" starting and ending at the center of the pocket. If defined with a value greater than the one assigned to the tool in the tool table, it assumes the table value. If programmed with a 0 value, the cycle assumes the value assigned to the tool in the tool table (Ae data). If the table value is also 0, they cycle will perform a vertical penetration, without any inclination (90° angle).



F Surface and side milling feedrate.

S Spindle speed.

T Finishing tool.

If programmed T=0, there is no finishing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

Activating or deactivating the preparation of the next tool.

The editor will only show this option if the user has configured the editor to allow programming the next tool.

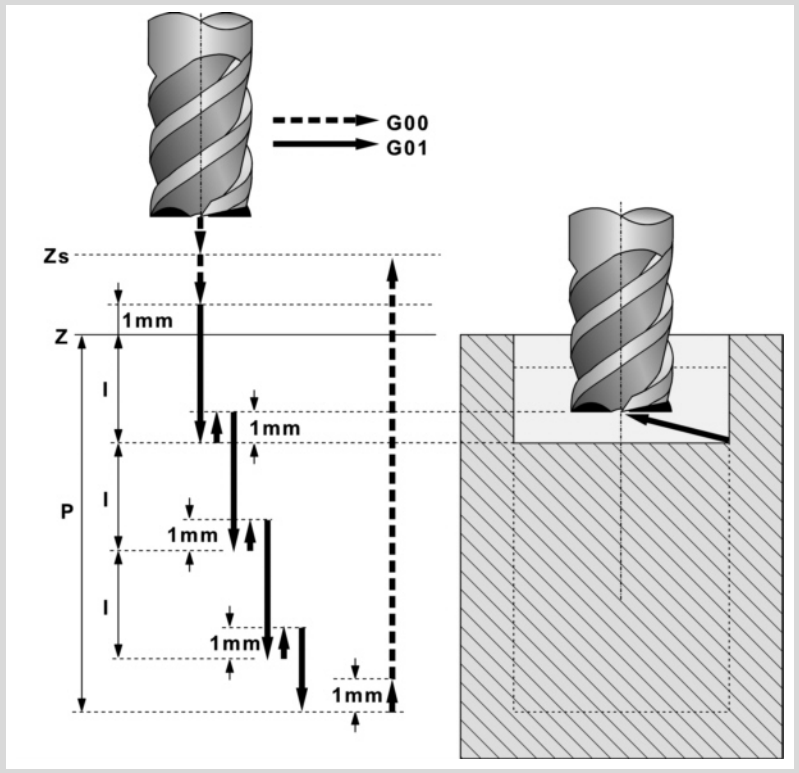
FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

6.3.1 Basic operation.

1 It selects the roughing tool and starts the spindle in the requested direction.

2 Rapid movement (G0) to the center of the pocket and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.

3 Rapid movement (G0) up to the approach plane.



4 Roughing operation.

It is carried out in layers, until reaching the total depth minus the finishing stock at the bottom "δz".

First, penetration "I" at feedrate "Fz" at an angle "β". Then, milling of the pocket surface up to a distance "δ" from the pocket wall. It is carried out at feedrate "F" and, if necessary, it recalculates the pass (Δ) so all the passes are identical. And last, rapid withdrawal (G0) to the center of the pocket, 1 mm off the machined surface.

5 Rapid withdrawal (G0) up to the safety plane (Zs).

6 It selects the finishing tool and it approaches in rapid (G0) down to 1 mm from the roughed-out bottom.

7 Finishing of the bottom of the pocket.

Penetration at feedrate "Fz" at an angle "θ". Milling of the bottom of the pocket up to a distance "δ" from the pocket wall. It is carried out at finishing feedrate "F" and, if necessary, it recalculates the finishing pass (Δ) so all the passes are identical.

8 Rapid withdrawal (G0) to the center of the pocket in the approach plane.

9 Finishing of the side walls. The finishing operation is carried out in "N" passes at the finishing feedrate "F" and with tangential entry and exit.

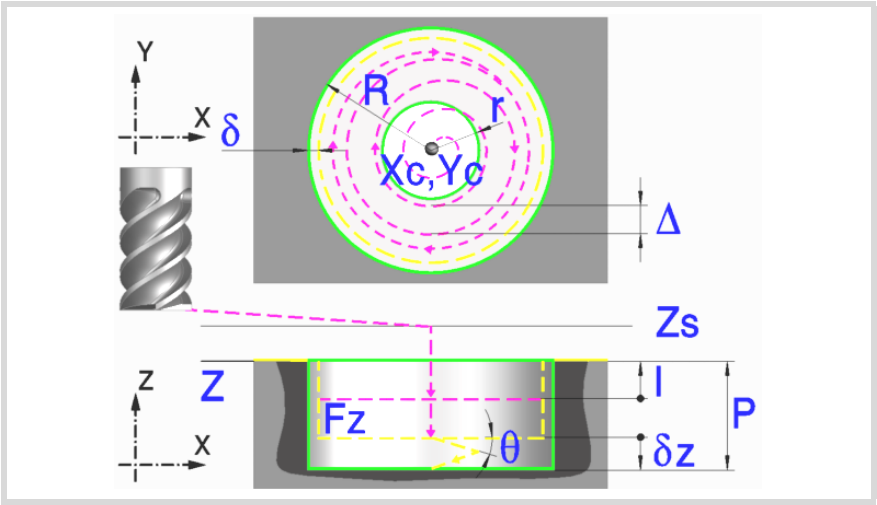
10 Rapid withdrawal (G0) to the center of the pocket in the safety plane (Zs).

If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

11 Rapid movement (G0) to the next point.

12 Repeats steps 3, 4, 5, 6, 7, 8, 9, 10.

6.4 Pre-emptied circular pocket.



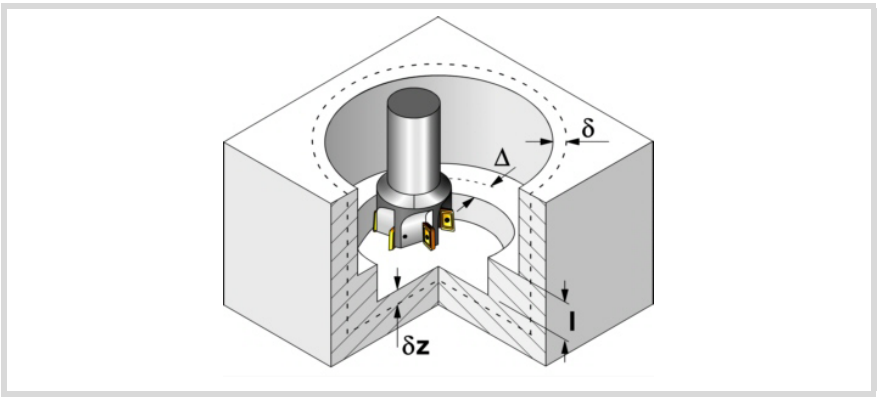
Geometric parameters:

Xc, Yc	Center of the pocket.
R	Pocket radius.
r	Pre-emptying radius.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
P	Total depth.

Roughing parameters:

The roughing operation empties the pocket leaving the following finishing stocks. Both stocks are defined as finishing parameters.

δ	Finishing stock on the side walls.
δz	Finishing stock at the bottom of the pocket.



The roughing operation defining parameters are:

Δ	Maximum milling pass or width. The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

I Penetration step.

- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the pocket is machined with the given pass (step) except the last pass that machines the rest.
- When programming with a 0 value, the cycle assumes the step as the cutting length assigned to the tool in the tool table (Lc data). If the cutting length defined by the tool table is also set at 0, the cycle executes the pocket in a single penetration.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

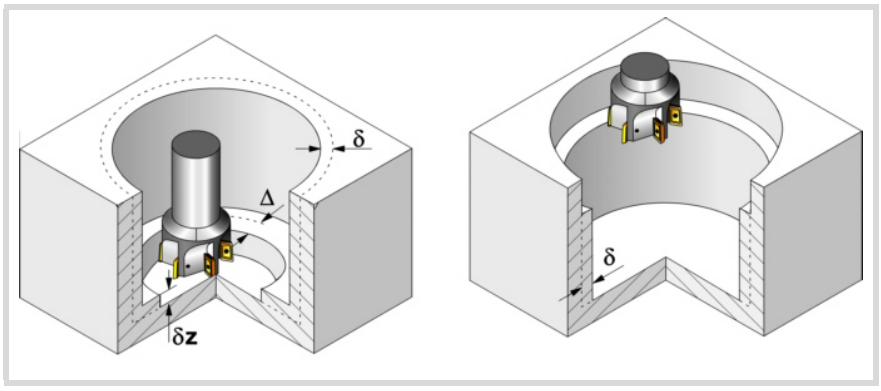
Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Finishing parameters:

The finishing operation is carried out in two stages. First, it machines the bottom of the pocket and then the side walls, with tangential entry and exit.



The finishing operation defining parameters are:

δ Finishing stock on the side walls.

δz Finishing stock at the bottom of the pocket.

Δ Milling pass or width at the bottom of the pocket.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

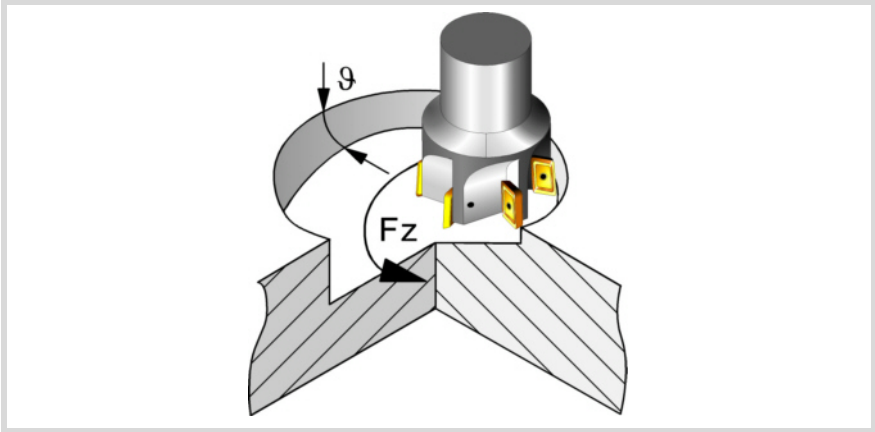
N Number of penetration passes (steps) for the side finishing. If the resulting step is greater than the cutting length assigned to the table in the tool table, the step will be limited to that value. When programming with a 0 value, it carries out the least amount of passes possible, considering the cutting length assigned to the tool in the tool table (Lc data). If the table value is also 0, they cycle checks the roughing and finishing tools.

- If both tools are the same, the wall finishing is carried out with tangential entry and exit at each penetration after the roughing operation.
- If both tools are different, the CNC will prompt the corresponding error message.

Fz Penetration feedrate. If programmed with a 0 value, the cycle uses half of the programmed F feedrate.

θ Penetrating angle.

The penetration is carried out along a helical path at the feedrate set by finishing parameter "Fz" starting and ending at the center of the pocket. If defined with a value greater than the one assigned to the tool in the tool table, it assumes the table value. If programmed with a 0 value, the cycle assumes the value assigned to the tool in the tool table (Ae data). If the table value is also 0, they cycle will perform a vertical penetration, without any inclination (90° angle).



F Surface and side milling feedrate.

S Spindle speed.

T Finishing tool.

If programmed T=0, there is no finishing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

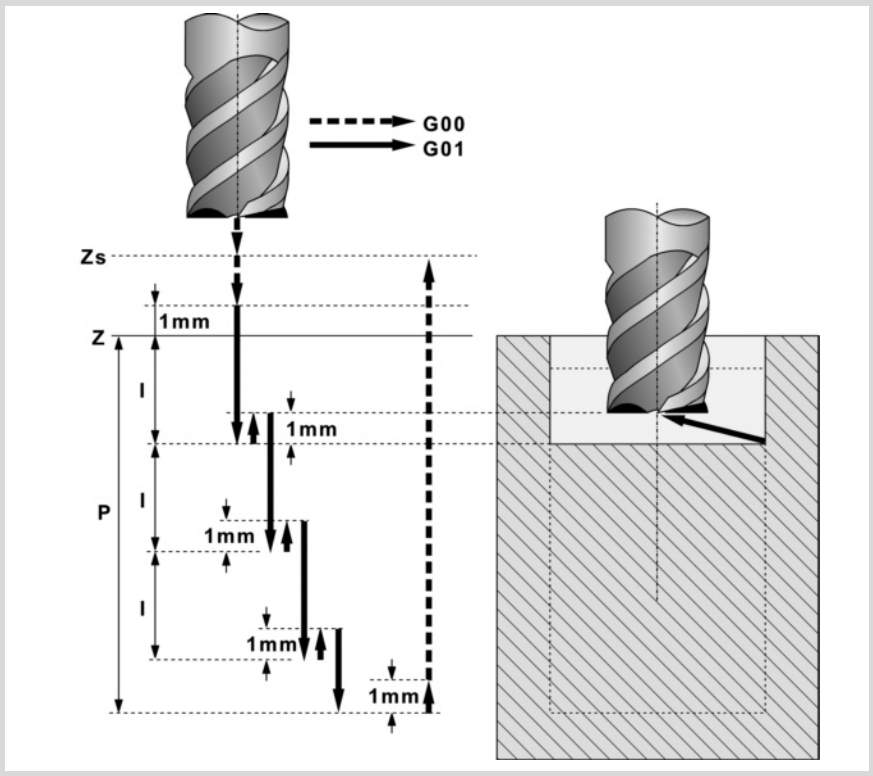
Activating or deactivating the preparation of the next tool.

The editor will only show this option if the user has configured the editor to allow programming the next tool.

6.4.1 Basic operation.

1 It selects the roughing tool and starts the spindle in the requested direction.

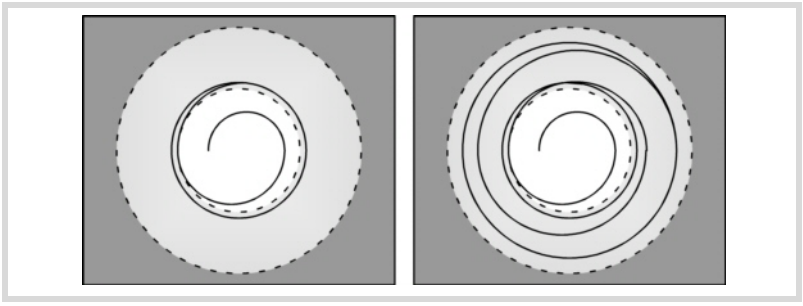
2 Rapid movement (G0) to the center of the pocket and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.



3 Rapid movement (G0) up to the approach plane.

4 Roughing operation. It is carried out in layers, until reaching the total depth minus the finishing stock at the bottom "δz".

First, "I" penetration and approach to the pre-emptied side with tangential entry. Then, milling of the pocket surface up to a distance "δ" from the pocket wall. It is carried out at feedrate "F" and, if necessary, it recalculates the pass (Δ) so all the passes are identical. And last, rapid withdrawal (G0) to the center of the pocket, 1 mm off the machined surface.



5 Rapid withdrawal (G0) up to the safety plane (Zs).

6 It selects the finishing tool and it approaches in rapid (G0) down to 1 mm from the roughed-out bottom.

7 Finishing of the bottom of the pocket.

Penetration at feedrate "Fz" at an angle "θ". Milling of the bottom of the pocket up to a distance "δ" from the pocket wall. It is carried out at finishing feedrate "F" and, if necessary, it recalculates the finishing pass (Δ) so all the passes are identical.

8 Withdrawal, in rapid (G0), to the center of the pocket in the approach plane (1 mm off the "Z" surface).

9 Finishing of the side walls. The finishing operation is carried out in "N" passes at the finishing feedrate "F" and with tangential entry and exit.

10 Rapid withdrawal (G0) to the center of the pocket in the safety plane (Zs).

If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

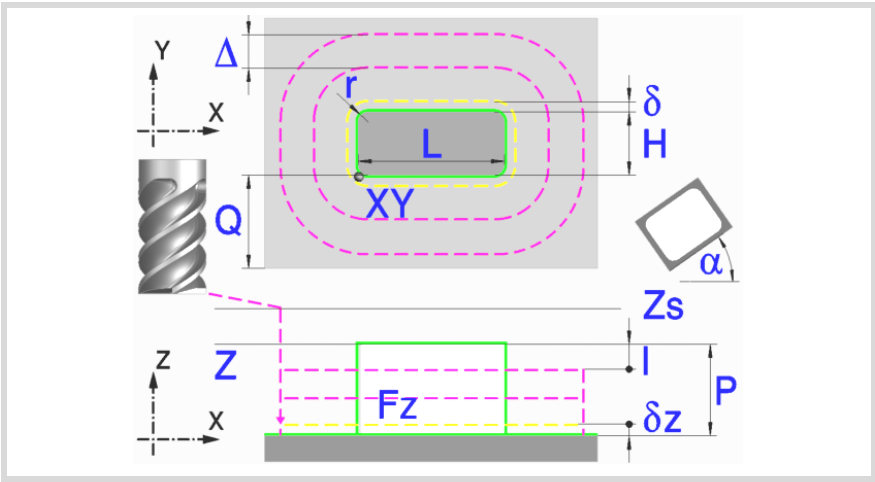
11 Rapid movement (G0) to the next point.

12 Repeats steps 3, 4, 5, 6, 7, 8, 9, 10.

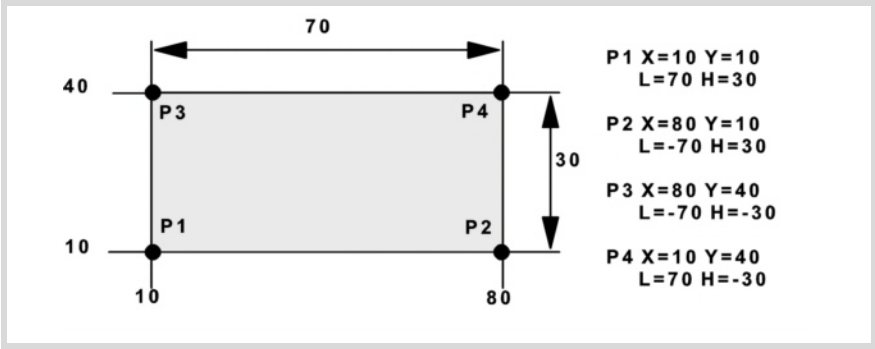
FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

6.5 Rectangular boss.

The way the roughing and finishing blocks of this cycle are joined will be the one previously set by the user with the instructions #HSC, G5, G50 or G7. We recommend to use #HSC or G5 controlling the shape of the corner with the instruction #ROUNDPAR.



Geometric parameters:



X, Y	Corner of the boss.
L, H	Boss dimensions. The sign indicates the orientation referred to the XY point.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
P	Total depth.
α	Angle, in degrees, between the boss and the abscissa axis. The turn is carried out on the defined corner, X,Y point.
Q	Amount of stock to be removed.

Type of corner (icon).

Square corner with icon.

Rounded corner with icon.

Chamfered corner with icon.

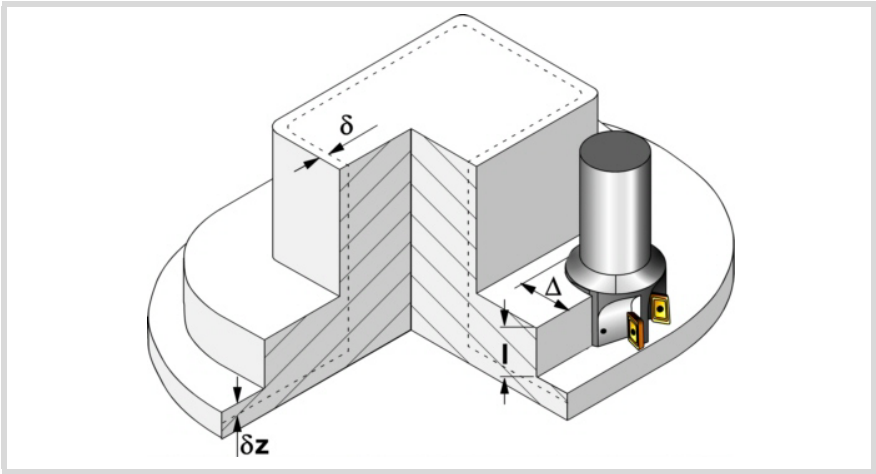
r	Rounding radius or chamfer size.

Roughing parameters:

The roughing operation machines the boss leaving the following finishing stocks. Both stocks are defined as finishing parameters.

δ Finishing stock on the side walls.

δz Finishing stock at the base of the boss.



The roughing operation defining parameters are:

Δ Maximum milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

I Penetration step.

- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the boss is machined with the given pass (step) except the last pass that machines the rest.
- When programming with a 0 value, the cycle assumes the step as the cutting length assigned to the tool in the tool table (Lc data). If the cutting length defined by the tool table is also set at 0, the cycle executes the pocket in a single penetration.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

Fz Penetration feedrate. If programmed with a 0 value, the cycle uses half of the programmed F feedrate.

F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

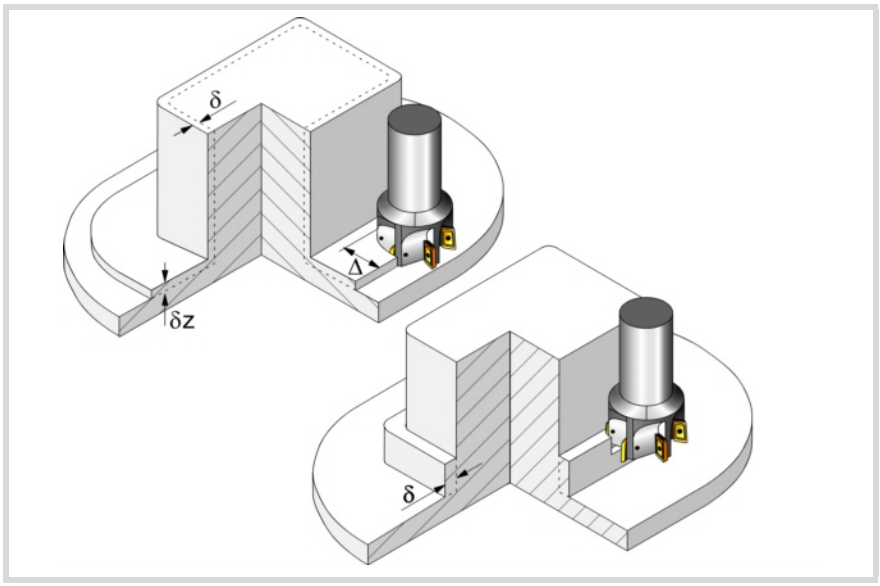
Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Finishing parameters:

The finishing operation is carried out in two stages. First, it machines the base of the boss and then the side walls, with tangential entry and exit.



The finishing operation defining parameters are:

δ	Finishing stock on the side walls.
δz	Finishing stock at the base of the boss.
N	Number of penetration passes (steps) for the side finishing. If the resulting step is greater than the cutting length assigned to the table in the tool table, the step will be limited to that value. When programming with a 0 value, it carries out the least amount of passes possible, considering the cutting length assigned to the tool in the tool table (Lc data). If the table value is also 0, they cycle checks the roughing and finishing tools. • If both tools are the same, the wall finishing is carried out with tangential entry and exit at each penetration after the roughing operation. • If both tools are different, the CNC will prompt the corresponding error message.
F	Surface and side milling feedrate.
S	Spindle speed.
T	Finishing tool. If programmed T=0, there is no finishing.
D	Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

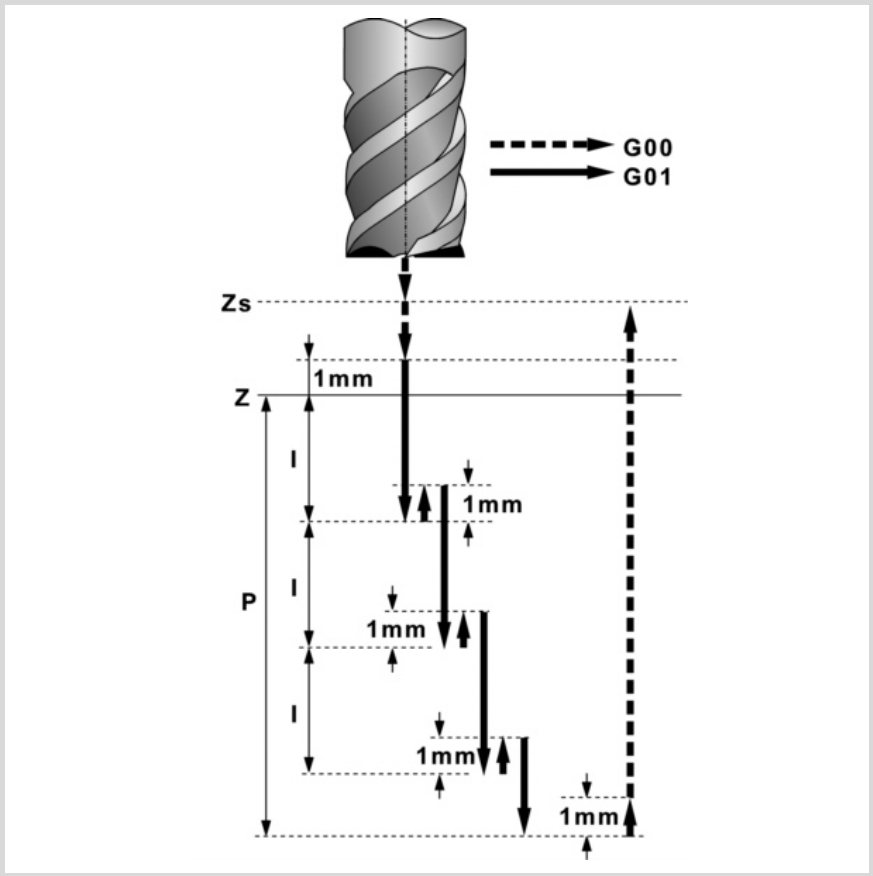
Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

6.5.1 Basic operation.

1 It selects the roughing tool and starts the spindle in the requested direction.

2 Rapid movement (G0) to the roughing starting point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.

3 Rapid movement (G0) up to the approach plane.



4 Roughing operation. It is carried out in layers, until reaching the total depth minus the finishing stock at the base "δz".

First, penetration "I" at feedrate "Fz". Then, milling of the boss surface up to a distance "δ" from the side wall. It is carried out at feedrate "F" and, if necessary, it recalculates the pass (Δ) so all the passes are identical. And last, rapid withdrawal (G0) to the starting point.

5 Rapid withdrawal (G0) up to the safety plane (Zs).

6 It selects the finishing tool and it approaches in rapid (G0) down to 1 mm from the last roughing operation.

7 Finishing of the base of the boss.

Penetration at feedrate "Fz". Milling of the base of the boss up to a distance "δ" from the side wall. It is carried out at the finishing feedrate "F" and with the roughing pass.

8 Rapid withdrawal (G0) to the starting point in the approach plane.

9 Finishing of the side walls. The finishing operation is carried out in "N" passes at the finishing feedrate "F" and with tangential entry and exit.

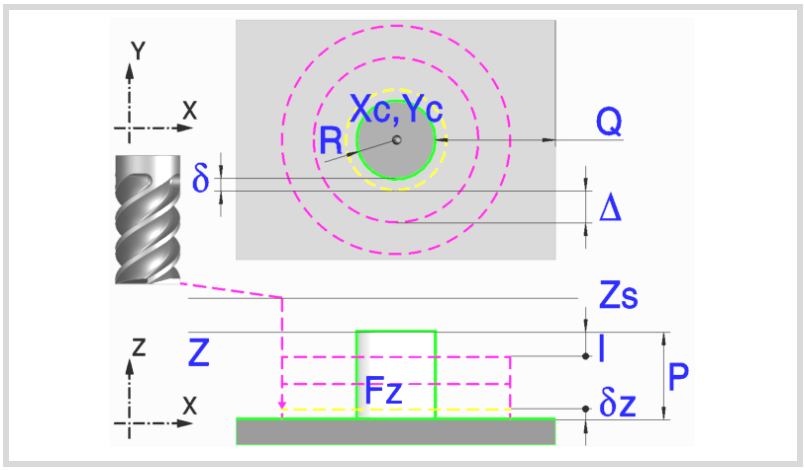
10 Rapid withdrawal (G0) up to the safety plane (Zs).

If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

11 Rapid movement (G0) to the next point.

12 Repeats steps 3, 4, 5, 6, 7, 8, 9, 10.

6.6 Circular boss.



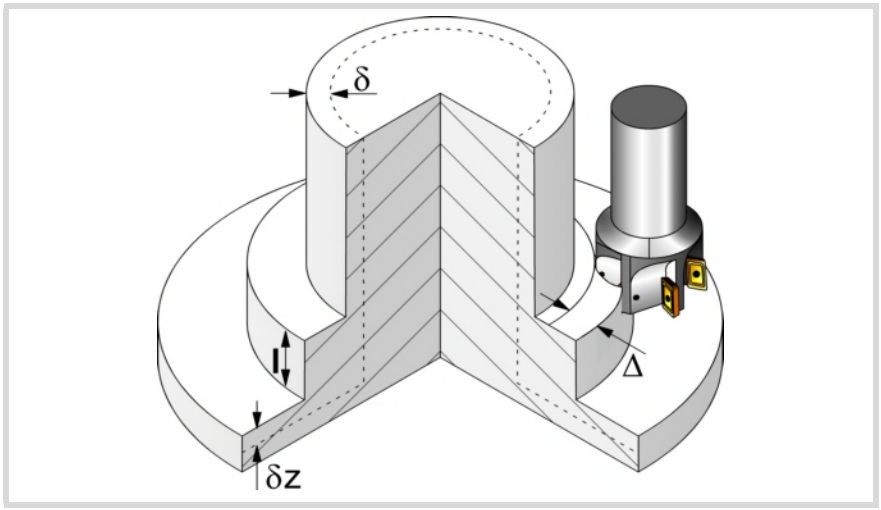
Geometric parameters:

Xc, Yc	Center of the boss.
R	Boss radius.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
P	Total depth.
Q	Amount of stock to be removed.

Roughing parameters:

The roughing operation machines the boss leaving the following finishing stocks. Both stocks are defined as finishing parameters.

δ	Finishing stock on the side walls.
δz	Finishing stock at the base of the boss.



The roughing operation defining parameters are:

Δ	Maximum milling pass or width. The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

I Penetration step.

- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the boss is machined with the given pass (step) except the last pass that machines the rest.
- When programming with a 0 value, the cycle assumes the step as the cutting length assigned to the tool in the tool table (Lc data). If the cutting length defined by the tool table is also set at 0, the cycle executes the pocket in a single penetration.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

Fz Penetration feedrate. If programmed with a 0 value, the cycle uses half of the programmed F feedrate.

F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

Counterclockwise.

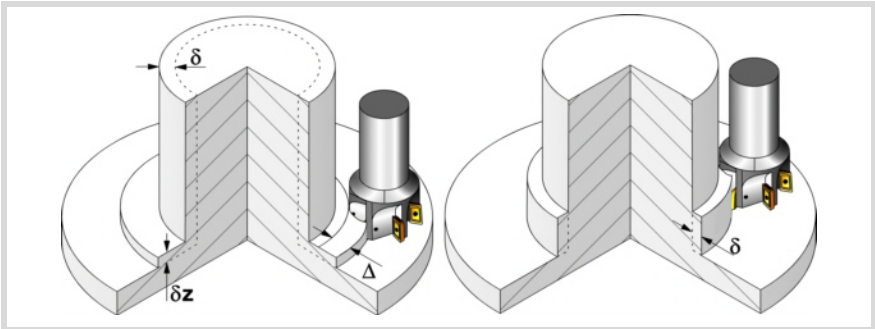
Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Finishing parameters:

The finishing operation is carried out in two stages. First, it machines the base of the boss and then the side walls, with tangential entry and exit.



The finishing operation defining parameters are:

δ Finishing stock on the side walls.

δz Finishing stock at the base of the boss.

N Number of penetration passes (steps) for the side finishing. If the resulting step is greater than the cutting length assigned to the table in the tool table, the step will be limited to that value. When programming with a 0 value, it carries out the least amount of passes possible, considering the cutting length assigned to the tool in the tool table (Lc data). If the table value is also 0, they cycle checks the roughing and finishing tools.

- If both tools are the same, the wall finishing is carried out with tangential entry and exit at each penetration after the roughing operation.
- If both tools are different, the CNC will prompt the corresponding error message.

F Surface and side milling feedrate.

S Spindle speed.

T Finishing tool.

If programmed T=0, there is no finishing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

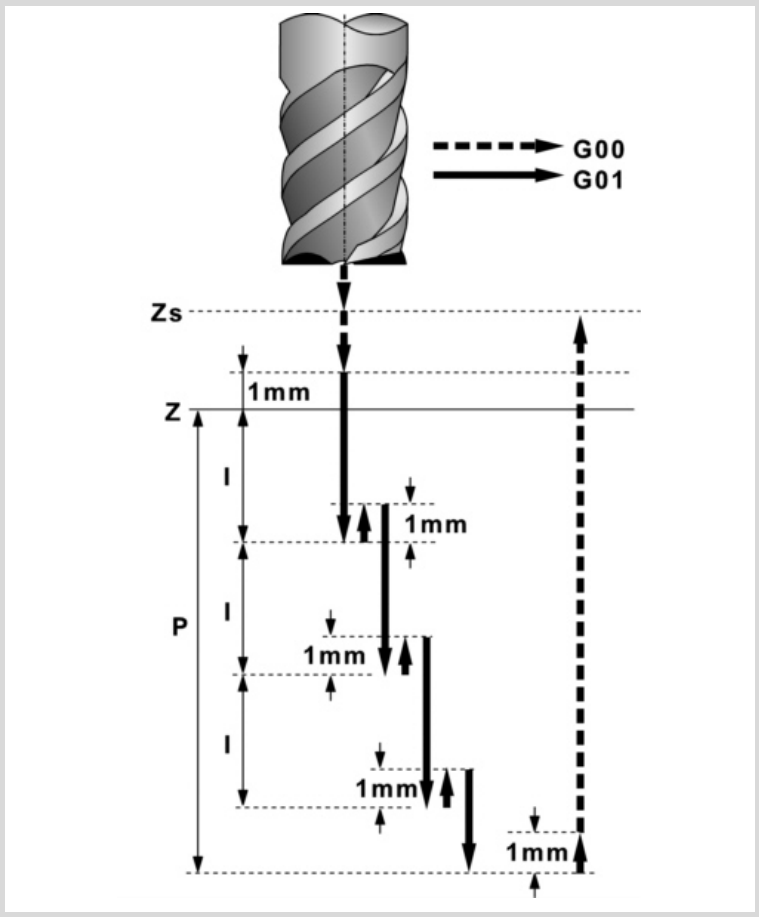
Next tool.

Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

6.6.1 Basic operation.

1 It selects the roughing tool and starts the spindle in the requested direction.

2 Rapid movement (G0) to the roughing starting point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.



3 Rapid approach (G0) up to 1 mm off the surface "Z".

4 Roughing operation. It is carried out in layers, until reaching the total depth minus the finishing stock at the base "δz".

First, penetration "I" at feedrate "Fz". Then, milling of the boss surface up to a distance "δ" from the side wall. It is carried out at feedrate "F" and, if necessary, it recalculates the pass (Δ) so all the passes are identical. And last, rapid withdrawal (G0) to the starting point.

5 Rapid withdrawal (G0) up to the safety plane (Zs).

6 It selects the finishing tool and it approaches in rapid (G0) down to 1 mm from the last roughing operation.

7 Finishing of the base of the boss.

Penetration at feedrate "Fz". Milling of the base of the boss up to a distance "δ" from the side wall. It is carried out at the finishing feedrate "F" and with the roughing pass.

8 Rapid withdrawal (G0) to the starting point in the approach plane.

9 Finishing of the side walls. The finishing operation is carried out in "N" passes at the finishing feedrate "F" and with tangential entry and exit.

10 Rapid withdrawal (G0) up to the safety plane (Zs).

If it has a multiple machining operation associated with it, it executes the following steps as often as necessary:

11 Rapid movement (G0) to the next point.

12 Repeats steps 3, 4, 5, 6, 7, 8, 9, 10.

6.

CANNED CYCLES OF THE EDITOR. POCKETS / BOSSES.

Circular boss.

FAGOR

FAGOR AUTOMATION

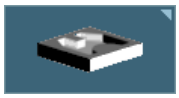
CNCelite
8058 8060
8065

REF. 2109

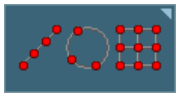
CANNED CYCLES OF THE EDITOR. 2D/3D PROFILE POCKETS.

7

When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



- 2D profile pocket.
- 3D profile pocket with islands.

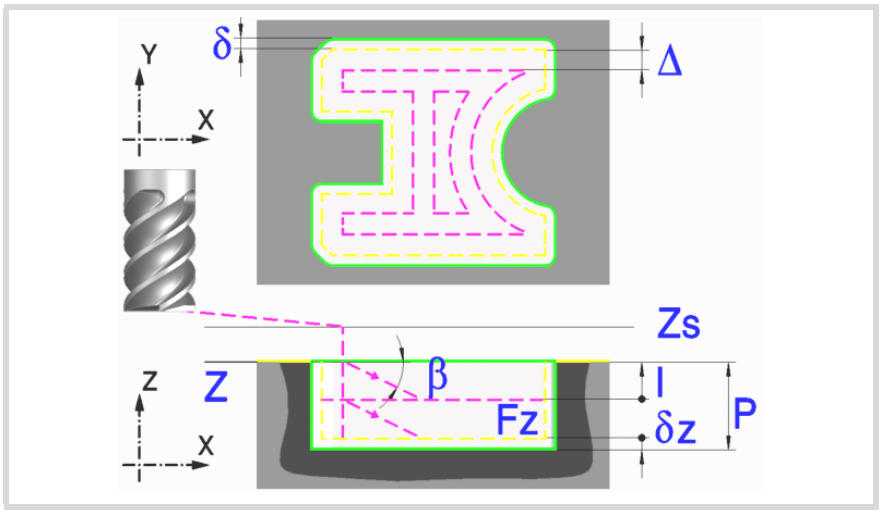


Multiple machining may be added to pocket cycles so the cycle is repeated in several points. The available multiple machining operations are the following. See chapter *"9 Canned cycles of the editor. Multiple positioning."*.

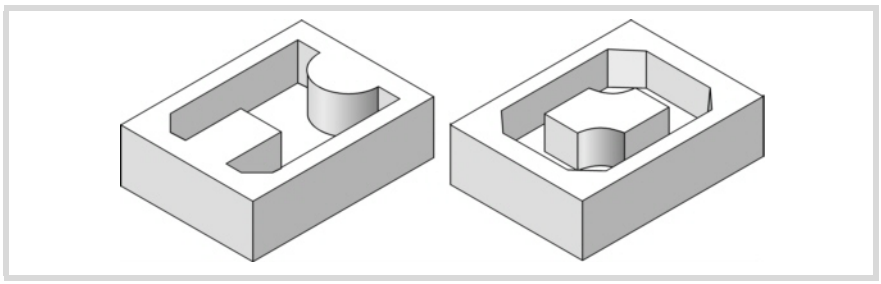
- Points in line.
- Points in arc.
- Points in rectangle.
- Points in grid.
- Random points (several points defined by the user).

To associate multiple machining to a cycle, first select and define a machining cycle from those allowed. Then, without quitting the editing of the cycle, press the softkey associated with multiple machining and select one of them.

7.1 2D profile pocket.



A pocket consists of an outside contour and a number of inside contours called islands. All the walls of 2D pockets are vertical.



It is recommended to previously define the #ROUNDPAR instruction in order to obtain a good finish because the finishing passes are carried out in G05.

Geometric parameters:

The composition of the pocket and the profile in the plane is stored in \ Cnc8070\ Users\ Profile.

pocket.P2D	Pocket composition.
profile.PXY	Plane profile.

P.2D	Name of the 2D pocket. Once the pocket configuration has been validated, the CNC associates the geometry of the pocket to its name.
P.XY	Name of the plane profile. The profile must indicate the pocket's outside contour and those of the islands.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
P	Total depth.

Drilling (icon).

It indicates whether drilling(a) takes place before machining the pocket or not. It should be used when the roughing tool cannot machine downwards.

Press the "Drilling" softkey to access the drilling cycle and after defining it, press the "End" softkey to return to the 2D pocket cycle.

The cycle calculates the drilling point, depending on the programmed profile.

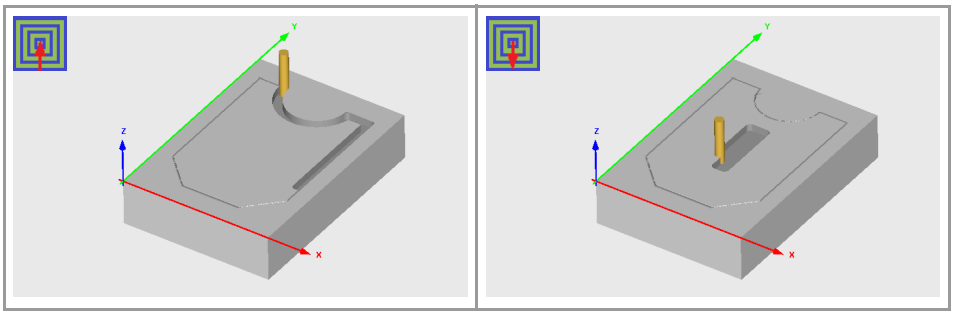
Machining strategy (icon).

Outward-inward concentric machining. The wall of the pocket is machined in the first pass. Depending on the geometry, this option may be faster, but the wear on the tool is greater.

- The starting point is near the initial point of the outer contour, and varies very little if the radius of the roughing tool is changed and does not vary at all if the roughing pass is changed.
- If a drilling operation is programmed before the roughing (or the bottom finishing if there is no roughing operation), the CNC chooses the optimal drilling point, as close as possible to the initial point of the outer contour.

Inward-outward concentric machining. The wall of the pocket is machined in the last pass. Depending on the geometry, this option maybe slower, but there is less wear on the tool. The CNC machines the pocket, attempting to minimize the number of sections where the tool machines over the largest surface, the number of penetrations in part sections and the machining time.

- The starting point does not depend on the initial point of the outer contour and can vary if either the tool radius or the machining pass is changed (milling width). In a pocket without islands, the initial point is found near the center of the pocket. In a pocket with islands, where different areas can be formed that require roughing, the initial point will be located in the center of the largest area.
- If a drilling operation is programmed before the roughing (or the bottom finishing, if there is no roughing operation), the CNC chooses the optimal drilling points (there may be more than one point).

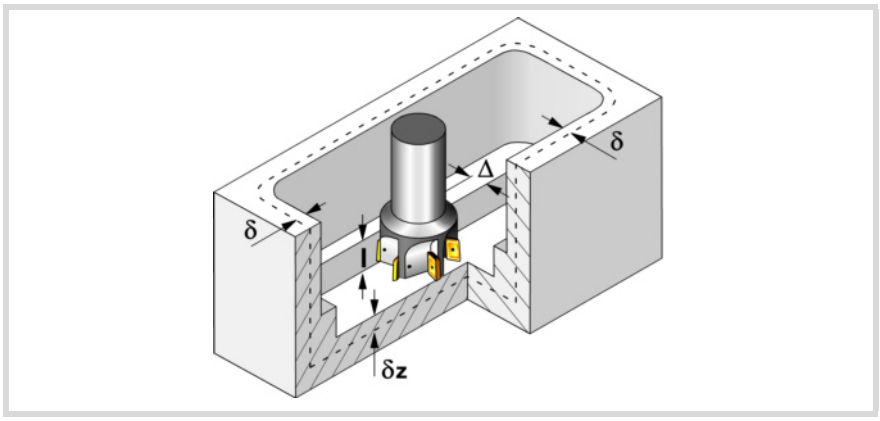


Roughing parameters:

The roughing operation empties the pocket leaving the following finishing stocks. Both stocks are defined as finishing parameters.

δ Finishing stock on the side walls.

δz Finishing stock at the bottom of the pocket.



The roughing operation defining parameters are:

Δ Maximum milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

I Penetration step.

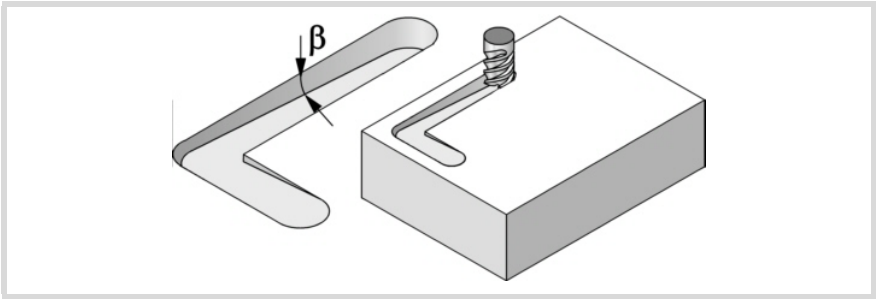
- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the pocket is machined with the given pass (step) except the last pass that machines the rest.
- When programming with a 0 value, the cycle assumes the step as the cutting length assigned to the tool in the tool table (Lc data). If the cutting length defined by the tool table is also set at 0, the cycle executes the pocket in a single penetration.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

Fz Penetration feedrate. If programmed with a 0 value, the cycle uses half of the programmed F feedrate.

β Penetrating angle.

The penetration is carried out maintaining this angle until the corresponding depth is reached. If defined with a value greater than the one assigned to the tool in the tool table, it assumes the table value. If programmed with a 0 value, the cycle assumes the value assigned to the tool in the tool table (Ae data). If the table value is also 0, they cycle will perform a vertical penetration, without any inclination (90° angle).



F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

For pockets with islands, the machining of the outer profile is done in the direction indicated by the icon, where as the islands are machined in the opposite direction.

Clockwise.

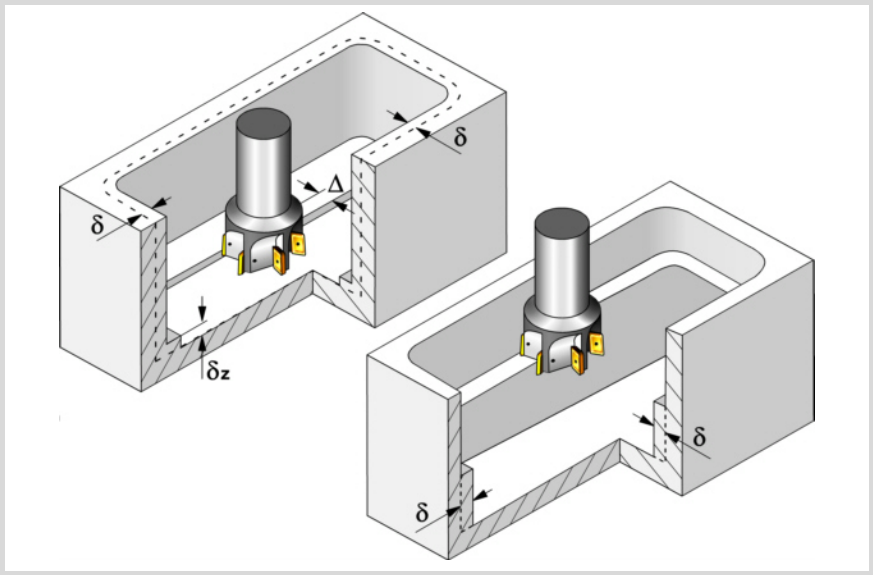
Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Finishing parameters:

The finishing operation is carried out in two stages. First, it machines the bottom of the pocket and then the side walls, with tangential entry and exit.



The finishing operation defining parameters are:

δ Finishing stock on the side walls.

δz Finishing stock at the bottom of the pocket.

Δ Milling pass or width at the bottom of the pocket.

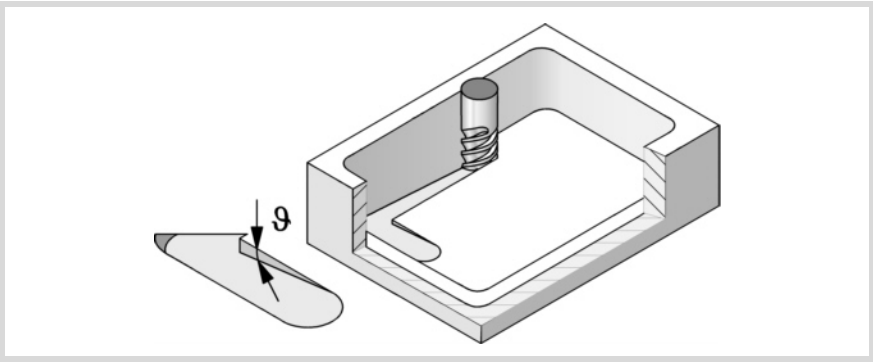
The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

N Number of penetration passes (steps) for the side finishing.

When programming a 0 value, it carries out the least amount of passes possible, considering the cutting length assigned to the tool in the tool table.

θ Penetrating angle.

The penetration is carried out at the feedrate set by roughing parameter "Fz" maintaining this angle until reaching the corresponding depth. If defined with a value greater than the one assigned to the tool in the tool table, it assumes the table value. If programmed with a 0 value, the cycle assumes the value assigned to the tool in the tool table (Ae data). If the table value is also 0, they cycle will perform a vertical penetration, without any inclination (90° angle).



F Surface and side milling feedrate.

S Spindle speed.

T Finishing tool.

If programmed T=0, there is no finishing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

For pockets with islands, the machining of the outer profile is done in the direction indicated by the icon, where as the islands are machined in the opposite direction.

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

Activating or deactivating the preparation of the next tool.

The editor will only show this option if the user has configured the editor to allow programming the next tool.

7.1.1 Executable pocket file

To simulate or execute this type of pockets, the CNC uses an executable file with geometry information. This file is generated the first time the pocket is simulated or executed. If from the editor, any data of the pocket geometry or the used tool, is modified, the CNC will generate this file again.

In versions prior to V2.00, the user generated the executable file from the editor before inserting the cycle. From version V2.00 on, it is no longer necessary, the CNC is in charge of generating the executable file when necessary.

The executable files are stored in the directory CNC8070 \Users \Pocket with the name of the pocket (parameter P.2D) and the extension C2D. These files must not be deleted, moved to another location or tampered with in any way. If when executing or simulating the pocket, the CNC cannot find these files, it will generate them.

Overall, a 2D pocket consists of the following files.

pocket.P2D	Pocket composition.
profile.PXY	Plane profile.
pocket.C2D	Executable file.

The executable file is also updated after a software update and when executing or simulating a pocket,

7.1.2 Basic operation.

The CNC calculates the initial coordinate, depending on the geometry of the pocket, the machining strategy and the tool radius.

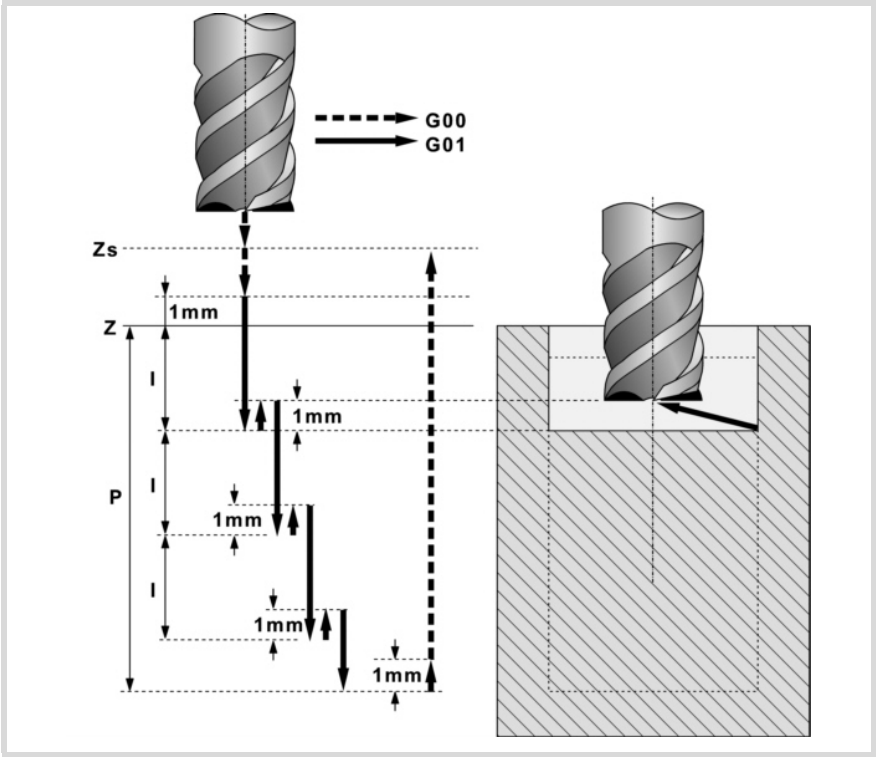
1 Drilling operation. Only if it has been programmed.

The CNC selects the optimal drilling point, as close as possible to the initial point of the outer contour.

The CNC selects the optimal drilling points (there may be more than one point).

2 It selects the roughing tool and starts the spindle in the requested direction.

3 Rapid movement (G0) to the roughing starting point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.



4 Rapid movement (G0) up to the approach plane.

5 Roughing operation. It is carried out in layers, until reaching the total depth minus the finishing stock at the bottom "δz".

First, penetration "I" at feedrate "Fz" at an angle "β". It then proceeds with the milling of the pocket surface up to a distance "δ" from the pocket wall, at feedrate "F".. And last, rapid withdrawal (G0) up to 1 mm off the machined surface.

The pocket is machined following paths concentric to the profile, in the programmed direction. The islands are machined in the opposite direction.

Outward-inward concentric machining. The wall of the pocket is machined in the first pass.

Inward-outward concentric machining. The wall of the pocket is machined in the last pass.

Clockwise.

Counterclockwise.

When the tool must go around or avoid an island, it increases in G0 up to the safety level (Zs). If the movement does not involve going around an island, the tool is raised to the working level + 1 mm.

6 Rapid withdrawal (G0) up to the safety plane (Zs).

7 It selects the finishing tool and it approaches in rapid (G0) down to 1 mm from the roughed-out bottom.

8 Finishing of the bottom of the pocket.

First, penetration at feedrate "Fz" at an angle "θ". It then proceeds with the milling of the bottom of the pocket up to a distance "δ" from the pocket wall, at feedrate "F".

The pocket is machined following paths concentric to the profile, in the programmed direction. The islands are machined in the opposite direction.

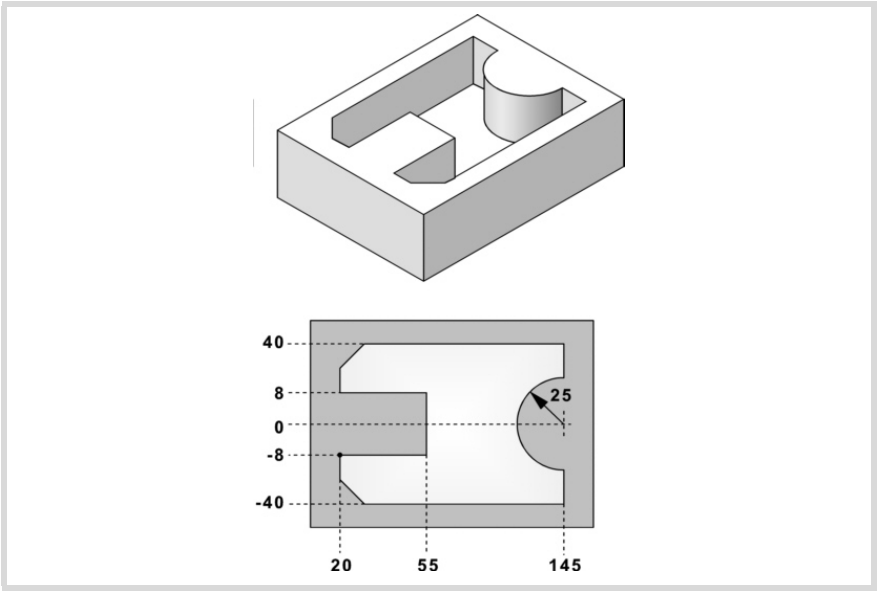
9 Rapid withdrawal (G0) up to the approach plane.

10 Finishing of the side walls. The finishing operation is carried out in "N" passes at the finishing feedrate "F" and with tangential entry and exit. The cycle executes the outside profile in the same direction that was defined and the islands in the opposite direction.

11 Rapid withdrawal (G0) up to the safety plane (Zs).

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

7.1.3 Examples of how to define 2D profiles



Profile P.XY	FAGOR 101	[RECALL]

Configuration:

Abscissa axis: X	Ordinate axis: Y
Autozoom: Yes	Validate

Profile:

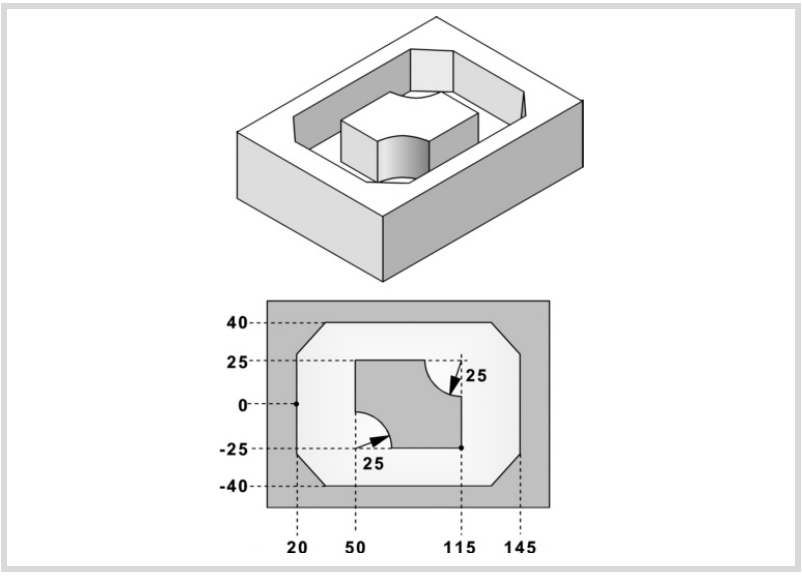
Starting point	X 20	Y -8		Validate
Straight	X 20	Y -40		Validate
Straight	X 145	Y -40		Validate
Straight	X 145	Y -25		Validate
Clockwise arc	Xf 145	Yf 25	R 25	Validate
Straight	X 145	Y 40		Validate
Straight	X 20	Y 40		Validate
Straight	X 20	Y 8		Validate
Straight	X 55	Y 8		Validate
Straight	X 55	Y -8		Validate
Straight	X 20	Y -8		Validate

Corners

Chamfer	
Select the lower left corner	[ENTER]
Chamfer 15	[ENTER]
Select the upper left corner	[ENTER]
Chamfer 15	[ENTER]
	[ESC]

End:

Save profile



Profile P.XY	FAGOR 102	[RECALL]

Configuration:

Abscissa axis: X	Ordinate axis: Y
Autozoom: Yes	Validate

Profile (outside profile):

Starting point	X 20	Y 0	Validate
Straight	X 20	Y -40	Validate
Straight	X 145	Y -40	Validate
Straight	X 145	Y 40	Validate
Straight	X 20	Y 40	Validate
Straight	X 20	Y 0	Validate

Corners

Chamfer	
Select the lower left corner	[ENTER]
Chamfer 15	[ENTER]
Select the lower right corner	[ENTER]
Chamfer 15	[ENTER]
Select the upper right corner	[ENTER]
Chamfer 15	[ENTER]
Select the upper left corner	[ENTER]
Chamfer 15	[ENTER]
	[ESC]

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

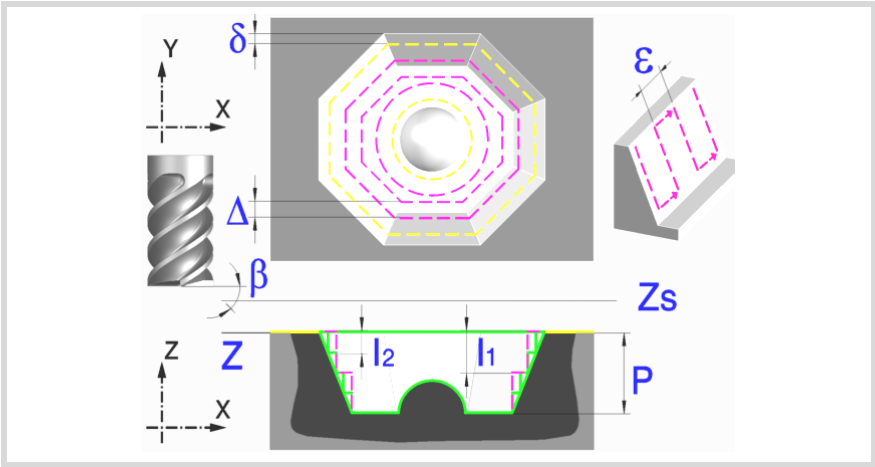
New profile (island):

Starting point	X 115	Y -25		Validate
Straight	X 115	Y 0		Validate
Clockwise arc	Xf 90	Yf 25		
	Xc 115	Yc 25	R 25	Validate
Straight	X 50	Y 25		Validate
Straight	X 50	Y 0		Validate
Clockwise arc	Xf 75	Yf -25		
	Xc 50	Yc -25	R 25	Validate
Straight	X 115	Y -25		Validate

End:

Save profile

7.2 3D profile pocket with islands.

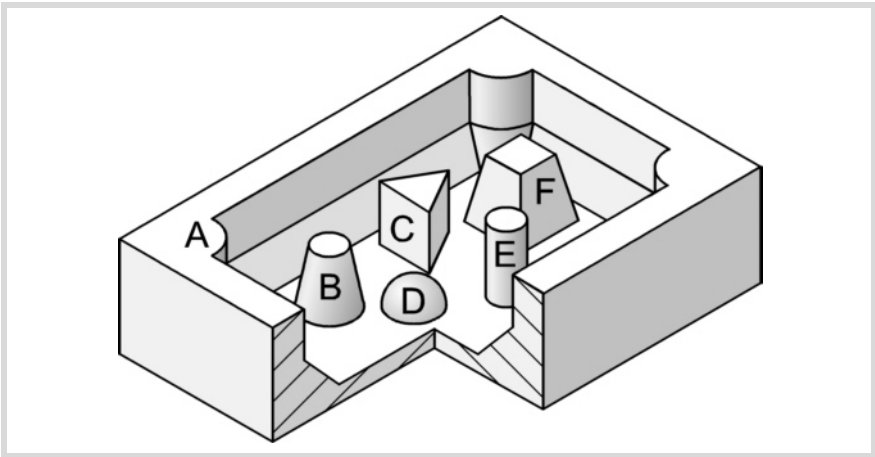


A pocket consists of an outside contour and a number of inside contours called islands.

As opposed to 2D pockets, whose walls are vertical, 3D pockets may be defined with a depth profile different for each contour (up to a maximum of 4 different ones).

The surface profile defines all the contours, the outside one and the inside ones (islands).

The first 4 contours defined in the surface profile may be assigned their own depth profiles. The rest of the profiles will be vertical.



The 3D pocket of the figure has 2 contours with "vertical profile" (C and E) and 4 contours with "non-vertical profile" (A, B, D and F).

Since only 4 contours may be defined with "non-vertical profile", contours A, B, D, F must be defined first and contours C, E at the end.

It is recommended to previously define the #ROUNDPAR instruction in order to obtain a good finish because the finishing passes are carried out in G05.

Geometric parameters:

The composition of the pocket and the plane and depth profiles are stored in \ Cnc8070\ Users\ Profile.

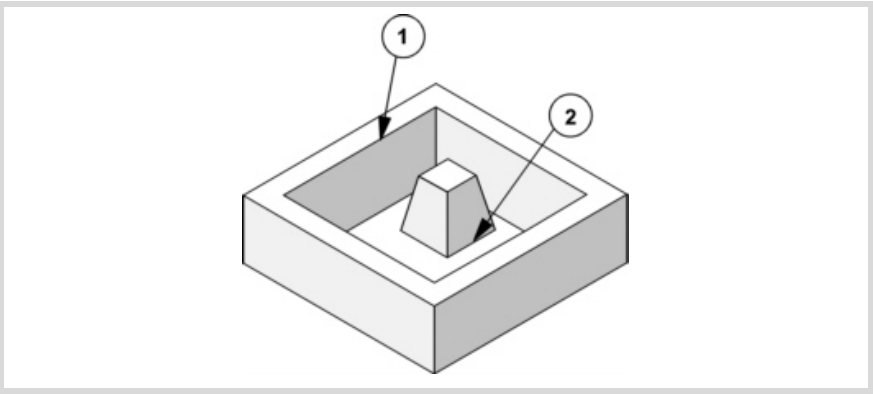
pocket.P3D	Pocket composition.
profile.PXY	Plane profile.
profile.PXZ	Depth profile.

P.3D Name of the 3D pocket.

Once the pocket configuration has been validated, the CNC associates the geometry of the pocket to its name (surface profile and depth profiles).

P.XY Name of the surface profile or plane profile.

It must indicate all the contours.
For the outside contour, the one for the surface (1).
For the islands, the one for the base (2).
All the contours must be closed and must not intersect themselves.
Remember that the order in which the contours are defined is very important.



P.Z1 P.Z2 P.Z3 P.Z4

Name of the depth profiles.

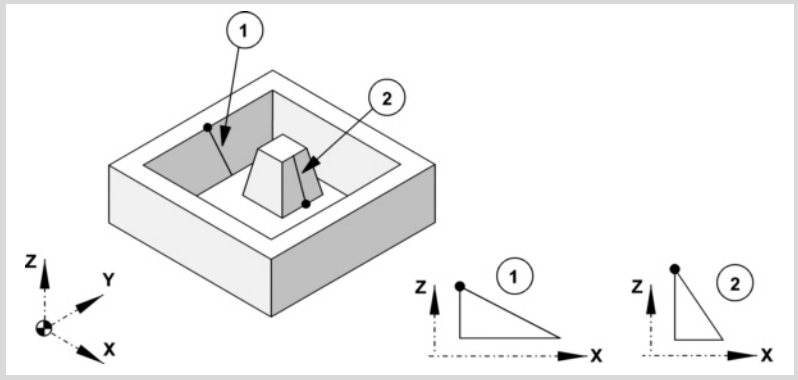
They corresponds to the first 4 contours defined in the surface profile, the number indicates the order.

To define the depth profile, use one of the axis of the plane and the perpendicular axis.

Use the same point to define the beginning of the contour and the beginning of the depth contour.

For the outside contour, one for the surface (1).

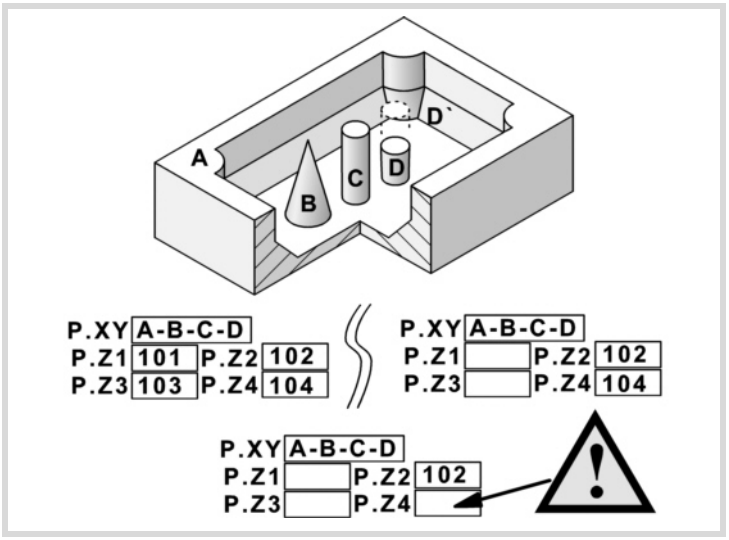
For the islands, one for the base (2).



All the profiles must be open and without direction changes along their travel (not zigzagging).

Vertical depth profiles for the outside contour and for the islands that reach the surface plane need not be programmed.

The figure shows three programming examples.



When defining the contours in the surface profile, all these cases follow the sequence A-B-C-D.

The top left-hand example defines all the depth profiles: Z1(A), Z2(B), Z3(C), Z4(D).

The top right-hand example has left out all the vertical depth profiles: Z1(A), Z3(C).

The lower example is programmed wrong because none of the vertical profiles have been defined.

If the profile of the island (D) is not defined, the cycle interprets that the island reaches the surface plane and will machine the island (D').

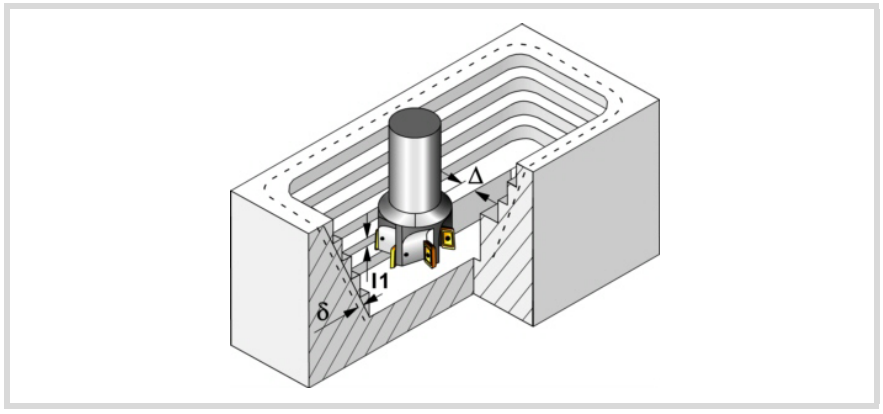
Z Part surface coordinate.

Zs Safety plane coordinate.

P Total depth.

Roughing parameters:

The roughing operation empties the pocket leaving the finishing stock δ on the side walls: This stock is defined as finishing parameter.



The roughing operation defining parameters are:

Δ Maximum milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

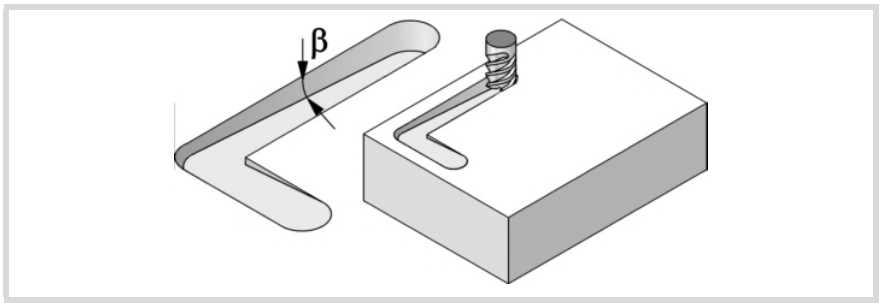
I1 Penetration step.

- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the pocket is machined with the given pass (step) except the last pass that machines the rest.
- When programming with a 0 value, the cycle assumes the step as the cutting length assigned to the tool in the tool table (Lc data). If the cutting length defined by the tool table is also set at 0, the cycle executes the pocket in a single penetration.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

Fz Penetration feedrate. If programmed with a 0 value, the cycle uses half of the programmed F feedrate.

β Penetrating angle.
The penetration is carried out maintaining this angle until the corresponding depth is reached. If defined with a value greater than the one assigned to the tool in the tool table, it assumes the table value. If programmed with a 0 value, the cycle assumes the value assigned to the tool in the tool table (Ae data). If the table value is also 0, they cycle will perform a vertical penetration, without any inclination (90° angle).



F Surface milling feedrate.

S Spindle speed.

T Roughing tool.
If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

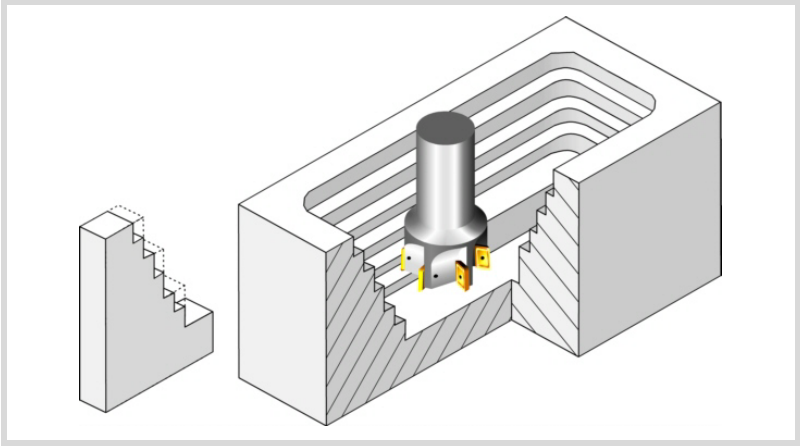
Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Pre-finishing parameters:

This operation minimizes the ridges remaining on the side walls after the roughing operation while maintaining the finishing stock δ.



The pre-finishing operation defining parameters are:

I2 Penetration step.

- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the pocket is machined with the given pass (step) except the last pass that machines the rest.
- When programming with a 0 value, the cycle assumes the step as the cutting length assigned to the tool in the tool table (Lc data). If the cutting length defined by the tool table is also set at 0, the cycle executes the pocket in a single penetration.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

F Milling feedrate.

S Spindle speed.

T Pre-finishing tool.

If programmed T=0, there is no pre-finishing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Finishing parameters:

The finishing operation takes into account the geometry of the tool tip. It compensates the tool tip radius defined in the table.

δ Finishing stock on the side walls.

ε Milling pass or width for the side walls.

Machining direction for the side walls (icon).

Always downwards.

Always upwards.

In zig-zag.

F Milling feedrate.

S Spindle speed.

T Finishing tool.

If programmed T=0, there is no finishing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

7.2.1 Executable pocket file

To simulate or execute this type of pockets, the CNC uses an executable file with geometry information. This file is generated the first time the pocket is simulated or executed. If from the editor, any data of the pocket geometry or the used tool, is modified, the CNC will generate this file again.

In versions prior to V2.00, the user generated the executable file from the editor before inserting the cycle. From version V2.00 on, it is no longer necessary, the CNC is in charge of generating the executable file when necessary.

The executable files are stored in the directory CNC8070 \Users \Pocket with the name of the pocket (parameter P.3D) and the extension C3D. These files must not be deleted, moved to another location or tampered with in any way. If when executing or simulating the pocket, the CNC cannot find these files, it will generate them.

Overall, a 2D pocket consists of the following files.

pocket.P3D	Pocket composition.
profile.PXY	Plane profile.
profile.PXZ	Depth profile.
pocket.C3D	Executable file.

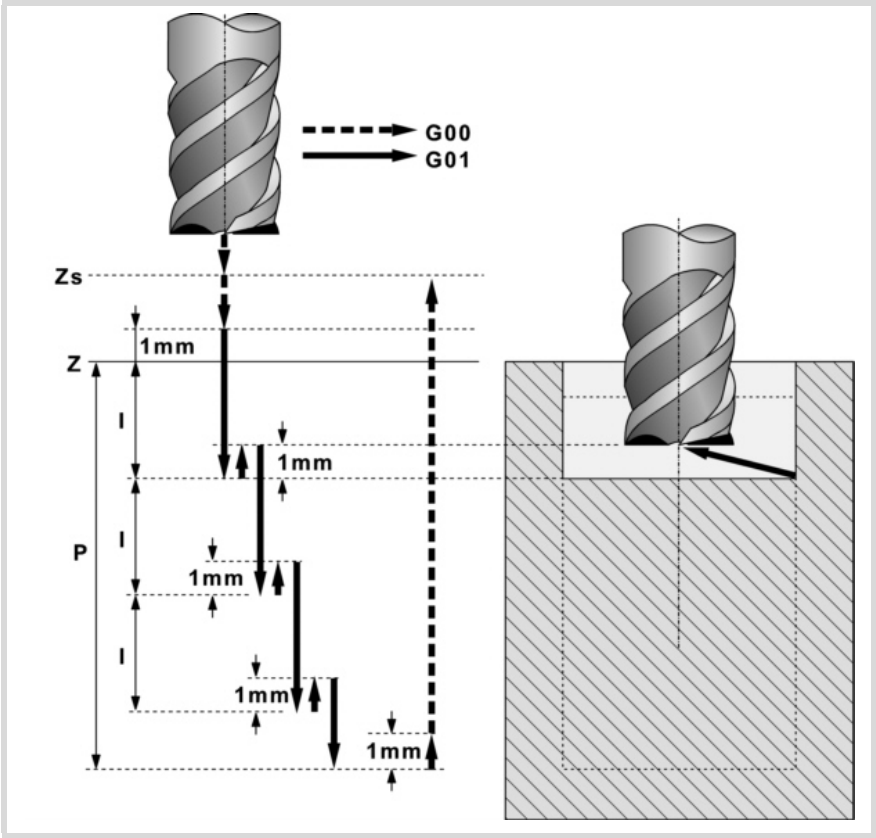
The executable file is also updated after a software update and when executing or simulating a pocket,

7.2.2 Basic operation.

The CNC calculates the initial coordinate depending on the geometry of the pocket and the tool radius.

1 It selects the roughing tool and starts the spindle in the requested direction.

2 Rapid movement (G0) to the roughing starting point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.



3 Rapid movement (G0) up to the approach plane.

4 Roughing operation. It is carried out in layers until the total depth is reached.

First, penetration "I1" at feedrate "Fz" at an angle "β". Then, milling of the pocket surface up to a distance "δ" from the pocket wall. It is carried out at feedrate "F" and, if necessary, it recalculates the pass (Δ) so all the passes are identical.

The pocket is machined following paths concentric to the profile, in the same direction as the outside profile was defined. The islands are machined in the opposite direction. When the tool must go around or avoid an island, it increases in G0 up to the safety level (Zs). If the movement does not involve going around an island, the tool is raised to the working level + 1 mm.

And last, rapid withdrawal (G0) up to 1 mm off the machined surface.

5 Rapid withdrawal (G0) up to the approach plane.

6 It selects the pre-finishing tool and starts the spindle in the requested direction.

7 Pre-finishing operation for the side walls. It is carried out in layers until the total depth is reached. The cycle will not run the pre-finishing passes that coincide with any previous roughing pass.

It is carried out with the pass indicated by "I2" and at the pre-finishing feedrate "F". The outside profile in the same direction that was defined and the islands in the opposite direction.

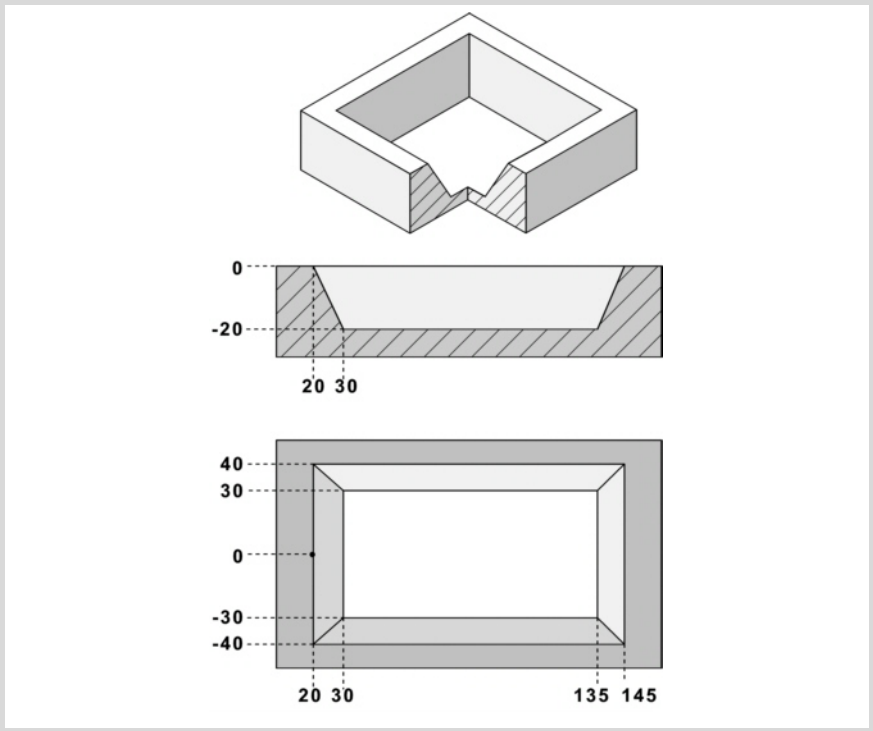
8 Rapid withdrawal (G0) up to the approach plane.

9 It selects the finishing tool and starts the spindle in the requested direction.

10 Finishing of the side walls. It is carried out with the pass "ε" and direction indicated by the icon.

Rapid withdrawal (G0) up to the safety plane (Zs).

7.2.3 Examples of how to define 3D profiles



Pocket P.3D	FAGOR-A	
Profile P.XY	FAGOR 110	[RECALL]

Configuration:

Abscissa axis: X	Ordinate axis: Y
Autozoom: Yes	Validate

Profile (outside profile):

Starting point	X 20	Y 0	Validate
Straight	X 20	Y -40	Validate
Straight	X 145	Y -40	Validate
Straight	X 145	Y 40	Validate
Straight	X 20	Y 40	Validate
Straight	X 20	Y 0	Validate

End:

Save profile

Profile P.Z1	FAGOR 211	Recall

Configuration:

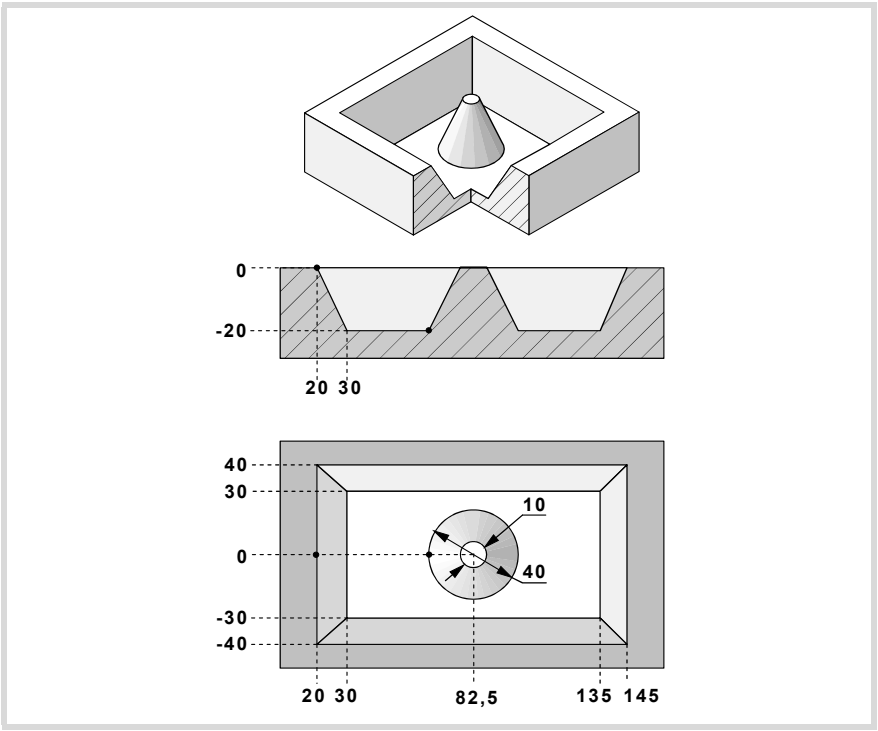
Abscissa axis: X	Ordinate axis: Z
Autozoom: Yes	Validate

Profile (depth profile):

Starting point	X 20	Z0	Validate
Straight	X 30	Z-20	Validate

End:

Save profile



Pocket P.3D	FAGOR-B	
Profile P.XY	FAGOR 120	[RECALL]

Configuration:

Abscissa axis: X	Ordinate axis: Y
Autozoom: Yes	Validate

Profile (outside profile):

Starting point	X 20	Y 0	Validate
Straight	X 20	Y -40	Validate
Straight	X 145	Y -40	Validate
Straight	X 145	Y 40	Validate
Straight	X 20	Y 40	Validate
Straight	X 20	Y 0	Validate

New profile (island):

Circle	X 62,5	Y0	Xc 82,5	Yc 0	Validate

End:

Save profile

Profile P.Z1	FAGOR 221	[RECALL]

Configuration:

Abscissa axis: X	Ordinate axis: Z
Autozoom: Yes	Validate

Profile (outside depth profile):

Starting point	X 20	Z0	Validate
Straight	X 30	Z-20	Validate

End:

Save profile

Profile P.Z2	FAGOR 222	Recall

Configuration:

Abscissa axis: X	Ordinate axis: Z
Autozoom: Yes	Validate

Profile (island depth profile):

Starting point	X 62,5	Z-20	Validate
Straight	X 77,5	Z0	Validate

End:

Save profile

7.

CANNED CYCLES OF THE EDITOR. 2D/3D PROFILE POCKETS.

3D profile pocket with islands.

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

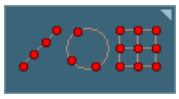
REF. 2109

CANNED CYCLES OF THE EDITOR. ROUGHING.

8

When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.

- Point-to-point profile milling.
- Free profile milling.
- Surface milling.
- Slot milling.



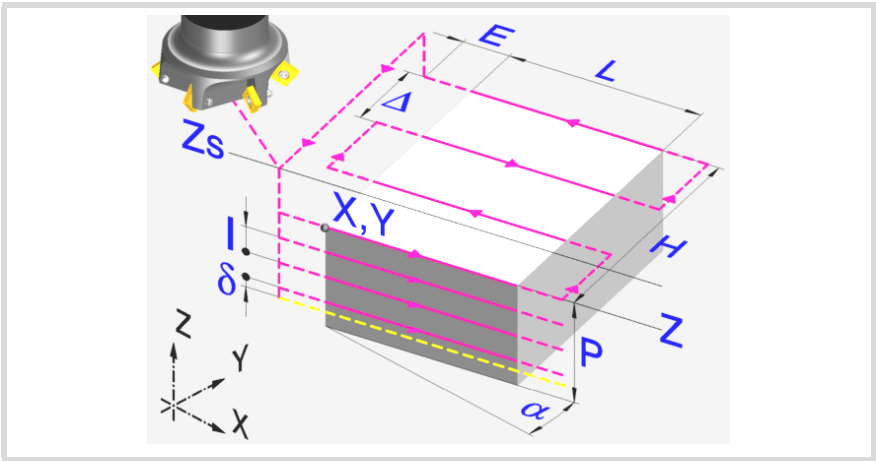
Multiple machining may be added to pocket cycles so the cycle is repeated in several points. The available multiple machining operations are the following. See chapter *"9 Canned cycles of the editor. Multiple positioning."*.

- Points in line.
- Points in arc.
- Points in rectangle.
- Points in grid.
- Random points (several points defined by the user).

To associate multiple machining to a cycle, first select and define a machining cycle from those allowed. Then, without quitting the editing of the cycle, press the softkey associated with multiple machining and select one of them.

8.1 Surface milling.

The way the roughing and finishing blocks of this cycle are joined will be the one previously set by the user with the instructions #HSC, G5, G50 or G7. We recommend to use #HSC or G5 controlling the shape of the corner with the instruction #ROUNDPAR.



Geometric parameters:

Machining direction (icon).

Bidirectional machining along the abscissa axis.

Bidirectional machining along the ordinate axis.

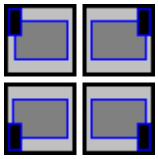
Unidirectional machining along the abscissa axis.

Unidirectional machining along the ordinate axis.

Spiral machining along the abscissa axis.

Spiral machining along the ordinate axis.

Corner where the surface milling begins (icon).



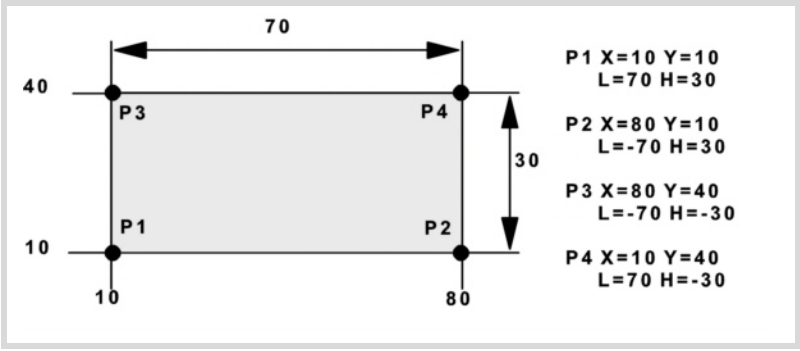
Any of the 4 corners may be selected.

X Y Corner to begin machining.

The (X, Y) point needs not coincide with the corner selected to begin machining.

L H Surface to be milled.

The sign of L and H indicates the orientation with respect to the XY point.



Z Part surface coordinate.

Zs Safety plane coordinate.

P Total depth.

α Angle, in degrees, between the surface and the abscissa axis. The turn is carried out on the defined corner, X,Y point.

Roughing parameters:

The roughing operation leaves a finishing stock δz defined as finishing parameter. The roughing operation defining parameters are:

Δ Maximum milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

E Overshooting distance of the tool off the surface being milled.

Fz Penetration feedrate. If programmed with a 0 value, the cycle uses half of the programmed F feedrate.

I Penetration step.

- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the milling is carried out with the given pass (step) except the last pass that machines the rest.
- When programming with a 0 value, the cycle assumes the step as the cutting length assigned to the tool in the tool table (Lc data). If the cutting length defined by the tool table is also set at 0, the cycle executes the pocket in a single penetration.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Finishing parameters:

δz Finishing stock.

Δ Maximum milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

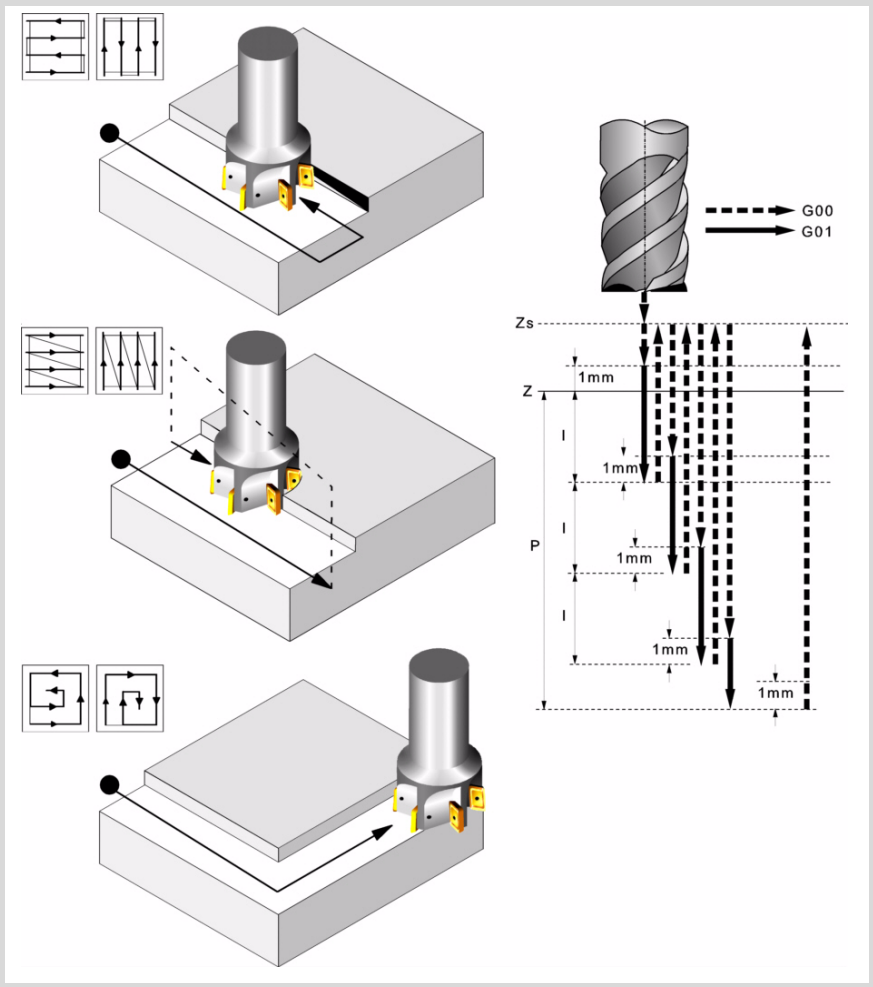
Next tool.

Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

8.1.1 Basic operation.

1 It selects the roughing tool and starts the spindle in the requested direction.

2 Rapid movement (G0) to the roughing starting point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.



3 Rapid movement (G0) up to the approach plane.

4 Roughing operation. It is carried out in layers, until reaching the total depth minus the finishing distance "δz".

First, penetration "I" at feedrate "Fz". Then, milling at feedrate "F" and, if necessary, it recalculates the pass (Δ) so all the passes are identical.

- In bidirectional and spiral milling, all the movements are at feedrate "F".
- In unidirectional milling, the movements between two consecutive milling passes are carried out in rapid and in the safety plane Zs.

And last, rapid movement (G0), first up to the safety plane (Zs) and then to the starting point.

5 Rapid movement (G0) up to 1 mm above the last pass.

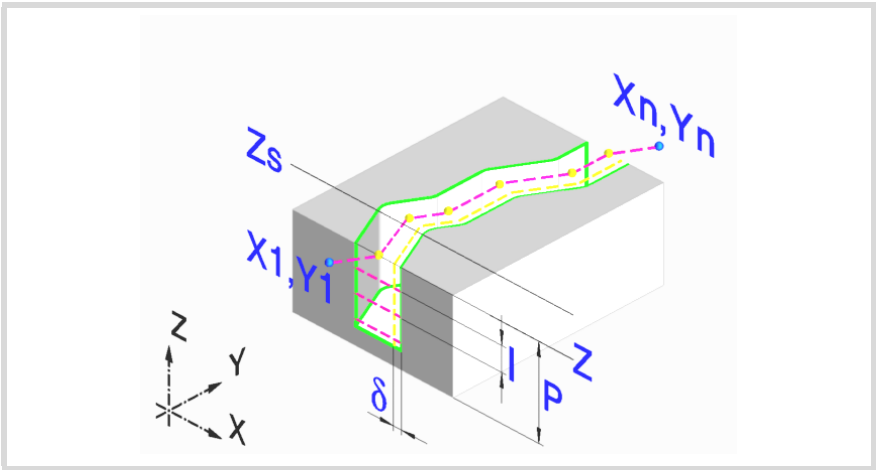
6 Finishing.

Penetration at feedrate "Fz". Milling at finishing feedrate "F" and, if necessary, it recalculates the finishing pass (Δ) so all the passes are identical.

7 Rapid withdrawal (G0) up to the safety plane (Zs).

8.2 Point-to-point profile milling.

The way the roughing and finishing blocks of this cycle are joined will be the one previously set by the user with the instructions #HSC, G5, G50 or G7. We recommend to use #HSC or G5 controlling the shape of the corner with the instruction #ROUNDPAR.

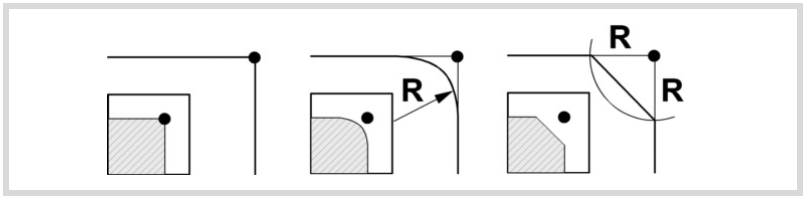


Geometric parameters:

X1, Y1	Profile entry point
R1	Radius of the tangential entry to the profile
P1..P25	Points of the profile.

All intermediate points P2 to P24 have an icon to indicate the type of corner; square, rounded or chamfered. For rounded or chamfered corners, indicate the rounding radius or chamfer size.

When not using all 25 points, define the first unused point with the same coordinates as those of the last point of the profile.

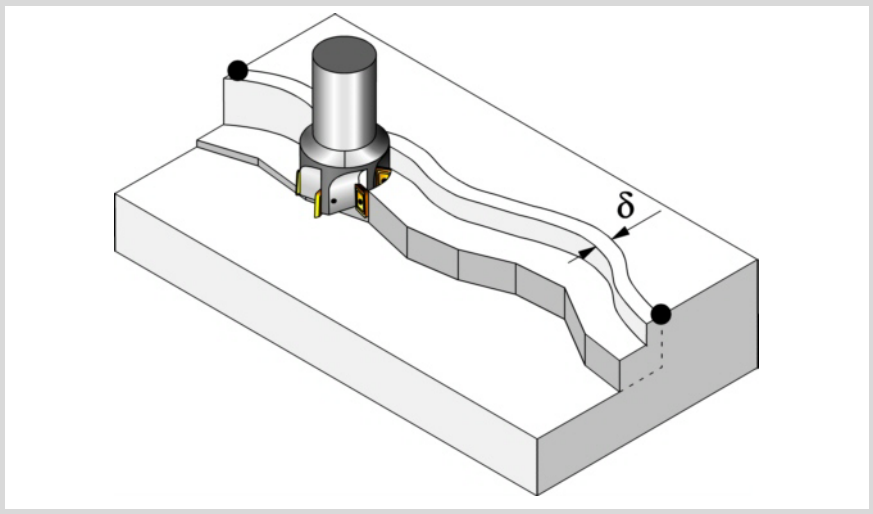


Delete all the points of the profile. Select this icon and press [DEL] to delete all the points of the table.

Rn	Radius of the tangential exit from the profile
Xn, Yn	Profile exit point
Z	Part surface coordinate.
Zs	Safety plane coordinate.
P	Total depth.

Roughing parameters:

The roughing operation mills the profile leaving the finishing stock δ. This stock is defined as finishing parameter.



The roughing operation defining parameters are:

Fz Penetration feedrate. If programmed with a 0 value, the cycle uses half of the programmed F feedrate.

I Penetration step.

- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the milling is carried out with the given pass (step) except the last pass that machines the rest.
- When programming with a 0 value, the cycle assumes the step as the cutting length assigned to the tool in the tool table (Lc data). If the cutting length defined by the tool table is also set at 0, the cycle executes the pocket in a single penetration.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Tool radius compensation (icon).

Without compensation.

Left-hand compensation.

Right-hand compensation.

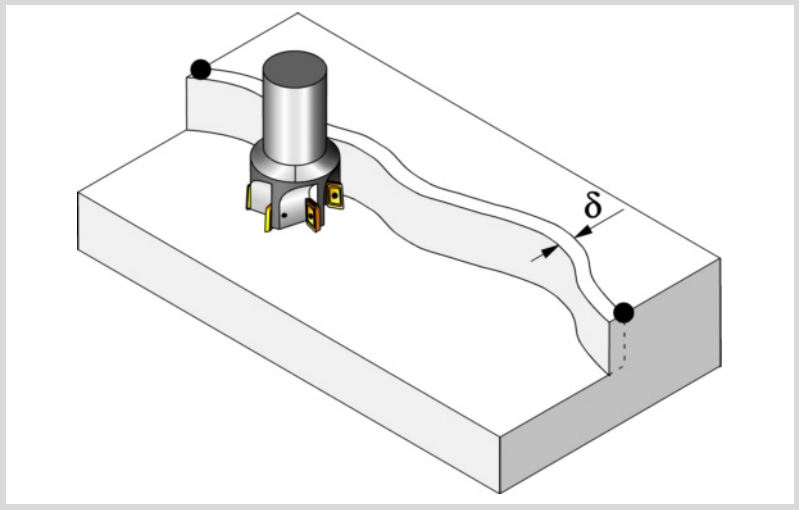
Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Finishing parameters:

In order to carry out the finishing operation, the roughing must be defined with tool radius compensation. The operation removes the finishing stock (δ).



The roughing operation defining parameters are:

δ	Finishing stock on the side walls. When working without tool radius compensation, there is no finishing operation, the finishing stock (δ) is ignored.
F	Milling feedrate.
S	Spindle speed.
T	Finishing tool. If programmed T=0, there is no finishing.
D	Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

8.2.1 Basic operation.

1. It selects the roughing tool and starts the spindle in the requested direction.
2. Rapid movement (G0), up to the XY point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.
3. Rapid movement (G0) up to the approach plane.
4. Roughing operation. It is carried out in layers until the total depth is reached.

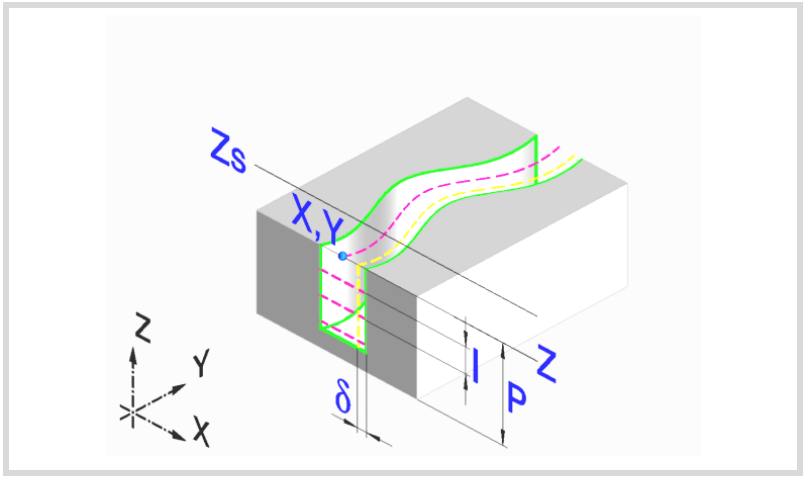
   First, penetration "I" at feedrate "Fz". Then, profile milling at feedrate "F" and tangential entry if it has been programmed. If roughing was defined with tool radius compensation, the milling is carried out at a "δ" distance from the wall. After done roughing, exit to point XnYn with tangential exit if it has been programmed. And last, rapid movement (G0), first up to the safety plane (Zs) and then to the starting point X1Y1.
5. It selects the finishing tool and starts the spindle in the requested direction.
6. Finishing operation.
7. Penetration to the bottom at feedrate "Fz".

   Profile milling at feedrate "F" and tangential entry if it has been programmed. Exit to point XnYn with tangential exit if it has been programmed.
8. Rapid withdrawal (G0) up to the safety plane (Zs).

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

8.3 Free profile milling.

The way the roughing and finishing blocks of this cycle are joined will be the one previously set by the user with the instructions #HSC, G5, G50 or G7. We recommend to use #HSC or G5 controlling the shape of the corner with the instruction #ROUNDPAR.



Geometric parameters:

X, Y Profile entry point

Name of the profile.

To machine with tangential entry and exit, define these values inside the profile.

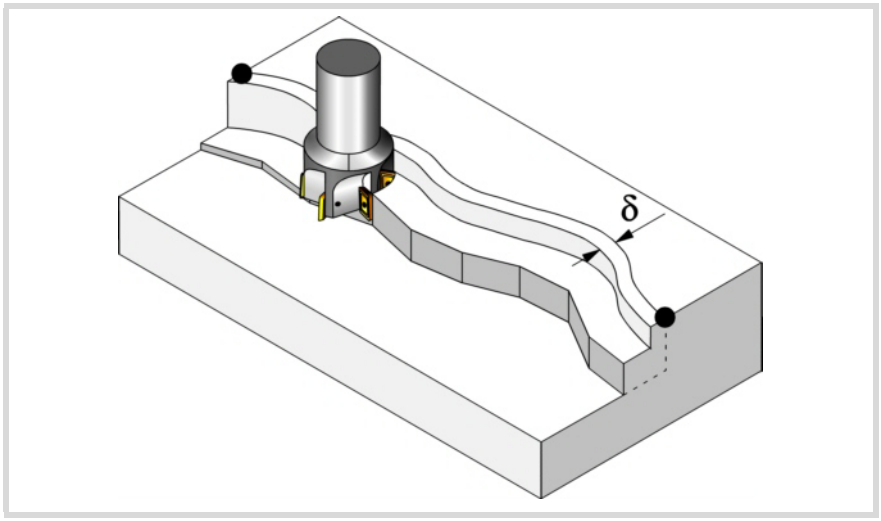
Z Part surface coordinate.

Zs Safety plane coordinate.

P Total depth.

Roughing parameters:

The roughing operation mills the profile leaving the finishing stock δ. This stock is defined as finishing parameter.



The roughing operation defining parameters are:

Fz Penetration feedrate. If programmed with a 0 value, the cycle uses half of the programmed F feedrate.

I Penetration step.

- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the milling is carried out with the given pass (step) except the last pass that machines the rest.
- When programming with a 0 value, the cycle assumes the step as the cutting length assigned to the tool in the tool table (Lc data). If the cutting length defined by the tool table is also set at 0, the cycle executes the pocket in a single penetration.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Tool radius compensation (icon).

Without compensation.

Left-hand compensation.

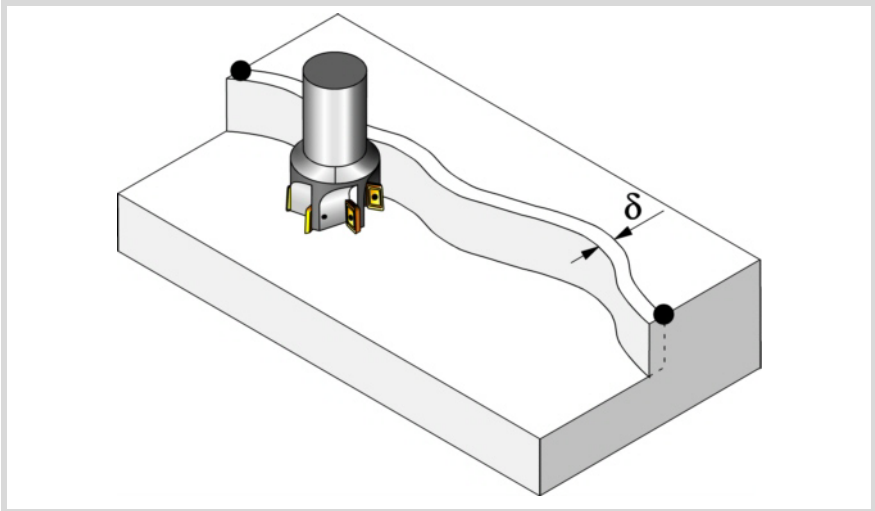
Right-hand compensation.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Finishing parameters:

In order to carry out the finishing operation, the roughing must be defined with tool radius compensation. This operation removes the finishing stock (δ).



The roughing operation defining parameters are:

δ	Finishing stock on the side walls. When working without tool radius compensation, the stock (δ) is ignored . In this case, the tool center travel is the same when roughing as when finishing.
F	Milling feedrate.
S	Spindle speed.
T	Finishing tool. If programmed T=0, there is no finishing.
D	Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

8.3.1 Basic operation.

1 It selects the roughing tool and starts the spindle in the requested direction.

2 Rapid movement (G0), up to the XY point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.

3 Rapid movement (G0) up to the approach plane.

4 Roughing operation. It is carried out in layers until the total depth is reached.

First, penetration "I" at feedrate "Fz". Then, profile milling at feedrate "F". If roughing was defined with tool radius compensation, the milling is carried out at a "δ" distance from the wall. And last, rapid movement (G0), first up to the safety plane (Zs) and then to the starting point X1Y1.

5 It selects the finishing tool and starts the spindle in the requested direction.

6 Finishing operation.

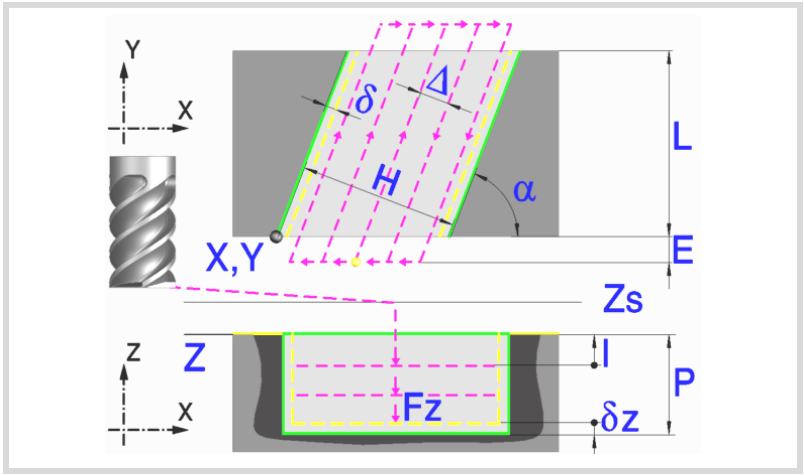
7 Penetration to the bottom at feedrate "Fz". Profile milling at feedrate "F".

8 Rapid withdrawal (G0) up to the safety plane (Zs).

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

8.4 Slot milling.

The way the roughing and finishing blocks of this cycle are joined will be the one previously set by the user with the instructions #HSC, G5, G50 or G7. We recommend to use #HSC or G5 controlling the shape of the corner with the instruction #ROUNDPAR.



Geometric parameters:

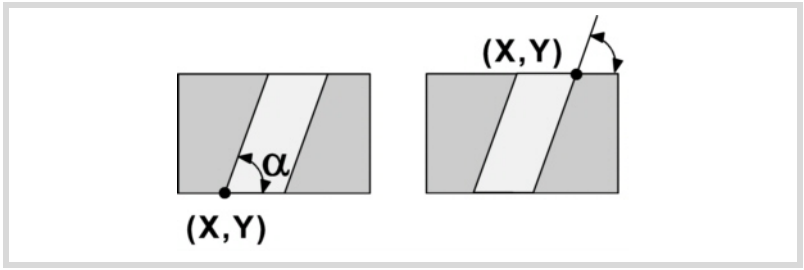
Type of slot milling (icon).

There are 6 possible types.

4 for slot mill each corner of the part.

2 for milling a slot across the part.

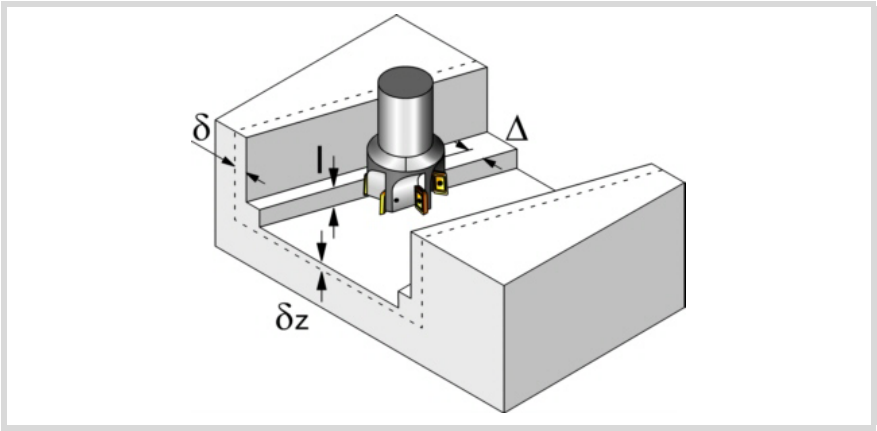
X, Y	Corner where the slot is to be milled.
L, H	Slot dimensions. The sign indicates the orientation referred to the XY point.
Z	Part surface coordinate.
Zs	Safety plane coordinate.
P	Total depth.
α	Angle, in degrees, between the slot and the abscissa axis. The turn is carried out on the defined corner, X,Y point.



Roughing parameters:

The roughing operation leaves the following finishing stocks. Both stocks are defined as finishing parameters.

δ	Finishing stock on the side walls.
δz	Finishing stock at the bottom of the pocket.



The roughing operation defining parameters are:

Δ Maximum milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

E Overshooting distance of the tool off the surface being milled.

Fz Penetration feedrate. If programmed with a 0 value, the cycle uses half of the programmed F feedrate.

I Penetration step.

- If programmed with a positive sign (I+), the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If programmed with a negative sign (I-), the slot milling is carried out with the given pass (step) except the last pass that machines the rest.
- When programming with a 0 value, the cycle assumes the step as the cutting length assigned to the tool in the tool table (Lc data). If the cutting length defined by the tool table is also set at 0, the cycle executes the pocket in a single penetration.

In either case, the cycle limits the step to the cutting length assigned to the tool in the tool table.

F Surface milling feedrate.

S Spindle speed.

T Roughing tool.

If programmed T=0, there is no roughing.

D Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

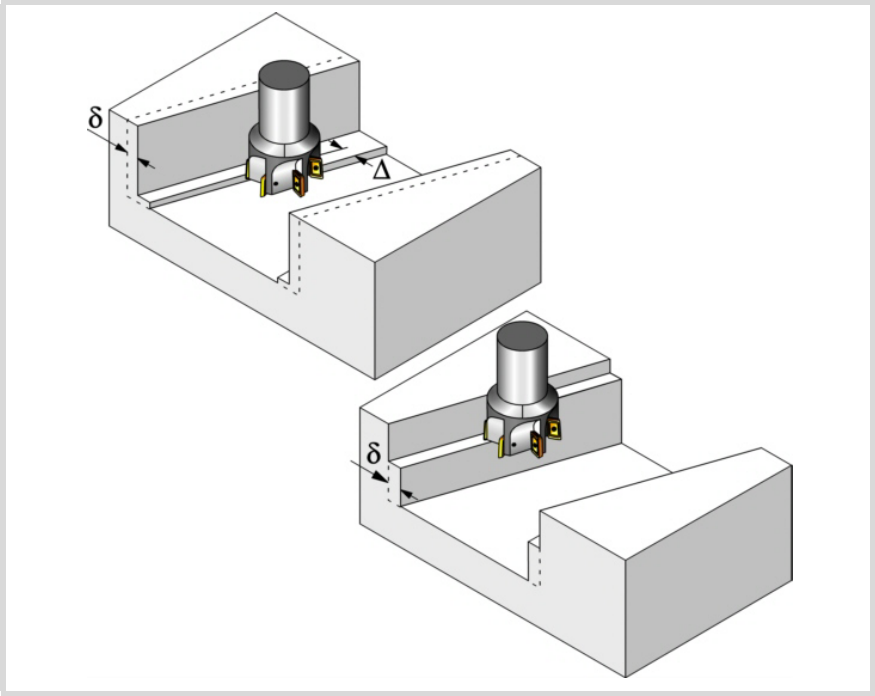
Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Finishing parameters:

The finishing operation is carried out in two stages. First, it machines the bottom of the slot and then the side walls, with tangential entry and exit.



The finishing operation defining parameters are:

δ	Finishing pass on the side walls.
δz	Finishing pass at the bottom.
Δ	Milling pass or width at the bottom of the slot. The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.
N	Number of penetration passes (steps) for the side finishing. If the resulting step is greater than the cutting length assigned to the table in the tool table, the step will be limited to that value. When programming with a 0 value, it carries out the least amount of passes possible, considering the cutting length assigned to the tool in the tool table (Lc data).
F	Surface and side milling feedrate.
S	Spindle speed.
T	Finishing tool. If programmed T=0, there is no finishing.
D	Tool offset.

Spindle turning direction (icon).

Clockwise.

Counterclockwise.

Machining direction (icon).

Clockwise.

Counterclockwise.

Programming of M functions.

Activating or deactivating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only some of them, define them first and leave the rest of the data unprogrammed.
The editor will only show this option if the user has configured the editor to allow programming M functions.

Next tool.

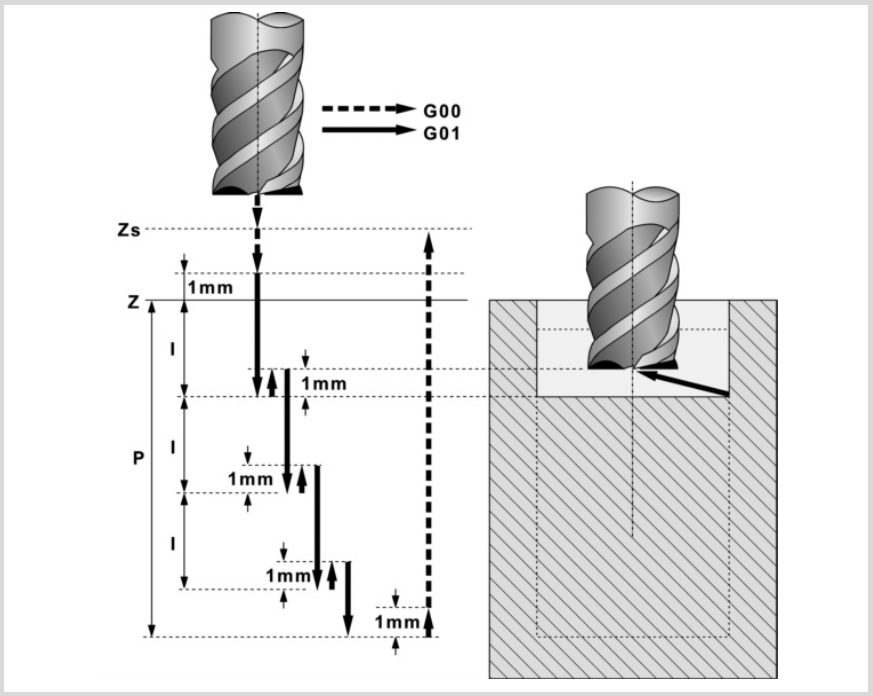
Activating or deactivating the preparation of the next tool.
The editor will only show this option if the user has configured the editor to allow programming the next tool.

8.4.1 Basic operation.

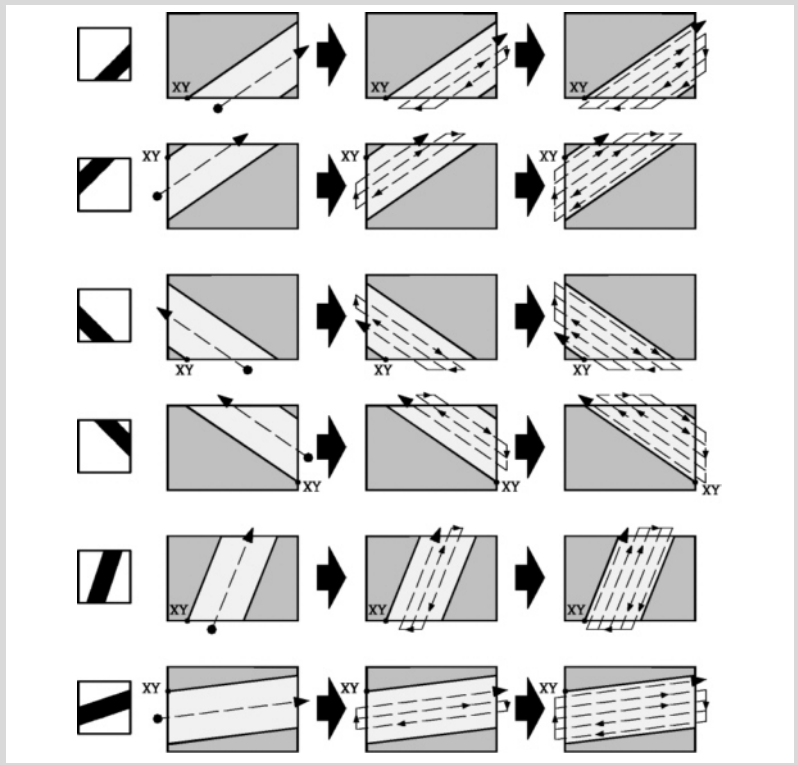
1 It selects the roughing tool and starts the spindle in the requested direction.

2 Rapid movement (G0) to the roughing starting point and the safety plane (Zs). Depending on the starting plane, it first moves in XY and then in Z or vice versa.

3 Rapid movement (G0) up to the approach plane.



4 Roughing operation. It is carried out in layers, until reaching the total depth minus the finishing distance "δz".



First, penetration "I" at feedrate "Fz". Then, slot milling of the boss surface up to a distance "δ" from the side wall. The slot is carried out at feedrate "F" and, if necessary, it recalculates the pass (Δ) so all the passes are identical. After done roughing, rapid movement (G0), first up to the safety plane (Zs) and then to the starting point. And last, rapid approach (G0) up to 1 mm off the machined surface.

5 Rapid withdrawal (G0) up to the safety plane (Zs).

6 It selects the finishing tool and it approaches in rapid (G0) down to 1 mm from the roughed-out bottom.

7 Finishing of the bottom of the slot.

Penetration at feedrate "Fz". Milling of the bottom of the slot up to a distance "δ" from the pocket. It is carried out at finishing feedrate "F" and, if necessary, it recalculates the finishing pass (Δ) so all the passes are identical.

8 Rapid withdrawal (G0) up to the safety plane (Zs).

9 Finishing of the side walls. Finishing is carried out in "N" passes at the finishing feedrate "F".

10 Rapid withdrawal (G0) up to the safety plane (Zs).

8.

CANNED CYCLES OF THE EDITOR. ROUGHING.
Slot milling.

FAGOR

FAGOR AUTOMATION

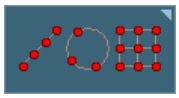
CNCelite
8058 8060
8065

REF. 2109

CANNED CYCLES OF THE EDITOR. MULTIPLE POSITIONING.

9

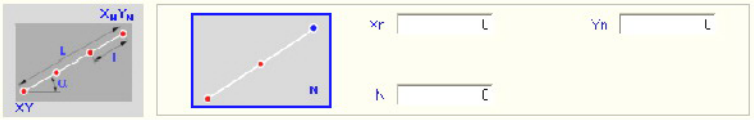
When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



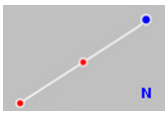
- Multiple machining in straight line
- Multiple machining in arc pattern
- Multiple machining in rectangular pattern
- Multiple machining in grid pattern
- Random multiple machining (several points defined by the user).
- Without multiple machining.

9.1 Multiple machining in straight line.

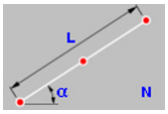
There are 5 different ways to define the machining operation. Use the icon to select the desired option.



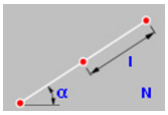
Positioning options.



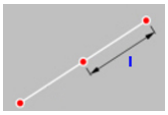
End point and number of machining operations.



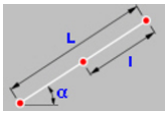
Length, angle and number of machining operations.



Distance between machining operations, angle and number of machining operations.



End point and distance between two machining operations.

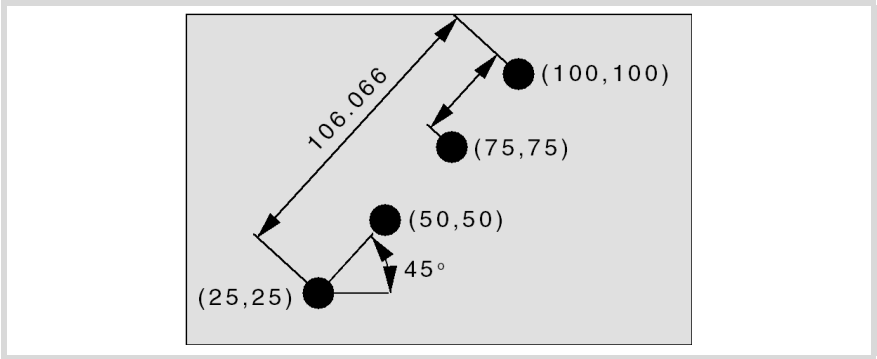


Length, angle and distance between two machining operations.

Xn, Yn	Coordinates of the end point of the positioning movement.
N	Number of machining operations including that of the cycle definition point.
α	Angle of the line with the abscissa axis.
L	Total length of the line.
I	Distance between two consecutive machining operations.

9.1.1 Programming example.

The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.



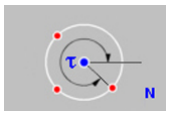
We now show the 5 possible ways to define it.

1)	Coordinates of the end point.	Xn 100, Yn 100
	Total number of machining operations.	N 4
2)	Angle of the path.	α 45
	Length of the line.	L 106.066
	Total number of machining operations.	N 4
3)	Angle of the path.	α 45
	Total number of machining operations.	N 4
	Distance between machining operations.	I 35.3553
4)	Coordinates of the end point.	Xn 100, Yn 100
	Distance between machining operations.	I 35.3553
5)	Angle of the path.	α 45
	Length of the line.	L 106.066
	Distance between machining operations.	I 35.3553

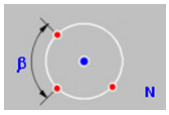
9.2 Multiple machining in arc pattern.

There are 9 ways to define the machining operation. Use the icon to select the desired option.

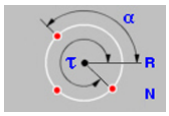
Positioning options.



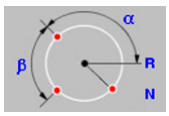
Arc center coordinates, number of machining operations and final angle of the arc.



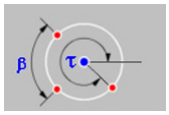
Arc center coordinates, number of machining operations and angular distance between two machining operations.



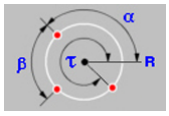
Arc radius, number of machining operations and starting and final angles of the arc.



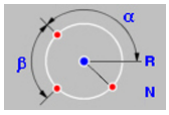
Arc radius, number of machining operations, starting angle of the arc and angular distance between two machining operations.



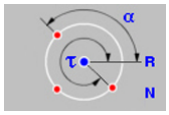
Arc center coordinates, final angle of the arc and angular distance between two machining operations.



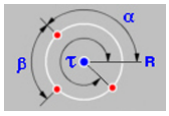
Arc radius, angular distance between two machining operations and starting and final angles of the arc.



Arc center coordinates, arc radius, number of machining operations, starting angle of the arc and angular distance between two machining operations.



Arc center coordinates, arc radius, number of machining operations and starting and final angles of the arc.

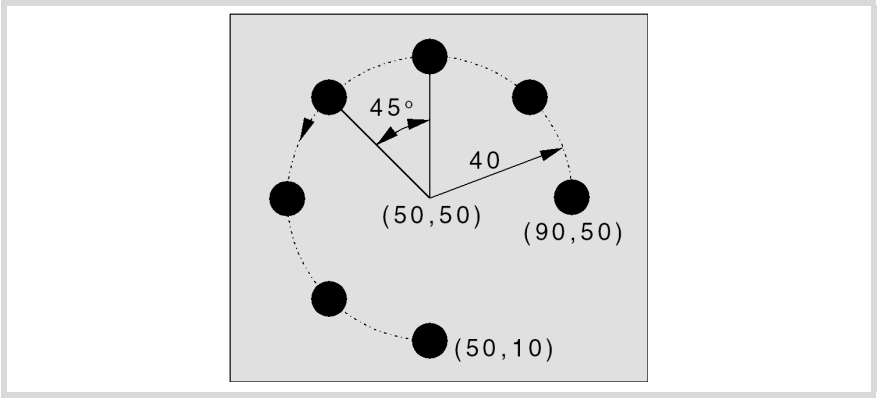


Arc center coordinates, arc radius, angular distance between two machining operations and starting and final angles of the arc.

Xa, Ya	Arc center coordinates.
R	Radius of the arc.
N	Number of machining operations including that of the cycle definition point.
α	Starting angle of the arc.
τ	Final angle of the arc.
β	Angular distance between two machining operations.
	The cycle moves counter-clockwise; to move clockwise, define the angular distance between machining operations with a negative sign.

9.2.1 Programming example.

The canned cycle defined at point X90, Y50 is to be repeated at the rest of the points.

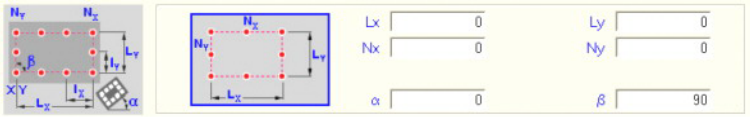


We now show the 9 possible ways to define it.

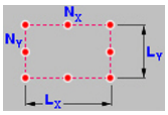
1)	Arc center coordinates.	Xa 50, Ya 50
	Total number of machining operations.	N 7
	Angle of the end point.	τ 270
2)	Arc center coordinates.	Xa 50, Ya 50
	Total number of machining operations.	N 7
	Angular distance between machining operations.	β 45
3)	Radius of the arc.	R 40
	Total number of machining operations.	N 7
	Angle of the starting point.	α 0
	Angle of the end point.	τ 270
4)	Radius of the arc.	R 40
	Total number of machining operations.	N 7
	Angle of the starting point.	α 0
	Angular distance between machining operations.	β 45
5)	Arc center coordinates.	Xa 50, Ya 50
	Angle of the end point.	τ 270
	Angular distance between machining operations.	β 45
6)	Radius of the arc.	R 40
	Angle of the starting point.	α 0
	Angle of the end point.	τ 270
	Angular distance between machining operations.	β 45
7)	Arc center coordinates.	Xa 50, Ya 50
	Radius of the arc.	R 40
	Total number of machining operations.	N 7
	Angle of the starting point.	α 0
	Angular distance between machining operations.	β 45
8)	Arc center coordinates.	Xa 50, Ya 50
	Radius of the arc.	R 40
	Total number of machining operations.	N 7
	Angle of the starting point.	α 0
	Angle of the end point.	τ 270
9)	Arc center coordinates.	Xa 50, Ya 50
	Radius of the arc.	R 40
	Angle of the starting point.	α 0
	Angle of the end point.	τ 270
	Angular distance between machining operations.	β 45

9.3 Multiple machining in rectangular pattern.

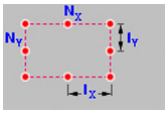
There are 3 ways to define the machining operation. Use the icon to select the desired option.



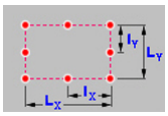
Positioning options.



Length of each side and number of machining operations.



Distance between two machining operations and number of machining operations.



Length of each side and distance between two machining operations.

Lx, Ly — Length of the rectangle, on each axis.

Nx, Ny — Number of machining operations (including that of the cycle definition point) on each axis.

Ix, Iy — Distance between two machining operations, on each axis.

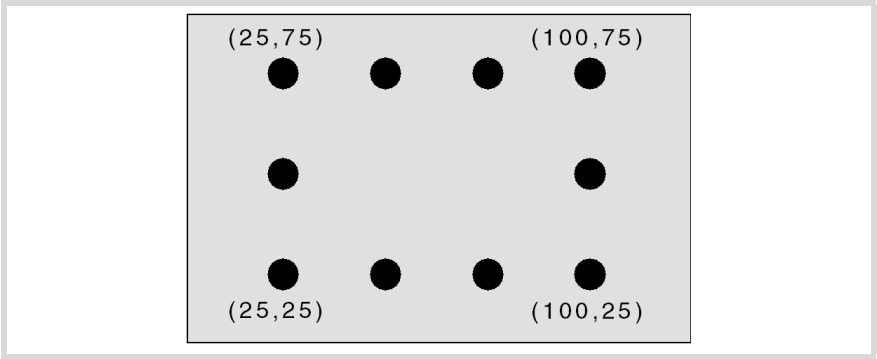
α — Angle of the rectangle with the abscissa axis.

β — Angle between both sides.

The cycle assumes as starting point the lower left one; if it is not, define the machining length (Lx, Ly) with the proper sign or the distance between machining operations (Ix, Iy).

9.3.1 Programming example.

The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.

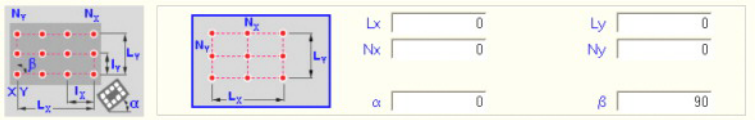


We now show the 3 possible ways to define it.

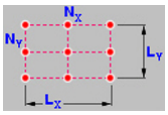
1)	Length of each side of the rectangle.	Lx 75, Ly 50
	Number of machining operations.	Nx 4, Ny 3
	Angle with respect to the abscissa axis.	α 0
	Angle between paths.	β 90
2)	Number of machining operations.	Nx 4, Ny 3
	Distance between machining operations.	Ix 25, Iy 25
	Angle with respect to the abscissa axis.	α 0
	Angle between paths.	β 90
3)	Length of each side of the rectangle.	Lx 75, Ly 50
	Distance between machining operations.	Ix 25, Iy 25
	Angle with respect to the abscissa axis.	α 0
	Angle between paths.	β 90

9.4 Multiple machining in grid pattern.

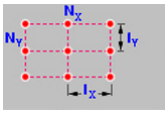
There are 3 ways to define the machining operation. Use the icon to select the desired option.



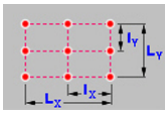
Positioning options.



Length of each side and number of machining operations.



Distance between two machining operations and number of machining operations.



Length of each side and distance between two machining operations.

Lx, Ly — Length of the grid, on each axis.

Nx, Ny — Number of machining operations (including that of the cycle definition point) on each axis.

Ix, Iy — Distance between two machining operations, on each axis.

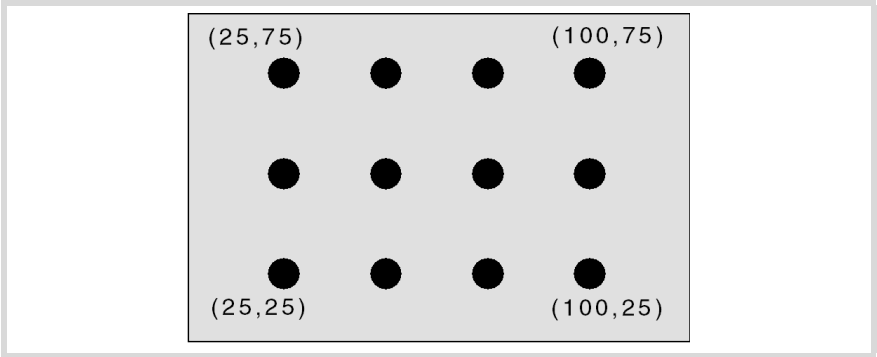
α — Angle of the grid with the abscissa axis.

β — Angle between both sides.

The cycle assumes as starting point the lower left one; if it is not, define the machining length (Lx, Ly) with the proper sign or the distance between machining operations (Ix, Iy).

9.4.1 Programming example.

The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.

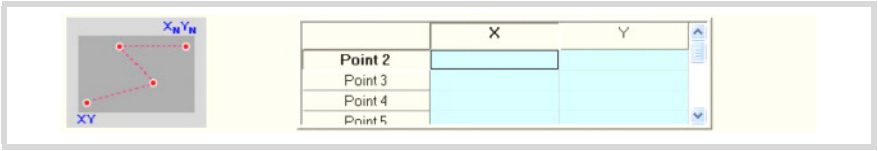


We now show the 3 possible ways to define it.

1)	Length of each side of the grid.	Lx 75, Ly 50
	Number of machining operations.	Nx 4, Ny 3
	Angle with respect to the abscissa axis.	α 0
	Angle between paths.	β 90
2)	Number of machining operations.	Nx 4, Ny 3
	Distance between machining operations.	Ix 25, Iy 25
	Angle with respect to the abscissa axis.	α 0
	Angle between paths.	β 90
3)	Length of each side of the grid.	Lx 75, Ly 50
	Distance between machining operations.	Ix 25, Iy 25
	Angle with respect to the abscissa axis.	α 0
	Angle between paths.	β 90

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

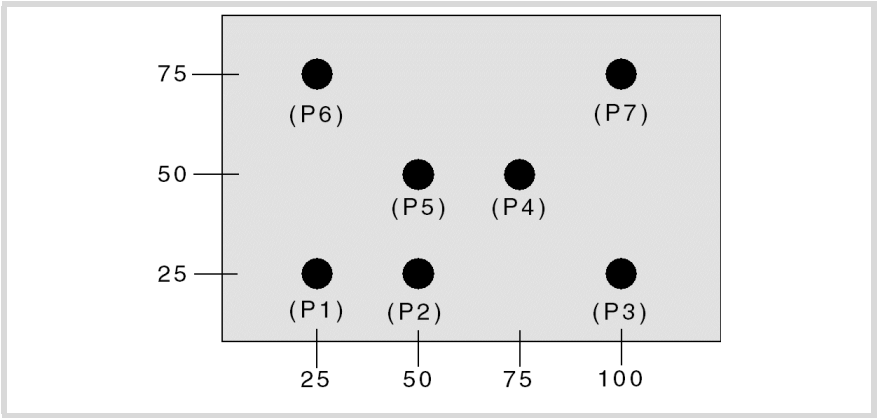
9.5 Random multiple machining.



The starting point is the cycle defining point. The rest of the points (P2) to (P12) must be defined in the area for multiple machining.

9.5.1 Programming example.

The canned cycle defined at point X25, Y25 is to be repeated at the rest of the points.



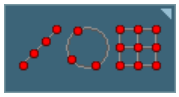
The canned cycle is defined at point (P1) X25, Y25. The rest of the points (P2) to (P7) must be defined in the area for multiple machining.

(P2)	X 50	Y 25
(P3)	X 100	Y 25
(P4)	X 75	Y 50
(P5)	X 50	Y 50
(P6)	X 25	Y 75
(P7)	X 100	Y 75

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES. LONGITUDINAL MACHINING. 10

When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.

- Multiple center-punching.
- Multiple drilling.
- Multiple tapping.
- Multiple reaming.
- Multiple boring.
- Multiple bore milling.
- Multiple thread milling.
- Multiple slot milling.



When these cycles are executed in planes with C axis or Y axis, multiple machining may be added to them so the cycle is repeated in several points. The available multiple machining operations are the following. See chapter *"9 Canned cycles of the editor. Multiple positioning."*.

- Points in line.
- Points in arc.
- Points in rectangle.
- Points in grid.
- Random points.

To associate multiple machining to a cycle, first select and define a machining cycle from those allowed. Then, without quitting the editing of the cycle, press the softkey associated with multiple machining and select one of them.

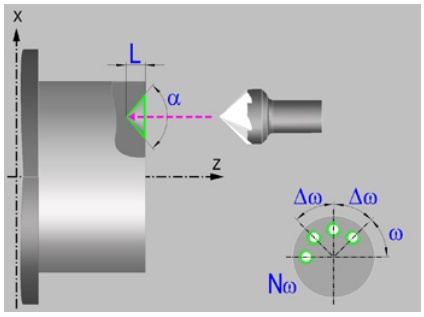
Selecting the positioning rotary axis for multiple machining.

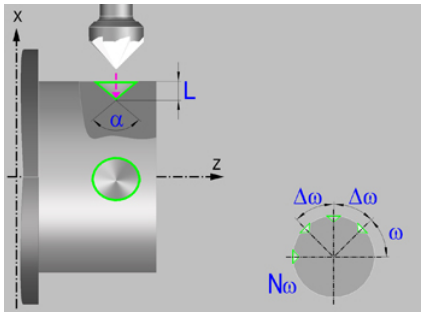
The cycle uses the following criteria for selecting the positioning rotary axis for multiple machining.

1. The C axis that was active before executing the cycle, programmed with #CAX if it is a spindle or #FACE or #CYL if it is an axis.
2. The master spindle before the cycle, if its machine parameter CAXIS=YES.
3. The first non-master spindle whose machine parameter CAXIS=YES.
4. The first axis whose machine parameter CAXIS=YES.
5. M19 with master spindle.

10.1 Multiple center-punching.

The cycle allows multiple center punching on the side of the part or on its face.





Geometric parameters.

Machining on the face or side of the part.

Machining on the face of the part.

Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

Work plane.

Positioning on the XZ axes.

Positioning on the A axis.

Positioning on the B axis.

Positioning on the C axis.

Positioning on the Y axis.

Coordinates of the profile's starting and final points.

If the starting and end point coordinates are not programmed, the cycle assumes the point where the axes are located when executing the cycle.

X, Z — Coordinates of the initial starting point for the machining. In the ZX plane, the X coordinate must be given in the active units, radius or diameter. With rotary axis or Y axis, the X coordinate must always be programmed in radii.

Y — Y coordinate of the first positioning movement.

A,B,C — Coordinate of the first positioning movement, on the rotary axis.

Coordinates of the starting point.

Xi, Yi, Zi — Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

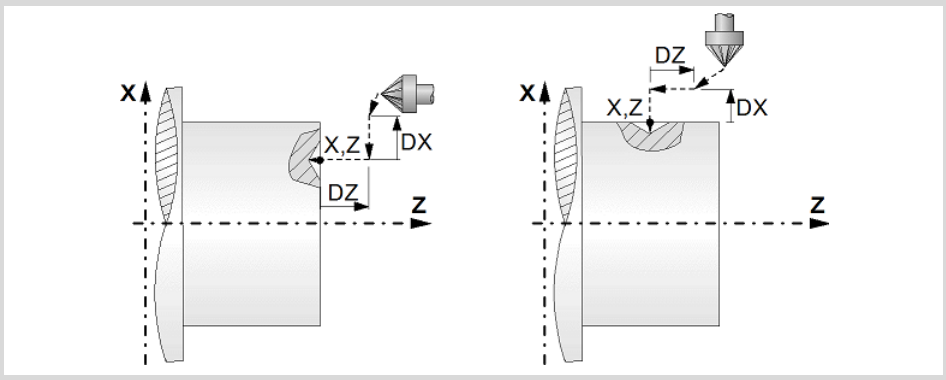
Coordinates of the end point.

Xf, Yf, Zf — Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.



- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.

Programming the center punching operation.

Programming the depth of the center punching operation.

Programming the angle and the diameter of the center punching operation.

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

α Center-punching angle.

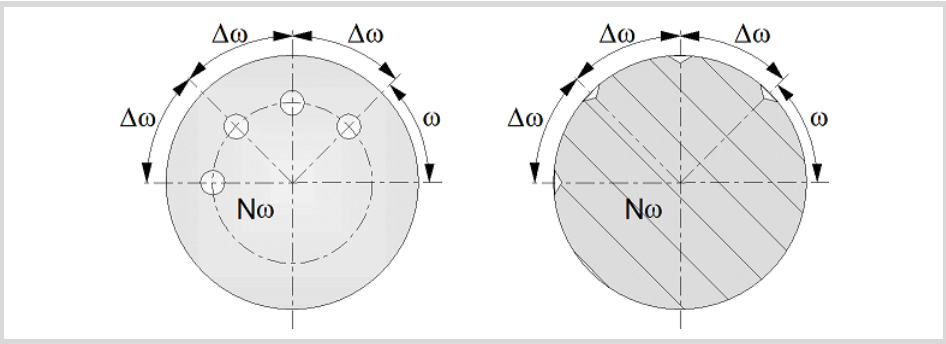
ϕ Center-punching diameter.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

$\Delta\omega$ Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

$N\omega$ Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Machining parameters.

Dwell at the bottom.

t The waiting time in seconds, after center punching, until the withdrawal movement starts.

Penetration feedrate.

F Penetration feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Tool data.

T Tool.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

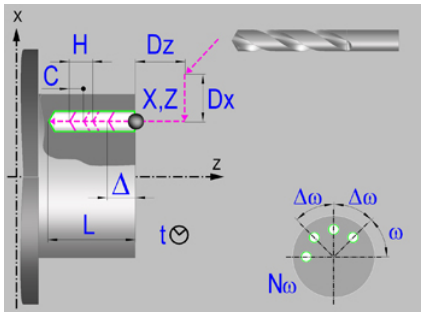
10.1.1 Basic operation.

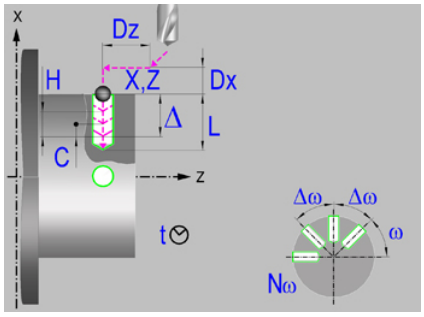
The machining steps in this cycle are as follows:

1. If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
2. The cycle turns the live tool at the indicated rpm.
3. The cycle orients the spindle to the angular position corresponding to the initial machining operation.
4. The tool approaches in rapid feedrate up to the safety point, at a distance of Dx and Dz of the starting point.
5. Rapid movement to the machining position, in X for the face and in Z for the side of the part (approach point).
6. Center punching of the part in work feedrate, until the programmed depth is reached.
7. Dwell at the bottom, if it has been programmed.
8. Fast backwards to the approach point.
9. Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in point 6.
10. Once the operation or cycle has ended, the tool will return to the safety point.
11. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

10.2 Multiple drilling.

The cycle allows multiple drilling on the side of the part or on its face.





Geometric parameters.

Machining on the face or side of the part.

Machining on the face of the part.

Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

Work plane.

Positioning on the XZ axes.

Positioning on the A axis.

Positioning on the B axis.

Positioning on the C axis.

Positioning on the Y axis.

Coordinates of the profile's starting and final points.

If the starting and end point coordinates are not programmed, the cycle assumes the point where the axes are located when executing the cycle.

X, Z — Coordinates of the initial starting point for the machining. In the ZX plane, the X coordinate must be given in the active units, radius or diameter. With rotary axis or Y axis, the X coordinate must always be programmed in radii.

Y — Y coordinate of the first positioning movement.

A,B,C — Coordinate of the first positioning movement, on the rotary axis.

Coordinates of the starting point.

Xi, Yi, Zi — Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

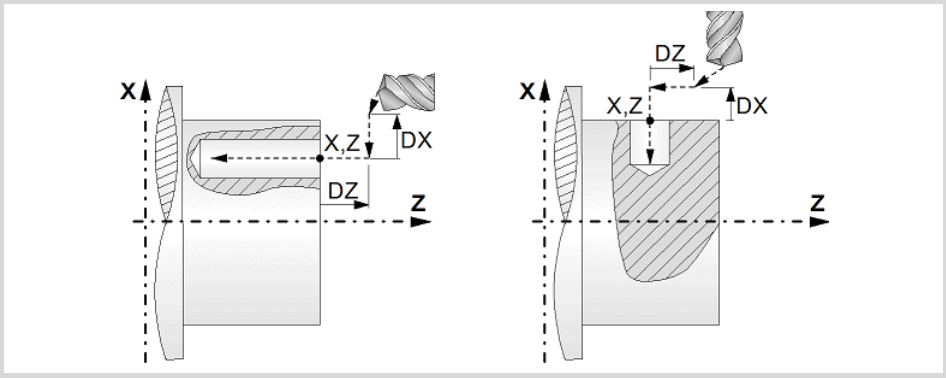
Xf, Yf, Zf — Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.



Programming of the drilling operation.

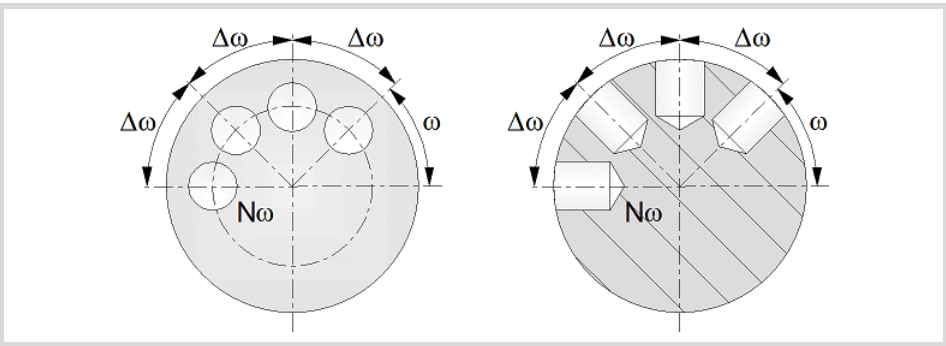
L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Machining parameters.

Drilling step (peck); constant-peck or variable-peck drilling.

Δ Maximum drilling peck (step).

KΔ Reduction factor of the drilling peck; if not programmed, the cycle will assume a factor of 1.

δ Minimum drilling step (peck); constant-peck or variable-peck drilling.

The first drilling peck will be "Δ", the second one "KΔ D", the third one "KΔ (KΔ D)" and so on until reaching the minimum peck; in other words, from the second peck on, the new peck will be the result of multiplying the previous peck by KΔ.

If the minimum drilling peck is greater than the maximum peck (δ > Δ), the cycle carries out a drilling at a constant peck equal to the minimum peck "δ". If the minimum drilling peck is smaller than the maximum peck (δ < Δ), the cycle carries out a drilling at a variable peck.

Withdrawal and approach distance in each penetration.

H Withdrawal distance after each peck; if not programmed, the cycle will assume a value of 1 mm. When programming H=0, it will withdraw up to the Z coordinate of the safety point.

C Approach distance in each penetration. If parameter C is not assigned a value or 0 is assigned, it assumes 1 mm by default.

Dwell at the bottom.

t The waiting time in seconds, after drilling, until the withdrawal movement starts.

Penetration feedrate.

F Penetration feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Tool data.

T Tool.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

10.2.1 Basic operation.

The machining steps in this cycle are as follows:

1. If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
2. The cycle turns the live tool at the indicated rpm.
3. The cycle orients the spindle to the angular position corresponding to the initial machining operation.
4. The tool approaches in rapid feedrate up to the safety point, at a distance of Dx and Dz of the starting point.
5. Rapid movement to the machining position, in X for the face and in Z for the side of the part (approach point).
6. Drilling loop. Depending on the minimum and maximum pecks defined, the cycle will carry out a drilling operation at a constant peck or at a variable peck.

   If the minimum drilling peck is greater than the maximum peck ($\delta > \Delta$), the cycle carries out a drilling at a constant peck equal to the minimum peck "δ". If the minimum drilling peck is smaller than the maximum peck ($\delta < \Delta$), the cycle carries out a drilling at a variable peck.

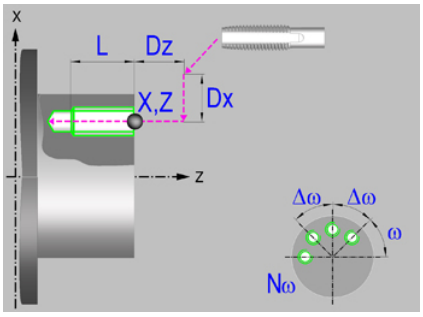
   In a drilling operation at variable peck, parameter Δdefines the drilling peck and KΔ the reduction factor for this peck. The first drilling peck will be "Δ", the second one "KΔ D", the third one "KΔ (KΔ D)" and so on until reaching the minimum peck; in other words, from the second peck on, the new peck will be the result of multiplying the previous peck by KΔ.

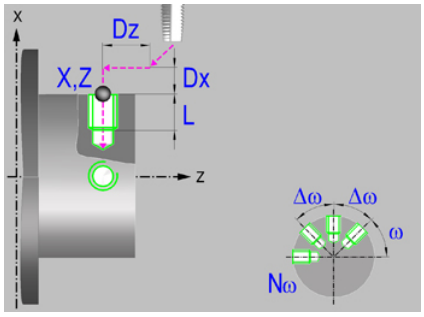
   In either case, the cycle repeats the following steps until reaching the programmed depth.

   First, rapid approach to the distance indicated by parameter C with respect to the previous penetration. Than, drilling down to the next penetration. And last, rapid withdrawal to the distance indicated by parameter H. When programming H=0, it will withdraw up to the Z coordinate of the safety point.
7. Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in point 6.
8. Once the operation or cycle has ended, the tool will return to the safety point.
9. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

10.3 Multiple tapping.

The cycle allows multiple tapping on the side of the part or on its face.





Geometric parameters.

Machining on the face or side of the part.

Machining on the face of the part.

Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

Work plane.

Positioning on the XZ axes.

Positioning on the A axis.

Positioning on the B axis.

Positioning on the C axis.

Positioning on the Y axis.

Coordinates of the profile's starting and final points.

If the starting and end point coordinates are not programmed, the cycle assumes the point where the axes are located when executing the cycle.

X, Z — Coordinates of the initial starting point for the machining. In the ZX plane, the X coordinate must be given in the active units, radius or diameter. With rotary axis or Y axis, the X coordinate must always be programmed in radii.

Y — Y coordinate of the first positioning movement.

A,B,C — Coordinate of the first positioning movement, on the rotary axis.

Coordinates of the starting point.

Xi, Yi, Zi — Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

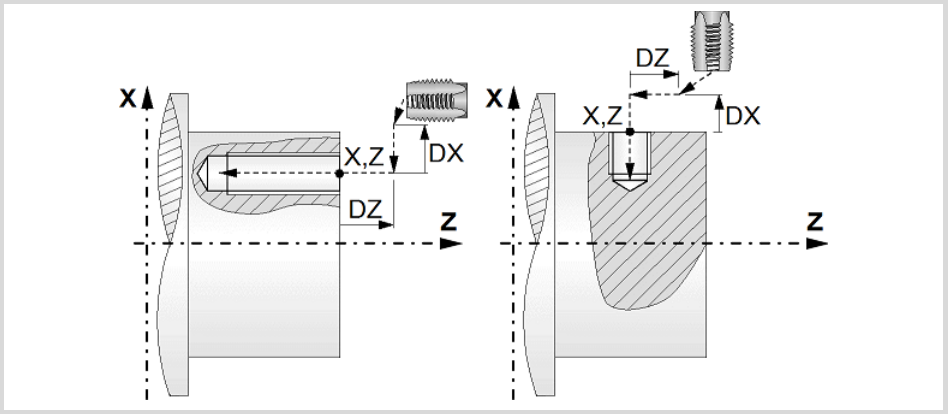
Xf, Yf, Zf — Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.



Tapping type.

Tapping with a clutch.

Rigid tapping. The spindle must have a servo-motor-drive system and an encoder.

Programming of the tapping operation.

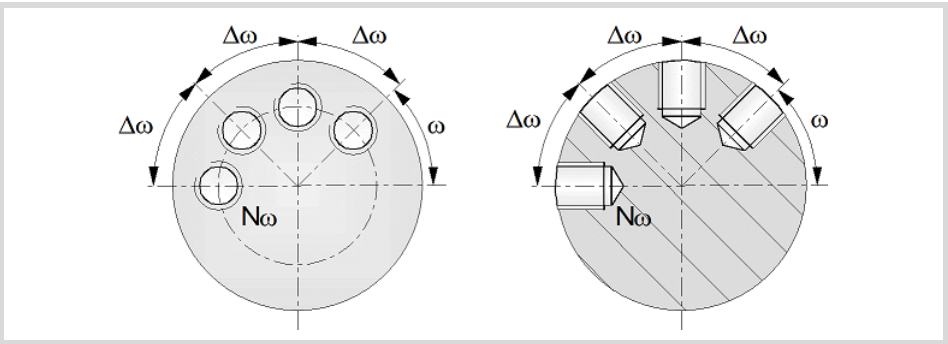
L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Machining parameters.

Define the thread.

Define the thread with the feedrate.

Define the thread with the pitch.

F Penetration feedrate.

P Thread pitch.

Dwell at the bottom.

t The waiting time in seconds, after tapping, until the withdrawal movement starts.

Live-tool data.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Tool data.

T Tool.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

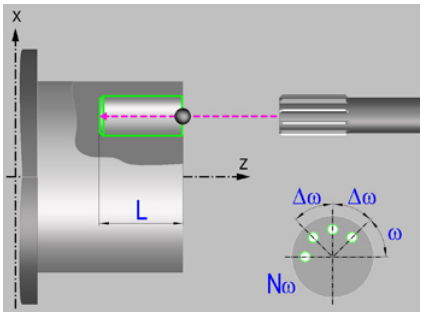
10.3.1 Basic operation.

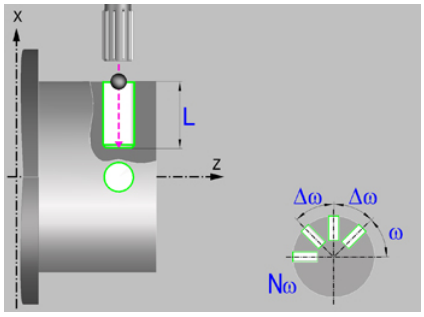
The machining steps in this cycle are as follows:

1. If the spindle is working in open loop (RPM or CSS mode) the CNC stops the spindle and performs a home search on the spindle (Io).
2. If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
3. The cycle turns the live tool at the indicated rpm.
4. The cycle orients the spindle to the angular position corresponding to the initial machining operation.
5. The tool approaches in rapid feedrate up to the safety point, at a distance of Dx and Dz of the starting point.
6. Rapid movement to the machining position, in X for the face and in Z for the side of the part (approach point).
7. Tapping of the part in work feedrate, until the programmed depth is reached.
8. Reversal of the turning direction of the live tool.
9. Fast backwards to the approach point.
10. Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in points 7, 8 and 9.
11. Once the operation or cycle has ended, the tool will return to the safety point.
12. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

10.4 Multiple reaming.

The cycle allows multiple reaming on the side of the part or on its face.





Geometric parameters.

Machining on the face or side of the part.

Machining on the face of the part.

Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

Work plane.

Positioning on the XZ axes.

Positioning on the A axis.

Positioning on the B axis.

Positioning on the C axis.

Positioning on the Y axis.

Coordinates of the profile's starting and final points.

If the starting and end point coordinates are not programmed, the cycle assumes the point where the axes are located when executing the cycle.

X, Z — Coordinates of the initial starting point for the machining. In the ZX plane, the X coordinate must be given in the active units, radius or diameter. With rotary axis or Y axis, the X coordinate must always be programmed in radii.

Y — Y coordinate of the first positioning movement.

A,B,C — Coordinate of the first positioning movement, on the rotary axis.

Coordinates of the starting point.

Xi, Yi, Zi — Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

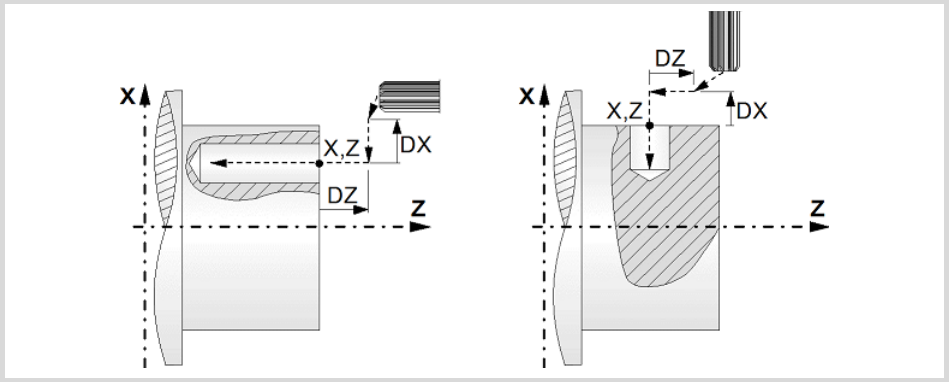
Xf, Yf, Zf — Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.



Programming the reaming operation.

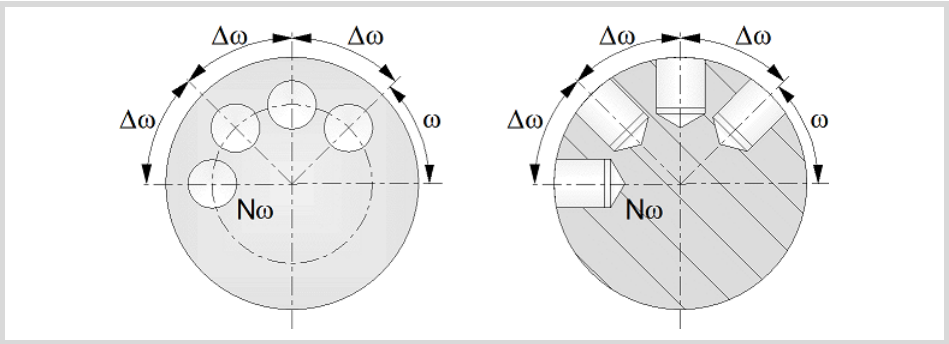
L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Machining parameters.

Dwell at the bottom.

t The waiting time in seconds, after reaming, until the withdrawal movement starts.

Penetration feedrate.

F Penetration feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Tool data.

T Tool.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

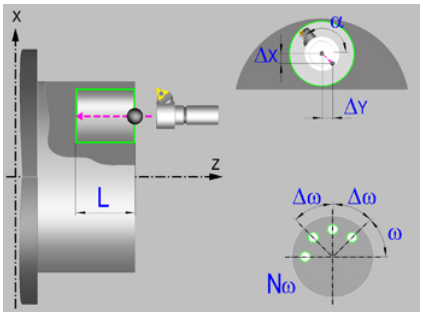
10.4.1 Basic operation.

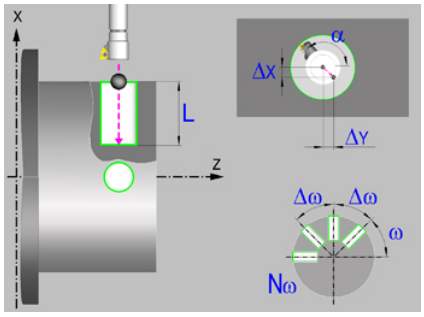
The machining steps in this cycle are as follows:

1. If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
2. The cycle turns the live tool at the indicated rpm.
3. The cycle orients the spindle to the angular position corresponding to the initial machining operation.
4. The tool approaches in rapid feedrate up to the safety point, at a distance of Dx and Dz of the starting point.
5. Rapid movement to the machining position, in X for the face and in Z for the side of the part (approach point).
6. Reaming of the part in work feedrate, until the programmed depth is reached.
7. Dwell at the bottom, if it has been programmed.
8. Fast backwards to the approach point.
9. Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in point 6.
10. Once the operation or cycle has ended, the tool will return to the safety point.
11. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

10.5 Multiple boring.

The cycle allows multiple boring on the side of the part or on its face.





Geometric parameters.

Machining on the face or side of the part.

Machining on the face of the part.

Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

Work plane.

Positioning on the XZ axes.

Positioning on the A axis.

Positioning on the B axis.

Positioning on the C axis.

Positioning on the Y axis.

Geometric parameters.

Coordinates of the profile's starting and final points.

If the starting and end point coordinates are not programmed, the cycle assumes the point where the axes are located when executing the cycle.

X, Z — Coordinates of the initial starting point for the machining. In the ZX plane, the X coordinate must be given in the active units, radius or diameter. With rotary axis or Y axis, the X coordinate must always be programmed in radii.

Y — Y coordinate of the first positioning movement.

A,B,C — Coordinate of the first positioning movement, on the rotary axis.

Coordinates of the starting point.

Xi, Yi, Zi — Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

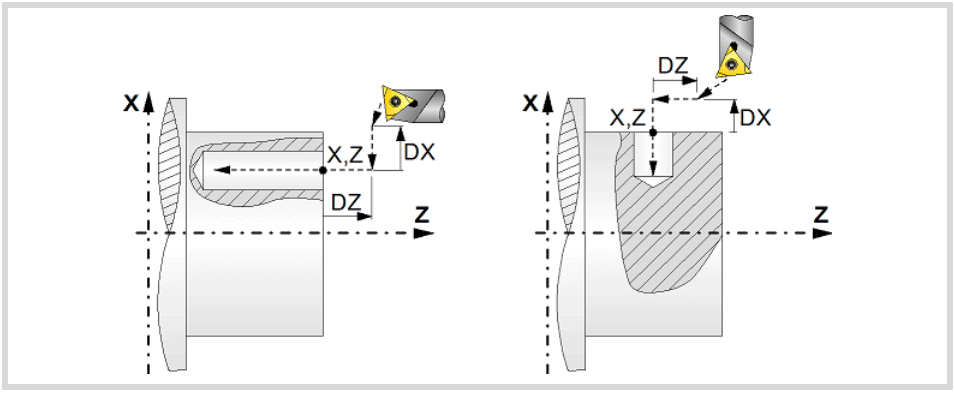
Xf, Yf, Zf Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.



Programming the boring operation.

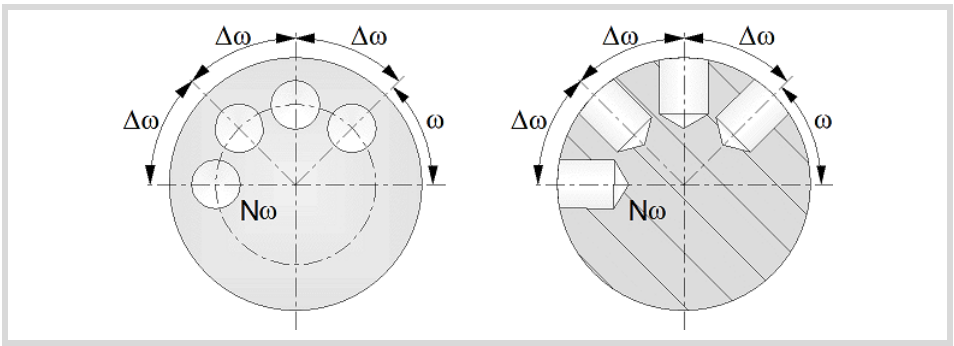
L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Machining parameters.

Type of withdrawal.

Withdrawal at work feedrate while the spindle is turning.

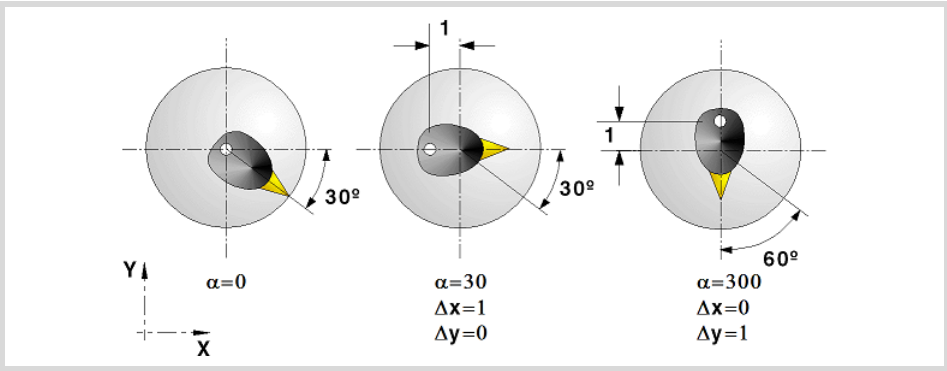
Rapid withdrawal while the spindle is stopped and the tool is touching the surface.

Rapid withdrawal with spindle orientation and the tool is not touching the surface. This type of withdrawal is not available for XZ positioning, nor for C axis positioning on the cylindrical side of the part.

α Angular position, in degrees, of the spindle of the live tool for the withdrawal.

ΔX X distance away from (off) the wall of the hole; if not programmed, the cycle will assume a 0 value.

ΔY Y distance away from (off) the wall of the hole; if not programmed, the cycle will assume a 0 value.



Dwell at the bottom.

t The waiting time in seconds, after boring, until the withdrawal movement starts.

Penetration feedrate.

F Penetration feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Tool data.

T Tool.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

10.5.1 Basic operation.

The machining steps in this cycle are as follows:

1. If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
2. The cycle turns the live tool at the indicated rpm.
3. The cycle orients the spindle to the angular position corresponding to the initial machining operation.
4. The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z (safety point).
5. Rapid movement to the machining position, in X for the face and in Z for the side of the part (approach point).
6. Boring of the part in work feedrate, until the programmed depth is reached.
7. Dwell at the bottom, if it has been programmed.
8. Tool withdrawal up to the approach point, according to the selected mode.

   Withdrawal at work feedrate while the spindle is turning.

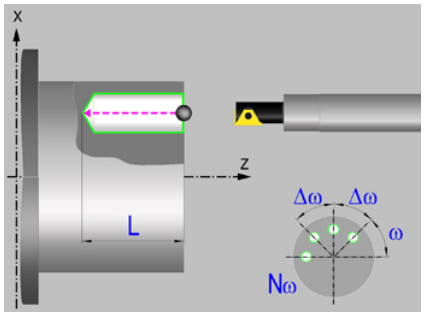
   Rapid withdrawal while the spindle is stopped and the tool is touching the surface.

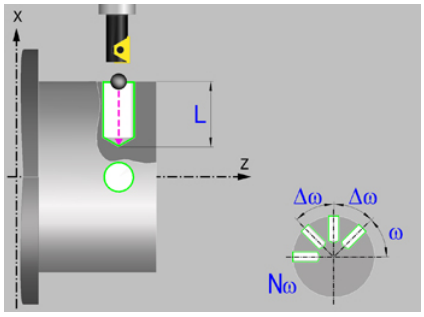
   Rapid withdrawal with spindle orientation and the tool has been retracted away from the surface.
9. Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in points 6, 7 and 8.
10. Once the operation or cycle has ended, the tool will return to the safety point.
11. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

10.6 Multiple bore milling.

The cycle allows multiple bore milling on the side of the part or on its face.





Geometric parameters.

Machining on the face or side of the part.

Machining on the face of the part.

Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

Coordinates of the profile's starting and final points.

If the starting and end point coordinates are not programmed, the cycle assumes the point where the axes are located when executing the cycle.

X X coordinate of the hole center of each machining operation. The coordinate must be given in the active units, radius or diameter.

Z Z coordinate of the hole center of each machining operation.

Coordinates of the end point.

Xf, Yf, Zf Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

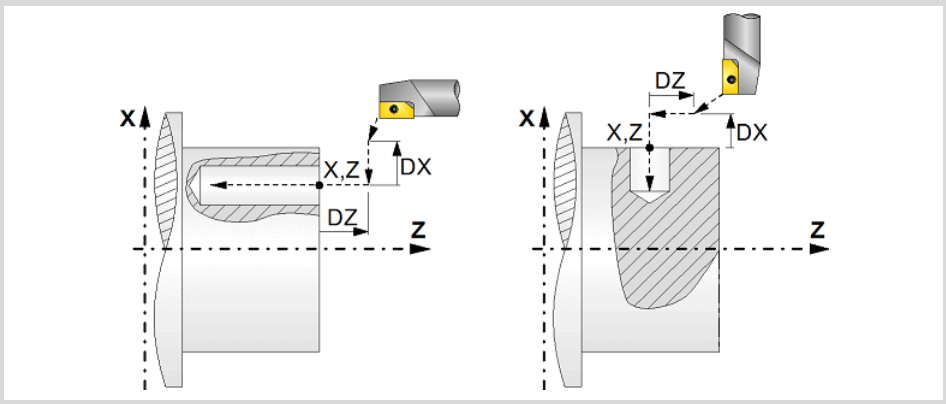
Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109



Programming the machining operation.

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

Ø Hole diameter.

Øk Pre-hole diameter.

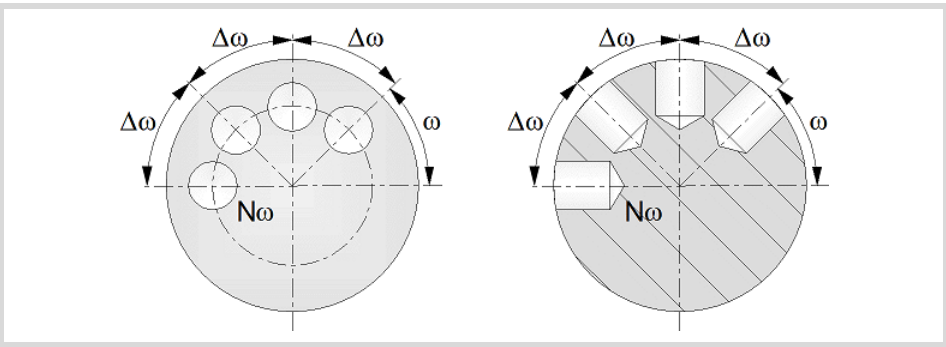
When starting from a hole that was machined earlier, parameter Øk defines the diameter of that hole; if not programmed or programmed with a 0 value, the cycle assumes that there is no previous hole.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Machining parameters.

Helical machining.

Counterclockwise machining.

Clockwise machining.

B Helical drilling step.

Bottom milling.

Straight finishing. The cycle redoes the bottom of the hole (blind hole).

Helical finishing. The cycle does not redo the bottom of the hole (thru hole).

Penetration feedrate.

F Penetration feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Tool data.

T Tool.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

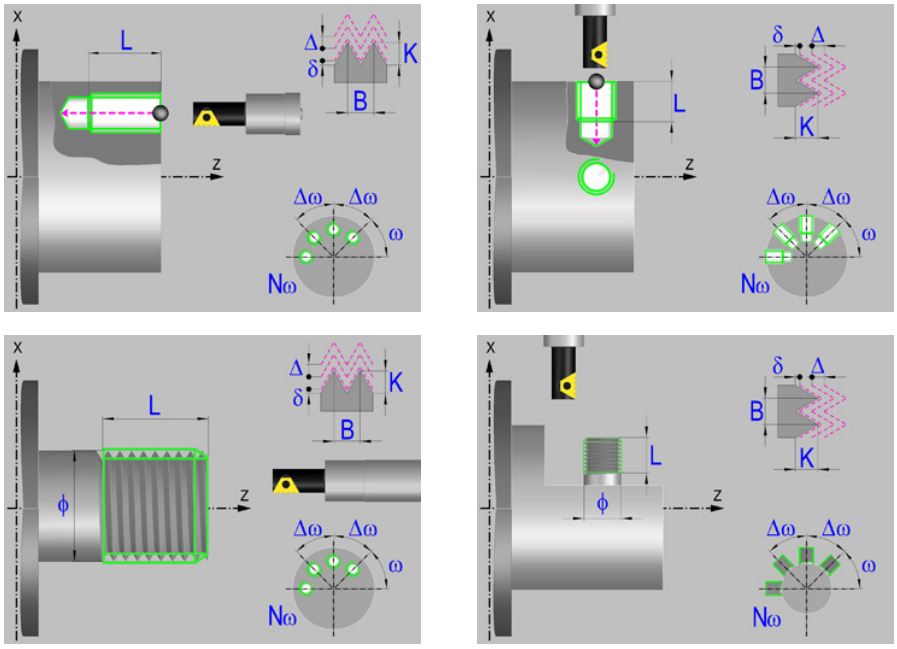
10.6.1 Basic operation.

The machining steps in this cycle are as follows:

1. If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
2. The cycle turns the live tool at the indicated rpm.
3. The cycle orients the spindle to the angular position corresponding to the initial machining operation.
4. The tool approaches in rapid up to the starting point, keeping the safety distance selected according to axes X and Z (safety point).
5. Rapid movement to the tangential entry point (approach point).
6. Tangential entry to the helical path of the drilling.
7. Helical movement, with the given pitch and in the given direction, down to the bottom of the hole.
8. Redo the bottom of the hole, if it has been programmed.
9. Tangential exit movement to the helical path of the drilling to the center of the hole.
10. Fast backwards to the approach point.
11. Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in points 6 to 10.
12. Once the operation or cycle has ended, the tool will return to the safety point.
13. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

10.7 Multiple thread milling.

The cycle allows multiple thread milling on the side of the part or on its face.



Geometric parameters.

Machining on the face or side of the part.

Machining on the face of the part.

Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

Inside or outside machining.

Inside machining.

Outside machining.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

Coordinates of the profile's starting and final points.

If the starting and end point coordinates are not programmed, the cycle assumes the point where the axes are located when executing the cycle.

X X coordinate of the hole center of each machining operation. The coordinate must be given in the active units, radius or diameter.

Z Z coordinate of the hole center of each machining operation.

Coordinates of the end point.

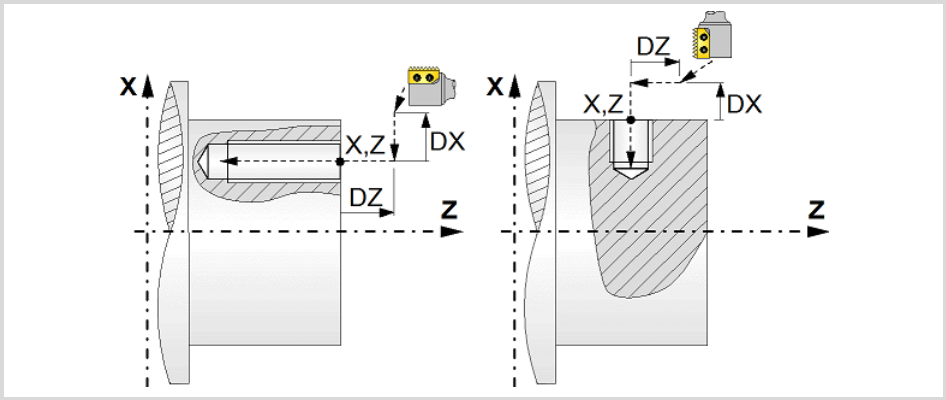
Xf, Yf, Zf Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.

- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.



Programming the machining operation.

L Machining depth. The parameter will be positive when machining towards negative coordinates of the penetrating axis and negative when machining towards positive coordinates.

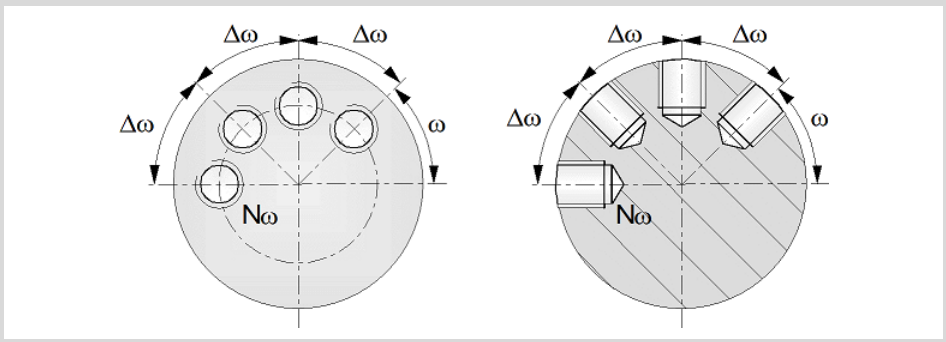
Ø Nominal thread diameter.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Machining parameters.

Helical machining.

Counterclockwise machining.

Clockwise machining.

Machining direction.

Machining from the part surface towards the bottom of the thread.

Machining from the bottom of the thread towards the part surface.

Tool type.

Tool for threading in a single pass.

Tool with a single edge cutter.

Tool with a multi-edge cutter. The number of edges of the tool must be defined.

N — Number of edges. If it is set to 0, the cycle execute a threading of a single pitch.

Thread data.

B — Thread pitch.

K — Depth of the thread (distance between the crest and the root of the thread).

Δ — Maximum penetration step of the thread. If this data is not programmed or programmed with a 0 value, the cycle will run a single pass down to the finishing stock.

δ — Finishing stock; if not programmed, the cycle assumes a value of 0.

α — Thread entry (start) angle. Angle (in degrees) of the segment formed by the center of the hole or of the boss and the thread entry point with respect to the abscissa axis. If not programmed, the cycle will take the value of 0.

Ds — Safety distance to the wall of the thread for the threading entry (start). If not programmed or programmed with a 0 value, the cycle will enter the thread from the center of the hole if it is an inside threading or it will issue an error message if it is an outside threading.

Penetration feedrate.

F — Penetration feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Tool data.

T Tool.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

10.7.1 Basic operation.

The machining steps in this cycle are as follows:

1. If the operation was programmed with another tool, the CNC will make a tool change moving to the change point if the machine so requires.
2. The cycle turns the live tool at the indicated rpm.
3. The tool approaches in rapid feedrate up to the safety point, at a distance of Dx and Dz of the starting point.
4. The cycle orients the spindle to the angular position corresponding to the initial machining operation.
5. Rapid movement to the machining point, in X for the face and in Z for the side of the part (approach point).
6. Rapid movement to the thread entry point, calculated by the cycle.
7. Machining depending on the selected tool.

- Tool for threading in a single pass.

  Helical movement, with the indicated direction and pitch, to the bottom of the thread (the movement will only be one revolution).

  Helical thread exiting movement, tangent to the previous helical path.

  It must be borne in mind that in the exit tangent to the helical path, the exit point will exceed the coordinate of the bottom of the thread along the longitudinal axis.

- Tool with a single edge cutter.

  Helical movement, with the indicated pitch and direction, to the bottom of the thread.

  Helical thread exiting movement, tangent to the previous helical path.

  It must be borne in mind that in the exit tangent to the helical path, the exit point will exceed the coordinate of the bottom of the thread along the longitudinal axis.

- Tool with a multi-edge cutter.

  Helical movement, with the indicated pitch and direction (the movement will be one revolution).

  Helical thread exiting movement, tangent to the previous helical path.

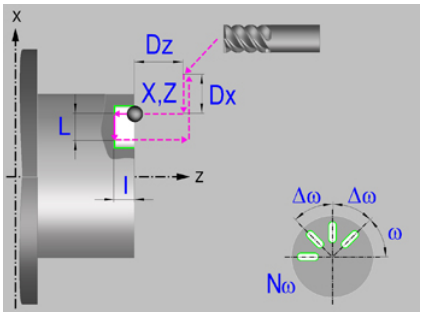
  Rapid movement to the thread entry point of the next threading path.

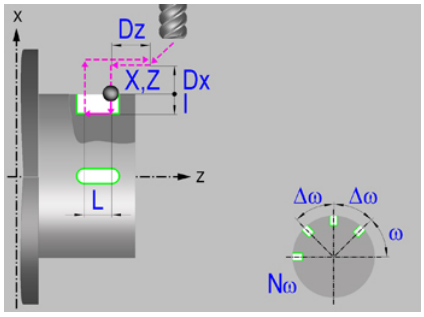
  Repetition of the previous 2 steps until reaching the bottom of the thread. It must be borne in mind that in the last helical exit, the exit point will exceed the coordinate of the bottom of the thread along the longitudinal axis.

8. Fast backwards to the approach point.
9. Depending on the number of machining operations programmed, the spindle orients to the second point and performs a new machining operation as described in points 6, 7 and 8.
10. Once the operation or cycle has ended, the tool will return to the safety point.
11. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

10.8 Multiple slot milling.

Multiple slots may be milled out on the side of the part or on its face.





Machining on the face or side of the part.

Machining on the face of the part.

Machining on the side of the part.

Each time the type of machining is changed the cycle modifies the icon and displays the corresponding geometrical help screen.

Work plane.

Positioning on the XZ axes.

Positioning on the A axis.

Positioning on the B axis.

Positioning on the C axis.

Positioning on the Y axis.

Coordinates of the profile's starting and final points.

If the starting and end point coordinates are not programmed, the cycle assumes the point where the axes are located when executing the cycle.

X, Z — Coordinates of the initial starting point for the machining. In the ZX plane, the X coordinate must be given in the active units, radius or diameter. With rotary axis or Y axis, the X coordinate must always be programmed in radii.

Y — Y coordinate of the first positioning movement.

A,B,C — Coordinate of the first positioning movement, on the rotary axis.

Coordinates of the starting point.

Xi, Yi, Zi — Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

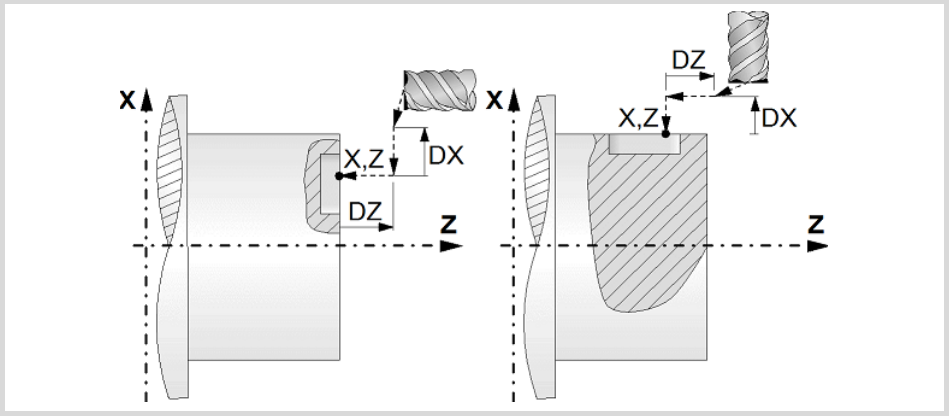
Xf, Yf, Zf — Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx, Dz Safety distance referred to the starting point.

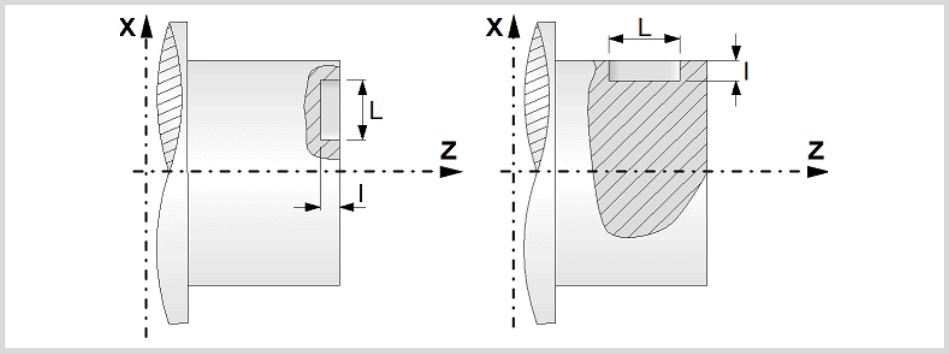
- Machining on the face of the part. The value of the safety distance on X is always defined in radius and it may be either positive or negative. The value of the safety distance in Z is always assumed with positive values.
- Machining on the side of the part. The value of the safety distance on X is always defined in radius and it is always assumed with positive values. The value of the safety distance in Z may be either negative or positive.



Programming of the drilling operation.

I Depth of the slot.

L Length of the slot.

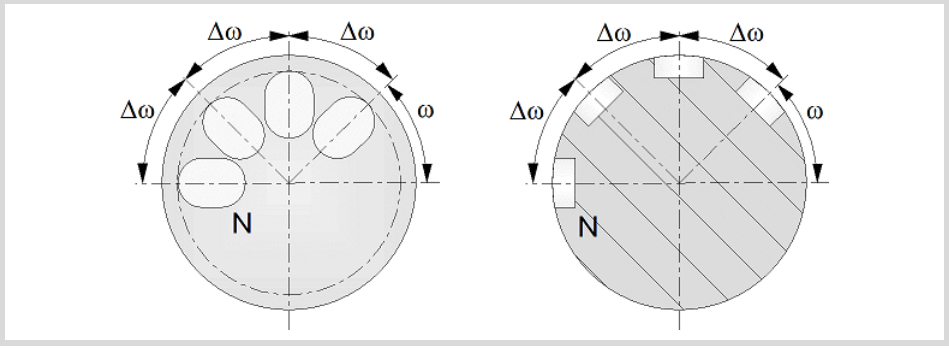


Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.



Live-tool parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Machining parameters. Roughing.

Penetration step.

Δ Maximum penetration step. If not programmed or programmed with a 0 value, it will run a single pass up to the finishing stock.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed.

Feedrate.

F Roughing feedrate.

Fp Penetration feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Tool data.

T Tool.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activating the execution of M functions before machining. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivating the execution of M functions before machining.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Machining parameters. Finishing.

Finishing stock at the bottom.

δ Finishing stock at the bottom.

Feedrate.

F Finishing feedrate.

Live-tool data.

S Live tool turning speed.

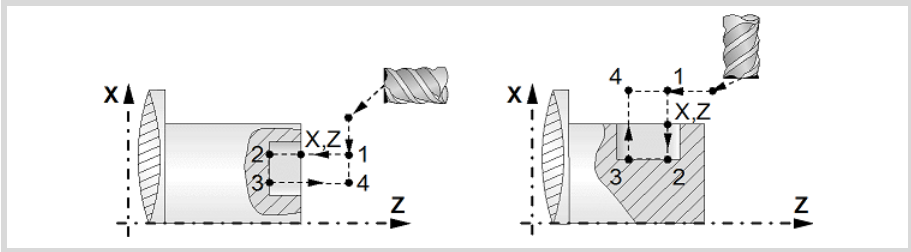
10.8.1 Basic operation.

The machining steps in this cycle are as follows:

1. If the operation was programmed with another tool the CNC will make a tool change, moving to the safety point.
2. The cycle turns the live tool at the indicated rpm.
3. The tool approaches in rapid feedrate up to the safety point, at a distance of Dx and Dz of the starting point.
4. The cycle orients the spindle to the angular position corresponding to the initial slot.
5. Rapid movement to the slot milling position, in X for the face and in Z for the side of the part (approach point).
6. Milling of the slot by following these steps:

   The cycle runs successive roughing passes until reaching the bottom of the slot respecting the finishing stocks. It then executes a single finishing pass eliminating the finishing stock.

   Each pass is executed as follows. First, penetration at programmed feedrate (segment 1-2). Then, mill the slot by moving the X or Z axis accordingly (segment 2-3). And last, withdrawal to the approach point (sections 3-4 and 4-1).



7. Depending on the number of machining operations programmed, the spindle orients to the second point and mills a new slot as described in point 6.
8. Once the operation or cycle has ended, the tool will return to the safety point.
9. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

10.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

Multiple slot milling.

FAGOR AUTOMATION

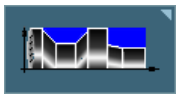
CNCelite
8058 8060
8065

REF. 2109

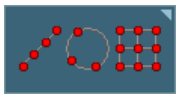
CANNED CYCLES OF THE EDITOR FOR ROTARY AXES. PROFILES.

11

When pressing the softkey, the editor shows the cycle used last. When pressing the same softkey again, the menu shows all the cycles of the group.



- Profile in the ZC/YZ plane.
- Rectangular pocket in the ZC/YZ plane.
- Circular pocket in the ZC/YZ plane.
- Circular pre-emptied pocket in the ZC/YZ plane.
- 2D profile pocket in the ZC/YZ plane.
- Profile in the XC/XY plane.
- Rectangular pocket in the XC/XY plane.
- Circular pocket in the XC/XY plane.
- Circular pre-emptied pocket in the XC/XY plane.
- 2D profile pocket in the XC/XY plane.

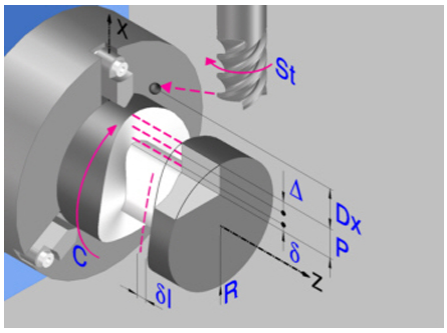


Multiple machining may be added to pocket cycles so the cycle is repeated in several points. The available multiple machining operations are the following. See chapter *"9 Canned cycles of the editor. Multiple positioning."*.

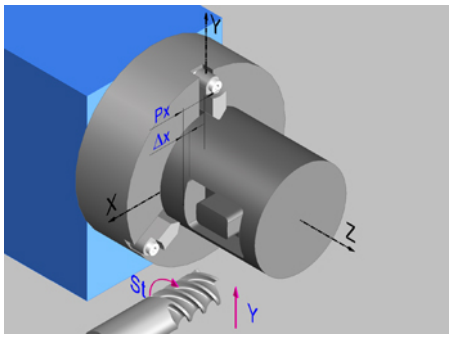
- Points in line.
- Points in arc.
- Points in rectangle.
- Points in grid.
- Random points.

To associate multiple machining to a cycle, first select and define a machining cycle from those allowed. Then, without quitting the editing of the cycle, press the softkey associated with multiple machining and select one of them.

11.1 ZC/YZ plane profiling cycle.



Profile in the ZC plane.



Profile in the YZ plane.

Geometric parameters.

Work plane.

ZC plane.

YZ plane.

Name of the program with the profile.

P Name of the program with the profile (between 0 and 999).

Radius of the cylinder (if ZC plane).

R Outside radius of the part.

Coordinates of the plane (if YZ plane).

X X coordinate of the work plane. The coordinate must always be programmed in radius.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

Coordinates of the starting point.

Xi, Yi, Zi Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

Xf, Yf, Zf Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Total depth.

P Total depth. This parameter must always be defined in radius and with a positive value.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point. The cycle also uses the safety distance for the withdrawal movement between successive roughing passes.

Dx X safety distance referred to the starting point. This parameter is always defined in radius.

Spindle angular position.

ω Angular position of the spindle, in degrees.

Define a profile.

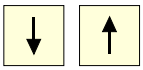
This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

Profile definition.

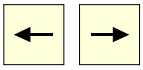
P Name of the program with the profile (between 0 and 999).

- Edit a new program with the profile. To edit a new program,write the program number (between 0 and 999) and press [RECALL]. The CNC shows the profile editor.
- To select a known program, key in the program number and press [ENTER].
- To select a program from the list, use the following keys.

Expand the program list.



Moves the cursor through the list of programs. To select a program, position the cursor on the desired program and press [ENTER].



Exit the list without selecting any program.

Modify an existing profile.

To edit an existing profile program, key in the program number and press [RECALL]. The profile editor of the CNC will show the selected profile, from where the following operations may be carried out.

- Add, delete or modify elements of the profile.
- Add, delete or modify chamfers, roundings, etc.
- Add new profiles to the program.

Spindle parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Tool radius compensation.

Without compensation

Left-hand tool radius compensation.

Right-hand tool radius compensation.

Penetration feedrate.

Fx Penetration feedrate.

Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the roughing operation.

Do not run the roughing operation.

Maximum penetration step.

Δ Maximum penetration step.

- If Δ is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If Δ is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the finishing operation.

Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

Number of penetration passes (steps) for the finishing operation.

N Number of penetration passes (steps) for the finishing operation. This parameter is only available when tool radius compensation is active.

Finishing stocks.

δl Side finishing stock. This parameter is only available when tool radius compensation is active.

δ Bottom finishing stock.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

11.1.1 Basic operation.

The machining steps in these cycles are the following:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
2. The live tool starts with the speed selected and in the indicated direction.
3. The tool approaches in rapid feedrate up to the safety point, at a distance of Dx of the starting point.
4. The spindle orients to the indicated "C" position.
5. Roughing operation (if it has been programmed).

   The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This cycle applies withdrawal between passes at the safety distance Dx. This operation is done with the conditions set for the roughing operation.
6. Finishing operation (if it has been programmed).

   If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

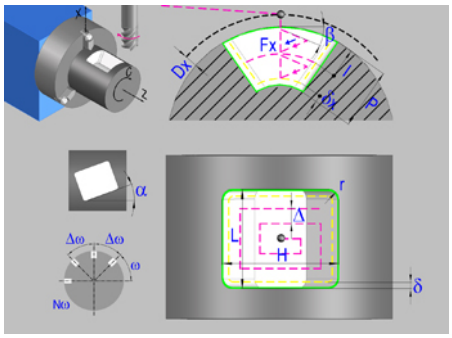
   The cycle eliminates the finishing stocks. If tool radius compensation is active, the cycle runs the indicated number of passes until the final profile is reached; if it is not active, the cycle runs a single pass. This operation is done with the conditions set for the finishing operation.
7. Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
8. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

Considerations.

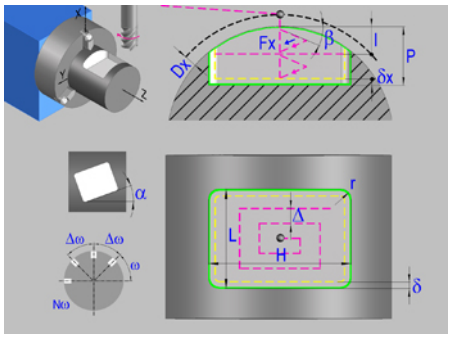
If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

11.2 Rectangular pocket cycle in the ZC/YZ plane.



Rectangular pocket in the ZC plane.



Rectangular pocket in the YZ plane.

Geometric parameters.

Work plane.

ZC plane.

YZ plane.

Starting point selection.

Starting point at the corner.

Starting point in the center.

Type of pocket corners.

Square (sharp) corner.

Rounded corner. The rounding radius must be defined.

Chamfered corner. The size of the chamfer must be defined.

r Rounding radius of the pocket corners.

c Size of the chamfer of the pocket corners.

Coordinates of the starting point.

Xc, Yc, Zc
Ac, Bc, Zc — Coordinates of the starting point. The CNC will show the associated coordinates for the selected plane.

Coordinates of the starting point.

Xi, Yi, Zi Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

Xf, Yf, Zf Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Pocket dimensions.

L Pocket length on the Z axis.

H Length of the pocket on the C axis (if ZC plane) or on the Y axis (if YZ plane).

Pocket orientation.

α Pocket angle referred to Z axis, in degrees.

Radius of the cylinder (if ZC plane).

R Outside radius of the part.

Coordinates of the plane (if YZ plane).

X X coordinate of the work plane. The coordinate must always be programmed in radius.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

Pocket depth.

P Pocket depth. This parameter is always defined in radius.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx X safety distance referred to the starting point. This parameter is always defined in radius.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.

Spindle parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Penetration feedrate.

Fx Penetration feedrate.

Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the roughing operation.

Do not run the roughing operation.

Maximum penetration step.

I Maximum penetration step.

- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

Sideways penetration angle.

β Sideways penetration angle.

Milling pass or width.

Δ Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction.

Clockwise.

Counterclockwise.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the finishing operation.

Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

Sideways penetration angle.

θ Sideways penetration angle.

Finishing stocks.

δ Side finishing stock.

δx Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction.

Clockwise.

Counterclockwise.

Number of side penetration passes (steps) for the finishing operation.

N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

11.2.1 Basic operation.

The machining steps in these cycles are the following:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
2. The live tool starts with the speed selected and in the indicated direction.
3. The tool approaches in rapid feedrate up to the safety point, at a distance of Dx of the starting point.
4. The spindle orients to the indicated "C" position.
5. Roughing operation (if it has been programmed).

   The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.
6. Finishing operation (if it has been programmed).

   If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

   The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

   The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed (δ, δx), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration pases as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.
7. Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
8. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

Considerations.

Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

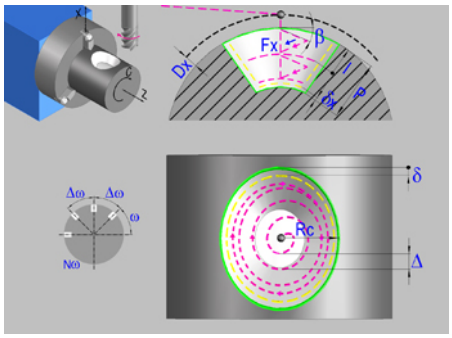
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

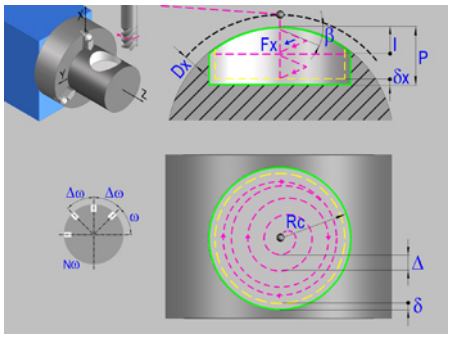
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

11.3 Circular pocket cycle in the ZC/YZ plane.



Circular pocket in the ZC plane.



Circular pocket in the YZ plane.

Geometric parameters.

Work plane.

ZC plane.

YZ plane.

Coordinates of the center of the pocket.

Xc, Yc, Zc
Ac, Bc, Zc — Coordinates of the center of the pocket. The CNC will show the associated coordinates for the selected plane.

Coordinates of the starting point.

Xi, Yi, Zi — Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

Xf, Yf, Zf — Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Pocket radius.

Rc — Pocket radius.

Radius of the cylinder (if ZC plane).

R — Outside radius of the part.

Coordinates of the plane (if YZ plane).

X — X coordinate of the work plane. The coordinate must always be programmed in radius.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

Pocket depth.

P — Pocket depth. This parameter is always defined in radius.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx X safety distance referred to the starting point. This parameter is always defined in radius.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.

Spindle parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Penetration feedrate.

Fx Penetration feedrate.

Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the roughing operation.

Do not run the roughing operation.

Maximum penetration step.

I Maximum penetration step.

- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

Sideways penetration angle.

β Sideways penetration angle.

Milling pass or width.

Δ Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction.

Clockwise.

Counterclockwise.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the finishing operation.

Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

Sideways penetration angle.

θ Sideways penetration angle.

Finishing stocks.

δ Side finishing stock.

δx Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction.

Clockwise.

Counterclockwise.

Number of side penetration passes (steps) for the finishing operation.

N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

11.3.1 Basic operation.

The machining steps in these cycles are the following:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
2. The live tool starts with the speed selected and in the indicated direction.
3. The tool approaches in rapid feedrate up to the safety point, at a distance of Dx of the starting point.
4. The spindle orients to the indicated "C" position.
5. Roughing operation (if it has been programmed).

   The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.
6. Finishing operation (if it has been programmed).

   If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

   The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

   The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed (δ, δx), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration pases as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.
7. Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
8. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

Considerations.

Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

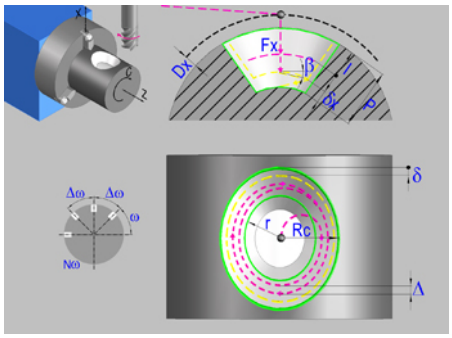
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

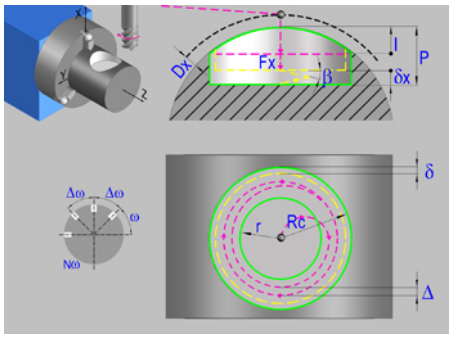
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

11.4 Circular pre-emptied pocket cycle in the ZC/YZ plane.



Circular pre-emptied pocket in the ZC plane.



Circular pre-emptied pocket in the YZ plane.

Geometric parameters.

Work plane.

ZC plane.

YZ plane.

Coordinates of the center of the pocket.

Xc, Yc, Zc
Ac, Bc, Zc — Coordinates of the center of the pocket. The CNC will show the associated coordinates for the selected plane.

Coordinates of the starting point.

Xi, Yi, Zi — Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

Xf, Yf, Zf — Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Pocket radius.

Rc — Pocket radius.

r — Inside radius of the pocket.

Radius of the cylinder (if ZC plane).

R — Outside radius of the part.

Coordinates of the plane (if YZ plane).

X — X coordinate of the work plane. The coordinate must always be programmed in radius.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

Pocket depth.

P — Pocket depth. This parameter is always defined in radius.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx X safety distance referred to the starting point. This parameter is always defined in radius.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.

Spindle parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Penetration feedrate.

Fx Penetration feedrate.

Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the roughing operation.

Do not run the roughing operation.

Maximum penetration step.

I Maximum penetration step.

- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

Sideways penetration angle.

β Sideways penetration angle.

If the tool fits in the pre-emptied area, it goes in straight in G0, ignoring the penetration angle. If the tool does not fit in the pre-emptied area, it goes in at the programmed penetration angle.

Milling pass or width.

Δ Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction.

Clockwise.

Counterclockwise.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the finishing operation.

Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

Sideways penetration angle.

θ Sideways penetration angle.

Finishing stocks.

δ Side finishing stock.

δx Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction.

Clockwise.

Counterclockwise.

Number of side penetration passes (steps) for the finishing operation.

N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

11.4.1 Basic operation.

The machining steps in these cycles are the following:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
2. The live tool starts with the speed selected and in the indicated direction.
3. The tool approaches in rapid feedrate up to the safety point, at a distance of Dx of the starting point.
4. The spindle orients to the indicated "C" position.
5. Roughing operation (if it has been programmed).

   The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.
6. Finishing operation (if it has been programmed).

   If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

   The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

   The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed (δ, δx), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration pases as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.
7. Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
8. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

Considerations.

Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

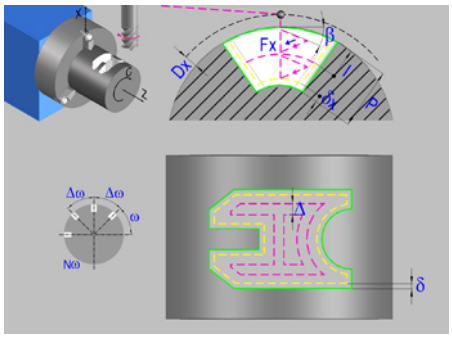
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

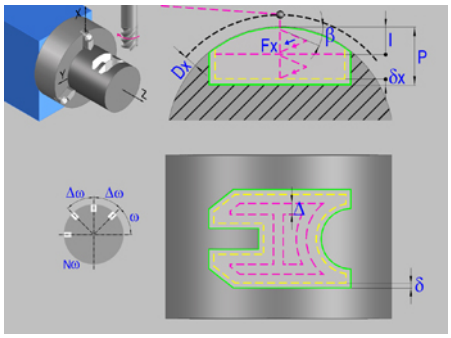
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

11.5 2D profile pocket cycle in the ZC/YZ plane.



2D profile pocket in the ZC plane.



2D profile pocket in the YZ plane.

Geometric parameters.

Work plane.

ZC plane.

YZ plane.

Name of the program with the profile.

P Name of the program with the profile (between 0 and 999).

Radius of the cylinder (if ZC plane).

R Outside radius of the part.

Coordinates of the plane (if YZ plane).

X X coordinate of the work plane. The coordinate must always be programmed in radius.

Coordinates of the starting point.

Xi, Yi, Zi Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

Xf, Yf, Zf Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Total depth.

P Total depth. This parameter must always be defined in radius and with a positive value.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dx X safety distance referred to the starting point. This parameter is always defined in radius.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.

Define a profile.

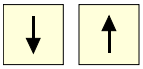
This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

Profile definition.

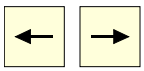
P Name of the program with the profile (between 0 and 999).

- Edit a new program with the profile. To edit a new program,write the program number (between 0 and 999) and press [RECALL]. The CNC shows the profile editor.
- To select a known program, key in the program number and press [ENTER].
- To select a program from the list, use the following keys.

Expand the program list.



Moves the cursor through the list of programs. To select a program, position the cursor on the desired program and press [ENTER].



Exit the list without selecting any program.

Modify an existing profile.

To edit an existing profile program, key in the program number and press [RECALL]. The profile editor of the CNC will show the selected profile, from where the following operations may be carried out.

- Add, delete or modify elements of the profile.
- Add, delete or modify chamfers, roundings, etc.
- Add new profiles to the program.

Spindle parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Penetration feedrate.

Fx Penetration feedrate.

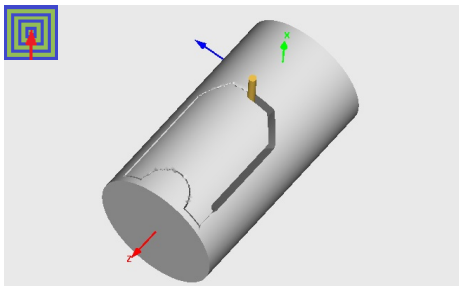
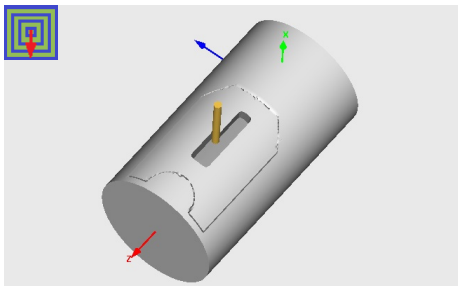
Machining strategy (icon).

Outward-inward concentric machining. The wall of the pocket is machined in the first pass. Depending on the geometry, this option may be faster, but the wear on the tool is greater.

- The starting point is near the initial point of the outer contour, and varies very little if the radius of the roughing tool is changed and does not vary at all if the roughing pass is changed.
- If a drilling operation is programmed before the roughing (or the bottom finishing if there is no roughing operation), the CNC chooses the optimal drilling point, as close as possible to the initial point of the outer contour.

Inward-outward concentric machining. The wall of the pocket is machined in the last pass. Depending on the geometry, this option maybe slower, but there is less wear on the tool. The CNC machines the pocket, attempting to minimize the number of sections where the tool machines over the largest surface, the number of penetrations in part sections and the machining time.

- The starting point does not depend on the initial point of the outer contour and can vary if either the tool radius or the machining pass is changed (milling width). In a pocket without islands, the initial point is found near the center of the pocket. In a pocket with islands, where different areas can be formed that require roughing, the initial point will be located in the center of the largest area.
- If a drilling operation is programmed before the roughing (or the bottom finishing, if there is no roughing operation), the CNC chooses the optimal drilling points (there may be more than one point).

2D profile pocket in the ZC/YZ plane.	2D profile pocket in the ZC/YZ plane.
	

Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the roughing operation.

Do not run the roughing operation.

Maximum penetration step.

I Maximum penetration step.

- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

Sideways penetration angle.

β Sideways penetration angle.

Milling pass or width.

Δ Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction (icon).

For pockets with islands, the machining of the outer profile is done in the direction indicated by the icon, where as the islands are machined in the opposite direction.

Clockwise.

Counterclockwise.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the finishing operation.

Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

Sideways penetration angle.

θ Sideways penetration angle.

Finishing stocks.

δ Side finishing stock.

δx Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction (icon).

For pockets with islands, the machining of the outer profile is done in the direction indicated by the icon, where as the islands are machined in the opposite direction.

Clockwise.

Counterclockwise.

Number of side penetration passes (steps) for the finishing operation.

N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

11.5.1 Basic operation.

The machining steps in these cycles are the following:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
2. The live tool starts with the speed selected and in the indicated direction.
3. The tool approaches in rapid feedrate up to the safety point, at a distance of Dx of the starting point.
4. The spindle orients to the indicated "C" position.
5. Roughing operation (if it has been programmed).

   The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.
6. Finishing operation (if it has been programmed).

   If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

   The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

   The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed (δ, δx), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration pases as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.
7. Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
8. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

Considerations.

Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

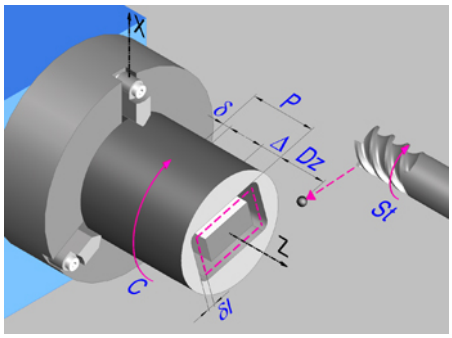
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

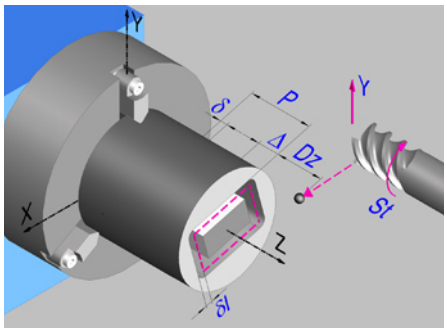
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

11.6 XC/XY plane profiling cycle.



Profile in the XC plane.



Profile in the XY plane.

Geometric parameters.

Work plane.

XA plane.

YB plane.

ZC plane.

YZ plane.

Name of the program with the profile.

P Name of the program with the profile (between 0 and 999).

Coordinates of the starting point.

Xi, Yi, Zi Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

Xf, Yf, Zf Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Coordinates of the starting point

Z Z coordinate of the penetration point into the part.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

Total depth.

P Total depth.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point. The withdrawal movement between successive roughing passes is done at 1 mm of the surface.

Dz X safety distance referred to the starting point.

FAGOR
FAGOR AUTOMATION
CNCelite
8058 8060
8065
REF. 2109

Spindle angular position.

ω Angular position of the spindle, in degrees.

Define a profile.

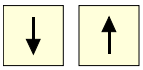
This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

Profile definition.

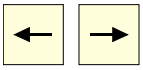
P Name of the program with the profile (between 0 and 999).

- Edit a new program with the profile. To edit a new program,write the program number (between 0 and 999) and press [RECALL]. The CNC shows the profile editor.
- To select a known program, key in the program number and press [ENTER].
- To select a program from the list, use the following keys.

Expand the program list.



Moves the cursor through the list of programs. To select a program, position the cursor on the desired program and press [ENTER].



Exit the list without selecting any program.

Modify an existing profile.

To edit an existing profile program, key in the program number and press [RECALL]. The profile editor of the CNC will show the selected profile, from where the following operations may be carried out.

- Add, delete or modify elements of the profile.
- Add, delete or modify chamfers, roundings, etc.
- Add new profiles to the program.

Spindle parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Tool radius compensation.

Without compensation

Left-hand tool radius compensation.

Right-hand tool radius compensation.

Penetration feedrate.

Fz Penetration feedrate.

Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the roughing operation.

Do not run the roughing operation.

Maximum penetration step.

Δ Maximum penetration step.

- If Δ is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If Δ is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the finishing operation.

Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

Number of penetration passes (steps) for the finishing operation.

N Number of penetration passes (steps) for the finishing operation. This parameter is only available when tool radius compensation is active.

Finishing stocks.

δl Side finishing stock. This parameter is only available when tool radius compensation is active.

δ Bottom finishing stock.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

11.6.1 Basic operation.

The machining steps in these cycles are the following:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
2. The live tool starts with the speed selected and in the indicated direction.
3. The tool approaches in rapid feedrate up to the safety point, at a distance of Dz of the starting point.
4. The spindle orients to the indicated "C" position.
5. Roughing operation (if it has been programmed).

   The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This cycle applies withdrawal between passes at 1 mm from the surface. This operation is done with the conditions set for the roughing operation.
6. Finishing operation (if it has been programmed).

   If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

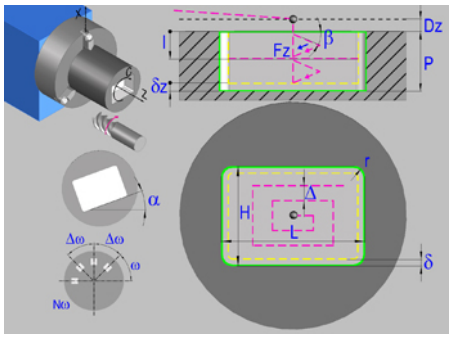
   The cycle eliminates the finishing stocks. If tool radius compensation is active, the cycle runs the indicated number of passes until the final profile is reached; if it is not active, the cycle runs a single pass. This operation is done with the conditions set for the finishing operation.
7. Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
8. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

Considerations.

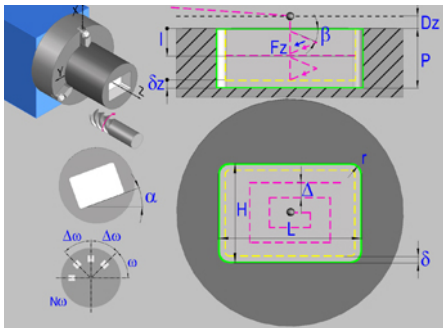
If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

11.7 Rectangular pocket cycle in the XC/XY plane.



Rectangular pocket in the XC plane.



Rectangular pocket in the XY plane.

Geometric parameters.

Work plane.

XA plane.

YB plane.

ZC plane.

YZ plane.

Starting point selection.

Starting point at the corner.

Starting point in the center.

Type of pocket corners.

Square (sharp) corner.

Rounded corner. The rounding radius must be defined.

Chamfered corner. The size of the chamfer must be defined.

r Rounding radius of the pocket corners.

c Size of the chamfer of the pocket corners.

Coordinates of the starting point.

X, Y, Z
A, B, Z

Coordinates of the starting point. The CNC will show the associated coordinates for the selected plane. The coordinate of the face axis must always be programmed in radii.

Coordinates of the starting point.

Xi, Yi, Zi Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

Xf, Yf, Zf Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Pocket dimensions.

L Pocket length on the X axis.

H Length of the pocket on the C axis (if XC plane) or on the Y axis (if XY plane).

Pocket orientation.

α Pocket angle referred to X axis, in degrees.

Coordinates of the plane.

Z Z coordinate of the work plane.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

Total depth.

P Total depth.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dz X safety distance referred to the starting point.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.

Spindle parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Penetration feedrate.

Fz Penetration feedrate.

Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the roughing operation.

Do not run the roughing operation.

Maximum penetration step.

I Maximum penetration step.

- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

Sideways penetration angle.

β Sideways penetration angle.

Milling pass or width.

Δ Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction.

Clockwise.

Counterclockwise.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the finishing operation.

Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

Sideways penetration angle.

θ Sideways penetration angle.

Finishing stocks.

δ Side finishing stock.

δx Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction.

Clockwise.

Counterclockwise.

Number of side penetration passes (steps) for the finishing operation.

N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

11.7.1 Basic operation.

The machining steps in these cycles are the following:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
2. The live tool starts with the speed selected and in the indicated direction.
3. The tool approaches in rapid feedrate up to the safety point, at a distance of Dz of the starting point.
4. The spindle orients to the indicated "C" position.
5. Roughing operation (if it has been programmed).

   The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.
6. Finishing operation (if it has been programmed).

   If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

   The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

   The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed (δ, δx), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration pases as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.
7. Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
8. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

Considerations.

Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

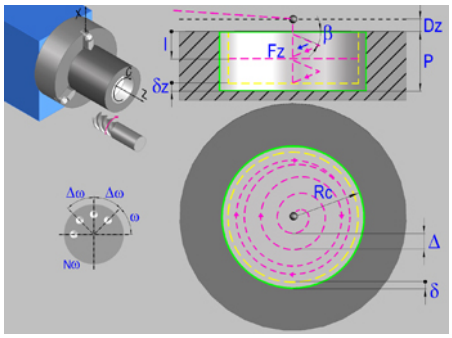
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

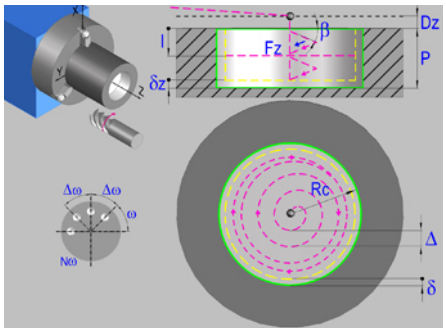
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

11.8 Circular pocket cycle in the XC/XY plane.



Circular pocket in the XC plane.



Circular pocket in the XY plane.

Geometric parameters.

Work plane.

XA plane.

YB plane.

ZC plane.

YZ plane.

Coordinates of the center of the pocket.

Xc, Yc, Zc
Ac, Bc, Zc — Coordinates of the center of the pocket. The CNC will show the associated coordinates for the selected plane. The coordinate of the face axis must always be programmed in radii.

Coordinates of the starting point.

Xi, Yi, Zi — Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

Xf, Yf, Zf — Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Pocket radius.

Rc — Pocket radius.

Coordinates of the plane.

Z — Z coordinate of the work plane.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

Total depth.

P — Total depth.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dz X safety distance referred to the starting point.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.

Spindle parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Penetration feedrate.

Fz Penetration feedrate.

Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the roughing operation.

Do not run the roughing operation.

Maximum penetration step.

I Maximum penetration step.

- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

Sideways penetration angle.

β Sideways penetration angle.

Milling pass or width.

Δ Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction.

Clockwise.

Counterclockwise.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the finishing operation.

Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

Sideways penetration angle.

θ Sideways penetration angle.

Finishing stocks.

δ Side finishing stock.

δx Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction.

Clockwise.

Counterclockwise.

Number of side penetration passes (steps) for the finishing operation.

N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

11.8.1 Basic operation.

The machining steps in these cycles are the following:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
2. The live tool starts with the speed selected and in the indicated direction.
3. The tool approaches in rapid up to the starting point, keeping the selected safety distance according to the Z axis.
4. The spindle orients to the indicated "C" position.
5. Roughing operation (if it has been programmed).

   The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.
6. Finishing operation (if it has been programmed).

   If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

   The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

   The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed (δ, δx), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration pases as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.
7. Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
8. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

Considerations.

Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

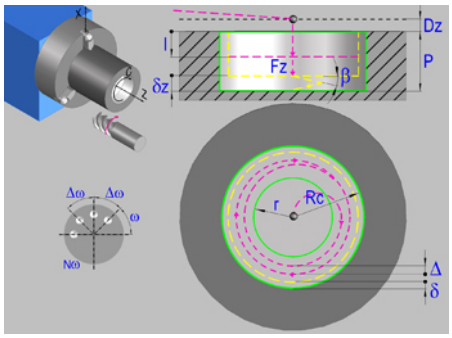
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

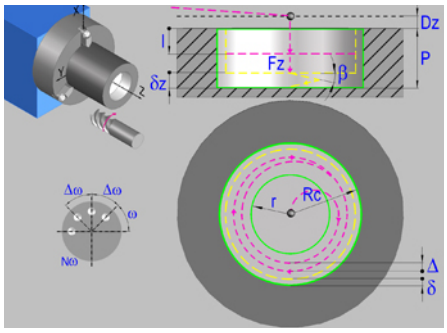
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

11.9 Circular pre-emptied pocket in the XC/XY plane.



Circular pre-emptied pocket in the XC plane.



Circular pre-emptied pocket in the XY plane.

Geometric parameters.

Work plane.

XA plane.

YB plane.

ZC plane.

YZ plane.

Coordinates of the center of the pocket.

Xc, Yc, Zc
Ac, Bc, Zc — Coordinates of the center of the pocket. The CNC will show the associated coordinates for the selected plane. The coordinate of the face axis must always be programmed in radii.

Coordinates of the starting point.

Xi, Yi, Zi — Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

Xf, Yf, Zf — Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Pocket radius.

Rc — Pocket radius.

r — Inside radius of the pocket.

Coordinates of the plane.

Z — Z coordinate of the work plane.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

Total depth.

P — Total depth.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dz X safety distance referred to the starting point.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.

Spindle parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Penetration feedrate.

Fz Penetration feedrate.

Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the roughing operation.

Do not run the roughing operation.

Maximum penetration step.

I Maximum penetration step.

- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

Sideways penetration angle.

β Sideways penetration angle.

If the tool fits in the pre-emptied area, it goes in straight in G0, ignoring the penetration angle. If the tool does not fit in the pre-emptied area, it goes in at the programmed penetration angle.

Milling pass or width.

Δ Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction.

Clockwise.

Counterclockwise.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the finishing operation.

Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

Sideways penetration angle.

θ Sideways penetration angle.

Finishing stocks.

δ Side finishing stock.

δx Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction.

Clockwise.

Counterclockwise.

Number of side penetration passes (steps) for the finishing operation.

N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

11.9.1 Basic operation.

The machining steps in these cycles are the following:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
2. The live tool starts with the speed selected and in the indicated direction.
3. The tool approaches in rapid feedrate up to the safety point, at a distance of Dz of the starting point.
4. The spindle orients to the indicated "C" position.
5. Roughing operation (if it has been programmed).

   The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.
6. Finishing operation (if it has been programmed).

   If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

   The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

   The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed (δ, δx), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration pases as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.
7. Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
8. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

Considerations.

Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

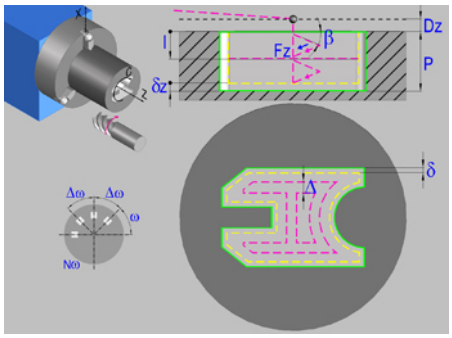
Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

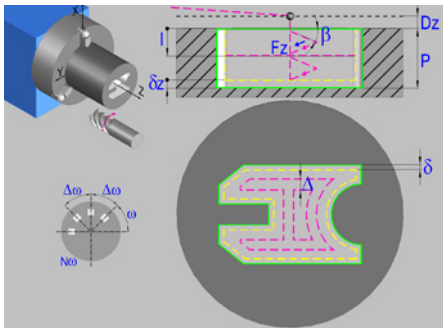
If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

11.10 2D profile pocket cycle in the XC/XY plane.



2D profile pocket in the XC plane.



2D profile pocket in the XY plane.

Geometric parameters.

Work plane.

XA plane.

YB plane.

ZC plane.

YZ plane.

Name of the program with the profile.

P Name of the program with the profile (between 0 and 999).

Coordinates of the starting point.

Xi, Yi, Zi Coordinate of the initial positioning; if not programmed, the axis maintains its position. For cycles where this axis is not involved, this parameter enables it to be positioned before executing the cycle; for example, after a tool change.

Coordinates of the end point.

Xf, Yf, Zf Coordinate to withdrawal the tool after the cycle has completed; if not programmed, there is withdrawal. If the cycle is machining in multiple mode, the tool will only be withdrawn after the last cycle.

Coordinates of the plane.

Z Z coordinate of the work plane.

To define the coordinates, edit the value manually or assign the current position of the machine using the Teach-in mode.

Total depth.

P Total depth.

Safety distance.

In order to prevent collisions with the part, the CNC allows a part approach point to be set. The safety distance indicates the position of the approach point referred to the starting point.

Dz X safety distance referred to the starting point.

Angular position of the machining operations.

ω Angular position of the spindle (in degrees) for the first machining operation.

Δω Angular step, in degrees, between machining operations. A positive step value means counterclockwise and a negative one clockwise.

Nω Total number of operations.

Define a profile.

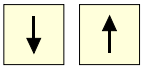
This icon lets define the profile using the profile editor; it may be an existing profile or a new one.

Profile definition.

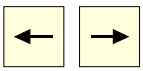
P Name of the program with the profile (between 0 and 999).

- Edit a new program with the profile. To edit a new program,write the program number (between 0 and 999) and press [RECALL]. The CNC shows the profile editor.
- To select a known program, key in the program number and press [ENTER].
- To select a program from the list, use the following keys.

Expand the program list.



Moves the cursor through the list of programs. To select a program, position the cursor on the desired program and press [ENTER].



Exit the list without selecting any program.

Modify an existing profile.

To edit an existing profile program, key in the program number and press [RECALL]. The profile editor of the CNC will show the selected profile, from where the following operations may be carried out.

- Add, delete or modify elements of the profile.
- Add, delete or modify chamfers, roundings, etc.
- Add new profiles to the program.

Spindle parameters.

Live tool turning direction.

Live tool clockwise.

Live tool counter-clockwise.

Coolant status.

Coolant ON. The CNC outputs the M8 function to the PLC.

Coolant OFF. The CNC outputs the M9 function to the PLC.

Once the operation or the cycle is completed or the part-program it belongs to, the CNC outputs the M9 function to the PLC.

Penetration feedrate.

Fz Penetration feedrate.

Machining strategy (icon).

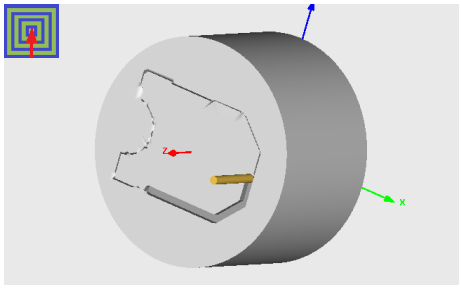
Outward-inward concentric machining. The wall of the pocket is machined in the first pass. Depending on the geometry, this option may be faster, but the wear on the tool is greater.

- The starting point is near the initial point of the outer contour, and varies very little if the radius of the roughing tool is changed and does not vary at all if the roughing pass is changed.
- If a drilling operation is programmed before the roughing (or the bottom finishing if there is no roughing operation), the CNC chooses the optimal drilling point, as close as possible to the initial point of the outer contour.

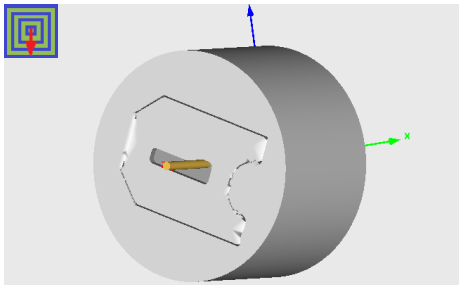
Inward-outward concentric machining. The wall of the pocket is machined in the last pass. Depending on the geometry, this option maybe slower, but there is less wear on the tool. The CNC machines the pocket, attempting to minimize the number of sections where the tool machines over the largest surface, the number of penetrations in part sections and the machining time.

- The starting point does not depend on the initial point of the outer contour and can vary if either the tool radius or the machining pass is changed (milling width). In a pocket without islands, the initial point is found near the center of the pocket. In a pocket with islands, where different areas can be formed that require roughing, the initial point will be located in the center of the largest area.
- If a drilling operation is programmed before the roughing (or the bottom finishing, if there is no roughing operation), the CNC chooses the optimal drilling points (there may be more than one point).

2D profile pocket in the XC/XY plane.



2D profile pocket in the XC/XY plane.



Machining parameters. Roughing.

To run the roughing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the roughing operation.

Do not run the roughing operation.

Maximum penetration step.

I Maximum penetration step.

- If the parameter is positive, the cycle recalculates the step so all the penetrations are identical with the same value as or smaller than the one programmed.
- If the parameter is negative, the cycle runs the passes with the programmed value, except the last one that machines the rest.

Sideways penetration angle.

β Sideways penetration angle.

Milling pass or width.

Δ Milling pass or width.

The cycle recalculates the pass so that all the passes are identical, with the same value as or smaller than the one programmed. If programmed with a 0 value, it assumes a value of 3/4 of the diameter of the selected tool.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction (icon).

For pockets with islands, the machining of the outer profile is done in the direction indicated by the icon, where as the islands are machined in the opposite direction.

Clockwise.

Counterclockwise.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the roughing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the roughing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the roughing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

Machining parameters. Finishing.

To run the finishing operation of the cycle, it must be activated in the corresponding verification checkbox.

Run the finishing operation.

Do not run the finishing operation. If the finishing operation is deactivated and you don't want to leave finishing stocks in the roughing operation, enter "0" in the corresponding boxes (δ, δl) (δ, δl).

Sideways penetration angle.

θ Sideways penetration angle.

Finishing stocks.

δ Side finishing stock.

δx Bottom finishing stock. If it is set to 0, the cycle does not execute the finishing operation at the bottom.

Machining feedrate.

F Machining feedrate.

Live-tool data.

Live tool data not programmed. The cycle hides the data related to the live tool and it ignores them. The cycle uses the second spindle of the channel as live tool.

Live tool data programmed. The cycle shows the data related to the live tool.

S Name of the spindle of the live tool, turning speed and gear.

To define the name, place the cursor on the "S" letter and enter the number of the spindle associated with the live tool; 1 for S1, 2 for S2 and so on.

If the spindle gear is not programmed, the cycles assumes 0. If the gear is defined with a value of 0, the cycle uses the range (gear) that corresponds to the programmed speed.

Machining direction (icon).

For pockets with islands, the machining of the outer profile is done in the direction indicated by the icon, where as the islands are machined in the opposite direction.

Clockwise.

Counterclockwise.

Number of side penetration passes (steps) for the finishing operation.

N Number of side penetration passes (steps) for the finishing operation. If defined with a value other than 0, the cycle executes N finishing passes, even if no side finishing stock has been programmed. If it is set to 0, the cycle does not execute the side finishing operation.

Tool data.

T Tool. If it is set to 0, the cycle does not execute the finishing operation.

D Tool offset; if not programmed, the cycle will assume the 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.

Machining M functions.

Activate the execution of M functions before the finishing operation. The cycle allows editing up to 4 M functions. To execute only one of them, define it first and leave the rest of the data unprogrammed.

Deactivate the execution of M functions before the finishing operation.

The editor will only show this option if the user has configured the editor to allow programming M functions.

11.10.1 Basic operation.

The machining steps in these cycles are the following:

1. If the roughing operation was programmed with another tool the CNC makes a tool change, moving to the change point if the machine so requires.
2. The live tool starts with the speed selected and in the indicated direction.
3. The tool approaches in rapid feedrate up to the safety point, at a distance of Dz of the starting point.
4. The spindle orients to the indicated "C" position.
5. Roughing operation (if it has been programmed).

   The cycle runs successive passes, up to a distance from the final profile equal to the finishing stock. This operation is done with the conditions set for the roughing operation.
6. Finishing operation (if it has been programmed).

   If the finishing operation was programmed with another tool, the CNC will make a tool change, moving to the change point if the machine so requires.

   The cycle eliminates the finishing stocks. This operation is done with the conditions set for the finishing operation.

   The cycle offers two finishing stocks; at the bottom and on the walls. If both stocks are programmed (δ, δx), the cycle executes the finishing operation at the bottom first and then on the walls. The finishing on the walls is carried out in as many penetration pases as indicated by parameter N. If there are no side finishing stocks, but parameter N is other than 0, the cycle executes N finishing passes.
7. Once the operation or cycle has ended, the tool will return to the safety point (starting point plus the safety distance). When executing a complete part, a combination of operations or cycles, the tool does not return to that point after executing each cycle.
8. The CNC keeps the machining conditions set for finishing selected; tool (T), axis feedrate (F) and tool speed.

Considerations.

Define a tool as T0.

If T0 is selected as roughing tool, the cycle does not run the roughing operation; after the approach movement, the cycle runs the finishing operation.

If T0 is selected as finishing tool, the cycle does not run the finishing operation; after the roughing operation, the tool will move to the safety point (starting point plus the safety distance).

Roughing and finishing in multiple pockets.

If the roughing tool and the finishing tool are different, the cycle executes first all the roughing operations and then all the finishing operations.

If the cycle uses the same tool for roughing and finishing, the order of the operations depends on the M functions assigned to each operation.

- If no M function has been defined, or they are the same for both operations, the cycle executes the whole pocket (roughing and finishing) before going on to the next one.
- If the M functions assigned to each operation are different, the cycle executes first all the roughing operations and then all the finishing operations.

11.

CANNED CYCLES OF THE EDITOR FOR ROTARY AXES.

2D profile pocket cycle in the XC/XY plane.

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

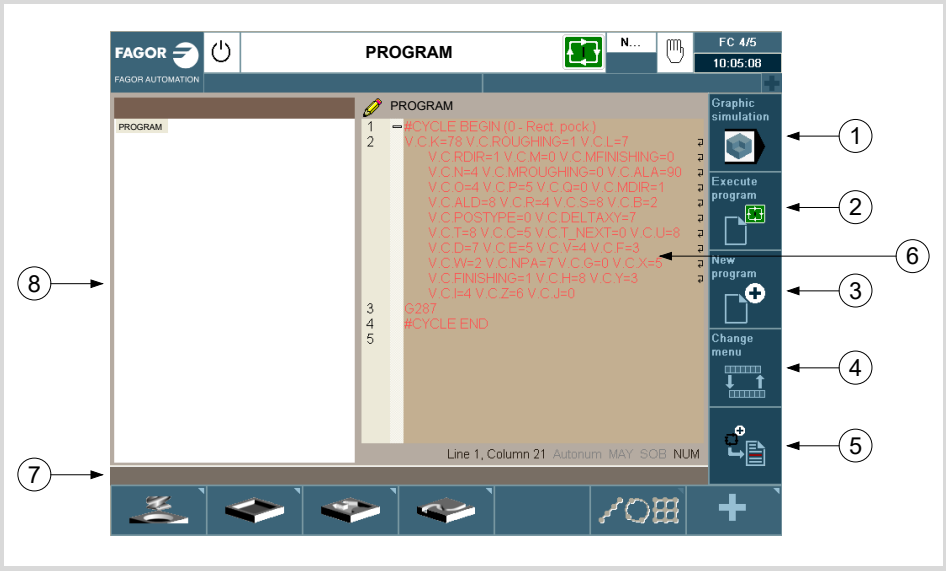
SAVING PROGRAMS

12

Part-programs may be edited, saved, simulated and executed. Each one of these programs is made up by concatenating simple operations or single cycles.

12.1 List of saved programs

Press [EDIT] to access the list of saved part-programs. The CNC will display the following screen:



1 Softkey to start simulating the selected program.

2 Softkey to start executing the selected program.

3 Softkey to create a new program.

4 Softkey that changes the horizontal softkey menu. The sodtkeys appearing when pressing this softkey are:

- Open program.
- Operations with blocks.
- Undo/Redo.
- Insert the cycle being edited.

5 Replace or insert the edited cycle.

6 Window that shows the cycles and ISO-coded blocks that make up the selected part. To edit a cycle, put the cursor on the cycle press [RECALL].

7 Message bar.

8 This softkey shows the list of saved programs. In in this window, it is possible to move through the list of programs. When selecting a program, the right window will show its contents. To change the working folder, press [RECALL] while the focus is on the list and a folder explorer will open to choose the new folder.

To toggle the focus between the list of programs and the part editor, press [CTRL][F2].

12.2 Edit a new part-program

To edit a new program, proceed as follows:

1. Press [EDIT] to access the list of saved part-programs.
2. Press the vertical softkey "New program". The following options will appear:
   - New: Entering a name in the dialog box and pressing the New button will create a new empty file with that name.
   - Copy: Entering a name in the dialog box and pressing the Copy button will create a file with that name and all the cycles and ISO-coded blocks of the program selected at the time on the list of programs will be copied into it.
   - Cancel: To cancel the creation of a new program.

Keyboard shortcuts [CTRL][C] and [CTRL][V] may be used on the programs list to copy and paste a program.

12.3 Delete a new part program

To delete a part-program, proceed as follows:

1. Press [EDIT] to access the list of saved part-programs.
2. Select, with the pointer, the part-program to be deleted from the left column. A program cannot be deleted if it is being executed or simulated.
3. Press the [DEL] key.

The bottom of the screen will show a message requesting confirmation of the deleting operation.

- If [ENTER] is pressed, the CNC will delete the selected program and will update the list of saved programs.
- If [ESC] is pressed, the program will not be deleted and it will quit deleting operation.

12.4 Inserting comments in any part-program.

The block or cycle may be added at the end of the program, after the last operation or insert it between 2 existing operations or replace it with one of the cycles of the program. To save the cycle, proceed as follows:

1. Define the desired cycle assigning the relevant data to it.
2. Press the softkey "Save cycle" to access the list of part-programs.
3. Select the desired program in the left window. The right window shows the contents of the part-program.
4. In the right window, place the cursor at the position where the edited cycle is to be inserted or replaced.
5. Press the vertical softkey to replace or insert the edited cycle and select the desired option.

12.5 Inserting a cycle in the current part-program.

The editor lets insert the cycle being edited directly into the current program in the cursor position.

1. Define the desired cycle assigning the relevant data to it.
2. Press the [INS] key to insert the cycle into the current program in the cursor position.

EXECUTION AND SIMULATION

13

The simulation may be used to graphically show a part-program or an operation with the data used to define it. This way, the simulation may be used to check the part-program or the operation before executing it or saving it and, therefore, correct or modify its data.

It is possible to execute or simulate a part-program or any operation. The simulation or execution may be done from beginning to end or step by step by pressing the [SINGLE] key.

13.1 Execute a part-program

Proceed as follows to execute a part-program:

1. Press [EDIT] to access the list of part-programs stored.
2. Select on the left column the program to be executed.
3. Press the vertical softkey "Execute program".

   After pressing this softkey, the CNC shows the standard screen for the MC work mode. When pressing the two-color key, the CNC shows the auxiliary screen for the MC mode. When pressing the vertical softkey "Graphics", the CNC shows the graphics screen in execution.
4. Once the desired screen has been selected, press [START].

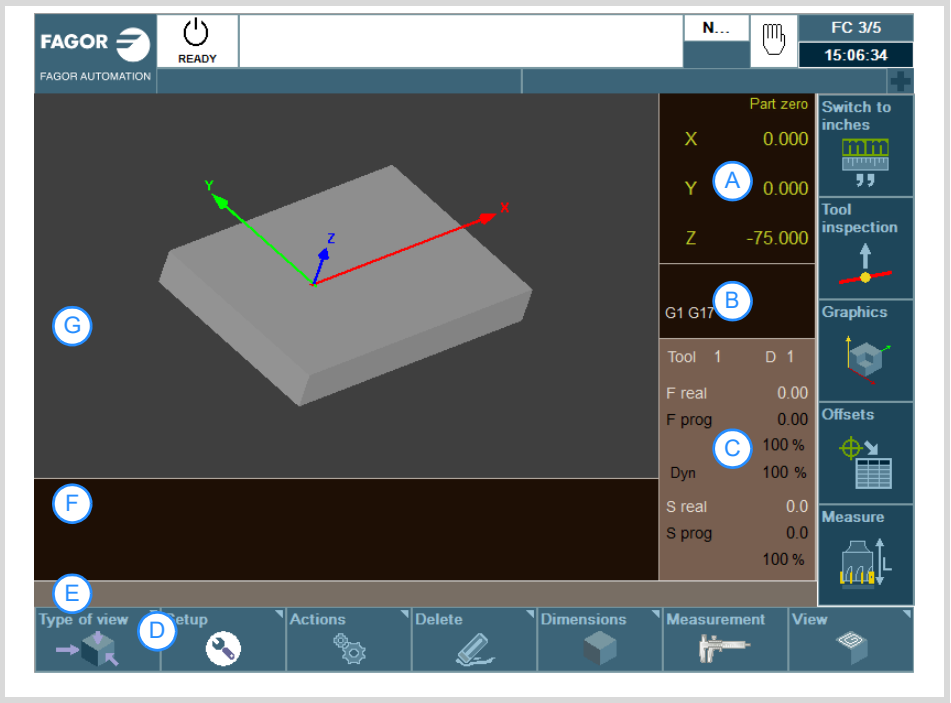
13.1.1 Execute a portion of a part-program

Proceed as follows to execute a portion of a part-program:

1. Press [EDIT] to access the list of part-programs stored.
2. Select the program from the left column and the operation (on the right column) from which to begin executing the part program.
3. Press the vertical softkey "Execute program".

   After pressing this softkey, the CNC shows the standard screen for the MC work mode. When pressing the two-color key, the CNC shows the auxiliary screen for the MC mode. When pressing the vertical softkey "Graphics", the CNC shows the graphics screen in execution.
4. Once the desired screen has been selected, press [START].

13.1.2 Graphics screen in execution.



A Window that informs about the position of the axes during the execution of the program.

B Window that shows the status of the G functions.

C Window that shows the selected tool T, the tool offset D, the real F and the programmed F as well as the real S and the programmed S.

D Softkeys with graphic display options.

- Type of view.
- Configuration.
- Actions.
- Delete.
- Dimensions.
- Measurement.

E Message bar.

F Window that shows the lines of the program being executed.

G Window that shows the lines of the program being executed through graphics.

13.2 Simulating a part-program

Proceed as follows to simulate a part-program:

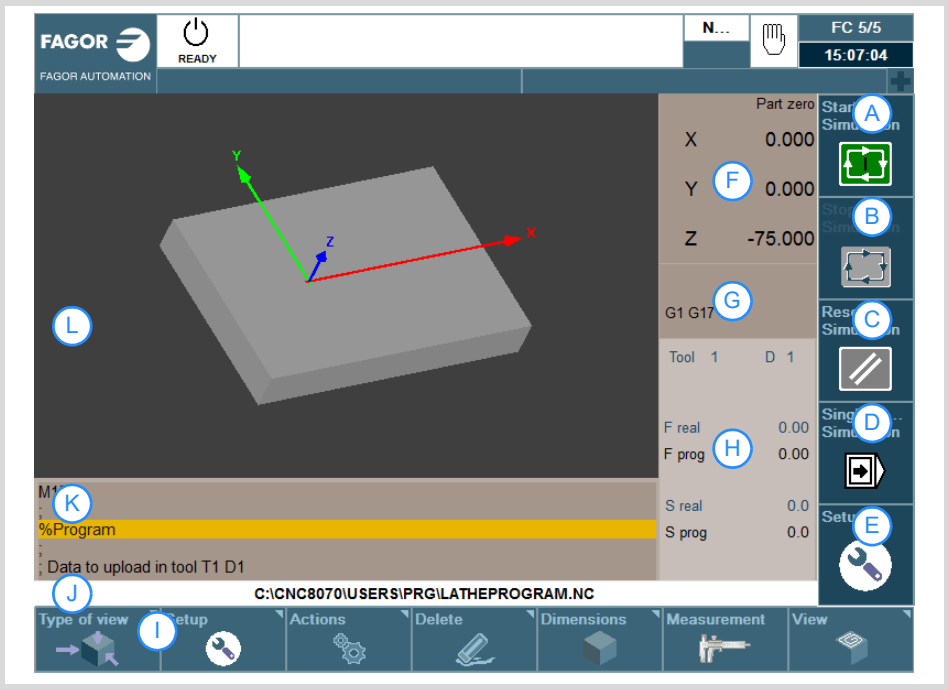
1. Press [EDIT] to access the list of part-programs stored.
2. Select on the left column the program to be simulated.
3. Press the vertical softkey "Graphic simulation". After pressing this softkey, the CNC shows the graphics screen in simulation.
4. Press the vertical softkey "Start simulation".

13.2.1 Simulate a portion of a part-program

Proceed as follows to simulate a portion of a part-program:

1. Press [EDIT] to access the list of part-programs stored.
2. Select the program from the left column and the operation (on the right column) from which to begin executing the part program.
3. Press the vertical softkey "Graphic simulation". After pressing this softkey, the CNC shows the graphics screen in simulation.
4. Press the vertical softkey "Start simulation".

13.2.2 Graphics screen in simulation.



Description of the page items:

- A Softkey to start simulating the selected program.
- B Softkey to stop simulating the selected program.
- C Softkey to reset the execution of the selected program.
- D Softkey to simulate the program block by block.
- E Simulation options.
- F Window that informs about the position of the axes during the execution of the program.
- G Window that shows the status of the G functions.
- H Window that shows the selected tool T, the tool offset D, the real F and the programmed F as well as the real S and the programmed S.
- I Softkeys with graphic display options.
- J Message bar.
- K Window that shows the lines of the program being executed.
- L Window that shows the lines of the program being executed through graphics.

13.2.3 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.

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).

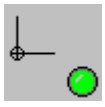
Cancel channel synchronization

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.

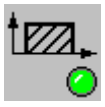
Synchronizing spindles.

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.

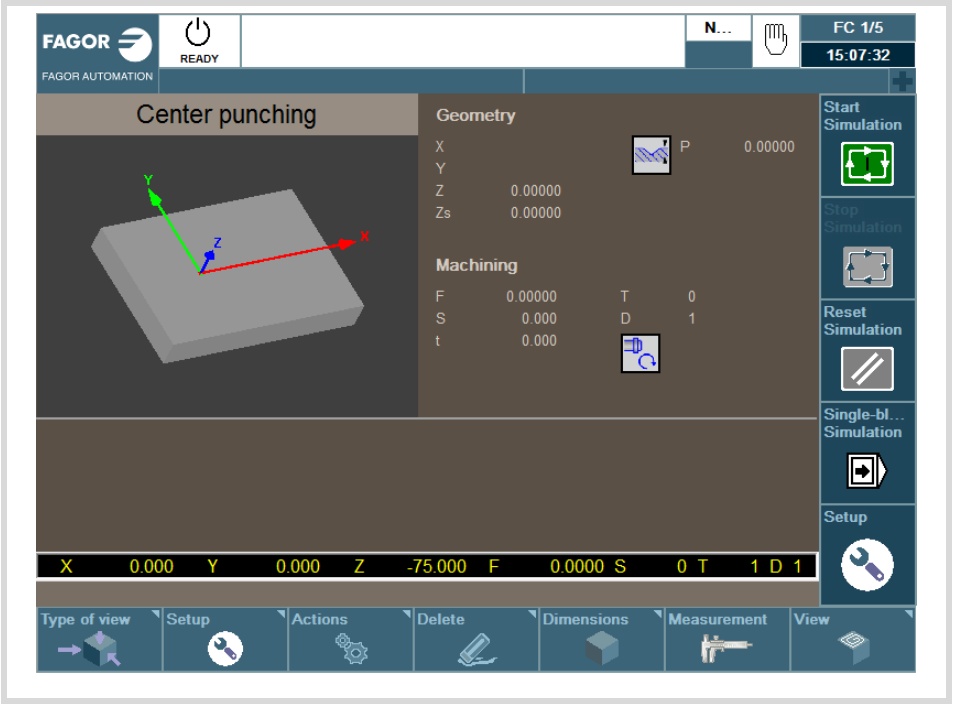
13.3 Simulating or executing an operation that has been saved

Proceed as follows to simulate or execute an operation that has been saved as a part of program:

1. Press [EDIT] to access the list of part-programs stored.
2. Select the program that contains it from the left column and the operation to be simulated or executed from the right column.
3. Press the [RECALL] key.
4. Press the vertical softkey "Simulate cycle" to simulate the operation and the vertical softkey "Execute cycle" to execute it.

13.3.1 Simulating a cycle

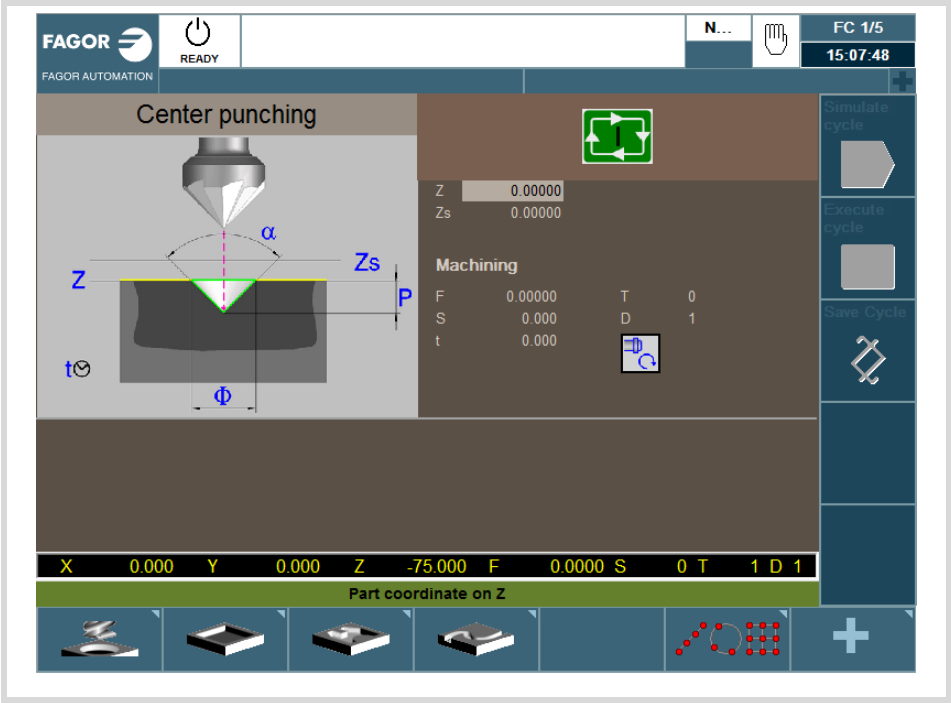
To simulate the edited cycle, press the vertical softkey "Simulate cycle".



For further information on simulating cycles, see the chapter *"2.1.4 Cycle simulation"*.

13.3.2 Cycle execution

To execute the edited cycle, press the vertical softkey "Execute cycle".



For further information on executing cycles, see the chapter *"2.1.5 Cycle execution"*.

User notes:

User notes:

FAGOR

FAGOR AUTOMATION

CNCelite
8058 8060
8065

REF. 2109

User notes:

FAGOR

FAGOR AUTOMATION

Fagor Automation S. Coop.

Bº San Andrés, 19 - Apdo. 144
E-20500 Arrasate-Mondragón, Spain
Tel: +34 943 039 800
Fax: +34 943 791 712
E-mail: info@fagorautomation.es
www.fagorautomation.com

AENOR
ER
Empresa Registrada
ER-0731/1994

CERTIFIED
IQNet
MANAGEMENT SYSTEM